KOLLOQUIUM ÜBER NEUERE ARBEITEN AUF DEM GEBIETE
DER MECHANIK UND STRÖMUNGSLEHRE
an der Technischen Universität Wien

EINLADUNG

zum Vortrag von Herrn

Ass.Prof. DI. Dr. Željko Tuković

University of Zagreb
Faculty of mechanical engineering and naval architecture

über

"OpenFoam implementation of moving mesh interface tracking method for simulating surface tension dominated interfacial fluid flow"

Zeit: Donnerstag, 3. März 2011, 16 Uhr c.t.

Ort: SEM 322
Institut für Strömungsmechanik und Wärmeübertragung
Resselg. 3, Stiege 2, 1. Stock, 1040 Wien

“OpenFOAM implementation of moving mesh interface tracking method for simulating surface tension dominated interfacial fluid flow”

Dr. Željko Tuković

Development of computational methods for numerical modeling of free-surface and interfacial fluid flow is a very active research area. The oldest and still the most popular methods use a fixed mesh to solve single set of governing equations for the whole fluid flow with different approaches to locate the interface between the phases. These methods are particularly attractive due to simply handling of large deformation and changing topology of the interface. The main difficulty in using these methods is the maintenance of a sharp interface between the different fluid phases and the computation of the surface tension forces.

The moving mesh methods, also known as moving mesh interface tracking methods use separate boundary-fitted moving mesh for each phase. These methods offer potentially the highest accuracy since the interface between phases is represented by the computational mesh boundary allowing very accurate calculation of surface tension force and direct implementation of the kinematic and dynamic conditions at the interface without any smoothing of the fluid phases properties. Although limited in their application, moving mesh interface tracking methods have important role in the numerical analyses of fundamental multiphase flows such as motion of isolated bubbles and droplets, where strong dependence between interface shape and near surface flow exists due to strong surface tension forces. Development of high accuracy moving mesh methods can also be justified by the fact that ” numerically exact” solutions generated by these methods can be used for validation of fixed mesh methods.

In this lecture I will present a moving mesh interface tracking method implemented in OpenFOAM framework for simulating three-dimensional, incompressible, and immiscible two-phase interfacial fluid flows with dominant surface tension forces. Collocated finite volume method is used for spatial discretization of Navier-Stokes equations on moving arbitrary polyhedral mesh. Adjustment of the mesh to the time-varying shape of the interface is done using vertex-based automatic mesh motion solver which calculates internal points motion based on the prescribed motion of interface points by solving variable diffusivity Laplace equation discretized using finite element method. Mesh consists of two partitions separated at the interface. Fluid flow is solved on each partition separately and coupling is accomplished in iterative manner by enforcing kinematic and dynamic conditions at the interface. Implementation of the dynamic condition also includes effects of variable surface tension caused by non-homogeneous distribution of surfactant along the interface. In order to allow numerical analysis of surfactant effects, a Finite Area Method is developed for the discretization of surface transport equation on the moving unstructured surface mesh. The overall solution procedure based on iterative PISO algorithm with modified Rhie-Chow interpolation has second-order accuracy in both space and time what is proven on several test cases.

In multiphase flows a multitude of interesting and sometimes unexpected flow phenomena can be encountered. Even in times of comfortable general purpose CFD software and ever increasing computer resources simulating these phenomena provides major challenges. Most multiphase flow situations are too complex for straight forward numerical modelling. Rather multiphase flow simulations should always be done with great care and most favourably in combination with dedicated experiments.

In this presentation some examples of multiphase flow modelling in the field of metallurgical flows, environmental flows and particulate flows are presented. Thereby, successful modelling approaches are presented as well as failures. All of these application driven research activities are further linked to their sometimes surprising multiphase flow core phenomenon.