Title: 3D Moving Mesh Technique for Microscale Two-Phase Flows

Authors: Gustavo Rabello dos Anjos   gustavo.rabello@epfl.ch
             Navid Borhani   navid.borhani@epfl.ch
             John R. Thome   john.thome@epfl.ch

Numerical simulation is employed to simulate two-phase flow phenomena using the continuum method for surface tension modeling [1]. The pure Lagrangian approach moves the mesh points with respect to the flow field, thus the particle's positions in the next time step are precisely calculated. In interface flows, the moving mesh approach leads to a highly deformable grid, which quickly adversely influences the quality of the computational elements. On the other hand, besides the fact that a fixed grid formulation allows an easier treatment of complex interface motion, the pure Eulerian description presents numerical difficulties due to the nonsymmetric character of the convection operator. To overcome these limitations, a generalized description capable of combining the best aspects of both classical descriptions is employed, namely the Arbitrary Lagrangian-Eulerian (ALE) framework [2]. The set of equations are based on the Navier-Stokes formulation but with the relative mesh velocity included in the convective term. By varying a single parameter, the formulation can be set to a fixed or a complete moving mesh technique. Another important issue in two-phase flow modeling is the definition of the interface between the fluids. In the classical Eulerian formulation, the interface is described implicitly by an equation which is convected by the fluid flow. Due to the discretization of such an equation, numerical diffusion may lead to low accuracies of the calculated surface tension term. In this work, the interface between fluids is described explicitly in a Lagrangian way by points and computational elements, thus a sharp and precise representation is successfully achieved. This geometrical procedure also ensures undesirable modes and spurious oscillations are damped out, thus leading to the convergence of the results. A Laplacian smoothing operator is applied to the whole grid and to the surface mesh to keep the points homogeneously distributed, thereby avoiding large concentrations of points in one specific region. Unfortunately, only using the Laplacian operator is not enough to preserve the mesh quality, thus other common geometric procedures also need to be applied [3]. The new methodology proposed here to simulate two-phase flows provides good accuracy to describe the interfacial forces and bubble dynamics. The employed formulation, the interface representation and the results will be presented. Focus will be on the treatment of the surface elements and how to preserve the mesh quality and the element aspect ratios. Furthermore, a 3D microscale simulation of a bubble flow will be presented.

[1] J.U.Brackbill, D.B. Kothe and C. Zemach, A Continuum Method for Modeling Surface Tension, Journal of Computational Physics, 100:335-354, 1992.

[2] J. Donea, A. Huerta, J.-Ph. Ponthot and A. Rodrìguez-Ferran, Arbitrary Lagrangian-Eulerian Methods, Encyclopedia of Computational Mechanics, Volume 1: Fudamentals, John Wiley and Sons, 2004.

[3] S. Negami, Diagonal flips in triangulation of surfaces, Discrete Mathematics, 135:225-232, 1994.