nano-tera.ch
swiss scientific initiative in health / security / environment systems

CMOSAIC RTD 2009

Moving Mesh Technique for Diabatic Two-Phase Flows

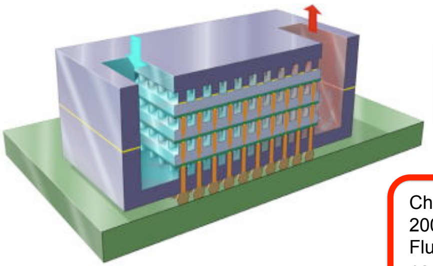
Gustavo Rabello dos Anjos, Navid Borhani, John Richard Thome

Heat and Mass Transfer Laboratory, École Polytechnique Fédérale de Lausanne, Switzerland

ÉCOLE POLYTECHNIQUE FÉDÉRALE DE LAUSANNE

1) Introduction:

The study of microscale two-phase flows is important in many areas of engineering. Since a full description of flow behavior in some situations is hard to predict experimentally, a numerical approach is often necessary to study particular cases. This work is part of the larger multi-disciplinary multi-laboratory CMOSAIC project which aims to study and design interlayer cooling system for the next generation of 3D stacked microprocessors, illustrated by:



Channel shapes: Rectangular, circular, square, trapezoidal.

Channel sizes: 50μm to 200μm.
Fluids: water, nanofluids, refrigerants

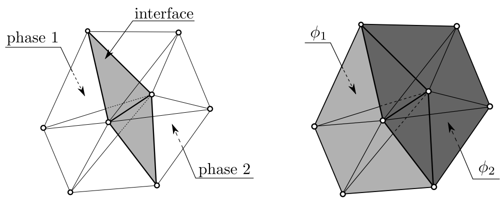
2) Objectives:

The goal of the present study is to:

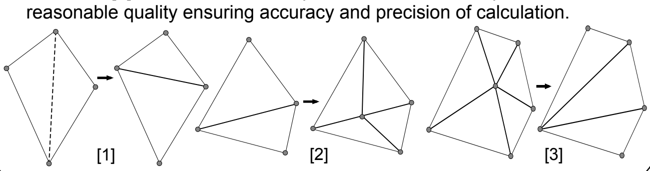
- Develop a 3D Arbitrary Lagrangian-Eulerian Finite Element code;
- Develop a platform for modeling two-phase flows;
- Predict flows in microscale geometries;
- Couple heat transfer and two-phase flow.

3) Surface Tension Force Model and Geometric Operations on the Interface:

Illustrated by the figure below, the Lagrangian approach differs from the standard approach by the addiction of points on the surface between the fluids. In fact, no artificial smoothing is required to deal with high properties ratio (Φ_1/Φ_2). This methodology leads to a sharp representation of interface and accurate results.



Due to insertion of points on the surface, it is mandatory to treat the mesh properly to preserve the aspect ratio of elements. Flipping operation is done where two elements don't present good shape, thus the edge is swapped as shown on [1]. Insertion of points are performed where the edge become larger then a referential edge length [2]. Deletion of point occurs where the concentration of elements is saturated [3]. All the illustrated operations tends to keep the mesh in a reasonable quality ensuring accuracy and precision of calculation.



4) Arbitrary Lagrangian-Eulerian Framework:

Shown in dimensionless vector form, the Navier-Stokes equations analytically represents the fluid flow, where $\boldsymbol{u}$ and $\boldsymbol{p}$ represent the velocity and pressure fields respectively, $\boldsymbol{\rho}$ and $\boldsymbol{\mu}$ stands for the density and viscosity of the phase Φ, N and Eo are dimensionless parameters to characterize the flow regime, t is time, $\boldsymbol{g}$ is gravity force and $\boldsymbol{f}$ is the surface tension that describes interfacial forces between two different fluids (colored red).

$$\frac{D(\rho \mathbf{u})}{Dt} + \nabla p = \frac{1}{N^{1/2}}\nabla\cdot[\mu(\nabla\mathbf{u}+\nabla\mathbf{u}^T)] + \underbrace{\rho\mathbf{g}}_{\text{gravity}} + \underbrace{\frac{1}{Eo}\mathbf{f}}_{\text{surface tension}}$$

$$\nabla\cdot\mathbf{u}=0$$

$$\frac{\partial\mathbf{u}}{\partial t} + (\mathbf{u}-\overbrace{\hat{\mathbf{u}}}^{\text{mesh velocity}})\cdot\nabla\mathbf{u}$$

$\hat{\mathbf{u}}=\mathbf{u} \rightarrow$ Lagrangian

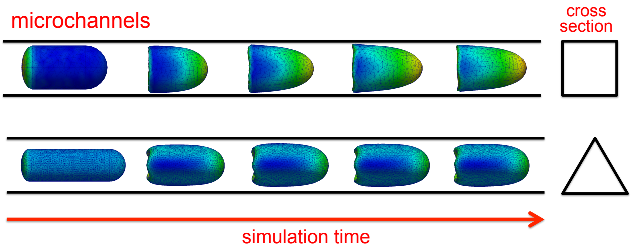
$\hat{\mathbf{u}}=0 \rightarrow$ Eulerian

5) 3-Dimensional Results:

The table presents the results of a 3D static droplet immersed in another fluid without the presence of external forces, i.e., gravity and velocity fields. This test is important to evaluate the pressure term coupled to the surface tension force. Pressure difference and calculation error, mass conservation and parasitic currents are measures of precision and accuracy of the model.

Surface Edge Length	Num. of elements e	Δp	Δp_{error}	Φ_{error}	Max{|u|,|v|,|w|}
h=0.10	205	4.03	0.09%	0.1%	1.2x10-5
h=0.088	220	4.03	0.09%	0.08%	1.1x10-5
h=0.06	312	4.01	0.02%	0.04%	4.0x10-6
h=0.04	408	4.01	0.02%	0.04%	3.7x10-6

The figures show 3D simulations of an important flow pattern. The confined bubble is found when the distance between walls (black line) is smaller than the bubble diameter. The simulations were carried out using the new implemented ALE Finite Element code using two different cross section channels. In the first example a bubble is flowing along the square channel and the second example an elongated bubble is inside a triangular channel. The simulations presented here show the high accuracy and flexibility of the developed code.



6) Conclusions:

The finite element method is a powerful and flexible way to discretize the numerical domain and to well represent the fluid dynamics. By applying the ALE technique to two-phase flows, we are able to take advantage of the best aspects of both reference frames (Lagrangian and Eulerian). The curvature calculation approach leads to faster accurate results, compared to classical distance function calculations. This showed that the methodology proposed to simulate two-phase flows provides good accuracy to describe the interfacial forces and bubble dynamics.