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

CMOSAIC RTD 2009

FNSNF

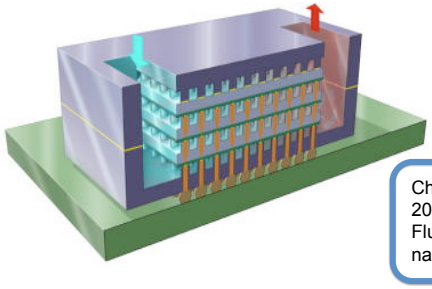
Numerical Simulation of Two-Phase Flows

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

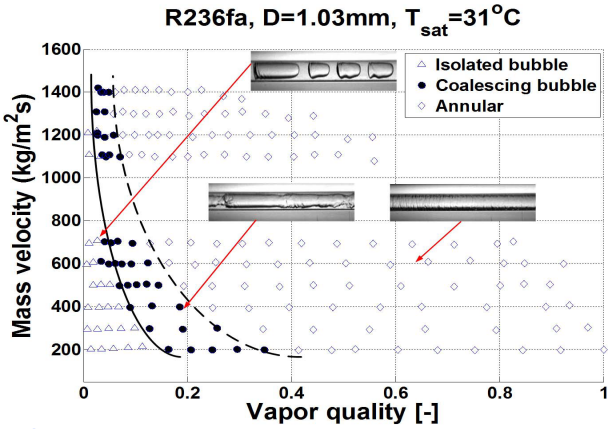
1) Introduction:

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 different 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 DNS-ALE Level-Set code
- Develop a platform for modeling two-phase flows
- Predict flows in microscale geometries
- Couple heat transfer and two-phase flow

3) Navier-Stokes Equations

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 $\boldsymbol{\mu}$ indicate density and viscosity of the fluid. $\boldsymbol{Re}$, $\boldsymbol{Fr}$ and $\boldsymbol{We}$ are dimensionless parameters to characterize the flow regime, $\boldsymbol{t}$ is time, $\boldsymbol{g}$ is gravity force and $\boldsymbol{f}$ is the surface tension that describes interfacial forces between two different fluids (colored red).

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

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

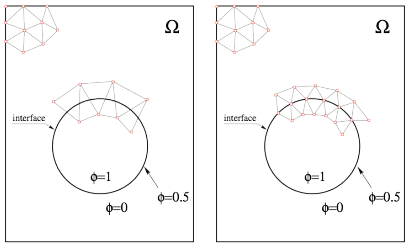
$$\frac{\partial\mathbf{u}}{\partial t} + (\mathbf{u}-\hat{\mathbf{u}})\cdot\nabla\mathbf{u}$$

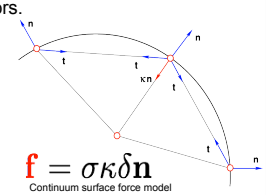
$\hat{\mathbf{u}} = \mathbf{u}$ → Lagrangian

$\hat{\mathbf{u}} = 0$ → Eulerian

4) Modelling: The Level-Set Function and the Surface Tension

The interface is computed explicitly and is part of the moving mesh. However, for the computation of the curvature, a level-set method is employed where the front is represented by the zero level-set of a pseudo-concentration function Φ: R^2 ➔ R for two-dimensional case and function Φ: R^3 ➔ R^2 for three-dimensional case.
The curvature κ is obtained geometrically as the variation of the normal or the tangent vector along the curve that defines the interface, namely: $\kappa n = \partial t/\partial s$ or $\kappa t = -\partial n/\partial s$, considering ∂s as an infinitesimal arc length. The Dirac delta is given by δ and the surface tension coefficient by σ. The computation of this term is easy to implement since the interface between the fluids is represented by computational nodes that are convected with the interface thus leading to reduced mass conservations errors.





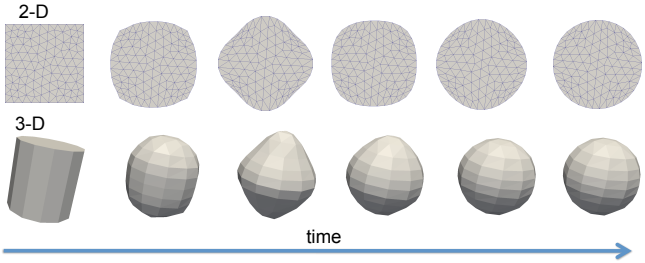
$$\mathbf{f} = \sigma\kappa\delta\mathbf{n}$$

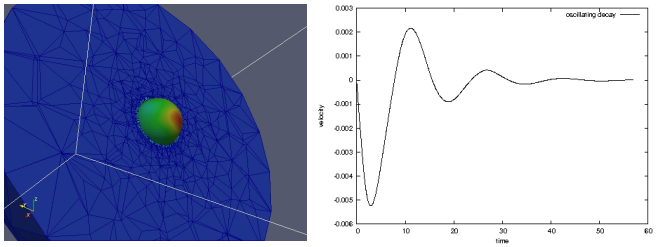
Continuum surface force model
Brackbill and Kothe (1992)

σ : Surface tension coefficient
κ : Curvature
δ : Dirac delta function
$\mathbf{n}$: Normal vector
$\mathbf{t}$: Tangent vector

5) Results

The perturbed domains relax into their equilibrium lowest potential energy shapes with time.





The left hand figure represents the initial perturbed condition. The bubble is immersed inside another fluid, colored blue. The red color over the bubble surface represents the velocity in z direction. The right hand figure shows the decay of the oscillation amplitude as a function of time.

6) Conclusions

The finite element method is a powerful and flexible way to discretize the numerical domain and to well represent of 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.