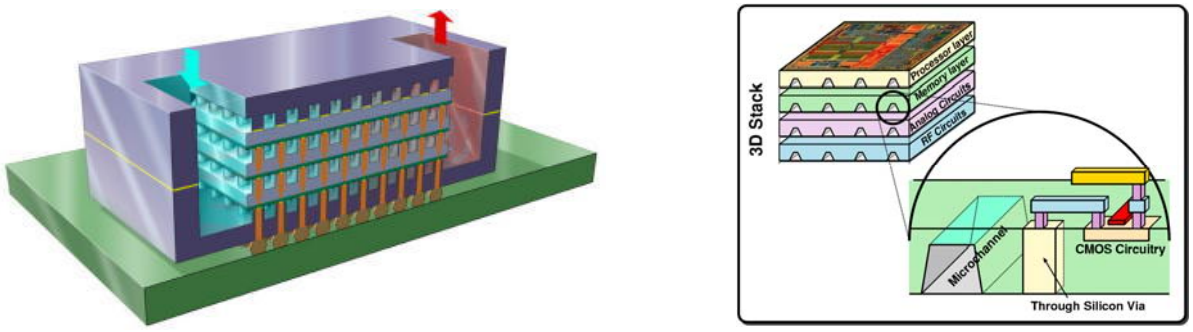
3D Moving Mesh Technique for Microscale Two-Phase Flows

G.R. Anjos, gustavo.rabello@epfl.ch
N. Borhani, navid.borhani@epfl.ch
J.R. Thome, john.thome@epfl.ch

Heat and Mass Transfer Laboratory (LTCM),
Swiss Federal Institute of Technology (EPFL), Lausanne, Switzerland

Today most of the cooling systems found in personal computers and datacenters use either air or water cooling mechanisms. With microprocessor performance increasing, the limits of these cooling systems are being reached and a new technique needs to be developed ([2]). It is known that the heat exchange of two-phase flow systems are much higher then those using single phase flow due to the nature of the thermal behavior of each phase in presence of an interface layer splitting both fluids. This work is part of the larger multi-disciplinary multi-laboratory Nano-Tera/CMOSAIC project which aims to study and design two-phase flow interlayer cooling systems for the next generation of 3D stacked microprocessors (figs. 1a and 1b).



(a) (b)

Figure 1: Design of the next generation of 3D stacked microprocessors. (a) Schematic drawing of the stacked microprocessors where the blue arrow represents the cooling system inflow and the red one represents the outflow after the remotion of the heat. (b) Microchannel shape representation and arrangement in which the two-phase flow will pass by.

In this work numerical simulation is employed to simulate two-phase flow phenomena using the finite element method. 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. In 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 ([4]). The set of equations (see eqs.1 and 2) 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.

$$\frac{\partial \mathbf{u}}{\partial t} + \mathbf{c} \cdot \nabla \mathbf{u} = \nabla p + \frac{1}{Re} \nabla \cdot [\nu(\nabla \mathbf{u} + \nabla \mathbf{u}^T)] + \frac{1}{Fr^2} \mathbf{g} + \frac{1}{We} \mathbf{f} \tag{1}$$

$$\nabla \cdot \mathbf{u} = 0 \tag{2}$$

On the left hand side of Eq. 1, the convective velocity c represents the relative velocity between the mesh and the flow field, given by the following expression: $c = \mathbf{u} - \hat{\mathbf{u}}$ where the last variable denotes the mesh displacement in one time step. The flow velocity is represented by $\mathbf{u}$. Pressure is define by p and time by t whilst Re, Fr and We are non-dimensional groups namely *Reynolds*, *Froud* and *Webber* numbers respectively. Using the Continuum Surface Model (CSF) model according to [1], the source term can be written as $f = \sigma \mathbf{n} \kappa \delta$, where σ is the surface tension coefficient, κ the curvature and $\mathbf{n}$ the surface unit outward normal vector. Additionally, δ represents the Dirac delta function with support on the interface. The density ρ and the viscosity ν are functions distributed along the phases to set up the properties of different working fluids.

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 appear leading to a low accurate calculation of the 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 need to be applied to maintain the mesh quality ([3]). The new methodology proposed here to simulate two-phase flows provides good accuracy to describe the interfacial forces and bubble dynamics.

References

[1] J.U. Brackbill and D.B. Kothe. A continuum method for modeling surface tension. *Journal of Computational Physics*, 100:335–354, 1992.

[2] Marcinichen J.B., Olivier J., and J.R. Thome. Reasons to use two-phase refrigerant cooling. ElectronicsCooling, 2011.

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

[4] Hughes T.J.R Stein E., de Borst R. *Encyclopedia of Computational Mechanics - Volume 1.* John Wiley and Sons, 1st. edition, 2004.