NUMERICAL ANALYSIS OF A GAS BUBBLE FILTER WITH AN APPLICATION IN A MICROCHANNEL

M. Muradoglu[§] and S. Nas[*]
[§]Department of Mechanical Engineering,
Koc University, Sariyer, Istanbul, Turkey
[*]Department of Aeronautical and Astronautical Engineering,
Istanbul Technical University, Istanbul, Turkey

Unsteady multiphase flow in a microchannel bubble filter is studied numerically by using a Finite Difference/Front Tracking method. In this method, one field formulation of the Navier-Stokes equations is written for the entire computational domain with different fluid properties for the ambient fluid and bubble gas. The surface tension forces acting on the bubble-fluid interface are treated as body forces and distributed over the neighboring grid points. The resulting field equations are solved time accurately by a projection method on a staggered grid. The numerical solutions are compared with the experimental observations for various flow parameters.

INTRODUCTION

Flows in microchannels are becoming more and more important as Micro Electro Mechanical Systems (MEMS) technology is advancing rapidly. The available measurement techniques are still too expensive if not impossible in analysis of flows in microchannels. In particular, it is very difficult to study multiphase flows experimentally in microchannels due to extremely small length scales. Therefore computational methods have a great potential in the analysis and design of such microfluidic systems in MEMS devices.

The Finite-Difference/Front-Tracking method has proven to be a useful tool for the analysis of multiphase flows in macroscale problems[1]. However, this methodology has not yet been applied to problems in microchannels although there is no conceptual difficulty to do this since the liquids and gases can be still treated as continuum in many flows in MEMS devices.

The Finite-Difference/Front-Tracking method is essentially based on the concept of Immersed Boundary Method[2], in which Navier-Stokes equations are written for both fluids as a single set of equations for the whole computational domain by properly treating the jumps in material properties, and carefully including the surface tension. In this method, the effects of surface tension are added to the Navier-Stokes equations as a body force by representing it as a delta function[1]. The bubbles are treated as fully deformable fluid particles. Fluid properties such as viscosity and density are smoothed near the interface to overcome the numerical difficulties in treating derivatives. The details of the method can be found in [1].

The bubble filter shown in figure 1 is approximated as two-dimensional problem and the bubbles are treated as cylinders. Both ambient fluid and bubble gas are assumed to be incompressible and the incompressible Navier-Stokes equations are solved by a projection method described in detail by Unverdi and Tryggvason[3]. The interface is represented by line segments and followed explicitly in a Lagrangian frame while the flow equations are solved in a regular fixed Eulerian grid. The information required to close the fluid and the front evolution equations are exchanged between Lagrangian and Eulerian grids by using an interpolation procedure as described in [1].

In figure 1, the flow field is plotted for the entire computational domain for the case of a single bubble. A uniform velocity ($u_{in} = 1m/s$) is prescribed at the inlet (west boundary) and the pressure is fixed to the atmospheric pressure at the outlet (east boundary). The north and south boundaries as well as the boundaries on the filter blocks are treated as no-slip boundaries. Bubbles enter from the inlet as a circular cylinder and move with local flow velocity. The material property ratios, bubble to ambient fluid, are $\mu_i / \mu_o = 1$, $\rho_i / \rho_o = 0.5$, where μ and ρ are viscosity and density, respectively. The subscripts o and i denote that the property is ambient or bubble property, respectively. The Reynolds number based on the inlet channel height, the inlet velocity and ambient fluid properties is $\mathrm{Re} = 4$. The computational domain is taken as $40ax10a$, where a is initial bubble radius, and is resolved by $512x128$ uniform orthogonal grid points. The blowout flow velocity field near the bubble is also shown in figure 2 for the same case to demonstrate the quality of the preliminary numerical solutions.

The preliminary results show that the Finite Difference/Front Tracking method is successful to resolve the flow field for various boundary conditions and governing parameters, and compare well with the experimental observations provided by Tsai and Lin [3].

The bubble filter will be analyzed in detail for various inlet boundary conditions, fluid property ratios, the governing parameters and geometric configurations for single and many bubbles with different sizes and the results will be reported in the final paper.

REFERENCES

1. Tryggvason, G., Bunner, B., Esmaeeli, A., Juric, D., Al-Rawahi, N., Tauber, W., Han, J., Nas, S. and Jan, Y. J. A Front-Tracking Method for the Computations of Multiphase Flow. *J. Comp. Phys*, 169:708--759, (2001).
2. Peskin, C. S. Numerical Analysis of Blood Flow in the Hearth. *J. Comp. Phys*, 25:220, (1977).
3. Unverdi, S. O. and Tryggvason, G. A Front-Tracking Method for Viscous, Incompressible, Multi-Fluid Flows. *J. Comp. Phys*, 100(1):25--37, (1988).
4. Tsai, J-H. and Lin, L. Active Microfluidic Mixer and Gas Bubble Filter Driven by Thermal Bubble Micropump. *Sensors and Actuators A,* 97—98:665-671, (2002).

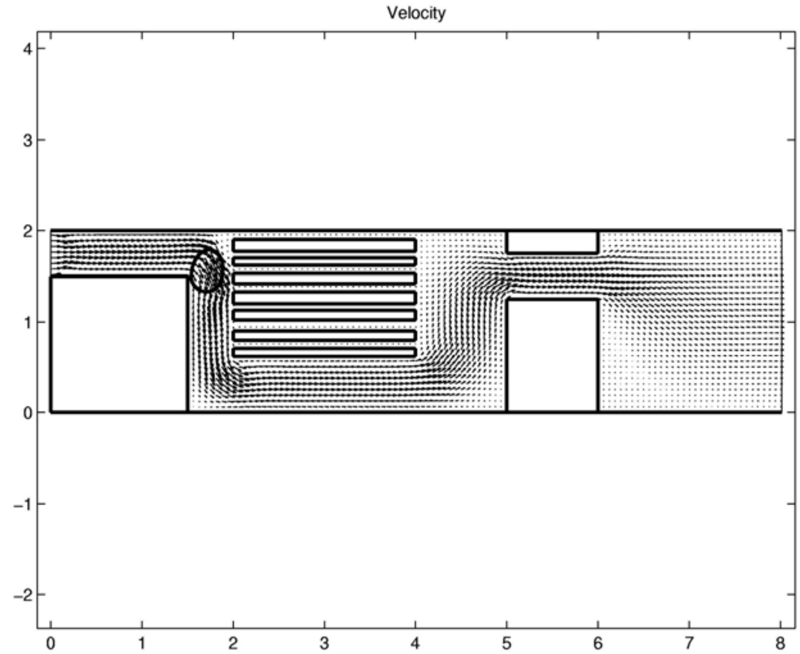

Figure 1: The velocity vectors in the entire computational domain. The flow parameters are specified in the text.

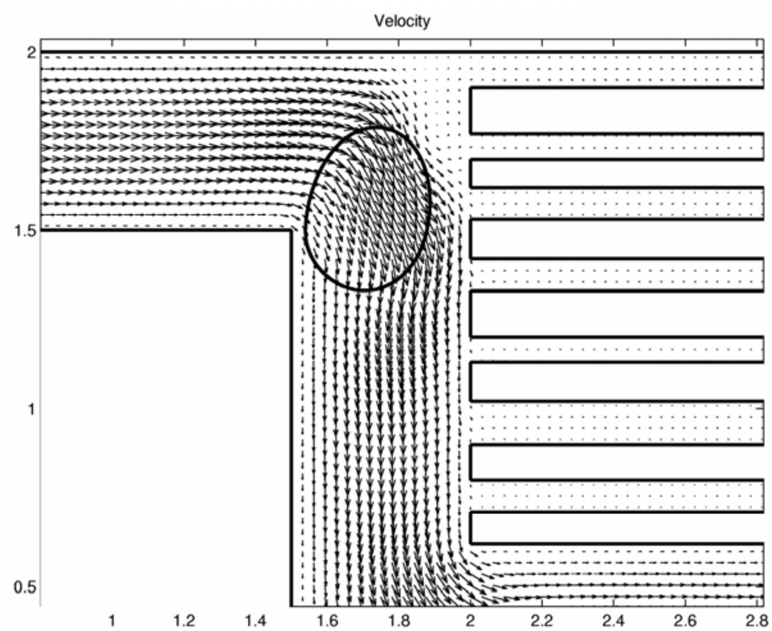

Figure 2: The velocity vectors near the bubble. The flow parameters are specified in the text.