SPH simulation method applied to micro-channel: multi-fluids and surface tension

[1,2]E.A. Patiño-Nariño, [2]D. S. de Lara, [2]Raluca Savu, [2]S. Moshkalev, [1]L.O.S. Ferreira.

eapatinon@fem.unicamp.br; dslara@unicamp.br; rsavu@unicamp.br; stanisla@unicamp.br; lotavio@fem.unicamp.br;

[1]FEM - Faculdade de Engenharia Mecânica, Unicamp
[2]CCS-Nano - Centro de Componentes Semicondutores e Nanotecnologias, Unicamp

Abstract: *Smoothed Particle Hydrodynamics (SPH) is employed for multiphase flow modeling. This method of Lagrangian formulation was used in immiscible fluid to represent interfaces in a self-adaptive way, without the need of complex algorithms to detect and trace the interfaces, as opposed to mesh Eulerian Methods. Thus, SPH implementation is used in conjunction with the Continuum Surface Force (CSF) method. With this methodology, the different phases (liquid-liquid) can be easily treated even with high deformations, dividing or breaking between the liquid phases. As validation, the example of drop deformation in a shear flow is presented. The SPH simulator together with the CSF approach shows promising results for the simulation of surface tension and flow of biphasic fluids.*

Key-words: *Continuum Surface Force method, Microfluids, Mesh-free, SPH.*

Introdution: Fluid microdevices often show fluid-structure, multi-fluid or multi-phase flow interactions. The non-linear effects present in these problems, especially in small dimensions (microscale) interface forces such as surface tension. This force become relevant in the flow behavior of the fluid compared to the inertial effects, which predominate in larger length scales (Macro-scale)[1]. Therefore, flow simulation with complex interfaces are precise, becoming a problem of interest and with practical relevance [2].

A meshfree method with lagrangean formalism and successful simulations of problems with multiscale, multiphase, and fluid-structure interaction is the SPH method [2]. Currently, the microfluidic simulation with SPH uses the Navier-Stokes equations, associated with the CSF method [3]. This approach has been successful in implementing the wettability, capillary and surface tension phenomena for fluid-fluid and fluid-structure [2], [3]. Thus, in this context the SPH method has been used in the last time in microfluidic problems, reaching success in applications of formation of droplets and emulsions [1]. This work presents the implementation of an SPH simulator for microfluidic systems, which can be used to model multiple fluids flowing in 2D micro-channels. The special interest is the behavior of emulsions (liquid-liquid) with fluids of similar density.

The example of deformation of drop with shear flow in a Couette flow condition is implemented with velocities in the top and bottom wall equal to 1 and -1 respectively [1], [2]. A circular droplet was considered in a shear flow with the same density. The radius of the drop is R = 1 and it is located in the middle of the channel. The domain size was 8R x 8R and the periodic boundary condition lies along the x-axis. The calculation was performed on 84 x 84 particles. The fluids were defined to have a $Re= LV_s/\mu = 0.5$. The drop in the shear flow was deformed until the equilibrium between the surface tension and the viscous stresses was reached [2]. Both forces can be expressed by the capillarity (Ca) number defined as $Ca = V_{max}(\mu/\alpha)$.

Results and discussio: Figure 1 (a) shows the comparison between the theoretical results of Taylor [4] and the SPH simulator for several numbers of Ca. The results in this work are in accordance with the empirical number D , which was calculated using the width and length of the deformed drop (D = [width - length] / [width + length]). **Figure 1** (b) shows the final equilibrium state for Ca = 0.05, Ca = 0.1, Ca = 0.15 and Ca = 0.2. It is noted that the deformation of the droplet has an ellipsoidal shape on the main axis by approximately 45 ° in the x-direction. This distribution of particles agrees with results similar to other works [1], [2].

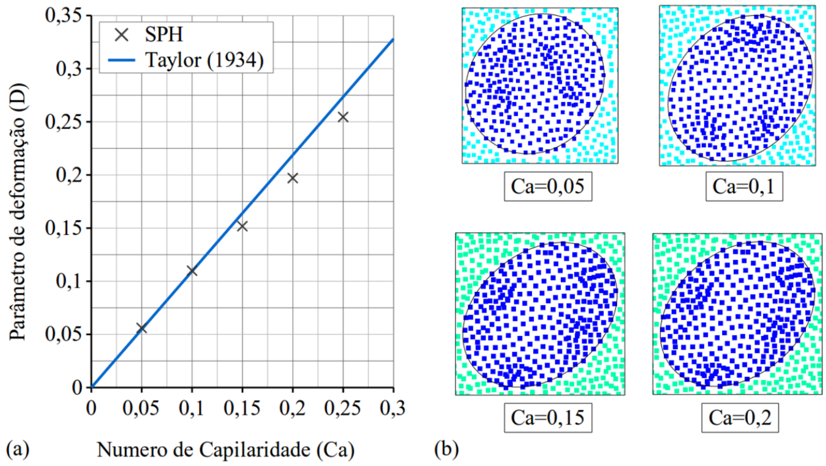

Figure 1. Deformation of drop in a shear flow: (a) deformation parameter D on the number of capillary (Ca); (b) deformation of the drop particles for Ca = 0,05, Ca = 0,1, Ca = 0,15 and Ca = 0,2.

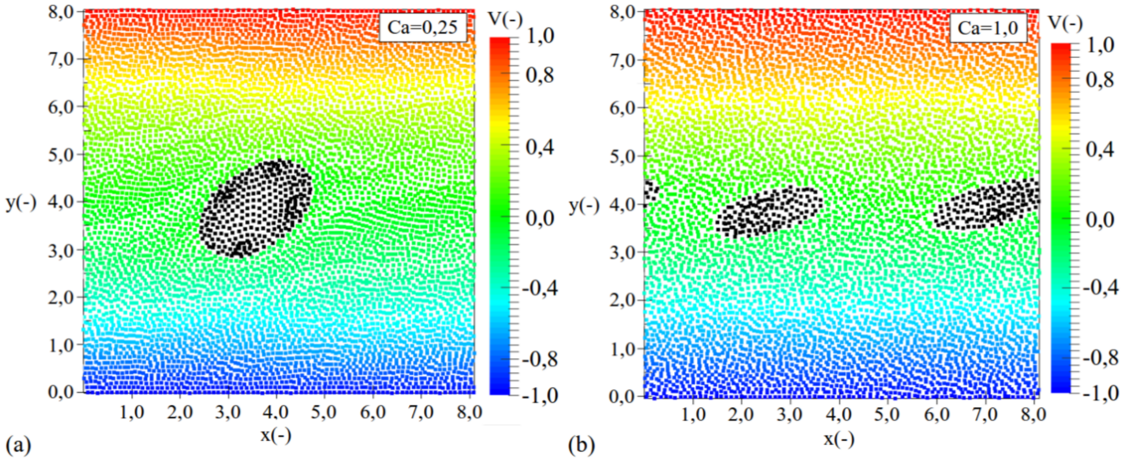

Figure 2. The deformation of the drop in the shear flow problem presenting the velocity field: (a) deformation of the drop particles for Ca = 0,25; (b) deformation of the drop particles for Ca = 1,0.

Figure 2 shows the comparison of the stable regime with Ca = 0.25 (**Figure 2.** (a)) and the non-linear deformed drop with Ca = 1.0 (**Figure 2.** (b)) in the presence of a velocity field for a Couette flow. **Figure 2** (b) shows the steady state for the results with Ca = 1.0, which initially deforms the drop and later it is divided to form two drops.

Conclusions: The simulations showed that high capillary number (Ca = 1.0) forms the dumbbell shape and the process of disintegration happens at Ca <0:25. Thus, SPH simulation coupled with the CSF methodology presents to be a novel tool in the simulation of droplets emulsions (liquid-liquid).

Reference:

[1] S. Adami, X. Y. Hu, and N. a. Adams, "A new surface-tension formulation for multi-phase SPH using a reproducing divergence approximation," *J. Comput. Phys.*, vol. 229, no. 13, pp. 5011–5021, Jul. 2010.

[2] X. Y. Hu and N. A. Adams, "A multi-phase SPH method for macroscopic and mesoscopic flows," *J. Comput. Phys.*, vol. 213, no. 2, pp. 844–861, Apr. 2006.

[3] J. P. Morris, "Simulating surface tension with smoothed particle hydrodynamics," *Int. J. Numer. Methods Fluids*, vol. 33, no. 3, pp. 333–353, Jun. 2000.

[4] G. I. Taylor, "The Formation of Emulsions in Definable Fields of Flow," *Proc. R. Soc. London A Math. Phys. Eng. Sci.*, vol. 146, no. 858, 1934.