Artificial Compressibility Based Method for Two-phase Surface Tension Dominated Flows

Sourabh P. Bhat and Jadav C. Mandal*

Indian Institute of Technology Bombay,
Powai, Mumbai - 400076, INDIA
Tel*: +91-22-25767129, Email*: mandal@aero.iitb.ac.in

Abstract

Two-phase flows are common in various engineering applications. An artificial compressibility based fully-coupled solver is developed for solving surface tension dominated flows using the volume of fluid (VOF) method. The surface tension force is applied as a source term using the continuum surface force (CSF) model. Few problems are presented to evaluate the efficacy of the solver.
Keywords: *Surface tension, Volume of fluid, Artificial compressibility method*

1. Introduction

Surface tension is an important phenomenon occurring due to interaction between two or more fluids with different intermolecular forces. In many multiphase flow applications, especially where capillary forces are important (eg. micro-channel flows) or in low gravity situations (eg. outer-space applications), surface tension forces become extremely important. In such flows accurate estimation of surface tension force is important as it determines the dominant flow features. The pressure based method for solving incompressible two-phase flow problems is a very common strategy observed in literature. In such methods the pressure and the velocity is linked by iteratively correcting the velocity and pressure field. Once the pressure and velocity fields are compatible, the interface is advected by freezing the velocity field. This results in a lockstep kind of method, where the velocity field is first calculated followed by the calculation of fluid interface between phases, for each time step. Thus, at any given instance the interface location lags or leads the velocity-field solution. In technical terms, the velocity field solution is loosely-coupled with the interface location in such an approach.

A density-based artificial compressibility method is known to be efficient in solving incompressible flow problems [1]. In this work, an additional volume fraction advection equation is added to the system of single phase equations to obtain two-phase governing equations. The surface tension force is added as a volumetric source term using the CSF model[2]. A tightly-coupled density based solver is developed to solve two phase flow problems. A few surface tension dominated flow problems are presented to test the efficacy of the solver.

2. Governing Equations and Numerical Formulation

The governing equations for a two-phase flow with surface tension forces can be written in a compact form as,

$$\frac{\partial \mathbf{U}}{\partial t} + \frac{\partial \mathbf{U}}{\partial \tau} + \frac{\partial \mathbf{F}}{\partial x} + \frac{\partial \mathbf{G}}{\partial y} = \frac{\partial \mathbf{F_v}}{\partial x} + \frac{\partial \mathbf{G_v}}{\partial y} + \mathbf{S} + \mathbf{F_s};$$

$$\mathbf{U} = \begin{bmatrix} p/(\rho\beta) \\ \rho u \\ \rho v \\ C \end{bmatrix}, \quad \mathbf{F} = \begin{bmatrix} u \\ \rho u^2 + p \\ \rho uv \\ uC \end{bmatrix}, \quad \mathbf{G} = \begin{bmatrix} v \\ \rho uv \\ \rho v^2 + p \\ vC \end{bmatrix}, \quad \mathbf{S} = \begin{bmatrix} 0 \\ \rho g_x \\ \rho g_y \\ 0 \end{bmatrix},$$

$$\mathbf{F_v} = \begin{bmatrix} 0 \\ \mu \frac{\partial u}{\partial x} \\ \mu \frac{\partial v}{\partial x} \\ 0 \end{bmatrix}, \quad \mathbf{G_v} = \begin{bmatrix} 0 \\ \mu \frac{\partial u}{\partial y} \\ \mu \frac{\partial v}{\partial y} \\ 0 \end{bmatrix}, \quad \mathbf{F_s} = \begin{bmatrix} 0 \\ \sigma\kappa \frac{\partial \widetilde{C}}{\partial x} \\ \sigma\kappa \frac{\partial C}{\partial y} \\ 0 \end{bmatrix}.$$

In the above equations, p is the relative pressure, u and v are the x- and the y-component of velocity respectively. The gravity vector $(g_x, g_y) = (0, -9.81)$. The volume fraction is denoted by C and it is used as an indicator for the location of the interface. The mollified volume fraction $\widetilde{C}$ (see [2]) is obtained by using a convolution kernel as given in [3]. The artificial compressibility parameter β is maintained as constant, chosen to be 1000 for all the simulations. The value of β does not change the solution but only aids the acceleration of the solution. The unsteady simulation is carried out by using a dual time stepping approach [4], where a steady state is obtained in pseudo-time τ for every real-time step. The real time is denoted as t and the pseudo-time is denoted as τ in the above equations. The equations are closed by using a relation of effective density (ρ) and effective viscosity (μ) based on the volume fraction function as,

$$\rho = C\ \rho_1 + (1 - C)\,\rho_2 \quad \text{and} \quad \mu = C\ \mu_1 + (1 - C)\,\mu_2.$$

The subscripts 1 and 2 indicate the density and viscosity of fluid-1 and fluid-2 respectively.

A second order finite volume method is used with Riemann solver based methodology. The standard HLL Riemann solver [5] is used to calculate the fluxes at cell interfaces. The second order reconstruction is performed by using linear solution dependent weighted least square method [6, 7].

3. Results and Discussion

3.1. *Equilibrium rod*

This is a simple test case which is designed to test that the curvature is calculated properly. Also, an analytical solution for pressure inside the bubble can be easily estimated. In this test case a fluid rod of uniform radius is placed in another fluid. Thus, due to application of only surface tension forces the pressure evaluated can be analytically calculated as $p = \sigma\kappa$. Here, p is the pressure inside the fluid rod, σ is the surface tension force per unit length of the interface and κ is the curvature. The curvature for a uniform radius circle can be easily evaluated as $\kappa = 1/r$, where, r is the radius of the circle. The dimensions of the square domain are 6 cm $\times$ 6 cm. The circular fluid rod of radius $r = 2$ cm is placed at the center of this domain. The curvature is therefore equal to $\kappa = 1/0.02 = 50\ \text{m}^{-1}$. With the surface tension, σ, for a water-air interface as 72.75×10^{-3} N/m, the pressure inside the rod can be evaluated analytically to be equal $50 \times 72.75 \times 10^{-3} = 3.6375\ \text{N/m}^2$. The results show a good agreement with this result as seen in figure 1, thus ascertaining the correctness of surface tension flux for a given curvature.

3.2. *Non-equilibrium rod*

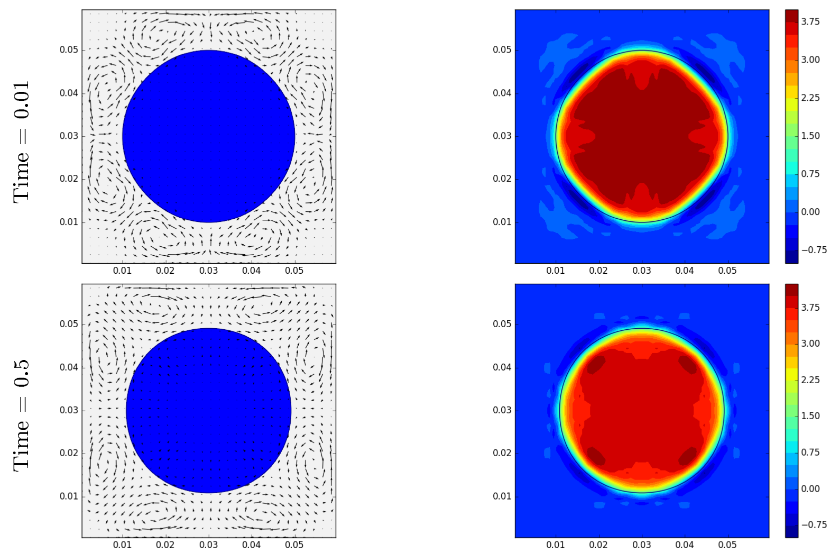
This test case is designed to evaluate the capability of the solver to calculate the curvatures properly and hence change the direction of surface tension forces as the curvature evolves from concave to convex. In this problem, a square shaped fluid of side length 3.75 cm is placed at the center of square domain of dimensions 7.5 cm $\times$ 7.5 cm. Without the effects of any other forces the bubble starts to oscillate under the influence of surface tension force. The solutions obtained by the solver agree well with the results from the literature [2]. The results obtained at various time level is displayed in figure 2.

3.3. *Acute wall contact angle*

The wall contact angle considered for this problem is 5°. Due to absence of any other force other than the surface tension and wall adhesion the surface distorts heavily to maintain the wall contact angle. The results are displayed in figure 4. The solutions obtained agree well with the results from the literature [2].

3.4. *Obtuse wall contact angle*

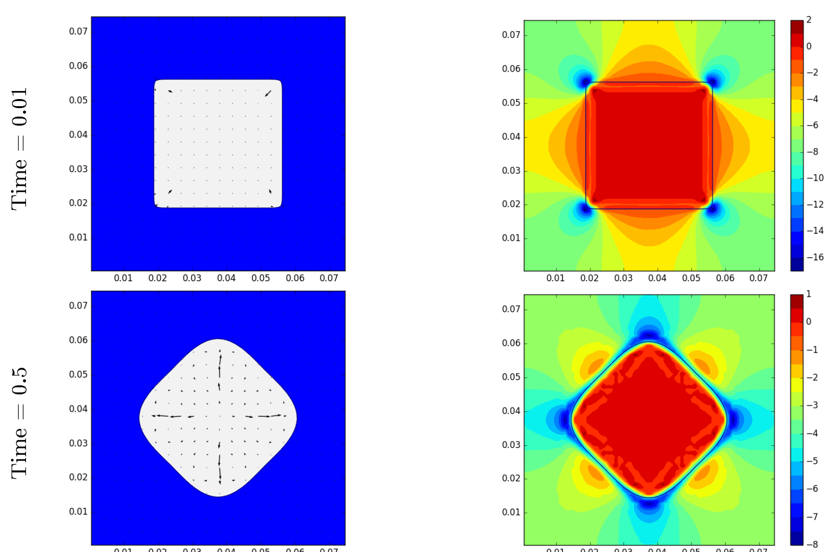
The wall contact angle considered for this problem is 175°. Again, as in the case of acute angle, due to absence of any other force other than the surface tension and wall adhesion the surface distorts heavily to maintain the wall contact angle. The results are displayed in figure 5. The solutions obtained agree well with



Volume Fraction (C=0.5) with velocity vectors

Pressure (Pa)

Figure 1: Equilibrium rod



Volume Fraction (C=0.5) with velocity vectors

Pressure (Pa)

Figure 2: Non-equilibrium rod

the results from the literature [2].

4. Conclusion

A tightly-coupled density based two-phase flow solver with surface tension forces is developed. The results obtained agree very well with the expected analytical results and results of test cases from the literature. Further investigation and validation is needed with practical problems and experimental results.

References

[1] J. C. Mandal and A. S. Iyer, "An upwind method for incompressible flow computations using pseudo-compressibility approach," in *19th AIAA Computational Fluid Dynamics Conference*, 2009.

[2] J. U. Brackbill, D. B. Kothe, and C. Zemach, "A Continuum Method for Modeling Surface Tension," *Journal of Computational Physics*, vol. 100, pp. 335–354, 1992.

[3] J. J. Monaghan, "Smoothed Particle Hydrodynamics," *Annual Review of Astronomy and Astrophysics*, vol. 30, pp. 543–574, sep 1992.

[4] A. L. Gaitonde, "A dual-time method for two-dimensional unsteady incompressible flow calculations," *International Journal for Numerical Methods in Engineering*, vol. 41, no. 6, pp. 1153–1166, 1998.

[5] A. Harten, P. D. Lax, and B. van Leer, "On Upstream Differencing and Godunov-Type Schemes for Hyperbolic Conservation Laws," *SIAM Review*, vol. 25, pp. 35–61, jan 1983.

[6] J. Mandal and J. Subramanian, "On the link between weighted least-squares and limiters used in higher-order reconstructions for finite volume computations of hyperbolic equations," *Applied Numerical Mathematics*, vol. 58, pp. 705–725, May 2008.

[7] J. C. Mandal and S. P. Rao, "High resolution finite volume computations on unstructured grids using solution dependent weighted least squares gradients," *Computers and Fluids*, vol. 44, pp. 23–31, May 2011.

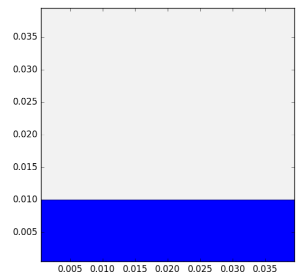
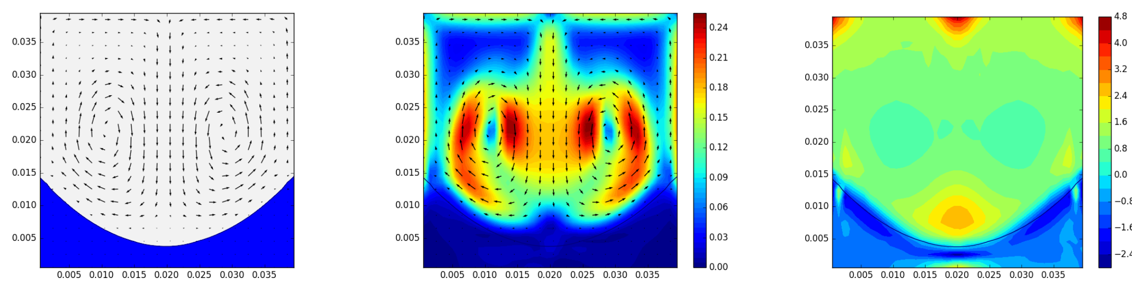


Figure 3: Wall contact angle: initial condition, time = 0.0 s

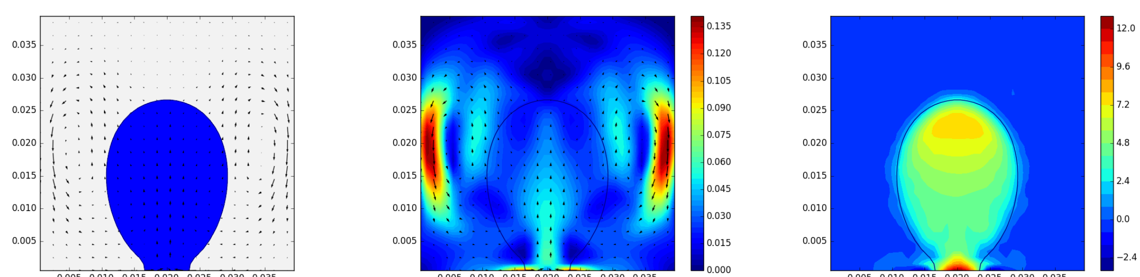


Volume Fraction (C=0.5) with velocity vectors

Velocity (m/s)

Pressure (Pa)

Figure 4: Acute wall contact angle without gravity, time = 0.5 s



Volume Fraction (C=0.5) with velocity vectors

Velocity (m/s)

Pressure (Pa)

Figure 5: Obtuse wall contact angle without gravity, time = 0.5 s