26th International Congress of Mechanical Engineering
COBEM 2021
ABCM Associação Brasileira de Engenharia e Ciências Mecânicas
November 22-26, 2021, Virtual Congress, Brazil

COBEM2021-1503 - Finite Element Simulation For Two-phase Flow with a Decoupled Fluid Interface

Daniel B. V. Santos
Gustavo R. Anjos
Federal University of Rio de Janeiro, Rio de Janeiro, Brazil
daniel.barbedo@coppe.ufrj.br, gustavo.rabello@coppe.ufrj.br

Abstract. *In this research, finite element numerical simulation is used to describe bubble dynamics in two-phase flows. The accurate simulation of interface dynamics in two-phase flows is of crucial relevance for numerical analysis of two-phase heat transfer. The simulation is achieved by solving the incompressible Navier-stokes and energy conservation equations for two-phase flows, discretized through the Finite Element Method (FEM), where the interface and the fluid meshes are not explicitly connected. Two meshes are used, a fixed one for the fluid flow, and a moving mesh which describes the interface movement between fluids. Fluid properties are smoothed out to avoid numerical instabilities in the transition from one fluid to another. Surface tension is implemented according to the well-known continuum surface tension model, using the Laplace-Beltrami operator for curvature computation. Surface tension force is added explicitly to the Navier-Stokes equations as a volume force through the gradient of a Heaviside function, therefore the momentum equation is solved using a one-fluid approach. To prove that such an interesting numerical scheme is stable and accurate, we present two test cases, consisting of a bubble rising due to gravity. The first case explores a small difference in viscosity and density and the second one presents a more aggressive difference in properties across both fluids, and a smaller surface tension, exposing a more difficult simulation. Both cases resulted in stable bubbles, with qualitative results agreeable to the literature.*

Keywords: *Two-phase Flow, Gravity Driven Bubble, Laplace-Beltrami, Decoupled Fluid Interface.*

1. INTRODUCTION

Two-phase flow is present in a broad range of industrial applications, such as cooling of electronic components like microprocessors and data banks, the extraction and refinement of oil and gas Jia Liu (2020), power electronics in hybrid cars, nuclear reactor components, medical X-ray equipment and more Chirag R. Kharangate (2017). It is also very physics rich phenomena, which can involve heat transfer, evaporation, condensation, capillarity effects, movement of bubbles, all in a single problem. As such, it is a complex phenomenon to model, be it experimentally or numerically.

Two-phase flow has several branches of research, with none elected the principal method of solving the problem. As such, there's many differing approaches to the problem, such as D.L. Sun (2010) who extended the VOSET method, which is a combination of the Level Set and Volume of Fluid, to three dimensions, using the Piecewise Linear Interface Construction (PLIC) to reconstruct the interface shape. Kong Ling (2015) expands on D.L. Sun (2010)'s work, extending PLIC to 3D cases and using a geometric approach to compute the level-set function, with the intent of simplifying the extension of VOSET from 2D to 3D. Andrea Ferrari (2017) proposed a method that combines both the Level Set (LS) and Volume of Fluid (VOF), called Flexible Coupled Level Set and Volume of Fluid (flexCLV), designed to benefit from the good interface topology offered by the Level Set method. Mathieu Labois (2017) presented a multiphase formulation with a pressure-based approach, using the 4+-equation model, for compressible flows. Tong Qin (2013) investigated the interaction between a deformable bubble and a rigid wall, through direct numerical simulation, using the ALE method to simulate the bubble interface.

E. Gros (2018) simulates a two-phase fluid flow with heat and mass transfer, using the finite element method combined with the ALE framework. Baolin Tian (2020) developed an ALE method for the five-equation model. The ALE model used is two-stage, with a Lagrangian phase and a rezone-remap phase. Zekang Cheng (2020) solved the incompressible Navier-Stokes equations for a two dimensional domain by discretizing them by Taylor-Hood elements, using an ALE finite element method, where the mesh conforms to the fluid interface, and remeshes when the interface suffers deformations, following its evolution. Anjos *et al.* (2020) presented an ALE finite element method for simulation of axisymmetric two-phase flows, with dynamic boundaries and interface tracking. In his work, the mesh points move to give a detailed description of the fluid interfaces while using adaptive mesh refining and remeshing to guarantee high quality mesh elements.

In this work, a two-phase fluid flow is simulated using the finite element method with an interface mesh decoupled from the fluid mesh, and a continuum surface force model suggested by Kothe and Zemach (1991), using the Laplace-Beltrami operator for calculating the curvature, to simulate the surface tension force. Two examples were evaluated, a bubble with density and viscosity close to the surrounding fluid, given by the ratio $\rho_1/\rho_2 = 10$ and $\mu_1/\mu_2 = 10$, and intense surface tension $We = 10$, and a bubble with a higher density and viscosity ratio between the bubble and surrounding fluid, given by $\rho_1/\rho_2 = 1000$ and $\mu_1/\mu_2 = 100$, and lesser surface tension, with $We = 125$.

2. METHODOLOGY

To describe the flow field, the Navier-Stokes equations for Newtonian fluids were utilized, with an added term to represent the surface tension forces, as shown in Anjos *et al.* (2020). The Navier-Stokes equations were then discretized by the finite element method and solved with two decoupled meshes, one for the fluid and one for the interface. The Navier-Stokes equations in their nondimensional form are given by,

$$\nabla \cdot \mathbf{v} = 0 \tag{1}$$

$$\rho(\mathbf{x})\left(\frac{\partial \mathbf{v}}{\partial t} + \mathbf{v} \cdot \nabla \mathbf{v}\right) = -\nabla p + \frac{1}{Re}\nabla \cdot \mu(\mathbf{x})\left(\nabla \mathbf{v} + \nabla \mathbf{v}^T\right) + \frac{1}{We}\mathbf{f_{st}} + \rho(\mathbf{x})\mathbf{g} \tag{2}$$

where $\mathbf{f_{st}}$ is the surface tension force due to the interface between phases, Re is the Reynolds number and We is the Weber number. $\rho(\mathbf{x})$ and $\mu(\mathbf{x})$ are the fluids density and viscosity, which are constant over time, but not over the domain and $\mathbf{g}$ is the gravitational force field.

The fluids are considered to be incompressible, and their viscosity is assumed to be constant over time. The Reynolds and Weber numbers are calculated using the denser fluid's properties. To avoid the discretization of the non-linear convective term of the Navier-Stokes equations, a semi-lagrangian scheme is utilized, as can be seen in Marques (2011).

The surface tension force term fst, is modeled after the continuum surface model suggested by Kothe and Zemach (1991), using the Laplace-Beltrami operator to obtain the curvature κ, as reported in Hysing (2005), and is given by

$$\mathbf{f_{st}} = \kappa \nabla H \tag{3}$$

where κ is the bubble or droplet interface curvature for each point of the interface mesh, and ∇H is the Heaviside function's gradient. The Heaviside function generates a value of 0 for points inside the reference (denser) fluid, values of 1 for the other fluid, and a smooth curve with values between 0 and 1 when close to the fluid's interface. The Heaviside function is given by

$$H = \begin{cases} 1, & \text{if } d > \epsilon \\ 0, & \text{if } d < -\epsilon \\ 1 - 0.5\left[1 + \dfrac{d}{\epsilon} + \dfrac{1}{\pi}sin\left(\pi d/\epsilon\right)\right], & \text{otherwise} \end{cases} \tag{4}$$

where ϵ is the interface length and d is the smallest distance from the evaluated point to the interface. $\rho*$ and $\mu*$ are calculated based on the Heaviside function values, and they are given as follow:

$$\rho(\mathbf{x}) = \rho_1 H(\mathbf{x}) + \rho_2(1 - H(\mathbf{x})) \qquad \mu(\mathbf{x}) = \mu_1 H(\mathbf{x}) + \rho_2(1 - H(\mathbf{x})) \tag{5}$$

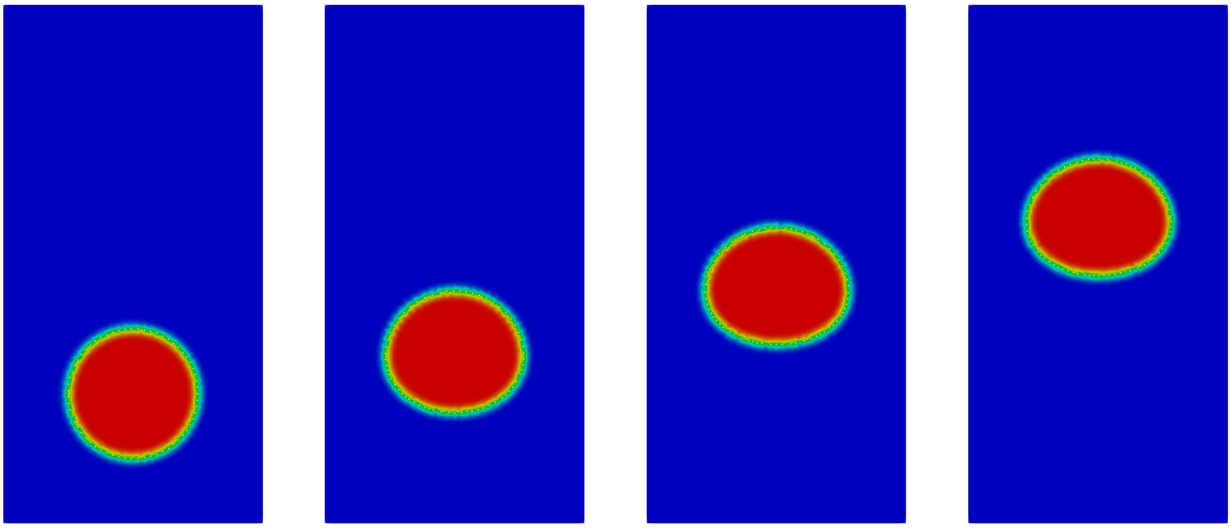

Figure 1. Test Case 1. Bubble shape at iteration 0, 1000, 2000 and 3000, with $\Delta t = 0.001$. The bubble changes position as it assumes an ellipsoid shape, with slightly flat bottom. $We = 10$ and $Re = 35$. 24313 nodes and 16090 elements, with $\rho_1/\rho_2 = 10$ and $\mu_1/\mu_2 = 10$

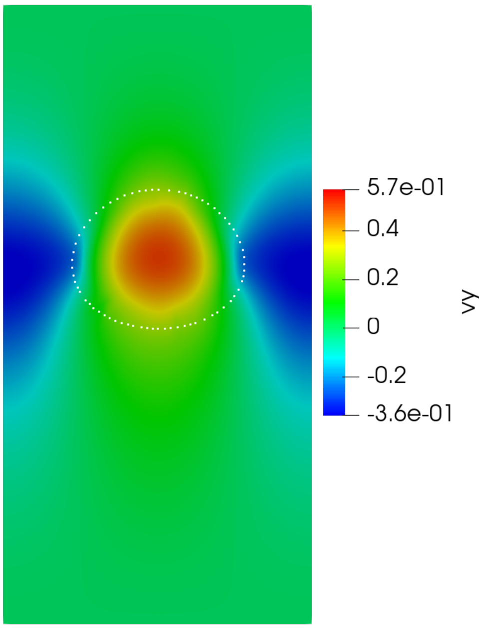

Figure 2. Vertical Velocity Field for Test Case 1. The white dots mark the bubble interface mesh position

3. NUMERICAL MODEL

To represent the two-phase fluid flow, two separate meshes were utilized. The first one is a regular finite element mesh, where the spatial domain is subdivided into a number of smaller, triangular elements. The mini element was utilized, which is comprised of four nodes, where the pressure is interpolated by the three edge nodes, and the velocity is interpolated by the three edge nodes, plus the fourth node located at the element's center of mass.

For the second mesh, a one dimensional, closed mesh where the last element links up with the first element, was utilized. This mesh is used as reference for the curvature, as well as Heaviside function calculations.

The two meshes are completely decoupled, and the points from the second mesh do not need to occupy the same position as points in the first mesh. Information however, is exchanged between the meshes. The Heaviside function is calculated using the interface mesh as reference, and is used to calculate the velocity fields. In turn, the interface mesh position is altered according to the velocity fields obtained by the fluid mesh.

The fluid properties are taken as constant inside an element, but an abrupt transition in the interface is avoided by the use of the smooth Heaviside function to calculate the properties at the interface proximity. The fluid properties, μ and ρ are interpolated over the a set length d perpendicular to the interface.

4. NUMERICAL RESULTS

This sections shows the two two-phase examples executed to demonstrate the accuracy and stability of the proposed methodology. The two examples are bubbles rising due to gravity effects, with different fluid properties.

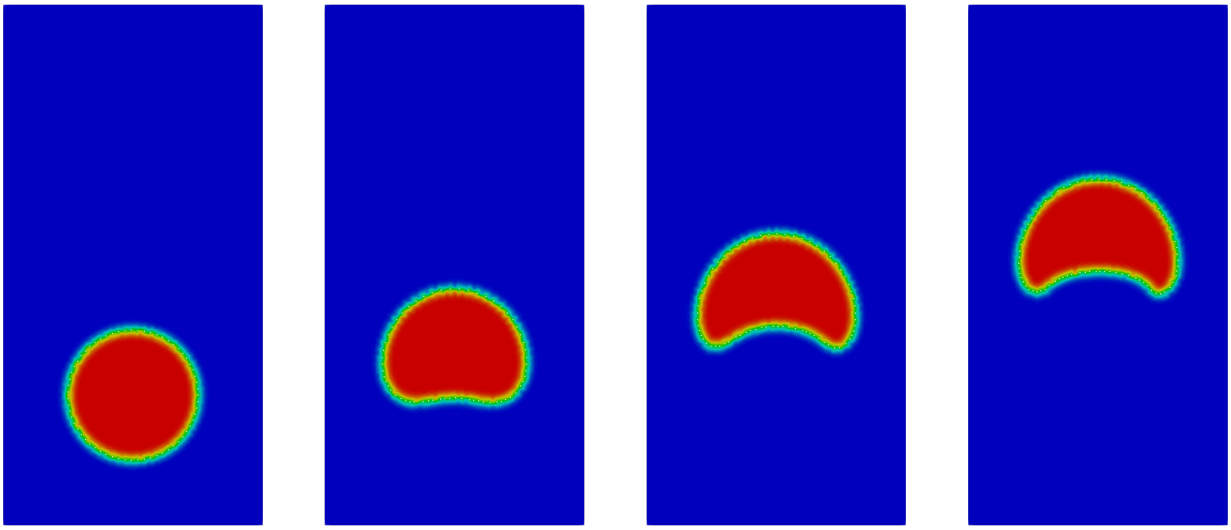

Figure 3. Test Case 2. Bubble shape at iteration 0, 1000, 2000 and 3000, with $\Delta t = 0.001$. The bubble changes position as it assumes an oblate ellipsoidal cap shape. $We = 125$ and $Re = 35$. 24313 nodes and 16090 elements, with $\rho_1/\rho_2 = 1000$ and $\mu_1/\mu_2 = 100$

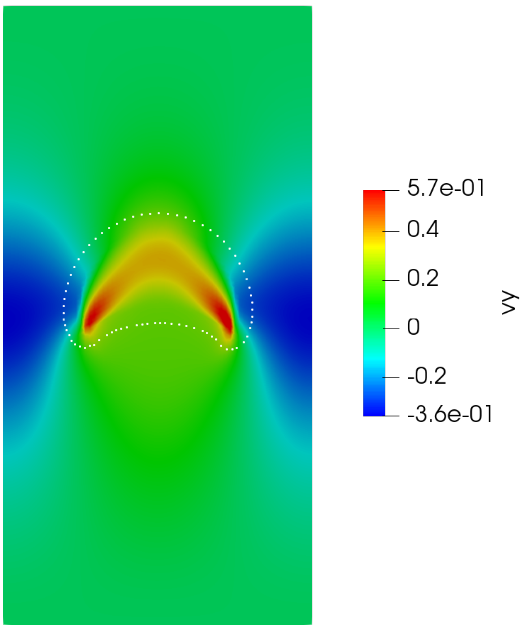

Figure 4. Vertical Velocity Field for Test Case 2. The white dots mark the bubble interface mesh position

4.1 Test Case 1

For this test case, a bubble of radius $r = 0.25$ was simulated, inserted in a fluid domain of length=1 and height=2. The fluid domain mesh is composed of 24313 nodes and 16090 elements. The bubble mesh, which represents the fluid interface has 62 nodes and 62 linear elements. The time step interval used was $\Delta t = 0.001$, and the simulation was run over 3000 iterations in total. The heaviside function was set to produce an interface width of 10% the bubble's diameter. Nondimensional values were utilized, and the heavier fluid is used as reference, assuming values of 1 for density and viscosity. For this test case, the density ratio is given by $\rho_1/\rho_2 = 10$ and the viscosity ratio by $\mu_1/\mu_2 = 10$. The surface tension is modulated by the Weber number, given by $We = 10$, and Reynolds number utilized was $Re = 35$. The results can be observed in Fig. 1, and the vertical velocity profile can be seen in Fig. 2.

4.2 Test Case 2

This test case uses the same geometry as the first test case, with the same number of nodes and elements, both for the fluid domain and the bubble interface mesh. The simulation was executed with the same time step of $\Delta t = 0.001$ and total number of iterations. The same interface width of 10% the bubble's diameter was utilized. For this test case, the density ratio utilized is given by $\rho_1/\rho_2 = 1000$ and the viscosity ratio by $\mu_1/\mu_2 = 100$. The Weber number is $We = 125$ for this case, and Reynolds number is also $Re = 35$. The results can be observed in 3 along the vertical velocity field in Fig. 4

5. CONCLUSION

The numerical methodology followed in this research, using the Continuum Surface Force model and the Laplace-Beltrami operator, showed stability for both test cases executed. The results obtained offer good qualitative results, with Test Case 1 showing an ellipsoidal shape with a slightly flat bottom and Test Case 2's bubble turning into an oblate ellipsoidal cap. Both results are coherent with the Reynolds and Weber numbers simulated, and further qualitative validation is currently in progress. The method offers ease of implementation and reasonable computational costs, with the possibility of extension to 3-dimensional two-phase flows.

6. REFERENCES

Andrea Ferrari, Mirco Magnini, J.R.T., 2017. "A flexible coupled level set and volume of fluid (flexclv) method to simulate microscale two-phase flow in non-uniform and unstructured meshes". *International Journal of Multiphase Flow*, Vol. 91, pp. 276–295. doi:http://dx.doi.org/10.1016/j.ijmultiphaseflow.2017.01.017.

Anjos, G., Mangiavacchi, N. and Thome, J., 2020. "An ale-fem method for two-phase flows with dynamic boundaries". *Computer Methods in Applied Mechanics and Engineering*, Vol. 362, p. 112820. ISSN 0045-7825. doi:https://doi.org/10.1016/j.cma.2020.112820. URL `http://www.sciencedirect.com/science/article/pii/S0045782520300013`.

Baolin Tian, L.L., 2020. "A five-equation model based global ale method for compressible multifluid and multiphase flows". *Computers and Fluids*, Vol. 214, p. 104756. doi:https://doi.org/10.1016/j.compfluid.2020.104756.

Chirag R. Kharangate, I.M., 2017. "Review of computational studies on boiling and condensation". *International Journal of Heat and Mass Transfer*, Vol. 108, pp. 1164–1196. doi:http://dx.doi.org/10.1016/j.ijheatmasstransfer.2016.12.065.

D.L. Sun, W.T., 2010. "A coupled volume-of-fluid and level set (voset) method for computing incompressible two-phase flows". *International Journal of Heat and Mass Transfer*, Vol. 53, pp. 645–655. doi: http://dx.doi.org/10.1016/j.ijheatmasstransfer.2009.10.030.

E. Gros, G. Anjos, J.T., 2018. "Moving mesh method for direct numerical simulation of two-phase flow with phase change". *Applied Mathematics and Computation*, Vol. 339, pp. 636–650. doi: https://doi.org/10.1016/j.amc.2018.07.052.

Hysing, S., 2005. "A new implicit surface tension implementation for interfacial flows". *International Journal for Numerical Methods in Fluids*, Vol. 51, pp. 659–672. doi:10.1002/fld.1147.

Jia Liu, X.L., 2020. "Numerical evaluation on multiphase flow and heat transfer during thermal stimulation enhanced shale gas recovery". *Applied Thermal Engineering*, Vol. 178, p. 155554. doi: https://doi.org/10.1016/j.applthermaleng.2020.115554.

Kong Ling, Z.H.L., 2015. "A three-dimensional volume of fluid & level set (voset) method for incompressible two-phase flow". *Computers & Fluids*, Vol. 118, pp. 293–304. doi:http://dx.doi.org/10.1016/j.compfluid.2015.06.018.

Kothe, J.U.B.D.B. and Zemach, C., 1991. "A continuum method for modeling surface tension". *Journal of Computational Physics*, Vol. 100, pp. 335–354. doi:https://doi.org/10.1016/0021-9991(92)90240-Y.

Marques, L., 2011. "Numerical study of bloodstream diffusion of the new generation of drug-eluting stents in coronary arteries". *Fluids*, Vol. 6, p. 71. ISSN 2311-5521. doi:https://doi.org/10.3390/fluids6020071.

Mathieu Labois, C.N., 2017. "Non-conservative pressure-based compressible formulation for multiphase flows with heat and mass transfer". *International Journal of Multiphase Flow*, Vol. 96, pp. 24–33. doi: http://dx.doi.org/10.1016/j.ijmultiphaseflow.2017.07.004.

Tong Qin, Saad Ragab, P.Y., 2013. "Axisymmetric simulation of the interaction of a rising bubble with a rigid surface in viscous flow". *International Journal of Multiphase Flow*, Vol. 52, pp. 60–70. doi: http://dx.doi.org/10.1016/j.ijmultiphaseflow.2013.01.001.

Zekang Cheng, Jie Li, C.Y.L., 2020. "An exactly force-balanced boundary-conforming arbitrary-lagrangian-eulerian method for interfacial dynamics". *Journal of Computational Physics*, Vol. 408, p. 109237. doi: https://doi.org/10.1016/j.jcp.2020.109237.

7. RESPONSIBILITY NOTICE

The authors are solely responsible for the printed material included in this paper.