# A Hybrid Grid Based Algebraic Volume of Fluid Method for Interfacial Flows

**Jai Manik, Amaresh Dalal and Ganesh Natarajan**

**Abstract** In the present work for numerically investigating the interfacial flows, an algebraic Volume of Fluid technique has been implemented over hybrid unstructured meshes. Following the work of Dalal et al. (Numer Heat Transf Part B 54 (2):238–259, 2008 [2]), the governing equations are discretised by cell centered finite volume method wherein pressure-velocity coupling has been achieved by momentum interpolation due to Rhie and Chow (AIAA J 21:1525–1532, 1983 [12]). The binary fluid problem is represented by a single fluid formulation with a fluid property jump at the interface. Two schemes namely NVD based GAMMA scheme (Jasak, Int J Numer Meth Fluids 31:431–449, 1999 [7]) and Convergent and Universally Bounded Interpolation Scheme for the Treatment of Advection (CUBISTA) (Alves et al., Numer Heat Transf 49:19–42, 2006 [1]) has been incorporated into an in-house fully coupled Navier-Stokes solver. These schemes are validated with the published results of collapse of water column also known as dam break problem by Martin and Moyce (Math Phys Sci 244:312–324, 1952 [9]) and Rayleigh-Taylor instability (Tryggvason, J Comput Phys 75:253–282, 1988 [13]). The results are found to be in good agreement.

**Keywords** Two phase flow · Volume of fluid method (VOF) · Free surface flows · Finite volume method · Advection schemes

Jai Manik · Amaresh Dalal (✉) · Ganesh Natarajan
Department of Mechanical Engineering, Indian Institute of Technology Guwahati, Guwahati 781 039, India
e-mail: amaresh@iitg.ernet.in

Jai Manik
e-mail: j.manik@iitg.ernet.in

Ganesh Natarajan
e-mail: n.ganesh@iitg.ernet.in



A.K. Saha et al. (eds.), *Fluid Mechanics and Fluid Power – Contemporary Research*, Lecture Notes in Mechanical Engineering,
DOI 10.1007/978-81-322-2743-4_105

## 1 Introduction

Interfacial flows arise in various practical and natural applications like water wave, liquid sloshing inside a tank, flame propagation and many other areas in engineering and science. The prediction of multi-phase flows is beneficial in process design or in the diagnosis of problems in the established process. A detailed analysis of immiscible fluid flows requires a precise representation of the interface which separates the two fluids. In the last few decades, a lot of work has been done in this direction and many methods have been proposed for defining sharp and accurate interface position. In general these methods can be broadly categorized under two categories: Lagrangian and Eulerian. In Lagrangian method either the mesh boundary coincides with the interface or internal mesh edges collectively forms an interface. So in order to capture the interface, continuous re-meshing of the domain is needed at each time step. This may require complex algorithm depending upon whether extra elements needs to be added or subtracted during the re-meshing operation. In addition to this an extra precaution has to be taken in order to enforce volume conservation of the moving cell. This method faces difficulty while simulating flows undergoing large deformations as the mesh elements will be highly deformed which on the other hand reduces the accuracy and sometimes leads to the numerical instabilities.

In Eulerian method, the interface needs not to coincide with the mesh boundary or with the edge of an internal element rather it is treated as a sharp front moving through the mesh elements. In general, two different approaches have evolved within this category: (a) surface methods and (b) volume methods. In surface methods special marker points are used to represent an interface and in order to approximate the points in between these marker points special interpolation techniques are used. The advantage of this method lies in the fact that interfaces remains sharp as it is advected with time across the domain. Glimm et al. [4] and Tryggvason et al. [14] are the major developers of this method. However, this method fails to give accurate results while simulating coalescence and breakup of the interface. The complexities further increases due to interaction between fixed grid and movable interface particles. Another front tracking method is the level set method introduced by Osher and Sethian [11], wherein a continuous level set function is defined throughout the domain by the shortest distance from the interface. Naturally, the interface is represented with the zero value of the level set function and is advected with the background flow field. This method is simple and easy to implement, but the limitation of mass loss occurs while simulating flows in which interface gets highly distorted.

An alternative to surface methods are volume methods or volume tracking methods. In this method, the presence of fluid is identified by some kind of indicators. One of the oldest method known as marker-and-cell (MAC) method proposed by Harlow and Welch [6], specifically for free surface flows, the presence of fluid is determined by the presence of massless particles. These particles are transported in a Lagrangian fashion over a fixed Eulerian grid. This method

simulated a variety of flow configurations including the famous dam break flow and Rayleigh Taylor instability. However, difficulty arises while simulating high shear flows, which needs large number of particles, which on the other hand increases the computational power. Additionally for simulating three dimensional flows, also large number of particles is needed which again requires high computational power.

Another way to identify a particular fluid is to use indicator function or color function. Volume of Fluid (VOF) method is one of the most widely used method which comes under this category. In this method the interface is tracked with the help of a phase indicator function $\phi$ (also known as volume fraction) whose value varies from 0 to 1. A value of $\phi = 1$ shows the complete presence of one fluid while $\phi = 0$ refers to the complete presence of other fluid in the respective grid element. For the values of $\phi$ in between 0 and 1, the grid element is partly filled by one fluid and partly by other, which indirectly represents the presence of interface in that particular grid element. The advection of volume fraction is governed by a hyperbolic equation, which may incur numerical diffusion of the step profile of the interface, while solving directly with the help of numerical differencing schemes. To avoid this numerical smearing of the interface, generally following two approaches are adopted: (1) interface reconstruction using line techniques [10, 17], (2) usage of high resolution differencing schemes. On unstructured grid it is difficult to use line reconstruction method, further the complicacy even more increases while extending it to three dimensional simulations. For this reason generally second approach is adopted for simulating interfacial flows over unstructured grid, wherein no explicit reconstruction of the interface is required and interface is captured by directly solving the hyperbolic equation. In the past various schemes have been devised [3, 15, 16] to discretize the advection equation which governs the volume fraction field. These schemes are generally blend of high resolution schemes and compressive schemes so as to reduce numerical diffusion and preserve boundedness of the solution at the interface. Gopala et al. [5] compared different interface capturing method over structured grids and discussed in detail about the merits and demerits of these methods. In the present work, GAMMA scheme due to Jasak and Weller [7] and CUBISTA scheme due to Alves et al. [1] have been implemented into an in house fully coupled Navier-Stokes solver over hybrid unstructured grid. Validation of different schemes has been carried out with the published results of dam break problem and Rayleigh-Taylor instability.

## *1.1 Governing Equations*

The two different fluids are modeled by single fluid formulation wherein the fluids separated by an interface are assumed to be viscous, incompressible and immiscible following the same form of equations accounting conservation of mass and momentum.

**Continuity Equation**

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

**Momentum Transport Equation**

$$\frac{\partial(\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) = -\nabla p + \nabla \cdot \tau + \rho \mathbf{g} \tag{2}$$

where $\rho$ is the density, $\mathbf{u}$ is the velocity, $\tau$ is the stress tensor, $\mathbf{g}$ is the gravitational acceleration and $p$ is the pressure.

In this study, the mixture of fluids is considered as a single continuum. Therefore, we can define a volume fraction $f$ (also called as the color or indicator function) whose value varies between 0 and 1. Values of volume fraction varies from cell to cell, one of the fluid (generally lighter fluid) is recognized with a volume fraction of 0 while the other one (generally heavier fluid) is recognized with unity. The cell through which interface is passing, will be recognized with a value of volume fraction varying in between 0 and 1 (Fig. 1). Thus the fluid properties of interest can be expressed as a function of the volume fraction $f$. The density and viscosity at any location (where the volume fraction is calculated) are evaluated as,

$$\rho = f\rho_1 + (1-f)\rho_2 \tag{3}$$

$$\mu = f\mu_1 + (1-f)\mu_2 \tag{4}$$

**Advection Equation**

The time dependence of $f$ is governed by

$$\frac{\partial f}{\partial t} + \mathbf{u} \cdot \nabla f = 0 \tag{5}$$

where $f$ is volume fraction.

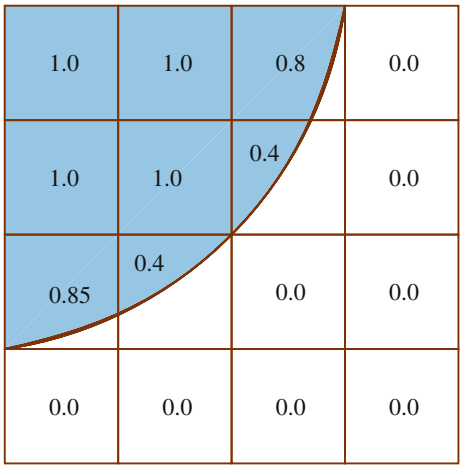


**Fig. 1** Volume fraction field

Using continuity equation, advection equation can be written in divergence form as

$$\frac{\partial f}{\partial t} + \nabla \cdot (\mathbf{u} f) = 0 \tag{6}$$

Following the work of Manik et al. [8] this advection equation is solved along with the conservation equations in a coupled fashion over unstructured grid in a collocated finite volume framework.

## 2 Results and Discussion

The performance evaluation of the two advection schemes has been done by simulating collapse of water column (dam break problem) and Rayleigh-Taylor instability and is shown in the following subsections.

### *2.1 Collapse of Water Column*

The problem definition consists of a rectangular domain of size 5 × 1.6 with a water column 1 × 1 placed at the bottom left corner as shown in Fig. 2. The grid comprises of 3000 hexahedral elements and simulation is carried out with a constant time step of 0.001 until $t$ = 2.5 and the contour plot of volume fraction is shown in Fig. 3. Figure 4 shows the variation of column height and evolution of the front along with the experimental results of Martin and Moyce [9]. It can be seen that the numerical results gives a satisfactory match with the experimental data.

### *2.2 Rayleigh-Taylor Instability*

It is a density driven problem where an interface separates two different fluid, heavier liquid on top is separated from a lighter liquid on the bottom. The

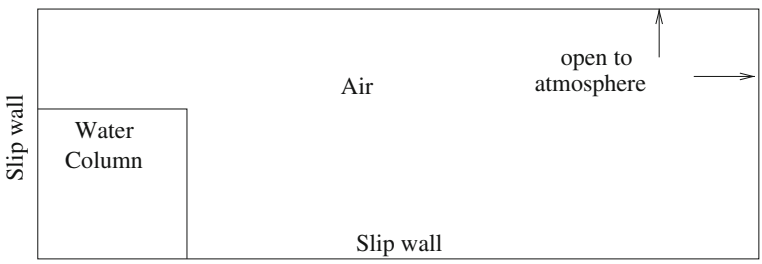


**Fig. 2** Schematic diagram for dam break flow simulation

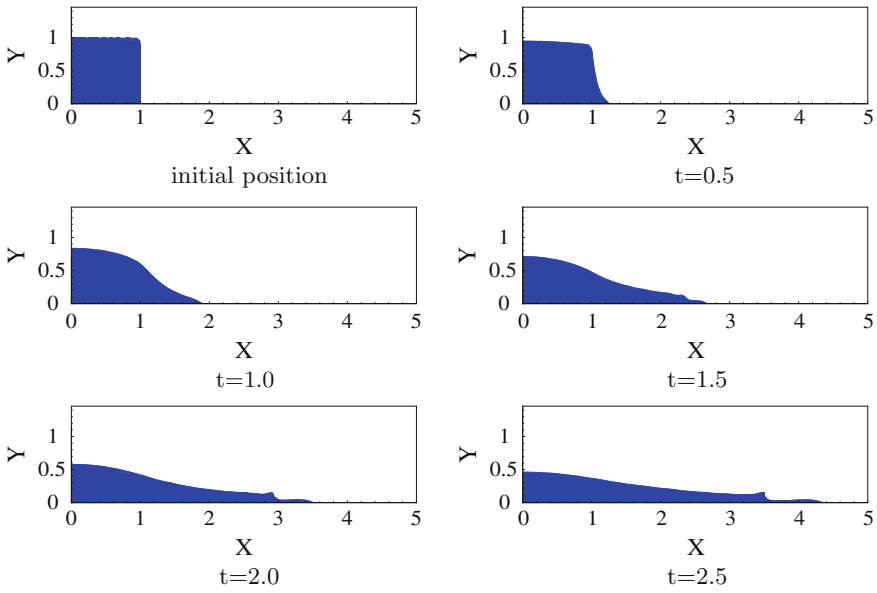


**Fig. 3** Numerical results of dam break problem using GAMMA scheme. Interface corresponds to volume fraction of 0.5

computational domain is a 1 × 4 rectangular domain and is discretised using 6000 hexahedral mesh elements. The top and bottom walls are prescribed with no-slip boundary condition and the remaining boundaries are assumed to be slip walls. The

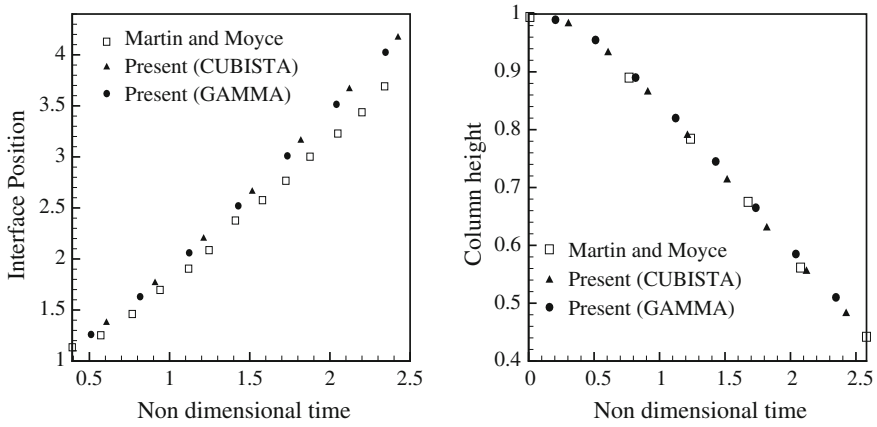


**Fig. 4** Comparision of interface position and column height with experiment results of Martin and Moyce [9]

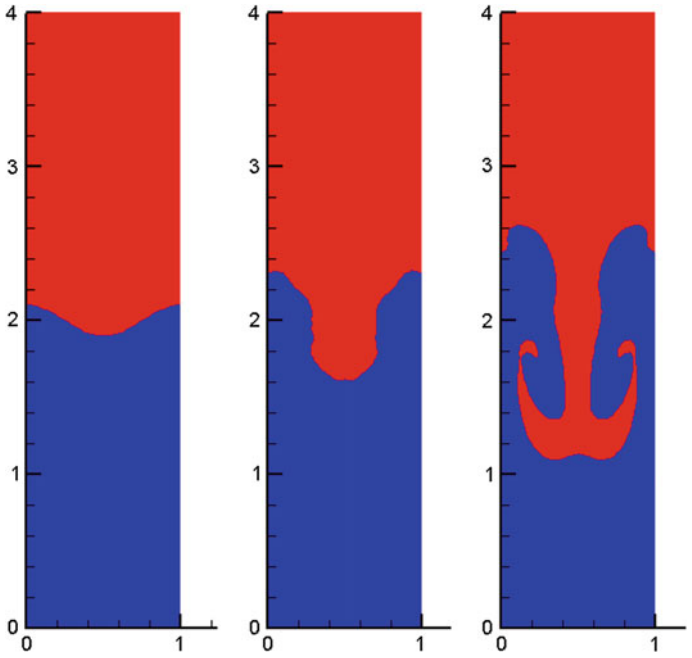


**Fig. 5** Interface position—initially and after few time steps

density and viscosity ratio are taken as 3 and 1 respectively so that the Atwood number (At $= \frac{\rho_{max}-\rho_{min}}{\rho_{max}+\rho_{min}}$) becomes 0.5. The interface is initially perturbed by the following relation.

$$y = 2.0 + 0.1\cos(2\pi x) \tag{7}$$

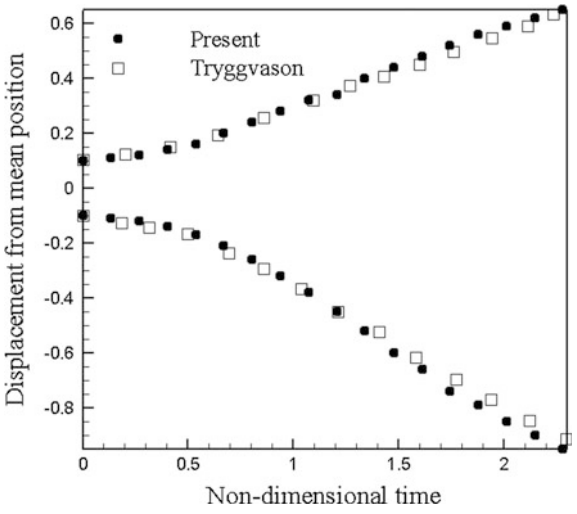


**Fig. 6** Position of rising and falling bubble with time

Temporal evolution of the interface position has been shown in Fig. 5. GAMMA scheme has been used for this simulation with a constant time step of 0.001. The results are compared with the analytical results of Trygggvason [13] as shown in Fig. 6 and found to be in well agreement with the published results.

## 3 Conclusion

In order to simulate interfacial flows, algebraic volume of fluid method has been adopted, wherein advection equation has been solved along with conservation equations in an in-house fully coupled Navier-Stokes solver. Two schemes namely GAMMA due to Jasak et al. [7] and CUBISTA by Alves et al. [1] has been used for discretisation of the advection equation. Validation has been done with the published results of dam break problem [9] and Rayleigh-Taylor instability [13] and a good qualitatively agreement between the numerical and experimental results depicts the potential of the algorithm to handle interfacial flows with large distortions. Further investigation can be done in simulating interfacial flows where surface tension force is dominant and for that we have plans to extend the present solver to simulate these kinds of problems as well.

**Acknowledgments** This study is funded by a grant from the DAE-BRNS, Government of India.

## References

1. Alves, M.A., Oliveira, P.J., Pinho, F.T.: A convergent and universally bounded interpolation scheme for the treatment of advection. Numer. Heat Transf. **49**, 19–42 (2006)
2. Dalal, A., Eswaran, V., Biswas, G.: A finite-volume method for Navier-Stokes equation on unstructured meshes. Numer. Heat Transf. Part B **54**(2), 238–259 (2008)
3. Darwish, M., Moukalled, F.: Convective schemes for capturing interfaces of free-surface flows on unstructured grids. Numer. Heat Transf. **49**, 19–42 (2006)
4. Glimm, J., Grove, J.W., Li, X.L., Oh, W., Sharp, D.H.: A critical analysis of Rayleigh-Taylor growth rates. J. Comput. Phys. **169**, 652–677 (2001)
5. Gopala, V.R., Berend, G.M., Wachem, V.: Volume of fluid methods for immiscible-fluid and free-surface flows. Chem. Eng. J. **141**, 204–221 (2008)
6. Harlow, F., Welch, J.E.: Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface. Phys. Fluids, vol. 12 (1965)
7. Jasak, H., Weller, H.G., Gosman, A.D.: High resolution NVD differencing scheme for arbitrarily unstructured meshes. Int. J. Numer. Meth. Fluids **31**, 431–449 (1999)
8. Manik, J., Parmananda, M., Dalal, A., Natarajan, G.: Development of 3-d Navier-Stokes solver over a hybrid unstructured grid. In: 22nd National and 11th International ISHMT-ASME Heat and Mass Transfer Conference. IIT Kharagpur, India, ASME (May 2013), hMTC1300353
9. Martin, J.C., Moyce, W.J.: An experimental study of collapse of liquid column on the rigid horizontal plan. Math. Phys. Sci. **244**, 312324 (1952)

10. Noh, W.F., Woodward, P.R.: SLIC (simple line interface method). In: van de Vooren, A.I., Zandbergen, P.J. (eds.) Lecture Notes. Physics, vol. 59, pp. 330–340 (1976)
11. Osher, S., Sethian, J.A.: Front propagation with curvature dependent speed: algorithm based on Hamilton-Jacobi formulations. J. Comput. Phys. **79**, 12–49 (1988)
12. Rhie, C.M., Chow, W.L.: Numerical study of the turbulent flow past an airfoil with trailing edge separation. AIAA J. **21**, 1525–1532 (1983)
13. Tryggvason, G.: Numerical simulations of the Rayleigh-Taylor instability. J. Comput. Phys. **75**, 253–282 (1988)
14. Tryggvason, G., Bunner, B., Esmaeeli, A., Juric, D., Al-Rawahi, N., Tauber, W., Han, J., Nas, S., Jan, Y.J.: A front-tracking method for the computations of multiphase flow. J. Comput. Phys. **169**, 701–759 (2001)
15. Tsui, Y., Lin, S., Wu, T.: Flux-blending schemes for interface capture in two-fluid flows. Int. J. Heat Mass Transf. **52**, 5547–5556 (2009)
16. Ubbink, O., Issa, R.: A method for capturing sharp fluid interfaces on arbitrary meshes. J. Comput. Phys. **153**, 26–50 (1999)
17. Youngs, D.L.: Time-dependent multi-material flow with large fluid distortion. In: Morton, K. W., Baines, M.J. (eds.) Numerical Methods for Fluid Dynamics, pp. 273–285 (1982)