International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:2, No:7, 2008

Numerical Simulation of Interfacial Flow with Volume-Of-Fluid Method

Afshin Ahmadi Nadooshan

***Abstract*—**In this article, various models of surface tension force (CSF, CSS and PCIL) for interfacial flows have been applied to dynamic case and the results were compared. We studied the Kelvin-Helmholtz instabilities, which are produced by shear at the interface between two fluids with different physical properties. The velocity inlet is defined as a sinusoidal perturbation. When gravity and surface tension are taking into account, we observe the development of the Instability for a critic value of the difference of velocity of the both fluids. The VOF Model enables to simulate Kelvin-Helmholtz Instability as dynamic case.

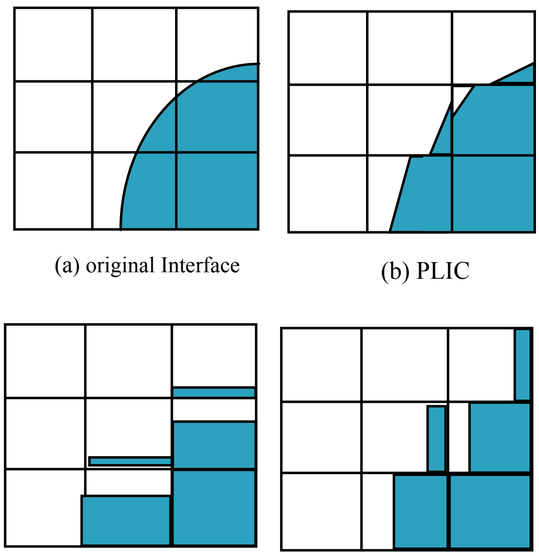
***Keywords*—**Interfacial flow; Incompressible flow; surface tension; Volume-Of-Fluid; Kelvin-Helmholtz.

I. Introduction

ONE of the important subjects that have been considered by fluid mechanics researchers is studying the interfacial flows. Theoretical studies of two phase viscous flows involving free surface instabilities or very strong interface tearing and stretching are difficult to perform. Numerical methods have begun to be used to simulate the flow dynamics of the problem. The numerical methods can be divided into two groups depending on the type of grids used: moving grid or fixed grid [1-11]. Two important approaches of fixed-grid methods, namely the volume-of-fluid and the level-set approaches, are among the most commonly used methods. The volume-of-fluid method [3-11], tracks the volume of each fluid in all cells containing portions of the interface, rather than the interface itself. The VOF method solves a non-diffusive solution of the advection equation, by a geometrically based calculation technique of the void fraction fluxes at the cell faces based on the reconstructed interface. Two early approaches are the SLIC algorithm of Noh and Woodward [3] and the volume-of-fluid algorithm of Hirt and Nichols [4], in which the interface is represented by a piecewise-constant line in each interfacial cell, either vertically or horizontally. A significant improvement of the interface representation was achieved by Youngs [5] by introducing a piecewise-linear method (see Fig. 1). The PLIC method approximate the interface is by a straight line of arbitrary orientation in each cell. Its orientation is found the distribution of one of the fluids in the neighbor cell. Generally, the influence of surface tension is incorporated into the momentum equation following the continuum surface force (CSF) model of Brackbill et al. [12, 13]. For doing so, the local curvature and the interface normal vector have to be calculated. This task is difficult, since the discontinuous void fraction function disallows the application of ordinary discretization schemes. The behavior of each algorithm under surface tension-dominant problems is discussed in the next section. In this work the VOF-PLIC method has been used for tracking surface and exerting the effect of surface tension.

Manuscript received July 20; 2008. This work was supported by Islamic Azad University, Majlessi Branch.

Afshin Ahmadi Nadooshan is with the Mechanical Engineering Department, Islamic Azad University, Majlessi Branch, Isfahan, Iran (phone:+983116643569; e-mail: afshin.ahmadi@gmail.com).



(a) original Interface (b) PLIC

(c) SLIC (Horizontal) (d) SLIC (Vertical)

Fig. 1 Interface Representation

II. Problem Formulation

We consider the unsteady, laminar, incompressible Navier-Stokes equations. A volume-of-fluid method is used to capture the fluid interfaces. It is assumed that the velocity field is continuous across the interface, but there is a pressure jump at the interface due to the presence of the surface tension.

The governing equations describing this problem are:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial F}{\partial t}+u_i\frac{\partial F}{\partial x_i}=0 \tag{2}$$

$$\frac{\partial \rho u_i}{\partial t}+\frac{\partial \rho u_i u_j}{\partial x_j}=-\frac{\partial p}{\partial x_i}+F_v^{st}\cdot\vec{i}+\mu\frac{\partial^2 u_i}{\partial x_j{}^2}+\rho g_i\,. \tag{3}$$

where ρ and μ are the average density and absolute viscosity in a cell, respectively, and they depend on the densities and viscosities of each fluid a the cell. That is:

$$\rho=\rho_2+F(\rho_1-\rho_2) \tag{4}$$

$$\mu=\mu_2+F(\mu_1-\mu_2) \tag{5}$$

Here we use pressure based numerical method, then the Poisson equation needs to be solved. The Poisson equation is:

$$\frac{\partial}{\partial x_i}\left(\frac{\partial p}{\partial x_i}\right)=-\frac{\partial}{\partial x_i}\left[\frac{\partial}{\partial x_j}\left(\rho u_i u_j-2\mu S_{ij}-\frac{F_v^{st}\cdot\vec{i}}{\rho}\right)\right] \tag{6}$$

One of the challenges in numerical simulation of interfacial flows is modeling of the surface tension force. In Piecewise Linear Interface Calculation methods using the Volume of Fluid technique, special techniques are developed to calculate the surface tension forces, which are later converted to a body force. One of attempts in this field, were done by Brackbill et al. [12, 13]. This method, which is called CSF method, reformulates surface tension into an equivalent volume force as follows:

$$\vec{\mathbf{F}}_{\mathbf{v}}{}^{\mathbf{st}}=\sigma\kappa\delta_s\mathbf{n} \tag{7}$$

This body force is added into the momentum equations and so the effect of surface tension is modeled. In this model For CSF model, F^v_{st} is corrected to decrease the intensity of spurious currents by applying a density correction term as follows [14]:

$$\mathbf{F_v}^{\mathbf{st}}=\sigma\kappa\boldsymbol{\delta}_{\mathbf{s}}\mathbf{n}=\sigma\kappa\mathbf{n}\left|\nabla\tilde{F}\right|\frac{\rho(x)}{[\rho]} \tag{8}$$

where [ρ] is the difference between the density of the heavier and the lighter fluids. The density correction term is added to reduce the force in the region with lighter fluid in momentum equation. This dampens the acceleration of the lighter fluid in cells near the interface that contains large amounts of lighter fluid. Another model was presented by Lafaurie et al. in 1999, and is called CSS model [15].

$$\mathbf{F_v}^{\mathbf{st}}=\sigma\nabla\cdot\left(\left|\nabla F\right|\mathbf{I}-\frac{\nabla F\otimes\nabla F}{\left|\nabla F\right|}\right) \tag{9}$$

PCIL model was presented to reduce the amounts of these currents and produce more accurate pressure jump across the interface. This model is based on VOF-PLIC in which surface tension force is obtained by modified CSS and CSF models by introducing a dimensionless parameter H. This parameter is the ratio of the cell face area occupied by heavier fluid to total cell face area [16].

$$\mathbf{F_v}^{\mathbf{st}}=H\sigma\kappa\mathbf{n}\boldsymbol{\delta}_{\mathbf{s}}=H\sigma\kappa\mathbf{n}\frac{\left|\nabla\tilde{F}\right|}{[F]} \tag{10}$$

$$\mathbf{F_v}^{\mathbf{st}}=-H\nabla.T=H\sigma\nabla\cdot\left(\left|\nabla F\right|\mathbf{I}-\frac{\nabla F\otimes\nabla F}{\left|\nabla F\right|}\right) \tag{11}$$

III. Numerical Method

A computer code is used to simulate the flow. The code is finite volume based and uses SIMPLEC algorithm. The code was modified to solve interfacial flows. The VOF_PLIC method was used for interface tracking and the CSF, CSS and PCIL models were exerted into the code to calculate the surface tension force.

IV. Results

Kelvin–Helmholtz instability can occur when velocity shear is present within a continuous fluid or when there is sufficient velocity difference across the interface between two fluids. One example is a wind blowing over a water surface, where the wind causes the relative motion between the stratified layers. The instability will manifest itself in the form of waves being generated on the water surface. The theory can be used to predict the onset of instability and transition to turbulent flow in fluids of different densities moving at various speeds. Hermann von Helmholtz studied the dynamics of two fluids of different densities when a small disturbance such as a wave is introduced at the boundary connecting the fluids. For some short enough wavelengths, if surface tension can be ignored, two fluids in parallel motion with different velocities and densities will yield an interface that is unstable for all speeds. The existence of surface tension stabilizes the short wavelength instability however, and theory then predicts stability until a velocity threshold is reached. The theory with surface tension included broadly predicts the onset of wave formation in the important case of wind over water. For a continuously-varying distribution of density and velocity, (with the lighter layers uppermost, so the fluid is RT-stable), the onset of the KH instability is given by a suitably-defined Richardson number (Ri). Typically the layer is unstable for Ri < 0.25. These effects are quite common in cloud layers. Also the study of this instability becomes applicable to inertial confinement fusion and the plasma–beryllium interface (see Fig. 2).

Fig. 2 Kelvin Helmholtz instability rendered visible by clouds over Mount Duval in Australia

We consider the flow of 2 fluids in 2 horizontal parallel infinite streams with different densities and different velocities. We consider a rectangle of 1 meters length and 1 meters height. In fluent, the edges AB and CD are defined as wall. The edge BC is defined as a outflow outlet. Finally, the edge AD is defined as a velocity inlet. For this study, we have defined two grids. The number of elements is the same for both: 100 elements on the x direction and 100 elements on the y direction (see Fig. 3).

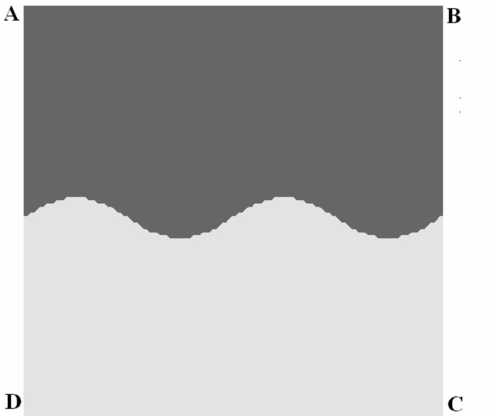


Fig. 3 Initial Geometry

We choose this model for study our 2 phase's problem. The physical restrictions of the VOF model are the following: the flow must be incompressible, Heat transfer is not available, species mixing and reacting flow cannot be modeled.

Therefore, it seems to be the best model for our study because we consider two incompressible flows, non miscible. Moreover, we choose the laminar solver in order to observe the development of the instability. Kelvin Helmholtz instability is difficult to visualize, therefore we choose two fluids with similar density. The parameters are given in Table I.

TABLE I
PHYSICAL PARAMETERS

Fluid Name	Viscosity (Kg/m s)	Density (Kg/m^3)	Surface tension (N/m)
Water	0.001	1000	0.073
Fuel Oil	0.05	900	

The velocity for fuel oil is 3m/s and the velocity for water is -3m/s. We use the regular grid because we do not know the height of the vortexes and we take a high velocity difference because we only want to visualize the instability. You can see the evolution of the instability in the time in Figs. (4) and (5). The first vortex which can be observed is not Kelvin-Helmholtz instability. Indeed, its form and height is not similar to following vortexes. Therefore, we can think that this first perturbation is linked to the VOF method.

V. Conclusion

In this article, various models of surface tension force for incompressible interfacial flows have been applied to dynamic case and the results were compared and reviewed. Kelvin-Helmholtz Instability is simulated as dynamic case. The VOF Model enables to simulate Kelvin-Helmholtz Instability. It is necessary to put a perturbation in order to observe the development of the Instability. The velocity inlet is defined as a sinusoidal perturbation. When gravity and surface tension are taking into account, we observe the development of the Instability for a critic value of the difference of velocity of the both fluids. This result is similarly to the theory of 2D Kelvin-Helmholtz Instability.

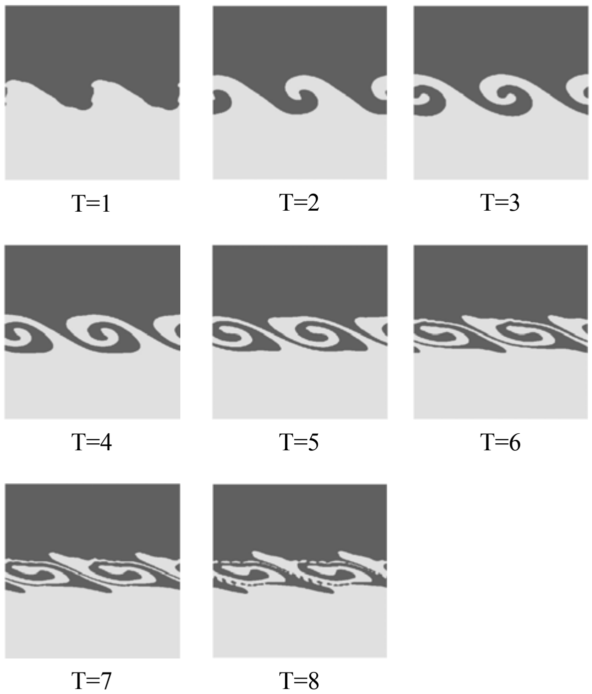


Fig. 4 Profile of density with CSF model in the time

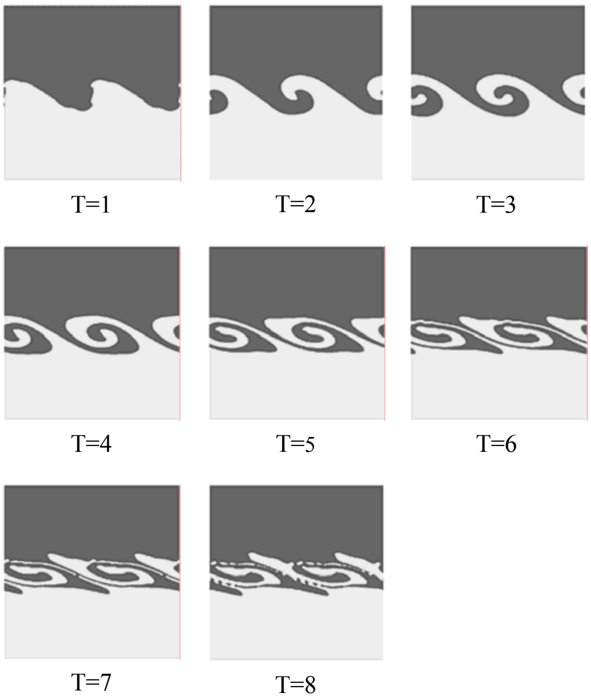


Fig. 5 Profile of density with CSS model in the time

REFERENCES

[1] D. Gerlach, G. Tomar, G. Biswas and F. Durst ,"Comparison of volume-of-fluid methods for surface tension-dominant two-phase flows", International Journal of Heat and Mass Transfer, 49, 740–754, 2006

[2] Osher, S., and Sethian, J., "A fronts propagating with curvature-dependent speed: algorithms based on Hamilton-Jacobi formulations", J. Comp. Phys. 79(1), 12(1988)

[3] Noh, W.F. and Woodward, P.R., "Slic (simple line interface method)", in Lecture Notes in Physics, 59, 1976.

[4] Hirt, C.W. and Nichols, B.D., "Volume of fluid (VOF) method for the dynamics of free boundaries", J. Comp. Phys., 39, 201-225, 1981.

[5] Youngs, D.L. “Time-dependent multi-material flow with large fluid distribution”, in Numerical methods for fluid dynamics, Morton and Norman,Editor,187-221,1996

[6] Ashgriz, N and Poo, J.Y., “FLAIR: Flux Line-segment model for advection and interface reconstruction”. J. Comp. Phys. , 93,449-468, 1991.

[7] Rider, W.J. and Kothe D.B., “Reconstruction volume tracking”, J. Comp. Phys., 14, 112, 1998.

[8] Pilliod J.E. and E.G. Puckett, "Second-order accurate volume-of-fluid algorithms for tracking material interfaces”, Lawrence Berkley Lab. Tech. Report, No.LBNL-40744, 1997.

[9] Welch S.W.J., T. Rachidi, Numerical computation of .lm boiling including conjugated heat transfer, Numer. Heat Transfer, Part B 42 (2002) 35–53.

[10] Agarwal D.K., S.W.J. Welch, G. Biswas, F. Durst, Planar simulation of bubble growth in .lm boiling in near-critical water using a variant of the VOF method, J. Heat Transfer (ASME) 126 (2004) 329–338.

[11] Renardy Y., M. Renardy, "PROST: a parabolic reconstruction of surface tension for the volume-of-fluid method", J. Comp. Phys. 183 (2002) 400–421.

[12] Jamet D., D. Torres, J.U. Brackbill, On the theory and computation of surface tension: the elimination of parasitic currents through energy conservation in the second-gradient method, J. Comp. Phys. 182 (2002) 262–276.

[13] Brackbill J.U., D.B. Kothe, C. Zemach, A continuum method for modeling surface tension, J. Comp. Phys. 100 (1992) 335–354.

[14] Kothe, D.B., W.J. Rider, S.J. Mosso, and J.S. Brock, “Volume tracking of interfaces having surface tension in two and three dimensions” AIAA 96-0859, 1996.

[15] Lafaurie B., C. Nardone, R. Scardovelli, S. Zaleski, G. Zanetti, Modeling merging and fragmentation in multiphase .ows with SURFER, J. Comp. Phys. 113 (1994) 134–147.

[16] Shirani, E., Ashgriz, N. and Mostaghimi, J., “Interface pressure calculation based on conservative of momentum for front tracking methods”, J. Comp. Phys., 203, 153-175, 2005.