**Proceedings of the 12th Asian Congress of Fluid Mechanics**
**18-21 August 2008, Daejeon, Korea**

# Numerical Simulation of Flow Instabilities During the Rise of a Bubble in a Viscous Liquid

Mohammad P-Fard*[1], Mehran M.Farhangi*[2], Hossein Moin*[3]
Department of Mechanical Engineering, Ferdowsi University of Mashhad,
*[1]mpfard@um.ac.ir, *[2]farhangi.mehran@gmail.com, *[3] h.moin.jnr@gmail.com

**Abstract:** In this paper, the flow instabilities during the rise of a single bubble in a narrow vertical tube are studied using a transient 2D/axisymmetric model. These instabilities include the oscillation of the bubble shape and formation of a wake behind it. In the model, the Navier-Stokes equations in addition to an advection equation for liquid volume fraction are solved. A modified Volume-of-Fluid (VOF) technique based on Youngs' algorithm is used to track the liquid/gas interface. The numerical results are validated by a comparison with available measurements. The effect of different parameters such as tube and bubble diameters, and liquid properties are investigated.

## 1. Introduction

The dynamic behavior of two-phase flows is of great importance in various processes ranging from engineering applications to environmental phenomena. The presence of air bubbles in hydrodynamic systems often reveals many undesirable effects such as early erosion, loss of efficiency or flow irregularities. Bubble oscillations are complicated phenomena that include the bubble rising trajectory and shape instabilities as well as the associated velocity and pressure fluctuations.

If a large bubble is confined in a narrow tube with a comparable cross-sectional diameter, the bubble will rise along the tube centerline, and path instabilities that might occur in unbounded domain will not appear. The problem seems to become somewhat simpler without trajectory oscillations and is therefore often neglected by researchers. Typically, for a large bubble in a narrow tube, a 'slug flow' will develop. Most studies on slug flows have focused on terminal velocity, steady shape and drag force. In a recent study [2], a universal correlation for the rise velocity of a long gas bubble in stagnant fluids contained in a vertical tube was obtained based on data collected from published literature. The velocity field in the liquid around the bubble has been investigated using Particle Image Velocimetry (PIV) by some researchers [3-4]. However, due to experimental difficulties, velocity profiles in the gas phase were seldom available. In the case of numerical simulations, the momentum equations and hence flow calculations were often ignored in the gas phase due to the large density ratio between liquid and gas [5]. Polonsky et al. [4] reported the oscillatory motion of the bubble bottom for a long gas bubble rising in a vertical tube while the nose of the bubble retained its shape. The amplitude of oscillations was found to increase with the bubble length, while the frequency remained constant.

In this study, the flow instabilities during the rise of a single bubble in a narrow vertical tube are simulated using a transient 2D/axisymmetric model. A modified Volume-of-Fluid (VOF) technique based on Youngs' algorithm is used to track the bubble deformation. To validate the model, numerical results are compared with those of the experiments for terminal rise velocity and bubble shape [1]. The velocity fields within the bubble and in the surrounding liquid are also examined and compared with those of the experiments [3] and other reported numerical results [5]. The effect of different parameters such as surface tension and viscosity of liquid are also investigated.

## 2. NUMERICAL METHOD

The main issue regarding the developed model is the advection of the bubble interface using VOF method. In this section, we present a brief account of the numerical method. The flow governing equations are:

$$\vec{\nabla} \cdot \vec{V} = 0 \tag{1}$$

$$\frac{\partial \vec{V}}{\partial t} + \vec{\nabla}(\vec{V}\vec{V}) = -\frac{1}{\rho}\vec{\nabla} p + \frac{1}{\rho}\vec{\nabla} \cdot \vec{\tau} + \frac{1}{\rho}\vec{F}_b \tag{2}$$

where $\vec{V}$ is the velocity vector, p is the pressure and $\vec{F}_b$ represents body forces acting on the fluid. The bubble interface is advected using VOF method by means of a scalar field $f$ whose value is unity in the liquid phase and zero in the gas. When a cell is partially filled with liquid, $f$ will have a value between zero and one. The discontinuity in $f$ is propagating through the computational domain according to:

$$\frac{df}{dt} = \frac{\partial f}{\partial t} + \vec{V} \cdot \vec{\nabla} f = 0 \tag{3}$$

For the advection of volume fraction $f$ based on Eq. 3, different methods have been developed such as SLIC, Hirt-Nichols and Youngs' PLIC [6]. The reported literature on the simulation of free-surface flows reveals that Hirt-Nichols method has been used by many researchers. In this study, however, we used Youngs' method [6], which is a more accurate technique. Assuming the initial distribution of $f$ to be given, velocity and pressure are calculated in each time step by the following procedure. The $f$ advection begins by defining an intermediate value of $f$,

$$\tilde{f} = f^n - \delta t \vec{\nabla} \cdot (\vec{V} f^n) \tag{4}$$

Then it is completed with a "divergence correction"

$$f^{n+1} = \tilde{f} + \delta t (\vec{\nabla} \cdot \vec{V}) f^n \tag{5}$$

A single set of equations is solved for both phases, therefore, density and viscosity of the mixture are calculated according to: $\rho = f\rho_l + (1-f)\rho_g$ and $\mu = f\mu_l + (1-f)\mu_g$, where subscripts $l$ and $g$ denote the liquid and gas, respectively. New velocity field is calculated according to the two-step time projection method as follows [6, 7]

$$\frac{\tilde{\vec{V}} - \vec{V}^n}{\delta t} = -\vec{\nabla} \cdot (\vec{V}\vec{V})^n + \frac{1}{\rho^n} \vec{\nabla} \cdot \vec{\tau}^n + \vec{g}^n + \frac{1}{\rho^n} \vec{F}_b^{\,n} \tag{6}$$

$$\vec{\nabla} \cdot [\frac{1}{\rho^n} \vec{\nabla} p^{n+1}] = \frac{\vec{\nabla} \cdot \tilde{\vec{V}}}{\delta t} \tag{7}$$

$$\frac{\vec{V}^{n+1} - \tilde{\vec{V}}}{\delta t} = -\frac{1}{\rho^n} \vec{\nabla} p^{n+1} \tag{8}$$

The continuum surface force (CSF) method [6, 7] is used to model surface tension as a body force ( $\vec{F}_b$ ) hat acts only on interfacial cells. Pressure Poisson equation is then solved to obtain the pressure field (Eq. 7), Next, new time velocities are calculated from Eq.(8) by considering the pressure field implicitly.

## 3. RESULT AND DISCUSSION

To validate the model, the results of simulations for terminal rise velocity and bubble shape are compared with those of the experiments. The measured data performed by Grace [1] for air bubbles in water is given as a diagram shown in Figure 1. The results of the model, shown in the same Figure, are predicted in the same region where observed by experiments. As seen from the figure, increasing the bubble diameter increases the rise velocity up to a certain limit after which the bubble starts to oscillate. In this regime, the rise velocity remains nearly constant. Further increase of the bubble diameter changes the deformation behavior to the spherical cap regime where the rise velocity again increases with diameter. The default material properties used in the simulations are given in Table 1. Bubbles with diameter ranged from 0.8 mm to 10 mm are simulated; larger ones tend to break up before they reach their terminal velocity. As it can be seen from Figure 1, for the bubbles smaller than the 0.5 mm, there is an increasing deviation of simulated to measured velocities, which occurs mainly because of the so-called parasitic currents. These currents are due to inaccuracies in the calculation of surface tension forces, in particular because of errors in the calculation of the interfacial normal vector and curvature.

Next, we studied the flow instabilities that occur during the rise of a bubble (20 mm in dia.) in a narrow vertical tube. Figure 2 displays the shape oscillation, velocity distribution and flow streamlines for this case. The wake formation and flow recirculation behind the bubble are clearly visible in the
figure. Driven by the buoyancy force, the bubble rises rapidly after its release. It is deformed from the initial spherical shape to the final bullet-like configuration. The bottom of the bubble moves rapidly upward and develops into a concave shape. It rebounds downward immediately into a convex shape. This up-and-down oscillatory movement of the bubble bottom continues as the bubble rises with decreasing amplitude. The top of the bubble (or bubble nose), on the contrary, remains a spherical cap shape with very little deformation as it ascends.

A better view of the bubble rise and oscillation is shown in Fig. 3 where a 3D view of the phenomenon for another case (ethylene as liquid) is displayed. A detailed quantitative comparison of the model predictions. predictions and experiments [3] is given in Fig. 4 for an axial location just above the bubble; the two results agree well in this region. The axial velocity profile in the fully developed falling film beside a rising single bubble is shown in Fig. 5 along with the theoretical profile developed by Brown [8] and the numerical predictions obtained by Bugg et al. [5]. The analysis of Brown [8] and the numerical model predict a gas–liquid interface location of r/R=0.75 and r/R =0.76, respectively. Finally the effect of different parameters on the oscillatory behaviors of bubble velocity and shape are investigated. Figures 6 and 7 show the variation of the rise velocity versus time for water and ethylene, respectively, where $d_b/D$ was 0.88. It can be seen that for the case of ethylene with a higher viscosity, the amplitude of oscillation is decreased as the bubble rises in the narrow tube. This implies that liquid viscosity has a damping effect on the oscillations, as was expected. The gas viscosity plays a negligible role in the bubble motion and is; therefore, can be ignored.

## Conclusion

In this paper, an axisymmetric VOF method was used to simulate the dynamics of a single gas bubble rising in a narrow vertical tube. The model was validated by a comparison between numerical results with available measurements for the bubble deformation and velocity during its rise in a liquid. The velocity field have been investigated and analyzed; the results obtained in the present study are in close agreement with experimental results.

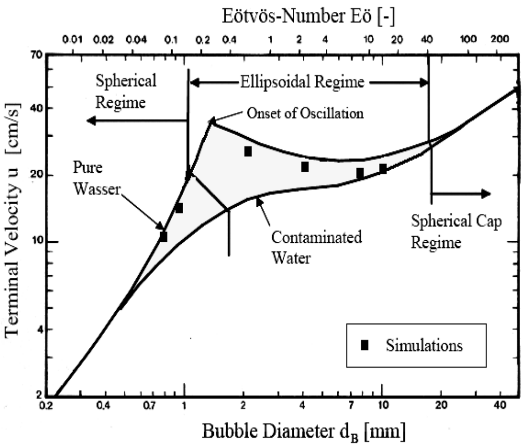


Fig 1. A comparison between the results of simulations with those of the experiments [1] for terminal rise velocity against initial bubble diameter. Based on the experiments, the rise velocity should be located in the region surrounded by the lines.

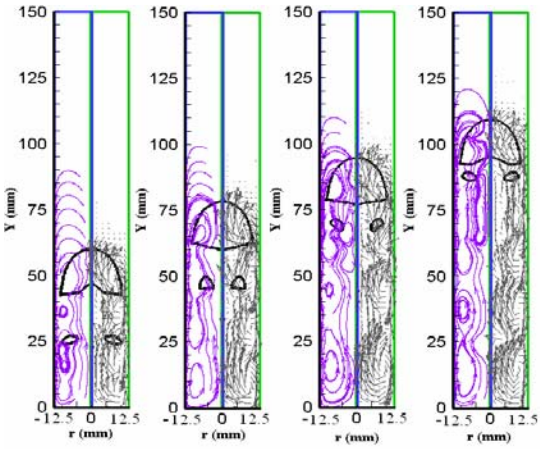


Fig 2. Shape oscillation, velocity distribution, and flow streamlines during the rise of a 20 mm dia. bubble in a narrow vertical tube.

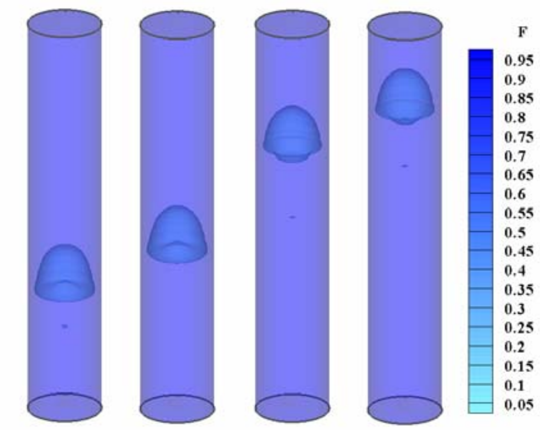


Fig 3. 3D shape oscillation during the rise of a 20 mm dia. bubble in a narrow vertical tube for ethylene.

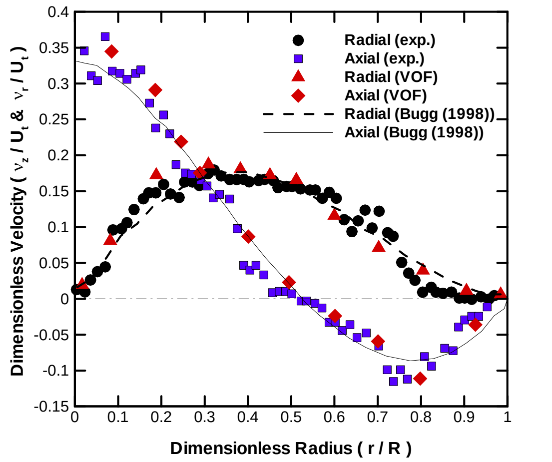


Fig 4. The axial and radial components of velocity at z/D= -0.111 (i.e. just above the bubble). The PIV measurements [16] are compared to the numerical results by Bugg et al. [17] and VOF method.

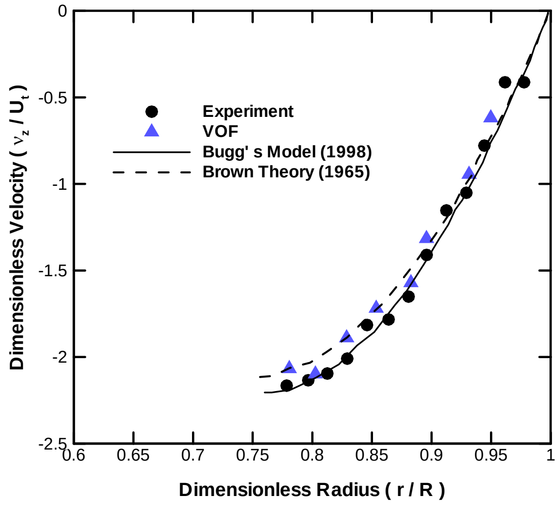


Fig 5. Velocity profile in the fully developed falling film beside a rising single bubble.

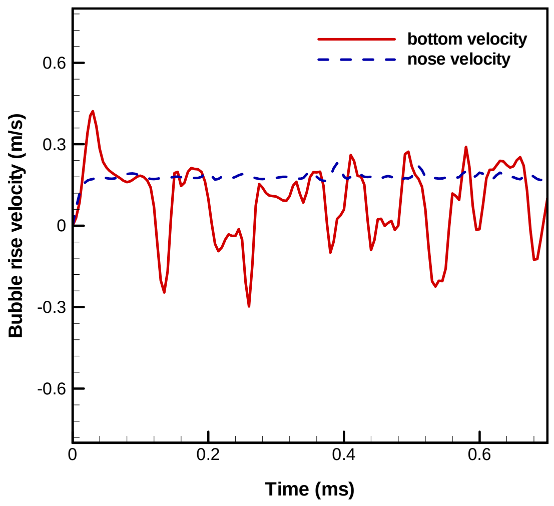


Fig 6. Bubble rise velocity versus time at the nose and bottom for water ($d_b/D$ = 0.88).

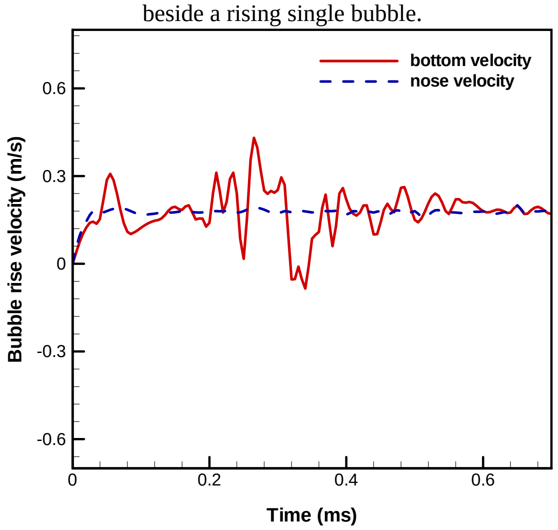


Fig 7. Bubble rise velocity versus time at the nose and bottom for ethylene ($d_b/D$ = 0.88).

**Table 1.** Material properties.

| **properties** | **water** | **air** |
|---|---|---|
| density | $\rho_l$ =998.2 kg/m$^3$ | $\rho_a$=1.1222 kg/m$^3$ |
| viscosity | $\mu_l$ =1002×10$^{-6}$ kg/(m.s) | $\mu_a$ =18.24×10-6 kg/(m.s) |
| surface tension | $\gamma$ =0.073 N/m | |

## References

[1] Grace JR. Trans ICHemE. 1973:51:116-120.

[2] Viana, F., Pardo, R., Yanez, R., Trallero, J.L. Joseph, D.D., 2003, "Universal correlation for the rise velocity of long gas bubbles in round pipes", J. Fluid Mech. Vol.494, pp. 379–398.

[3] Bugg, J.D., Saad, G.A., 2002, " The velocity field around a Taylor bubble rising in a stagnant viscous fluid: numerical and experimental results", Int. J. Multiphase Flow, Vol. 28, pp. 791–803.

[4] Polonsky, S., Shemer, L., Barnea, D., 1999, "The relation between the Taylor bubble motion and the velocity field ahead of it", Int. J. Multiphase Flow, Vol. 25, pp. 957–975.Polonsky, L., Shemer, D. Barnea, Int. J. Multiphase Flow 25 (1999) 957-975.

[5] Bugg, J.D., Mack, K., Rezkallah, K.S., 1998, "A numerical model of Taylor bubbles rising through stagnant liquids in vertical tubes", Int. J. Multiphase Flow, Vol. 24, pp. 271–281.

[6] Passandideh-Fard, M., and Roohi, E., 2008 (in print), Transient Simulations of Cavitating Flows using a Modified Volume-of-Fluid (VOF) Technique, Int. J. of Comput. Fluid Dynamics.

[7] Bussmann M., Mosthghimi J., and Chandra S., 1999 "On a Three-Dimensional Volume Tracking Model of Droplet Impact", Phys. Fluid, Vol. 11, p. 1406.

[8] Brown, R.A.S., 1965. The mechanics of large gas bubbles in tubes I. Bubble velocities in stagnant liquids. Can. J. Chem. Eng, Vol. 43, pp. 217–223.