Numerical Simulations of a Train of Air Bubbles Rising Through Stagnant Water

Hong Xu, Chokri Guetari

ANSYS INC.

Abstract

Transient numerical simulations of the rise of a train of gas bubbles in a liquid are presented. Computational Fluid Dynamics is used to simulate the axisymmetric two-dimensional flow in and around a set of bubbles separated by a fixed distance. The homogeneous multiphase model available in CFX was used. This model allows the simulation of the flow inside and outside the bubble with no additional empirical correlations for drag or slip velocity. The effect of surface tension is included through a continuous surface model by Brackbill et al.

Three bubble diameters were considered: 1 mm, 5mm and 40 mm. The first bubble is expected to maintain a spherical shape, the second is expected to become elliptical and the third is in the range of the spherical cap regime. The CFX results showed that the predicted bubble shape agreed well with data from Clift et al.

Terminal velocity as well as flow patterns and bubble shape are compared to available data.

Introduction

Bubbly flows are central to many industrial processes. Heat transfer through boiling is the preferred mode in most power plants. Bubble-driven circulation systems are used in metal processing operations such as steel making, ladle metallurgy, and the secondary refining of aluminum and copper. Similarly, many natural processes involve bubbles, such as propagation of sound in the ocean. Bubbles also play a major role in the interactions of the oceans with the atmosphere. Understanding the physics and details of bubbly flows is therefore of major technological as well as scientific interest.

Experimental and analytical studies relating to bubble hydrodynamics date back to the early 1900s. Recently, advanced measurement techniques and computational tools have allowed a much more detailed examination of bubble motion. The behavior of a single clean bubble is reasonably well understood. If the bubble is small, so that surface tension effects are high, or the bubble is in a very viscous liquid, it remains spherical and rises with a steady state velocity. As the bubble becomes larger, it deforms first into an ellipsoidal shape and eventually into a spherical cap shape. If the bubble is large enough, as it rises in a quiescent liquid it will deform and eventually break up into two bubbles. The breakup process is controlled by a balance between inertial and surface tension forces. As the inertial forces overcome the interfacial forces, breakup occurs. The Weber number is usually used to gauge the relative magnitude of these two effects.

The analysis of a single bubble rising in a vast pool of liquid is the simplest bubbly flow case. In this paper, an axisymmetric two-dimensional train of bubbles rising through a quiescent liquid is analyzed. The details of the flow field, bubble shape, and terminal velocity will be determined based on the CFX simulations.

Procedure

As a gas bubble moves through a liquid due to buoyancy, it develops a flow pattern both inside and outside the bubble. Computational Fluid Dynamics (CFD) can be used to elucidate the details of the flow field, the pressure distribution and to compute quantities that are important in industrial applications, like the terminal velocity and interfacial area.

The domains used for the CFD analyses are described in Figure 1. The top and bottom surfaces are treated as a periodic pair, which implies that the simulation is for a row of bubbles separated by a distance of 20 bubble diameters. Therefore, the results of these analyses will not correspond exactly to the motion of a single bubble in an infinite domain.

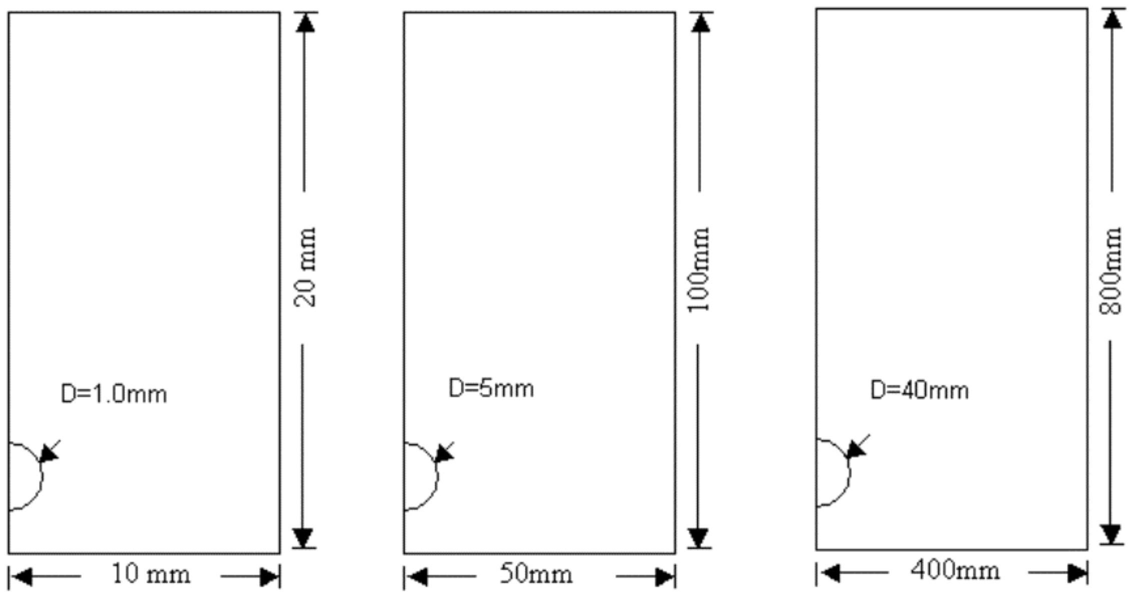

Figure 1. Dimensions used for the CFD model

Three bubble diameters were considered in this paper: 1 mm, 5 mm, and 40 mm. Air properties where used for the gas phase and water properties for the liquid phase. Table 1 summarizes the values used in this paper.

Table 1. Fluid properties used.

	Air	Water
Density (kg/m^3)	1.185	997.0
Viscosity (cP)	1.831*10^{-2}	0.8899
Surface Tension	0.074 dynes	

Analysis

Analytical Calculation of the Terminal Velocity

Surface tension acts to minimize surface area and forces small bubbles, on the order of a millimeter, to remain or become nearly spherical and to behave much like rigid spherical particles. The terminal velocity for a bubble of diameter 1.0 mm can be determined by writing a force balance that involves gravity, buoyancy, and drag on a perfect sphere.

The rise of a bubble is due to gravity and the difference in density between the gas and the liquid. A momentum balance on the bubble is simply $\Sigma F_x = ma_x$, where the sum of forces includes gravity acting on the bubble (F_g), buoyant force (F_b), the drag exerted by the fluid (F_D), and a lift force which is usually neglected. Additionally, a virtual or added mass force contributes to the balance when the bubble is accelerating and causes surrounding fluid to move. However, the virtual mass force goes to zero as a bubble reaches its terminal velocity and its acceleration goes to zero.

The momentum balance is (1):

$$\frac{g(\rho_l - \rho_g)\pi d^3}{6} - \frac{C_D \rho_g \pi d^2 V^2}{8} = \frac{\pi d^3(\rho_l + \rho_g/2)}{6}\frac{dV}{dt} \qquad [1]$$

Initially, the bubble is stationary and the drag force is zero. As the bubble starts to rise and accelerates, the drag force increases, which in turn reduces the acceleration. This process continues until the acceleration drops to zero, at which time the bubble rises at a constant velocity as the drag, gravity and buoyancy are balanced. By setting the acceleration terms equal to zero in eq. [1], the terminal velocity, V_t, can be written as:

$$V_t = \left[\frac{4\, g\, (\rho_l - \rho_g)\, d}{3\, \rho_l\, C_D} \right]^{1/2} \qquad [2]$$

The drag coefficient, which depends on the velocity, can be computed using a variety of expressions. Because of surface tension forces, very small drops and bubbles are nearly spherical and would behave much like rigid particles. However, larger fluid drops or bubbles may experience a considerably different behavior, because the shear stress on the drop surface is transmitted to the fluid inside the drop, which in turn results in circulation of the internal fluid. The internal circulation dissipates energy, which is extracted from the energy of the bubble motions and is equivalent to an additional drag force. For Reynolds numbers between 1 and 500, Rivikind and Ryskind proposed the following equation for the drag coefficient for spherical drops and bubbles (2):

$$C_D = \frac{1}{\kappa + 1}\left[\kappa\left(\frac{24}{\mathrm{Re}} + \frac{4}{\mathrm{Re}^{1/3}}\right) + \frac{14.9}{\mathrm{Re}^{0.78}}\right] \qquad [3]$$
$$\kappa = \mu_d / \mu_c$$

Where μ_d and μ_c are the viscosities of gas and liquid, respectively. Introducing eq. [3] into eq. [2], and based on the fluid properties specified in Table 1, the terminal velocity for the case of 1.0 mm bubble is 0.265 m/s and the value of Re is 300.

Numerical Methods

Since both gas and liquid were assumed to be continuous phases, the homogeneous multiphase model, i.e. a common fields for velocity, pressure and turbulence quantities are shared by all fluids, is used to simulate the two-phase flow. For a given transport process, the homogeneous model assumes that the transported quantities (with the exception of volume fraction) for that process are the same for all phases.

$$\Phi_\alpha = \Phi \quad 1 \le \alpha \le N_p$$

Hence, the individual phase continuity equation can be solved to determine the volume fractions, but the individual transport equations can be summed over all phases to give a single transport equation for Φ:

$$\frac{\partial}{\partial t}(\rho\phi) + \nabla \bullet (\rho U \phi - \Gamma \nabla \phi) = S$$

where:

$$\rho = \sum_{\alpha=1}^{Np} r_\alpha \rho_\alpha, \qquad U = \frac{1}{\rho}\sum_{\alpha=1}^{Np} r_\alpha \rho_\alpha U_\alpha, \qquad \Gamma = \sum_{\alpha=1}^{Np} r_\alpha \Gamma_\alpha$$

In particular, the homogeneous model for momentum transport assumes $U_\alpha = U$, so the momentum equation can be simplified as follows:

$$\frac{\partial}{\partial t}(\rho U) + \nabla \bullet (\rho U \otimes U - \mu(\nabla U + (\nabla U)^T)) = B - \nabla P$$

where

$$\rho = \sum_{\alpha=1}^{Np} r_\alpha \rho_\alpha, \qquad \mu = \sum_{\alpha=1}^{Np} r_\alpha \mu_\alpha$$

In addition, the surface tension can be included as a volumetric force at the interface of the two fluids. This model is based on the work by Brackbill et al [4].

The governing equations for this model were solved using the commercial CFD code CFX.

Analysis Results & Discussion

The terminal velocities predicted by CFX compared to values extracted from Figure 2 are summarized in Table 2.

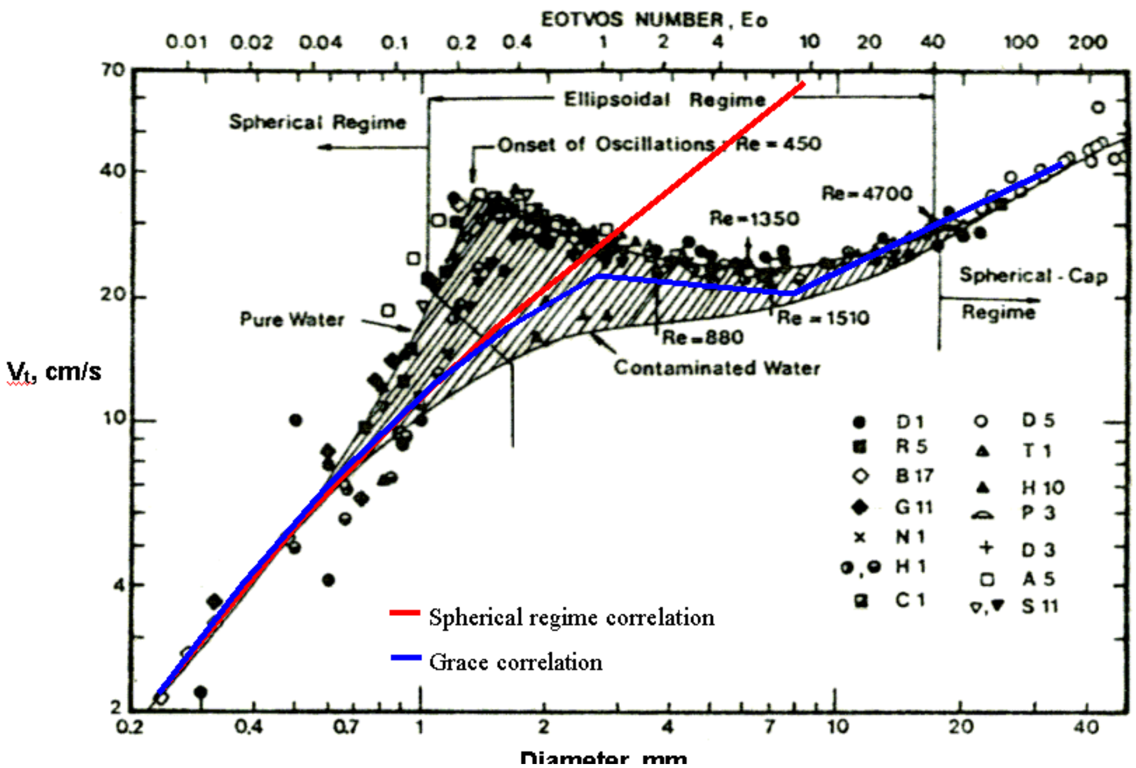

Figure 2. Terminal velocity as function of diameter. From Grace & Weber, 1982.

Based on Figure 2, bubbles on the order of 1 mm or less in diameter should remain spherical in shape as they rise in a quiescent fluid. Bubbles of diameters ranging between 1mm and 20 mm should deform and become ellipsoidal. If the bubble diameter is large enough, usually greater than 20 mm, it tends to deform into what is known as a spherical-cap.

Figures 3, 4, and 5 display the velocity of the bubble based on the movement of the top and bottom surfaces of the interface, Max Vel and Min Vel, respectively.

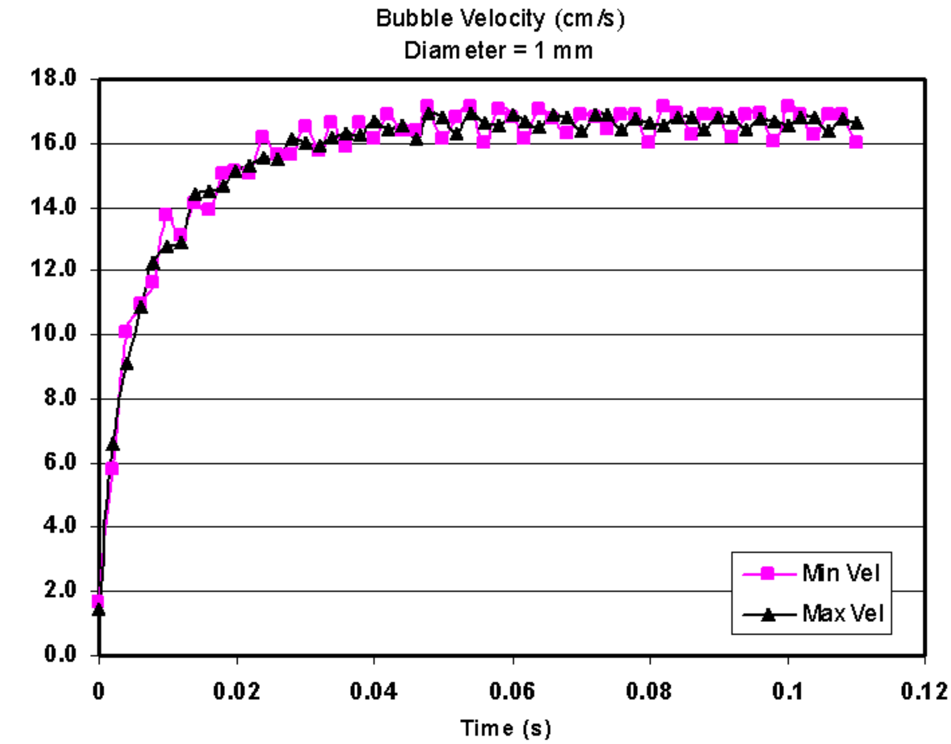

Figure 3. Bubble velocity as a function of time, 1 mm diameter

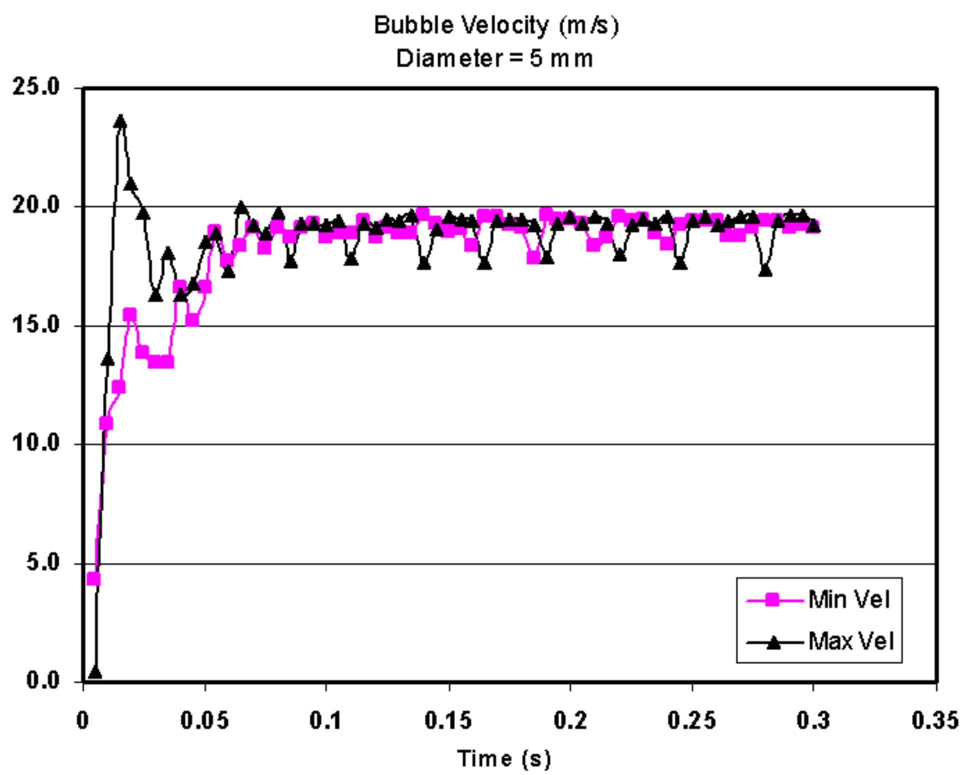

Figure 4. Bubble velocity as a function of time, 5 mm diameter

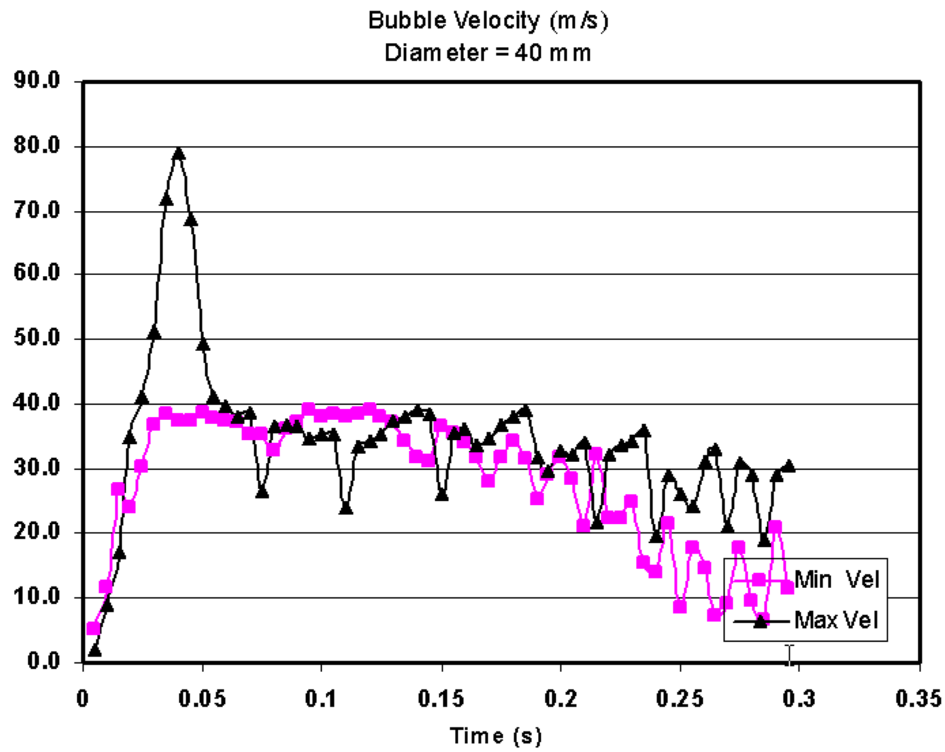

Figure 5. Bubble velocity as a function of time, 40 mm diameter

Figure 6 shows the volume fraction distribution as a function of time for the case of a 1mm bubble. The shape of the bubble remains spherical through out the simulation, which is consistent with Figure 2. In addition, the terminal velocity predicted by the CFD analysis is in perfect agreement with the value read from Figure 1. It is also important to note that the velocities at the top and bottom of the bubble are closely matched.

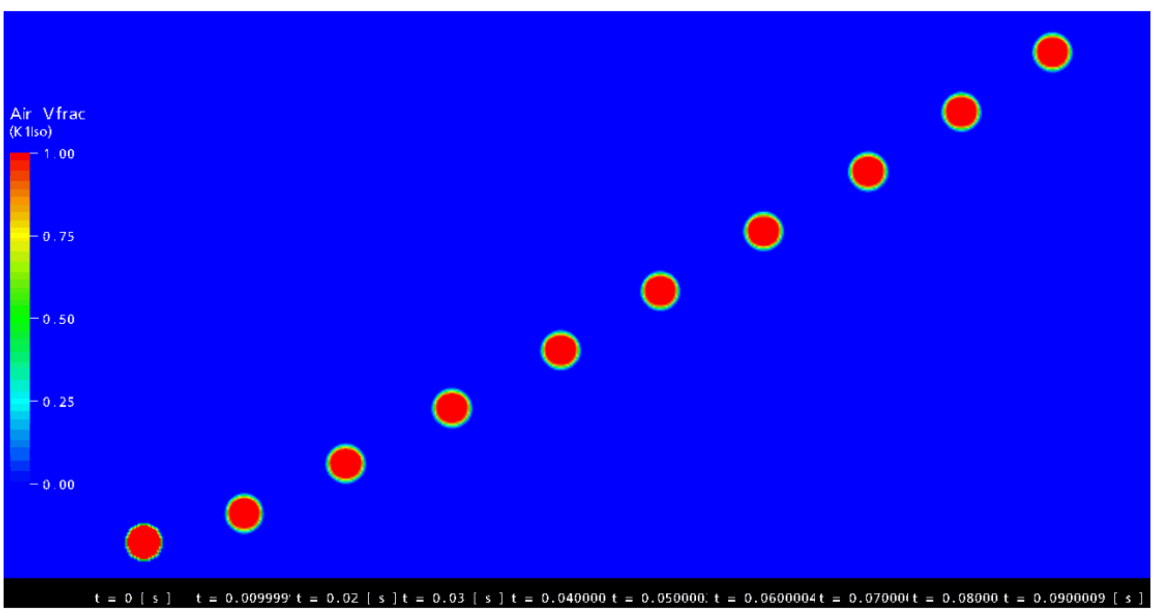

Figure 6. The rise of the 1 mm diameter bubble

Figure 7 displays the volume fraction distribution as a function of time for the case of a bubble of 5 mm in diameter. The bubble shape changes to an ellipse over a period of 0.05 seconds. Figure 4 clearly displays the difference in velocities between the top and bottom of the bubble. However, the top and bottom surfaces eventually reach a stable velocity once the final shape of the bubble has been established. The predicted terminal velocity for the 5mm diameter bubble is 20% lower than the value shown in Figure 2.

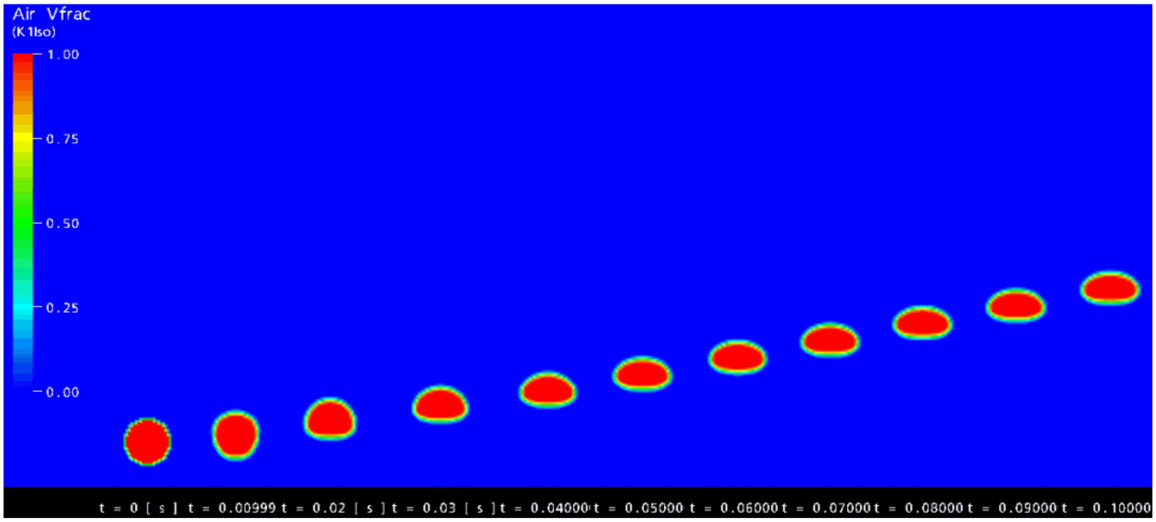

Figure 7. The rise of the 5 mm diameter bubble

Figure 8 shows the volume fraction distribution as a function of time for the case of a bubble 40 mm in diameter. This size bubble is clearly in the spherical cap regime and this is collaborated by the CFD analysis. In this case the Weber number is above the critical value, which means that the bubble will brake under the impact of the inertial forces. The CFD analysis does show that the break-up takes place after 0.2 seconds. However, this axisymmetry two-dimensional simulation is not suited for the analysis past break-up.

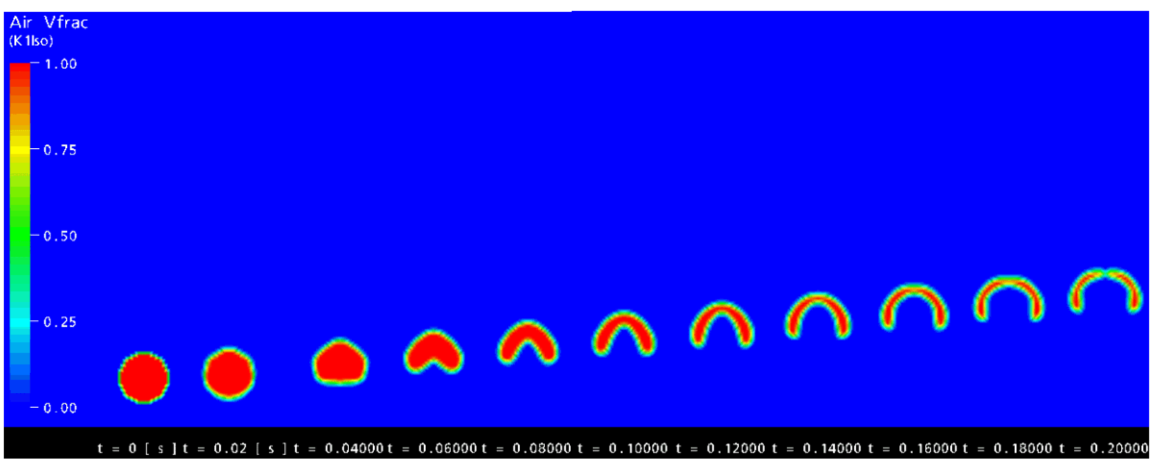

Figure 8. The rise of the 40 mm diameter bubble

Table 2. Terminal velocity.

Diameter (mm)	Terminal Velocity (cm/s)	
	CFX	Fig. 2
1	16.5	16.5
5	19.2	24.0
40	39.2	43.0

Table 2 summarizes the terminal velocities for the 3 bubble diameters considered in this paper. The values predicted by CFD are within 20% of the experimental data for a coarse grid. Additional improvements can be achieved by further refining the mesh.

Conclusion

CFX has been used successfully to model the rise of a stationary bubble in a quiescent liquid. Three bubble diameters where used to show that CFX is able to predict the final bubble shape.

The terminal velocities predicted by CFX were compared to available data and differences are explained based on the details of the setup and the resulting flow field.

References

1. Ron Darby. "Chemical Engineering Fluid Mechanics" Marcel Dekker, Inc. 2001.
2. Grace JR. Hydrodynamics of liquid drops in immiscible liquids. In: Cheremisinoff, NP, R Gupta, eds. Handbook of Fluids in Motion. Ann Arbor Science, 1983, Chap 38.
3. R. Clift, J. R. Grace and M. E. Weber. "Bubbles, Drops, and Particles" Academic Press, New York, 1978.
4. J.U Brackbill, D.B. Kothe and C. Zemach. "A Continuum Method for Modelling Surface Tension". Journal of Computational Physics 100:335-354, 1992.

Additional Figures

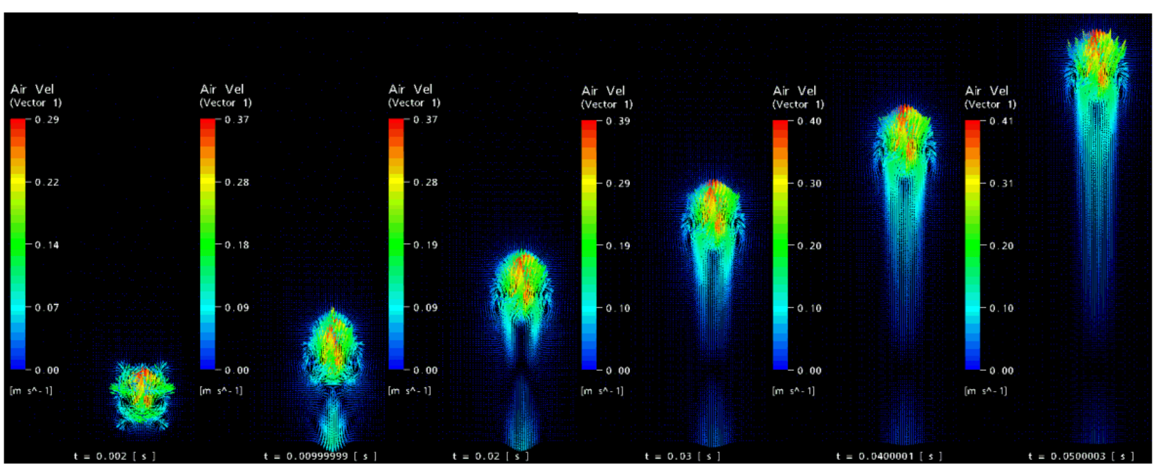

Figure 9. Development of the velocity field for a 1 mm bubble

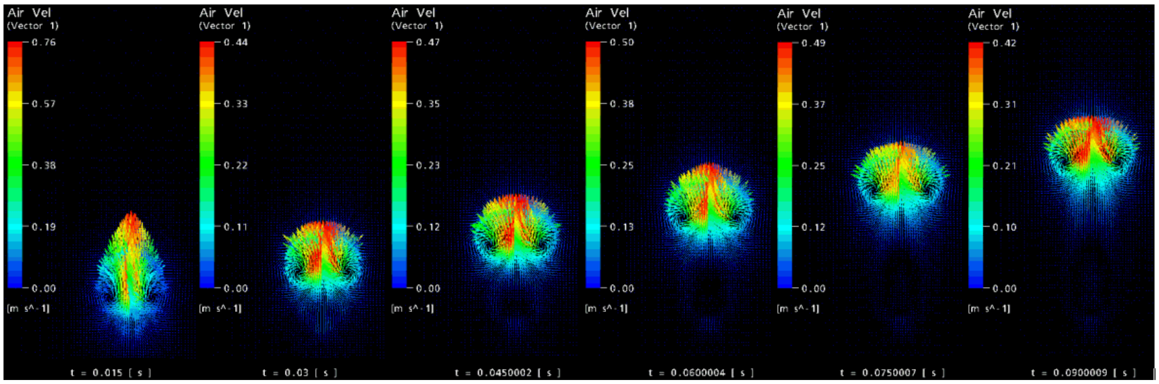

Figure 10. Development of the velocity field for a 5 mm bubble

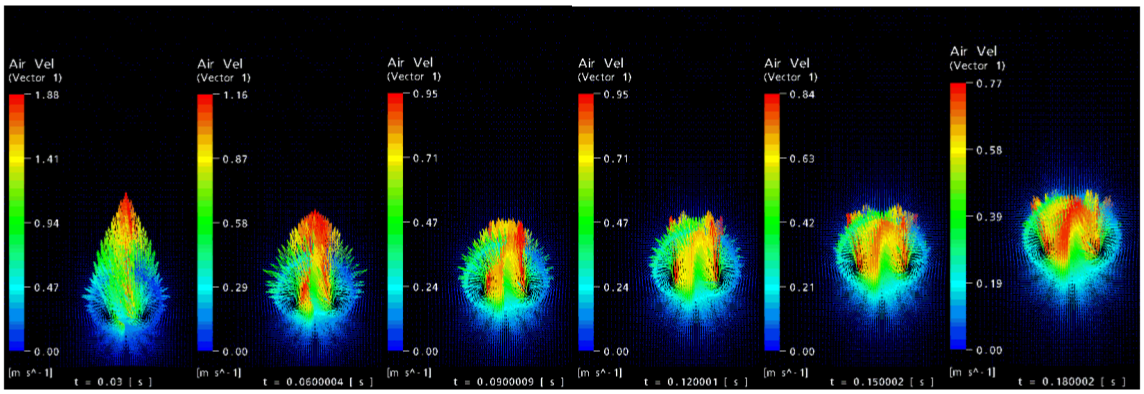

Figure 11. Development of the velocity field for a 40 mm bubble