

**UCLA**  
**COMPUTATIONAL AND APPLIED MATHEMATICS**

---

**Numerical Solution of the Navier-Stokes Equations  
for the Initial Motion of a Two Dimensional Bubble  
in a fluid**

**Emad Fatemi**

**May 1993**

**CAM Report 93-11**

---

**Department of Mathematics  
University of California, Los Angeles  
Los Angeles, CA. 90024-1555**

Numerical solution of the Navier-Stokes equations  
for the initial motion of a two dimensional bubble  
in a fluid

EMAD FATEMI <sup>1</sup>

University of California, Department of Mathematics,  
405 Hilgard Avenue  
Los Angeles, CA 90024-1555

---

<sup>1</sup>This research was supported by ONR grant #N00014-91-j-1034

## Abstract

Numerical solutions of the Navier-Stokes equations for the initial motion of a two dimensional bubble due to the buoyancy force are calculated. The dynamics of the gas inside the bubble is ignored but surface tension and viscosity effects are included. The results of the numerical calculations are compared with the theory and experiments of Walters and Davidson and are found to be in good agreement with the experiments, despite the fact that due to numerical difficulties the calculations are done with lower Reynolds numbers than the Reynolds number of the flow in the experiments. The position of center of mass of the bubble is calculated over a range of viscosities and is found out not to be very sensitive to the Reynolds number of the flow. The shape of the bubble and the size of the small bubbles that separate from the original bubble depend strongly on the Reynolds number. The numerical method is a modified volume of fluid (VOF) algorithm for solving two dimensional free-surface viscous flows. The VOF method is specially suited for continuing the solution even after the change of topology in the shape of the bubble and separation of the small bubbles.

## 1 Introduction

This paper deals with numerical calculation of the initial motion of a two dimensional bubble in a liquid. When a circular bubble is suddenly formed in a liquid it tends to rise under the buoyancy force. The shape of the bubble gets distorted during the rise and a jet of liquid from the bottom of the bubble is projected into the bubble. As the bubble moves upward two small bubbles are separated from the sides of the bubble. The separation of the small bubbles is the mechanism for change of flow from the initial irrotational motion to the fully separated flow.

This work was motivated by the experimental work of Walters and Davidson (1962), where they photographed the initial motion of a two dimensional bubble in water. Their theory was based on assumption of irrotational motion in the liquid and expansion of the potential as a series of harmonics. Surface tension was neglected and therefore pressure on the boundary was taken to be constant. Their theory, valid only for small displacements from the cylindrical form, predicts an initial upward acceleration of  $g$  and the original shape of the tongue projected from the bottom of the bubble.

In this work we solve the full Navier-Stokes equations using a VOF method. The dynamics of the gas inside the bubble is ignored, but effects of surface tension and viscosity are included. Reynolds number of the flow in the experiments was around 1000, but unfortunately that high Reynolds number is not achievable by our numerical method. We are able to calculate up to Reynolds numbers 100 and the results both qualitatively and quantitatively are very close to the experiments. We find that the theoretical predictions of Walters and Davidson are valid for

smaller time intervals that they have assumed. We suspect that this is partly because their theory is based on assumption of viscosity coefficient being zero. We have found out that distortion of the bubble is sensitive to variations in the viscosity coefficient.

## 2 Governing Equations

The Navier-Stokes equations describing the dynamics of incompressible fluids consists of conservation of momentum equations and incompressibility condition. In two dimensional space and including the gravitational force, the system in the scaled form is written as the following.

$$u_t + (u^2)_x + (uv)_y = -p_x + \frac{1}{Re} (\sigma_x^{11} + \sigma_y^{12}) \quad (1)$$

$$v_t + (uv)_x + (v^2)_y = -p_y + \frac{1}{Re} (\sigma_x^{21} + \sigma_y^{22}) - 1 \quad (2)$$

$$u_x + v_y = 0 \quad (3)$$

$$\sigma = \begin{pmatrix} \sigma^{11} & \sigma^{12} \\ \sigma^{21} & \sigma^{22} \end{pmatrix} = \begin{pmatrix} 2u_x & u_y + v_x \\ v_x + u_y & 2v_y \end{pmatrix} \quad (4)$$

The independent variables are time,  $t$ , and spatial coordinates,  $x = (x, y)$ . The dependent variables are velocity vector field,  $u = (u, v)$ , and ratio of pressure to density,  $p$ . Here we have used  $\sigma$  to represent the stress tensor defined in terms of the first derivatives of the velocity vector field. The non-dimensional parameters, Reynolds number(  $Re$ ), and capillary number(  $Ca$ ), are defined in the usual way. Here we have taken the gravity to be in the  $y$  direction.

The system has to be augmented with the boundary conditions for the solid boundaries and the free surfaces. At the solid boundaries we specify the normal velocity of the fluid to be zero, also we specify the tangential component of the stress vector to be zero (free slip boundary condition). We specify kinematic boundary condition,

$$s \cdot n = u \cdot n, \quad (5)$$

and stress boundary conditions,

$$\left(-pI + \frac{1}{Re}\sigma\right)n = \frac{1}{Re} \frac{1}{Ca} \kappa n. \quad (6)$$

The curvature of the boundary is denoted by  $\kappa$ , the unit vector normal to the boundary of the fluid is denoted by  $n$ , and the velocity of the boundary is denoted by  $s$ . The numerical method for tracking the boundary relies on a volume of fluid function,  $\psi$ . The function  $\psi$  is one inside the fluid and is zero outside. The

boundary is the level set defined implicitly by the equation  $\psi(x, y) = 0.5$ . The evolution of  $\psi$  satisfies conservation of mass equation.

$$\psi_t + (\psi u)_x + (\psi v)_y = 0 \quad (7)$$

In our scheme the kinematic boundary condition is automatically satisfied if equation (7) for  $\psi$  is discretized in a conservative form. The above numerical scheme for tracking of the boundary allows for changes of topology to be handled automatically.

### 3 Numerical Scheme

A modified volume of fluid technique is developed for solution of the Navier-Stokes equations and calculation of the free boundary. The VOF method has its roots in the original Marker and Cell method (MAC) developed by Harlow and Welch (1961). In the original MAC method marker particles are inserted in the fluid and are moved with the velocity field of the fluid. The boundary is defined implicitly as the boundary between the regions where there are no particles and where particles are present. This algorithm was extended to flows of two fluids separated by an interface and applied for calculation of the Rayleigh-Taylor instability by Daly (1967). Later Hirt and Shannon (1968) improved on the free surface boundary conditions by including the stress boundary conditions. An improved implementation of the free surface stress boundary condition by including the tangential stress condition was introduced by Hirt and Nichols (1971). Later the MAC approach was abandoned by them in favor of a marker function for tracking the position of the fluid (1981). To control the numerical diffusion of the marker function a donor-acceptor scheme was used by Hirt and Nichols (1981). Computer codes SOLA-VOF and NASA-VOF3D are written based on this algorithm.

The algorithm used for our calculations is the VOF method with a new method of implementing the free-surface stress boundary conditions. Hirt and Shannon (1968) implemented the stress boundary condition by assuming the boundary in the surface cell to be horizontal, vertical, or with 45° angle. They specified the pressure in the boundary cell to be equal to the normal viscous stress. Hirt and Nichols (1981) improved on that implementation by approximating the velocity just outside the boundary by using the tangential stress condition. In this work we do not specify the pressure in the boundary cells but we solve for the pressure in the boundary cells. The stress boundary conditions are treated as numerical boundary conditions for the momentum equation. Our method is close to the methods developed by Tryggvason and colleagues (1992) in the way that the free-surface stress boundary conditions are implemented, except we do not smooth out the jump in the viscosity or density over several grid points. In our algorithm the jumps are over one grid point.

We review the scheme for solving the Navier-Stokes equations and the equation for the volume of fluid function briefly here. The computational domain is divided into a uniform mesh of size  $\Delta x \times \Delta y$ . A staggered grid is used;  $\psi$  and  $p$  are defined in the center of the computational cell and  $u$  and  $v$  are defined at the middle of the cell boundaries. The values of  $\psi$  at the grid points are interpreted as the volume of the fluid in each cell. Components of the stress tensor are evaluated by finite difference formulas from  $u$  and  $v$ .  $\sigma^{12}$  and  $\sigma^{21}$  are evaluated on the corners of the cell.  $\sigma^{11}$ , and  $\sigma^{22}$  are evaluated in the center of the computational cell. Explicit time stepping is used to calculate the time dependent solution. One cycle of the computation starts with known values of  $\psi^n$ , and  $(u^n, v^n)$ . In the first step  $\psi^{n+1}$  is evaluated using the conservation of mass equation. The velocities in the newly filled cells are adjusted to satisfy incompressibility condition. From values of  $u^n$  and  $v^n$  intermediate values  $v^*$  and  $u^*$  are calculated by solving conservation of momentum equation without pressure terms. Pressure is calculated by imposing incompressibility condition on  $u^{n+1}$  and  $v^{n+1}$ . Finally  $u^{n+1}$  and  $v^{n+1}$  are evaluated using  $u^*, v^*$ , and the calculated pressure,  $p^{n+1}$ . The cycle is thus closed.

Solution of the conservation of mass equation, equation 7, requires special treatment to control numerical diffusion. Numerical diffusion is controlled by using limiters for calculating mass fluxes of the continuity equation. This scheme is known as donor-acceptor scheme and was designed by Hirt and Nichols (1981). The flux is defined such that the amount of fluid leaving the donor cell is equal or less than the available fluid. The above procedure introduces upwinding in the opposite direction and reduces numerical diffusion, thus the profile of the  $\psi$  function stays sharp during the calculation.

Imposing free-surface stress boundary conditions requires computing curvature of the boundary. For computation of the curvature we perform an extra computational step to reconstruct an approximate boundary. The boundary is not defined uniquely from the volume of fluid function. For our calculations we reconstruct three points on the surface using the algorithm used by Hirt and Nichols (1981). In their approach two approximate boundaries are constructed, one treating the boundary as a function of  $x$  and the other treating it as a function of  $y$ . The choice between the two is made by comparing the slopes of the two reconstruction. The reconstruction with the smallest absolute value of the slope is chosen. This is equivalent to deciding whether the surface is closer to vertical or horizontal. The slope contains information about position of the fluid with respect to the boundary and also sign of the curvature. If our surface is closer to horizontal, we choose the  $x$  reconstruction, otherwise the  $y$  reconstruction is chosen.

The curvature is evaluated by using the coordinates of the reconstructed points on the boundary. Consider the boundary to be a curve,  $x(s)$ , parameterized by  $s$ . Then the unit tangent vector,  $T$ , is defined as  $T = \frac{dx}{ds} / |\frac{dx}{ds}|$ . We

compute the curvature using the following formula.

$$\kappa = \frac{\left| \frac{dT}{ds} \right|}{\left| \frac{dx}{ds} \right|} = \frac{2(T(s + \frac{1}{2}\Delta s) - T(s - \frac{1}{2}\Delta s))}{x(s + \Delta s) - x(s - \Delta s)} + O(\Delta s^3) \quad (8)$$

In addition to the above, variety of different approaches were tried for calculating the curvature. The above simple discretization is superior to other formulae. Intuitively we suspect this is because the discretized version is coordinate free and also the discretized form is similar to central differencing of a second order differential operator.

The stress boundary conditions are implemented by adding a source term in the momentum equation for calculation of  $u^*$  and  $v^*$ . The pressure in the boundary cells is then calculated in the next step of the algorithm. The viscous part of the stress is automatically included in the calculation. There is inaccuracy, proportional to the mesh size, from implementing the boundary condition at the center of the boundary cells rather than exactly at the boundary. A more accurate implementation requires subcell resolution of the boundary which was not attempted.

The VOF scheme is an explicit scheme and the time steps have to satisfy stability conditions. Approximate stability conditions are obtained by considering stability of the corresponding linear problems. These conditions are as the following.

$$\Delta t < \frac{1}{\frac{\max|u|}{\Delta x} + \frac{\max|v|}{\Delta y}} \quad (9)$$

$$\Delta t < \frac{Re}{4} \min[\Delta x^2, \Delta y^2] \quad (10)$$

$$\Delta t < \sqrt{\frac{Ca \min(\Delta x^3, \Delta y^3)}{64}} \quad (11)$$

The first stability condition is the CFL condition necessary for stability of conservation of mass equation and the nonlinear part of the momentum equation. The second criteria corresponds to stability of the viscous part of the momentum equation and the third condition is to damp out spurious free surface waves. In practice these criteria are used and are sufficient for stability of the numerical calculations.

## 4 Numerical Results

For our numerical calculations we choose the parameters of the experiments of Walters and Davidson (1962). We scale the problem using the initial radius of the bubble,  $a$ , for length. Then we scale time with  $T = \sqrt{a/g}$  and velocity using

$U = \sqrt{ag}$ . We use the following values from the experiments for our constants.

$$a = 2.54\text{cm}, \quad \rho = 1\text{gr/cm}, \quad \sigma = 74, \quad g = 980\text{cm/sec}^2, \quad \mu = 0.013\text{gr/cmsec}$$

With the above values our non-dimensional parameters are:

$$Re = 9731, \quad Ca = 0.008765, \quad \frac{1}{ReCa} = 0.01170.$$

Unfortunately we are unable to perform numerical calculations with Reynolds number as high as 1000. The numerical scheme is unable to resolve all the frequencies present at that Reynolds number and it becomes unstable. We are able to do successful calculations up to Reynolds number 100. Our calculations are done with changing the value of  $Re$  while keeping the ratio  $\frac{1}{ReCa} = \frac{\sigma}{\rho ga^2}$  fixed. Physically this corresponds to changing the coefficient of viscosity only and keeping all other parameters unchanged.

The first two figures show the computed boundary of the rising bubble for Reynolds numbers 100 and 10. The computed boundaries for the Reynolds number 100 are in excellent agreement with the experimental results of Walter and Davidson (1962). The computed shapes are in one to one correspondence with plate one of the above paper. The computed results for Reynolds number 10 are qualitatively different. The separation of the small bubbles is delayed and the projected tongue of liquid in the bottom of the bubble is less pronounced. In the next figure we calculate the position of center of mass of the rising bubble as a function of time for Reynolds numbers 100, 50, 20, and 10. The dashed line is the motion of a bubble with acceleration  $g$  upward. The results indicate that only for the beginning of the motion the bubble is accelerated with acceleration  $g$ . The results suggest that the motion of the center of the mass is not very sensitive to the Reynolds number of the flow in the range that we considered.

In figure four we plot the position of center of the mass of the bubble for a smaller interval of time and also the theoretical dashed curve,  $s = \frac{1}{2}gt^2$ . This figure is in comparison with figure 4 of the paper of Walters and Davidson. Our calculated result shows a regular departure from the dashed curve and is in agreement with their experimental results. Walters and Davidson obtained closer fit with assumption that the movement of the bubble is delayed due to the mechanical apparatus that was used for the experiment. Our numerical result is in disagreement with their assumption and indicates that the theoretical curve is only correct asymptotically.

In the next figure we show the position of tip of the bubble, tip of the liquid tongue, and vertical diameter of the bubble as a function of time. Our calculated results are in good agreement again with the experimental results of Walters and Davidson except the motion of the lower part of the bubble is a little exaggerated in our calculations. Our results can be compared with figure 3 of the paper of Baker and Moore (1989) where they have performed similar calculations. In



the last figure we show the time history of the vertical diameter of the bubble calculated with different Reynolds numbers. The dashed line is the theoretical curve for inviscid fluid calculated by Walters and Davidson (1962). The calculated results indicate that the distortion of the bubble is sensitive to the Reynolds number and as the Reynolds number is increased our computed curves approach the theoretical inviscid limit. Again our calculated results are in agreement with the experimental results shown in figure 5 of Walters and Davidson (1962).

## 5 Conclusion

A VOF numerical method was used to calculate the initial motion of a rising 2D gas bubble in a slightly viscous fluid. The calculations are in good agreement with the experimental results of Walter and Davidson (1962). The distortion in shape of the bubble and the time of separation of small bubbles is sensitive to the Reynolds number of the flow. The movement of center of mass of the bubble seems to be not very sensitive to the Reynolds numbers in the range that was considered.

## 6 Acknowledgment

We thank Peter Smereka for suggesting this problem to us, also we thank Professor Stanley Osher for encouragement and his interest in this research.

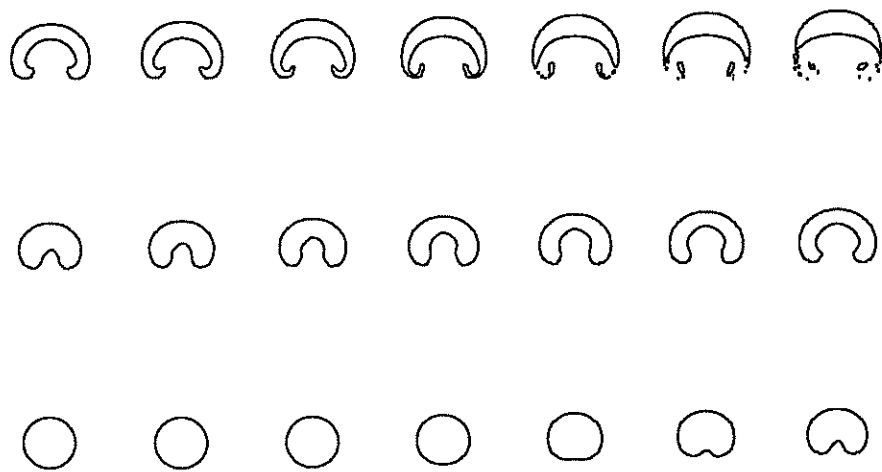


Figure 1: Motion of the bubble for  $Re=100$

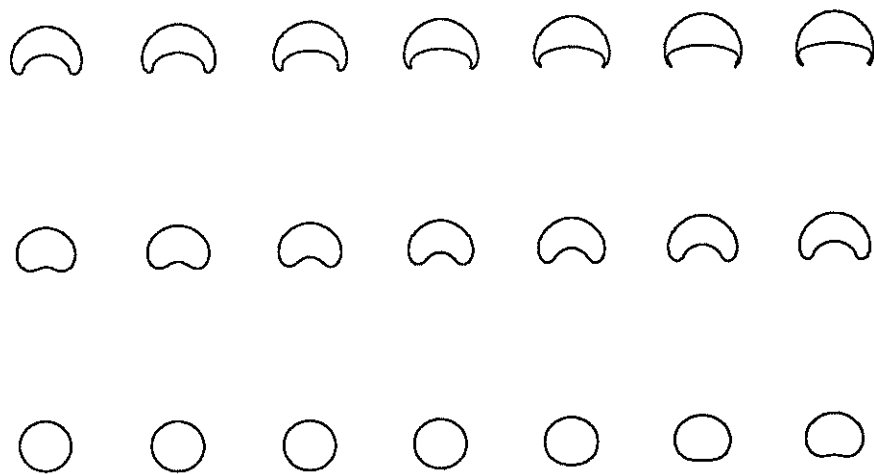


Figure 2: Motion of the bubble for  $Re=10$

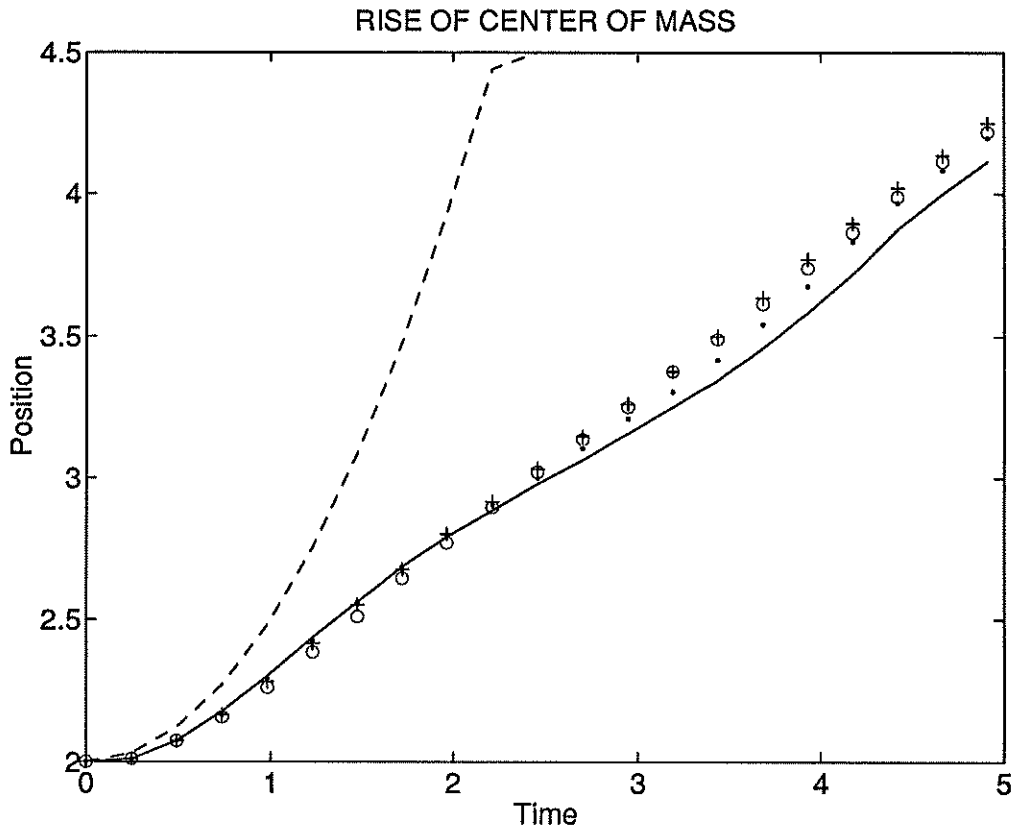


Figure 3: Position of center of mass of the bubble, o, Re=10; +, Re=20; ·, Re=50; solid line, Re=100; dashed line, theoretical curve.

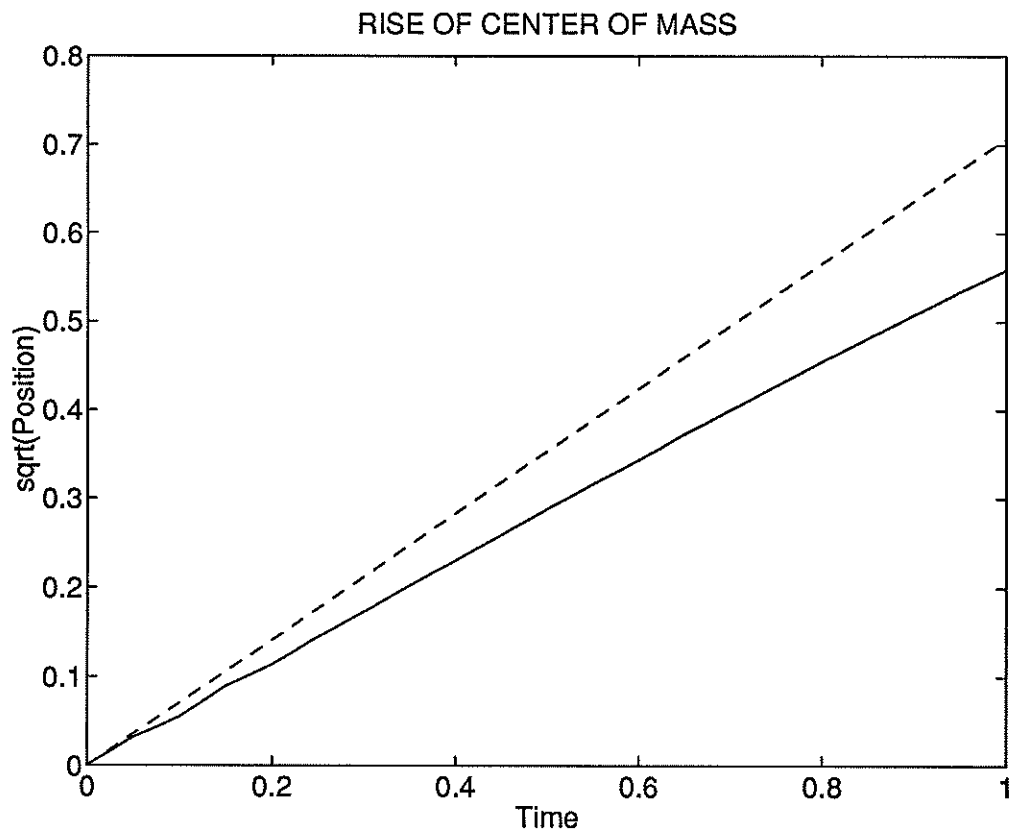


Figure 4: Position of center of mass of the bubble, initial period.

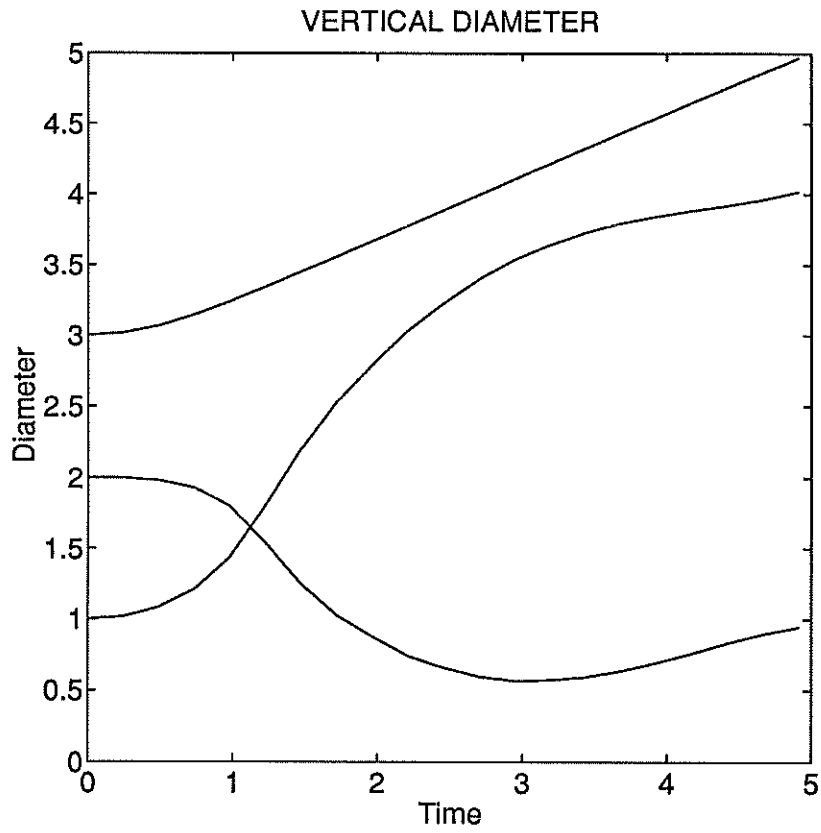


Figure 5: Position of tip of the bubble, tip of the liquid jet, and vertical diameter of the bubble.

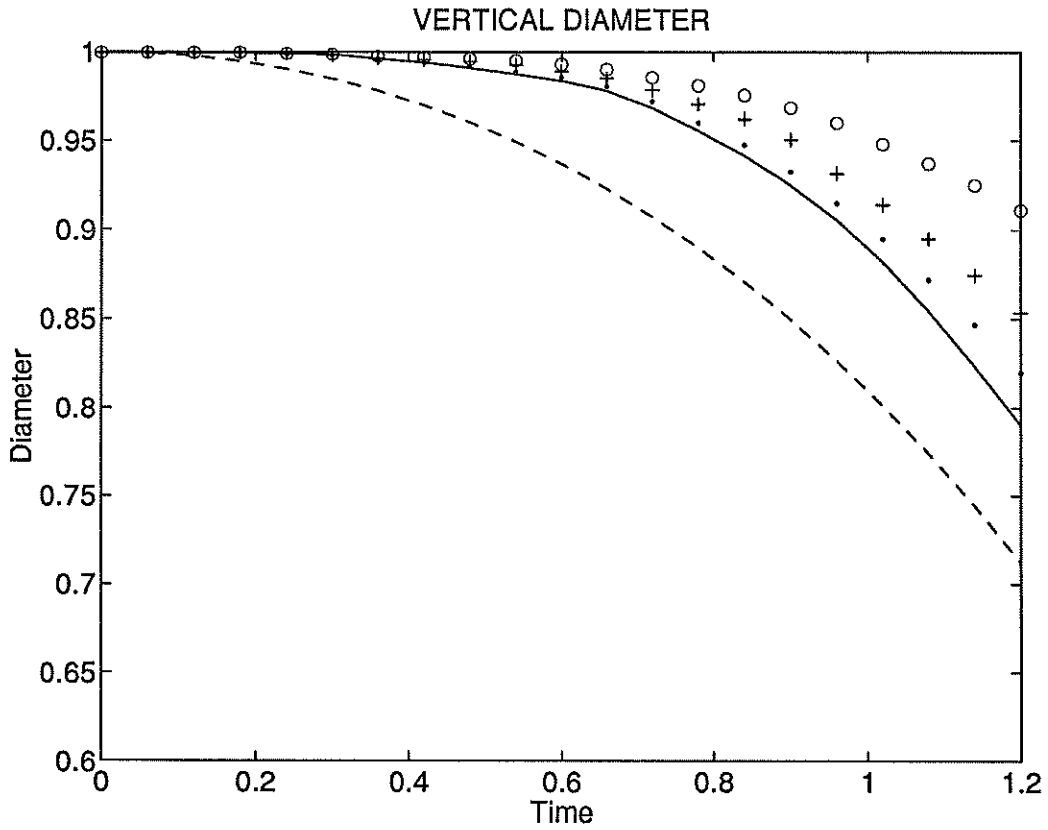


Figure 6: Vertical diameter of the bubble, initial period, o, Re=10; +, Re=20; ·, Re=50; solid line, Re=100; dashed line, theoretical curve.

## References

- [1] A. A. Amsden and F. H. Harlow, *J. Comput. Phys.*, 6 , 322 (1970).
- [2] G.R. Baker and D.W. Moore, The rise and distortion of a two dimensional gas bubble in an inviscid liquid, *Physics of Fluids A* (1989).
- [3] B. J. Daly, ‘Numerical study of two fluids Rayleigh-Taylor instability’, *Phys. Fluids*, 10, 297 (1967).
- [4] F. Harlow and E. Welch, ‘Numerical calculation of the time-dependent viscous incompressible flow of fluids with free surfaces’, *Phys. Fluids*, 8 , p. 2184 (1965).
- [5] C. W. Hirt and J. P. Shannon, ‘Free-surface stress condition for incompressible flow calculations’ *J. Comput. Phys.* 2, 403-411 (1968).
- [6] C. W. Hirt and B. D. Nichols, ‘Volume of fluid (VOF) method for the dynamics of the free boundaries’, *J. Comput. Phys.* 39, 201 (1981).
- [7] B. D. Nichols and C. W. Hirt, ‘Improved free surface boundary condition for incompressible flow calculations’, *J. Comput. Phys.* 8, 434-448 (1971).
- [8] B. D. Nichols, C. W. Hirt, and R. S. Hotchkiss, ‘SOLA-VOF: a solution algorithm for transient fluid flow with multiple free boundaries’, *LA-8355*(1980).
- [9] M. D. Torrey, R. C. Mjolsness, L. R. Stein, ‘NASA-VOF3D: a three-dimensional computer program for incompressible flows with free surfaces’, *LA 11009-MS* (1987).
- [10] S. O. Unverdi and G. Tryggvason, ‘A front-tracking method for viscous, incompressible, multi-fluid flows’, *J. Comput. Phys.* 99, to appear.
- [11] J.K. Walters and J.F. Davidson, The initial motion of a gas bubble formed in an inviscid liquid, *J. Fluid Mech.* 12, 408 (1962).