Multiphase Science and Technology, Vol. 18, No. 3, pp. 231-249, 2006

RECENT PROGRESS IN COMPUTATIONAL STUDIES OF DISPERSE BUBBLY FLOWS

G. Tryggvason[1], A. Esmaeeli[1], J. Lu[1], S. Homma[2] and S. Biswas[1]
[1]*Dept. of Mechanical Engineering, Worcester Polytechnic Institute, Worcester, MA, USA*
[2]*Dept. of Applied Chemistry, Saitama University, Saitama 338-8570, Japan*

Abstract. Direct numerical simulations of bubbly flows are reviewed and recent progress is discussed. Simulations of homogenious bubble distribution in fully periodic domains at relatively low Reynolds numbers have already yielded considerable insight into the dynamics of such flows. The challenge now is to examine bubbles at higher Reynolds numbers, bubbles in channels and confined geometries, bubble interactions with turbulent flows, as well as more complex flows. We discuss current studies of these problems, as well as recent results on bubble formation in boiling.

1. INTRODUCTION

Direct numerical simulations (DNS) of bubbly flows, where the flow around each bubble is fully resolved and viscosity, inertia, and surface tension are accounted for have already been used to study the behavior of systems containing up to O(100) bubbles. These simulations have yielded large amount of new insight into the dynamics of such flows, including the identification of preferred relative orientation of bubble pairs, as well as data for the bubble induced flow. Direct numerical simulations have also been used to explain the importance of deformability, both for the motion of a single bubble in a shear flow as well as the collective dynamics of many bubbles.

The goal of direct numerical simulations of multiphase flows is two fold. First of all, such simulations should generate insight into, and understanding of, the basic behavior of multiphase flow, such as the forces on a single bubble or a drop, how the bubbles and drops affect the flow, how many bubbles and drops interact in dense disperse flows. Secondly, the vast data generated by direct numerical simulations should be helpful in generating closure models for engineering simulations of the averaged flow field. As in turbulent flow of a single-phase fluid, multiphase flows generally possess a large range of scales, ranging from the sub-millimeter size of a bubble or an eddy to the size of the system under investigation. For an industrial bubble column, for example, the ratio of the largest to the smallest length scale is easily tens or hundreds of thousands. Assuming somewhat arbitrarily that we need about ten grid points per smallest scale, it is clear that the size of a grid needed for DNS of such systems exceeds the power of currently available computers. But even if the possibility of doing direct numerical simulation of a full size industrial system existed, it is unlikely that it will be desirable to do so, except perhaps in a few exceptional cases. Multiphase flows, like single-phase turbulent flows, exhibit a great deal of universality and it is almost certain that recomputing small-scale behavior that is already understood is not necessary.

Most of the work done so far has been for bubbles in periodic domains with relatively modest rise Reynolds numbers. While such systems are of relevance for high viscosity liquids, it is obviously important to understand what happens as the Reynolds number increases and how walls and preexisting turbulence interact with the bubbles. Here we review earlier investigations briefly and report on the current status of studies addressing these issues.

2. NUMERICAL METHOD

In the limit of high and low Reynolds numbers it is sometimes possible to simplify the flow description considerably by either ignoring inertia completely (Stokes flow) or by ignoring viscous effects and assuming a potential flow. Similarly, for dilute flow of small particles it is sometimes possible to treat the particles as non-interacting points. For the general situation, non-dilute flows at intermediate Reynolds numbers, it is necessary to solve the full unsteady Navier-Stokes equations. Most methods currently in use for DNS of multiphase flows are based on writing one set of the governing equations for the whole flow field, including both the liquid and the bubbles, by allowing for a discontinuous density and viscosity field and including a singular term representing the surface forces. The momentum equation is:

$$\rho\frac{\partial \mathbf{u}}{\partial t}+\rho\nabla\cdot\mathbf{uu}=-\nabla P+\nabla\cdot\mu(\nabla\mathbf{u}+\nabla\mathbf{u}^T)+\sigma\int_F \kappa_f\mathbf{n}_f\delta(\mathrm{x}-\mathrm{x}_f)dA_f\,. \tag{1}$$

and we usually assume that both fluids are incompressible:

$$\nabla\cdot\mathbf{u}=0\,. \tag{2}$$

Here, **u** is the velocity, P is the pressure, and ρ and μ are the discontinuous density and viscosity fields, respectively. δ is a three-dimensional delta-function constructed by repeated multiplication of one-dimensional delta-functions. κ is twice the mean curvature. **n** is a unit vector normal to the front. Formally, the integral is over the entire front, thereby adding the delta-functions together to create a force that is concentrated at the interface, but smooth along the front. **x** is the point at which the equation is evaluated and $\mathbf{x}_f$ is the position of the front.

Numerical implementations of equations (1) and (2) include the Volume of Fluid, Level Set, and CIP methods. Equations (1) and (2) are also used in the Front-Tracking/Finite Volume method of Unverdi and Tryggvason (1992), which is used for the simulations discussed later in this paper. In all these methods, the governing equations are solved by a projection method on a fixed grid. The way the phase boundary is tracked is, however, different. In most cases a marker function that identifies the different fluids is advected by the flow. In the method of Unverdi and Tryggvason, however, the phase boundary is tracked by connected marker points (the "front") and the marker function reconstructed from the location of the "front." This keeps the boundary between the phases sharp, and allows the accurate computation of the surface tension. The front points are advected by the flow velocity, interpolated from the fixed grid. As the front deforms, surface markers are dynamically added and deleted. The surface tension is represented by a distribution of singularities (delta-functions) located at the front. The gradients of the density and viscosity become delta functions when the change is abrupt across the boundary. To transfer the front singularities to the fixed grid, the delta functions are approximated by smoother functions with a compact support on the fixed grid. At each time step, after the front has been advected, the density and the viscosity fields are reconstructed by integration of the smooth grid-delta function. The surface tension is then added to the nodal values of the discrete Navier-Stokes equations. Finally, an elliptic pressure equation is solved by a multigrid method to impose a divergence-free velocity field. For a detailed description of the original method, including various validation studies, see Unverdi and Tryggvason (1992) and Tryggvason *et al.* (2001).

The original method of Unverdi and Tryggvason (1992) has been extreamly successful for relatively complex multiphase flows at modest Reynolds numbers. For higher Reynolds numbers and large properties ratios the calculations generally become more challenging. This is, of course, true of any flow simulation, whether involving many phases or not. While centered schemes for the advection terms, as used in the original method of Unverdi and Tryggvason (1992), are always the most accurate ones for fully resolved flows, they are not as robust for marginally resolved flows as high order upwind shemes. Thus, several authors have used the QUICK and ENO schemes, for example, to generate more robust methods for higher Reynolds numbers. We have also done so when needed.

As the density ratio increases, the computations generally become more difficult. Those difficulties include the appearance of high velocities near the interface when the fully conservative form of the advection terms is used, particularly at high Reynolds numbers and large density differences. These high velocities can be traced to an incompatible in the advection of the mass and momentum and can be overcome simply by using the nonconservative form of the advection terms. This is slightly counter-

intuitive since the conservative form is usually preferred for computations of shocks in compressible flows.

Another difficulty encountered when the density ration increases, is the solution of the elliptic equation for the pressure. Iterative solvers generally take longer to converge, but—as long as the density is positive—a simple iterative technique like SOR will always converge with the proper selection of the acceleration constant. We should, however, note that for a density ratio of 1000, say, a very minor error—relative to the density in the heavier liquid—can lead to negative values. More efficient multigrid solvers may, on the other hand, fail to converge. While some progress has been made, the development of efficient pressure solvers for high-density ratios remains high on the wishlist of nearly everybody engaged in the use of one-fluid methods for multiphase flow simulations. In many applications, relatively movest values of the density and the viscosity ratios can be used as approximations for larger differences, without changing the solution too much. This is particularly true when the dynamics of the low viscosity/density fluid is largly controlled by what the high viscosity/density fluid is doing, such as for bubbles and the short time motion of drops. In other cases, such as wind generated breakup and droplet suspensions, it is more important to use the correct properties of both fluids.

For high surface tension, methods based on the one-fluid formulation on fixed grids generally suffer from artificial currents induced by a mismatch between the resolution of the pressure and the surface tension term. In early implementation of surface tension into VOF methods using the Continous Surface Force (CSF) (Brackbill, Kothe and Zemach, 1992; Lafaurie, Nardone, Scardovelli, Zaleski and Zanetti, 1994) the parasitic current posed a serious limitation on the range of problems that could be solved. While the front tracking method of Unverdi and Tryggvason (1992) also showed parasitic currents at high surface tension, these were generally smaller. The origin of these current is now reasonably well understood and remedies are emerging. The key seems to be that the normal vector to the interface should be found as the gradient of the indicator function and discretized in exactly the same way as the pressure gradient. If the curvature is constant this results in no parasitic currents. If the curvature is not constant it must be found from information about the interface. In the PROST-VOF method of Renardy and Renardy (2002) the curvature is found by fitting a parabolic surface to the interface, resulting is significant reduction of the parasitic currents. Although comparable approaches for explict front tracking have not been published (in part, possibly, because the problem is less urgent), there are good reasons to believe that a similar approach will yield similar improvement.

For large viscosity ratios there are generally inaccuracies around the interface due to the smoothing of the transition zone. As the error is generally “pushed” to the less viscosus side, the error tends to be small when the flow in the less viscous liquid has relatively minor impact on the global behavior. This is usually the case for bubbles, and for short time evolution of drops, such as during drop collisions and splashing. For viscous drops suspended in a less viscous liquid the errors in the viscous terms are more serious and the convergence rate is worse. It has been known for quite some time, in the context of VOF methods, that taking the harmonic mean of the viscosities at points where the viscosity is not defined, instead of the arithmetic average, guaranties the

continuity of the viscosus stresses (Patankar, 1980). Ferziger (2003) expanded this approach to interfaces smoothed over several gridpoints.

With relatively few exceptions, fluid systems consist of "active" regions of concentrated vorticity (boundary and shear layers) and more "passive" regions where the flow is more uniform (often potential flow). Frequently, the "active" regions are only a small part of the whole flow field and using the fine grid required for the "active" regions everywhere results in excessive and unnecessarily fine grid in the passive regions. For large-scale simulations, considerable savings can be realized using adaptive gridding. The simplest "adaptive" gridding is the use of stretched grids to concentrate points near walls, for example. This can be done by simply making the grid-size a function of the grid index and performing the advection and interpolation/smoothing of front points in the mapped coordinate system. This technique should be easily extended to general mapped and body fitted grids. A more general approach is, however, the use of locally refined Cartesian grids. In this approach, the grid cells are rectangular but each cell can be refined by splitting it up into four (2D) or eight (3D) cells. The various levels of refinement are organized in a tree structure that allows each level to be accessed efficiently. This technique, usually referenced as Adaptive Mesh Refinement (AMR)— although the terminology could apply to other types of refinement— has been developed by several groups for compressible flow calculations. Multiphase flow applications are still rare, but include Agresar, Linderman, Tryggvason and Powell (1998), Roma, Peskin and Berger (1999), and Sussman *et al*. (1999).

3. RESULTS

As numerical methods for DNS of multiphase flow are refined further, the range of problems that are easily simulated will increase. However, while such development are important, it is also important to appreciate that a large number of systems are already "simulatable" and that the challenges in understanding the physics of multiphase flow are always likely to be greater than the numerical ones. In the next four sections we will examine the status of studies of several aspects of bubbly flows.

3.1 *Homogeneous bubbly flows*

Homogeneous bubbly flow with many buoyant bubbles rising together in an initially quiescent fluid is perhaps one of the simplest example of disperse flow. Such flows can be simulated using periodic domains where the bubbles in each period interact freely, but the configuration is repeated infinitely many times in each coordinate direction. As the number of bubbles in each period is increased the bubbles generally rise unsteadily, repeatedly undergoing close interactions with other bubbles. The behavior is, however, statistically steady and the average motion (averaged over long enough time) does not change. The hope is that once the size of the system is large enough, information obtained by averaging over each period will be representative of a truly homogeneous bubbly flow.

We have examined the motion of nearly spherical bubbles at moderate Reynolds numbers in a number of papers. Esmaeeli and Tryggvason (1998) examined a case where

the average rise Reynolds number of the bubbles remained relatively small (1-2) and Esmaeeli and Tryggvason (1999) looked at another case where the Reynolds number was 20-30. Bunner and Tryggvason (2002) simulated a much larger number of three-dimensional bubbles using a parallel version of the method used by Esmaeeli and Tryggvason. Their largest simulations followed the motion of 216 three-dimensional buoyant bubbles per periodic domain for a relatively long time. The effect of deformability was also studies by Bunner and Tryggvason (2002) who found that relatively modest deformability could lead to a streaming state where bubbles gathered in a stream or a chimney. These simulations suggest that we are well on our way to understand elementary behavior of homogeneous bubbly flow when the Reynolds number is relatively low.

In many practical situations the Reynolds number is, however, considerably higher and it is well known that bubbles at high enough Reynolds number rise unsteadily, either wobbling as they rise or rising along a spiral path. Recent experimental studies by Ellingsen and Risso (2001) suggest that the wobbling mode may be a transitionary phase and that wobbly bubbles would eventually rise along spiral paths, if one waited long enough. Computationally it is found that two-dimensional bubbles in periodic domains start to wobble at much lower rise Reynolds number than their three-dimensional counterparts. Esmaeeli, Ervin, and Tryggvason (1994) briefly examined bubbles that settle down into periodic wobbling and showed that the bubbles slow down significantly once they start to wobble. Göz, Bunner, Sommerfeld, and Tryggvason (2000) examined higher Reynolds numbers and found what looked like chaotic motion at high enough Reynolds numbers. They suggested that the results indicated that real (three-dimensional) deformable bubbles rising at high enough Reynolds number would exhibit chaotic motion. However, since air bubbles in water become spherical cap bubbles when their size (and rise Reynolds number) increases, such motion might not be observed under normal conditions. We are currently examining the buoyancy driven motion of many bubbles at high enough Reynolds numbers so that moderately deformable three-dimensional bubbles show wobbly motion. Given the streaming state for deformable bubbles found by Bunner and Tryggvason (2003), the first question is obviously whether wobbly bubbles stream. Since the streaming is sensitively dependent on the lift force and preliminary results for two-dimensional bubbles by Esmaeeli (1995) showed that wobbly bubbles exhibited essentially no drift in a linear shear, it is not obvious that wobbly bubbles will stream. This is, indeed, what we find. Wobbly bubbles initially put in a column (in a similar way as Bunner and Tryggvason, 2003, did) immediately disperse as spherical bubbles do. Thus, examining the properties of homogeneous distribution of wobbly bubbles is of relevance for realistic systems. Figure 1 shows one frame from a simulation of 14 bubbles at about 6% void fraction, Eo=4 and $M=10^{-6}$. The bubbles and the fluid velocity in a plane cutting through the middle of the domain are shown at a relatively late time. The vorticity in the same plane is also shown as color contours. While the bubbles occasionally "clump" together, on the average they remain essentially uniformly distributed. The rise Reynolds number for four bubbles at the same governing parameters is shown in figure 2. Since the bubbles are interacting freely, they start to move unsteadily relatively early, although the fluctuations in the velocity of each bubble are initially relatively small. After time 15 or so each bubble undergoes larger velocity fluctuations. Preliminary results suggest that the behavior of wobbly bubbles, including

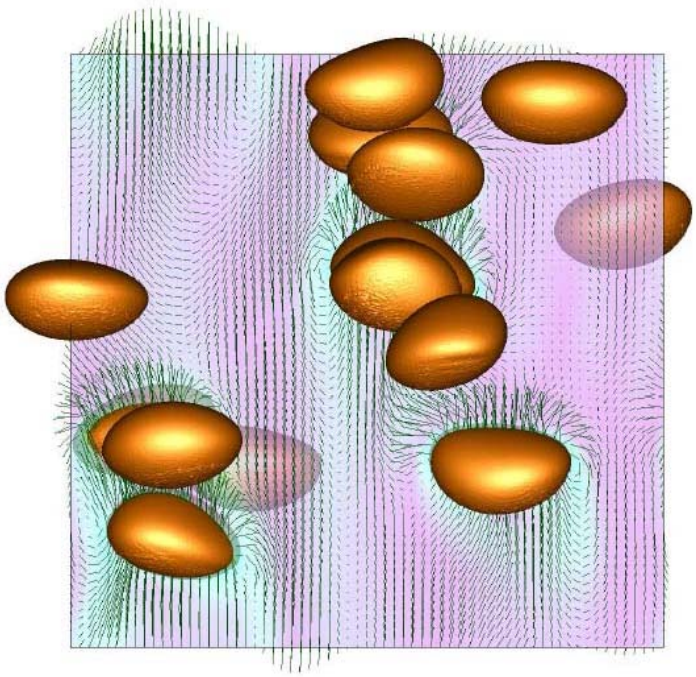

Figure 1 One frame from a simulation of 14 wobbly bubbles in fully periodic domains. The velocity field and the vorticity in a plane cutting through the domain is shown.

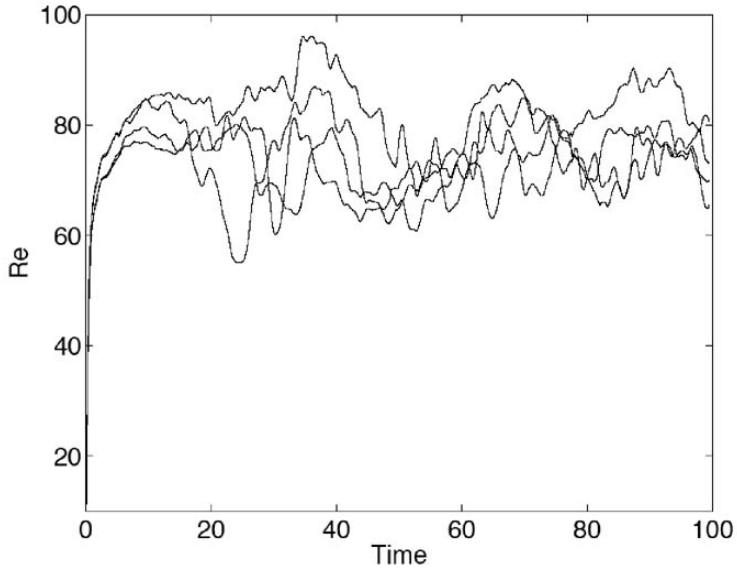

Figure 2 The rise Reynolds number versus time for four wobbly bubbles.

the velocity fluctuations in the liquid and the bubble dispersion, is significantly different from nearly spherical non-wobbly bubbles. More detailed studies are in progress.

Other studies of the motion and interactions of many bubbles have been done by several Japanese authors. Early work, using the VOF method to compute the motion of a single two-dimensional bubble can be found in Tomiyama *et al*. (1993) and more recent work on bubble interactions, using both VOF and the Lattice Boltzman Method, is presented in Takada *et al*. (2000, 2001). Major progress has also been made in the simulation of finite Reynolds number suspension of rigid particles. Feng, Hu, and Joseph (1994, 1995) simulated the two-dimensional, unsteady motion of one and two rigid particles. Hu (1996) computed the motion of a few hundred two-dimensional particles and a fully three-dimensional simulation of hundred particles were presented by Johnson and Tezduyar (1997). Recent papers include simulations of over 1000 spheres by Pan, Joseph, Bai, Glowinski, and Sarin (2002) and a study of the fluidization of 300 circular particles in plane Poiseuille flow by Choi and Joseph (2001).

3.2 *Bubbles in laminar channel flows*

While studies of homogeneous bubbly flows can yield considerable insight into many aspects of the interactions between freely rising bubbles, there are obviously many aspects of realistic flows that depend critically on the presence of walls. The pipe flow experiments by Kobayashi, Iida and Kanegae (1970) and Serizawa, Kataoka and Michiyoshi (1975) showed, for example, an increase in the concentration of bubbles near the walls in upflow in a vertical pipe. It is easy to see why nearly spherical bubbles rising in a vertical channel with upflow move toward the walls: The bubbles generally move faster than the liquid and the cross product of the slip velocity and the vorticity vector results in a lift force directed toward the wall (for down-flow, the same arguments suggest that the bubbles move away from walls, as seen experimentally by Oshinow and Charles, 1974, and Wang *et al*., 1987, for example). When the bubbles are very close to the wall, however, the fluid between the bubble and the wall results in a lubrication force that repels the bubble. Furthermore, as demonstrated by Ervin and Tryggvason (1997) and others, bubble deformation can result in a circulation around the bubble of the opposite sign to the fluid shear near the walls and cause a lift force away from the wall. The motion of bubbles near walls continues to be a topic of considerable interest as shown by the recent experimental study of de Vries, Biesheuvel, and van Wijngaarden (2002) who studied how the trajectories of a bubble near walls changes with the bubble size.

A few simulations of fully three-dimensional bubbles in a vertical channel with upflow were done by Bunner (2000) and we have recently started to examine this problem in more details. Figure 3 shows results from two preliminary simulations of two-dimensional systems where one frame from relatively late time in each simulation is shown. A plot of the averaged bubble distribution at this time and the averaged velocity distribution is shown figure 4. For both system the initial channel Reynolds number is about a thousand, the system contains 32 bubbles of diameter less than one tenth of the channel width, in a domain bounded by walls in the horizontal direction and periodic boundary in the vertical direction. The surface tension is selected such that the bubbles

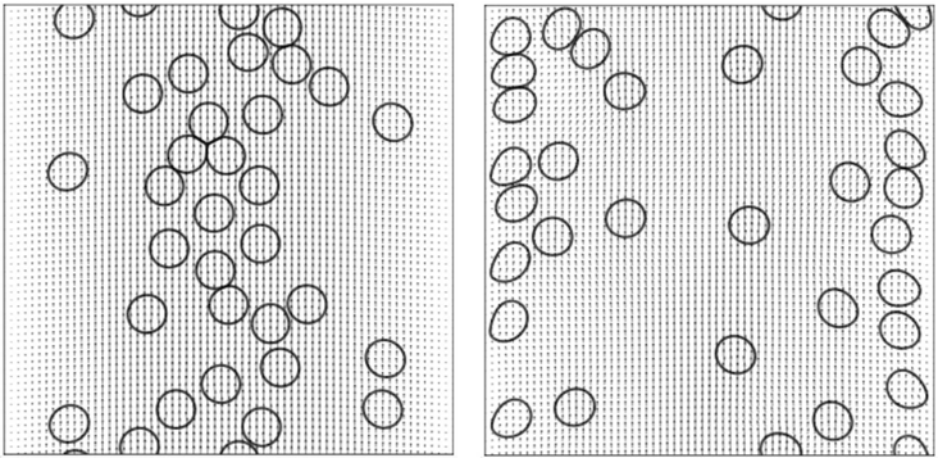

Figure 3 Two simulations of the rise of bubbles in upflow in a vertical channel. Gravity is zero in the left frame, but in the right frame the bubbles are buoyant and rise upward.

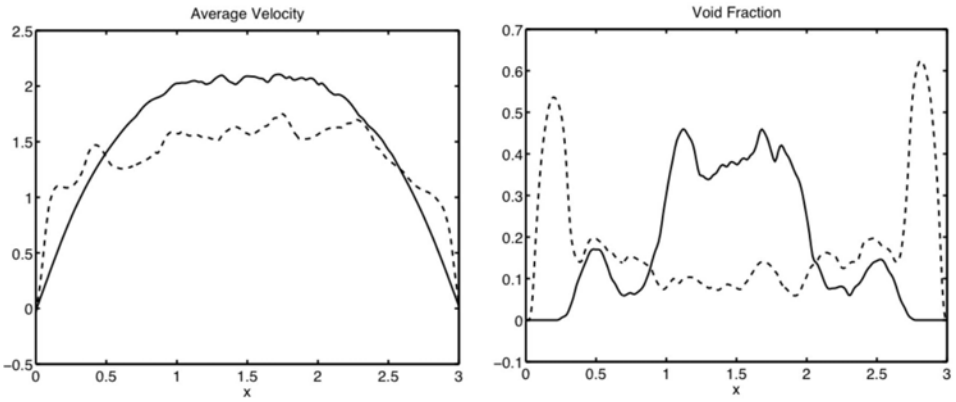

Figure 4 The average velocity (left) and the average void fraction (right) at one time for the simulations in figure 3. The solid lines correspond to the left frame in figure 3 and the dashed line corresponds to the right frame.

remains nearly spherical, except in the high shear regions near the walls. For the system on left, gravity is zero, but on the right the bubbles are buoyant and move upward relative to the liquid. It is clear that the phase distribution is very different, as expected. On the left where the bubbles move with the liquid velocity, they collect near the center of the channel. On the right, where the bubbles move faster than the liquid, while remaining nearly spherical except very close to the walls, the bubbles experience a lift force that pushes them out from the core, towards the walls. The wall layer does, however, lead to a relatively uniform velocity in the channel and the few bubbles left in the middle are now subject to a smaller shear and thus smaller lift. We therefore expect to see only a few bubbles remaining in the core. Laminar bubbly flows have been

examined experimentally by Nakoryakov *et al.* (1985) and Antal, Lahey, and Flaherty (1991) have developed and used a two-fluid model to analyse such flows. The model has been studied further by Azpitarte and Buscaglia (2003). The model showed good agreement with the experimental data and while the opportunities to refine the model by DNS is probably limited to the effect of deformable bubbles on the lift force and the wall repulsion force, this systems is an idela setting in which the bridging between DNS and two-fluid models can be studied. It should, in particular, be possible to examine the key question of how small a simulated system can be for the results of the model to apply. The results of figures 3 and 4 suggest that even a relatively small system is sufficient and we have recently started to compare the DNS results with two-fluid model predictions.

3.3 *Bubbles in turbulent channel flows*

Essentially all of our studies of bubbly flows have, so far, focused on relatively low Reynolds number flow where the flow is laminar or at least would be laminar in the absence of the bubbles. Yet, multiphase flows in practical applications are often turbulent. Direct numerical simulations, where the continuum length and time scales of turbulent flows are fully resolved are about twenty years old, and while DNS is limited to modest Reynolds numbers, such simulations have already yielded fundamentally new insight into the structure and behavior of turbulent flows. The potential for similar insight for turbulent bubbly flows therefore clearly exists. Indeed, many authors have examined the behavior of turbulent multiphase flow in the limit where the size of the disperse particles is small compared with smallest flow scales and the bubbles can be represented by a point particle. Although the interactions between the flow and the particle are modeled and not resolved, it is likely that the results are representative of the real behavior of very small particles. For bubbly flows, where bubble diameters of the order of a millimeter are common, the fundamental assumption of this approach fails and it is necessary to resolve fully the flow around each bubble, along with the turbulent flow.

We are currently conducting simulations of bubbles injected near the walls in a turbulent channel flow. The goal of this investigation is to cast some light on the mechanisms underlying drag reduction and to provide data that may be useful for the modeling of such flows. Experimental studies (see Merkle and Deutsch, 1990, Kato *et al.*, 1995, and Kodama *et al.,* 2002, for a review) show that the injection of a relatively small amount of bubbles into a turbulent boundary layer can result in a significant drag reduction. While the general believe seems to be that the bubbles should be as small as possible (a few wall units in diameter), drag reduction is found experimentally in situations where the bubbles are considerably larger (order of 100 wall units). Figure 5 shows one frame from a simulation of sixteen bubbles in the so-called "minimum turbulent channel" of Jimenez and Moin (1991). The dimensions of the channel are π units in the streamwise direction, $\pi/2$ in the spanwise direction and 2 in the wall normal direction. The channel is bounded by walls at the top and bottom and has periodic spanwise and streamwise boundaries. The wall Reynolds number is $Re^{+}=135$ and as initial conditions we use a fully turbulent flow computed using a spectral code by Professors Maxey and Karniadakis at Brown University. The computations were done using a grid of $256\times128\times192$ grid points, uniformly spaced in the streamwise and the spanwise direction but unevenly spaced in the wall normal direction. In addition to the

bubbles, isocontours of spanwise vorticity are also shown, with different color indicating positive and negative vorticity. The mean wall shear on the top and the bottom wall is plotted versus time in Fig. 2, for flow with and without bubbles. Since the flow structure is changing and the domain is fairly small, the total wall drag changes with time, even in the absence of bubbles. For flow without bubbles, however, the time-averaged value remains constant. After the bubbles are added at time zero, the drag gradually decreases and by time 40 the drag on both walls has been reduced by nearly 15%. Although the drag on the bottom wall continues to decrease, the drag on the top wall had increased again slightly near the end of the simulation. If the bubbles completely relaminarized the flow, the drag would be reduced to about one third of the total average drag in the absence of bubbles. While we frequently see drag reduction for deformable bubbles, nearly spherical bubbles generally lead to drag increase since they come closer to the wall and are slowed down by the viscous sublayer. The drag reduction induced by deformable bubbles seems to be primarily due to bubble-induced cancellation of streamwise vorticity with the wall bound vorticity of the opposite sign. We have repeated the simulation presented here on a coarser grid, continuing for a longer time, and find that eventually the bubbles drift away from the wall and the drag reduction disappears. While the results on the coarser grid are not as accurate, they agree reasonably well with the results on the finer grid for the early time and we thus believe that the coarse grid long time results are indicative of what would happen if the simulation in figure 5 and 6 were continued. To reduce the possibility that the results depend sensitively on the particular flow configuration at time zero, we have also inserted the bubbles at a later time (but using lower grid resolution) and found drag reduction. For other work on bubbles in a turbulent channel see Kanai and Miyata (2001) and Kawamura and Kodama (2002). Neither simulation did, however, show drag reduction.

We note that the bubbles that we are examining here are much larger than those studied by Ferrante and Elgobashi (2003), who recently simulate the effect of a large number of bubbles with a diameter of 2.4 wall units in a turbulent boundary layer using fully resolved flow and a point particle model for the bubbles. Such modeling, although clearly inappropriate for the larger bubbles examined here, may yield accurate results for such small bubbles.

3.4 *Generation of bubbles and complex flows*

Although opportunities for studies of two-fluid disperse systems are enormous, it is the longer-term development of the ability to compute the coupled motion of complex systems that will bring about the full impact of direct numerical simulations. Most engineering fluid systems include a large number of physical phenomena such as fluid flow, evaporation, solidification, and chemical reactions, and while the ability to examine each aspect in detail is important, simulations of the full system will allow unprecedented insight. Here, progress is just starting. A few investigators have, for example, simulated boiling flows (Son and Dhir, 1998, Juric and Tryggvason, 1998, Welch and Wilson, 2000, Shin and Juric, 2002, Esmaeeli and Tryggvason, 2003, Yoon, Koshizuka, and Oka, 2001, Kunugi, 2001, and Kunugi *et al*., 2002, Son, Ramanujapu, and Dhir, 2002), and the field is rapidly advancing. Understanding of boiling is particularly relevant for bubbly flows, as bubbles are frequently formed by boiling.

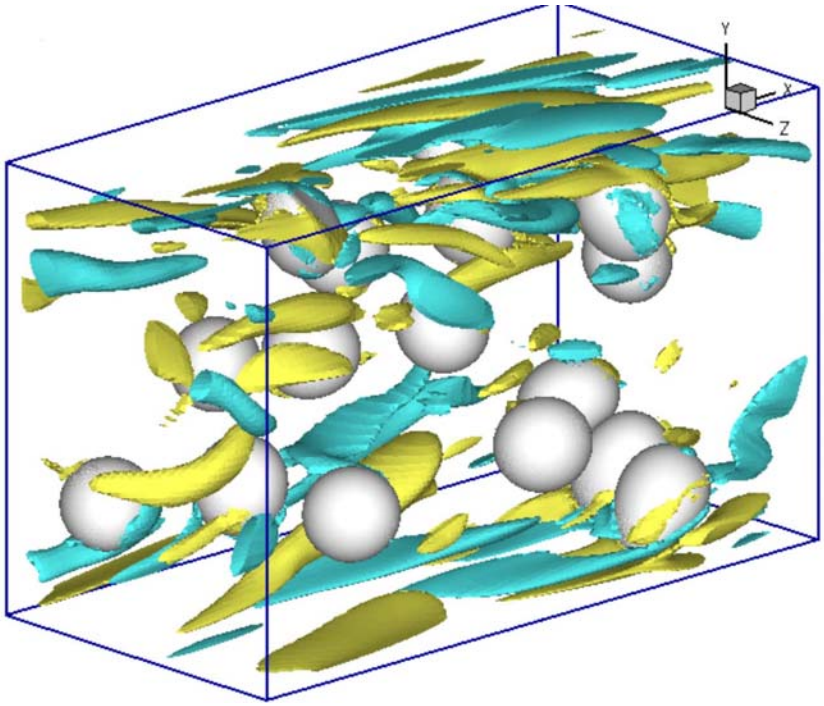

Figure 5 One frame from a simulation of the motion of 16 bubbles in a turbulent channel. The bubbles and isosurfaces of streamwise vorticity are shows.

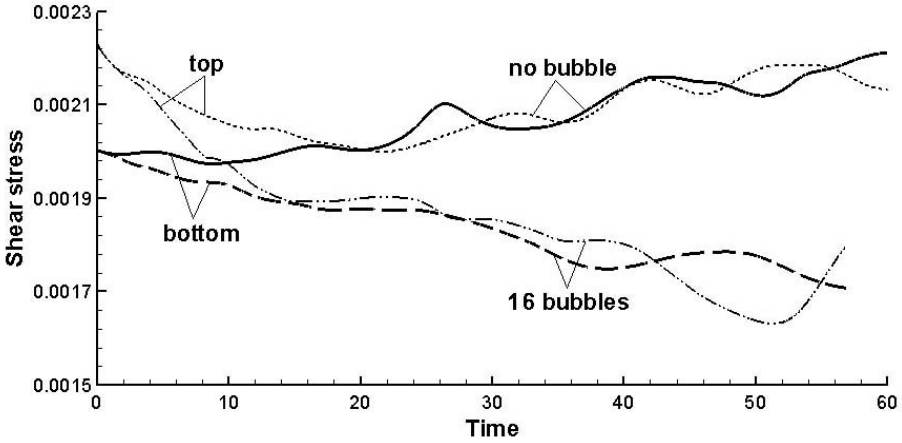

Figure 6 The mean wall shear on the top and the bottom wall versus time for turbulent channel flow with and without bubbles. For the no-bubbles case the average drag remains approximately constant but with bubbles there is significant reduction in the wall drag. The shear stress and the time are in computational units.

We have extended the method described in section 2 to boiling flows (Juric and Tryggvason, 1998; Esmaeeli and Tryggvason, 2003) and used it to examine a number of problems. Those include explosive boiling of a nucleus in initially superheated liquid (Esmaeeli and Tryggvason, 2003) and film boiling on a flat plate (Esmaeeli and Tryggvason, 2004). For film boiling we examined, in particular, how large systems, where bubbles broke off from the vapor film at the wall, differed from systems where the bubble growth was forced to take place at the linearly most-unstable wavelength. The main question was whether subharmonic instabilities would lead to competition between waves and the formation of larger bubbles. Such meargers are seen in many other systems, but here we found that such competition was relatively weak. Simulations with only one wave therefore generally resulted in heat transfer rates in good agreement with results for larger systems. We also examined the effect of the Jacob number on the boiling of a finite depth pool. For low wall superheat (but high enough to prevent wetting and a transition to nucleate boiling) bubbles broke away from the vapor film. At higher superheat the vapor production was sufficiently fast so that long vapor vents formed, sometimes reaching the surface of the pool. At even higher superheat the vents became unsteady, resulting in unsteady churn-like boiling. For moderate superheat where the flow time scale was much shorter than the time for total evaporation of the pool, the system reached a well-defined steady state, but at high superheat all the liquid quickly evaporated and no steady state emerged.

We have also recently started to study film boiling from one or more circular tubes. Since a heated wire or a rod is a common component in practical situations, a number of experimentalists have examined the problem. The ratio of the diameter of the wire to the capillary length scale defines limiting cases. As this ratio approaches infinity, we should approach results for flat walls (at least on the top of the cylinder) and when this ratio is very small, we expect the vapor layer to initially grow cylindrically, independent of the diameter of the cylinder. Figure 7 shows three frames from a simulation of vapor production from a heated cylinder. Initially, the cylinder is surrounded by a vapor film, whose radius is perturbed by a wave of the most-unstable capillary length. As the hot cylinder produces additional vapor and buoyancy lifts the vapor upward, a bubble is formed. Preliminary results indicate that the numerical predictions agree with experimental predictions for moderate sized cylinders. For more details see Esmaeeli and Tryggvason (2004).

Bubbles are, of course, also formed in other ways, such as by the injection of air through a hole in a wall or a needle sticking into the flow domain. Relatively little has, however, been done to investigate injection numerically. The exceptions include Takahira, Takahashi, and Banerjee (2004) who used a level set method to simulate the injection of air into a shear flow and the subsequent formation of a bubble, and Oguz and Prosperetti (1993) who simulated the injection of air through a needle, assuming inviscid flow and using a boundary integral method.

4. CONCLUSIONS

Direct numerical simulations of multiphase flows have come a long way during the last decade. It is now possible to follow the motion of hundreds of bubbles, drops, and

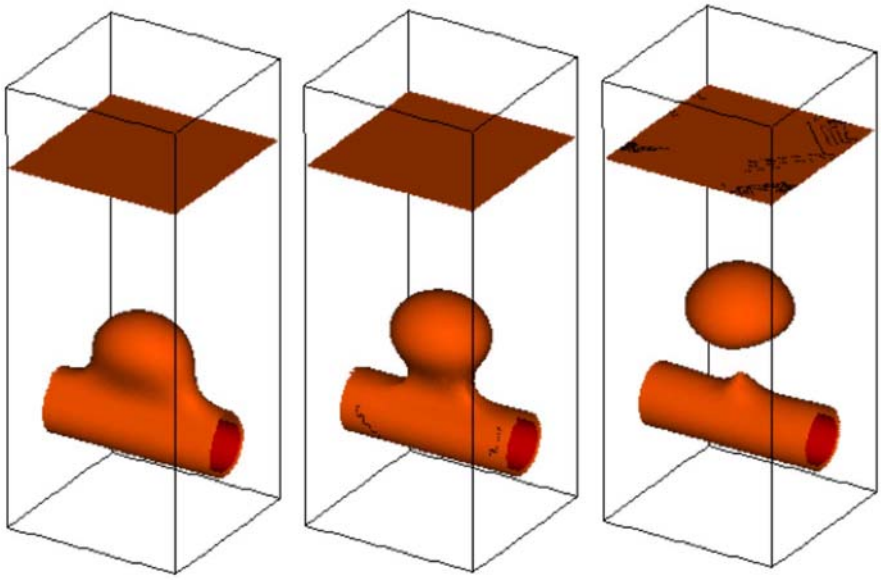

Figure 7 Three frames from a simulation of the formation of a vapor bubble from a hot cylinder.

particles at finite Reynolds numbers in simple geometries for sufficiently long time so that meaningful averages can be computed. Much remains to be done, however. Below we discuss a few of the critical issues that will have to be dealt with as we move forward.

There is still considerable need for continuing refinement and development of numerical methods. In addition to progress being made in dealing with parasitic currents, viscous stresses, advection at high Reynolds numbers, and the development of efficient pressure solvers for high density ratios, there has recently been considerable interest in developing so-called sharp interface methods where the interface is captured on a regular fixed grid without using a smooth approximation to the transition zone. These methods include the ghost fluid method (Fedkiw, Aslam, Merriman, and Osher, 1999; Fedkiw, 2002), the immersed interface method (Lee and LeVeque, 2003) and the method of Udaykumar and collaborators (Udaykumar, Mittal and Shyy, 1999). These methods hold the promise of higher accuracy, but as the improvement is likely to be more incremental than a transformation of the state-of-the-art, the wide acceptance of these methods is likely to depend critically on controlling their complexity. As the level-set method has shown, simplicity is an important factor in how widely methods are adopted.

In addition to major increase in the use of DNS for disperse two-fluid flows, the near future will also bring us many more simulations of much more complex multiphase systems. Those include systems where the phase boundary is more complex and the evolution of the system includes continuous breakup and coalescence, as well as systems with more complex physics, such as heat transfer and phase change, and the presence of contaminants and electric or magnetic fields. In many cases these problems are well within range and simply have not been addressed yet, such as three fluid problems with

both bubbles and drops (or solids) in the same fluid. In other cases further numerical development is needed. In addition to development of methods that allow us to simulate additional physics, it is also likely that the boundary between modeling and direct numerical simulations may become blurred. For boiling it is likely that thin films between the vapor bubble and the wall must be modeled (see Son and Dhir, 1998, for preliminary work) and for breakup problems very small drops may be best treated as point particles, even if everything else is well resolved.

The major difficulty in the wide spread adoption of DNS for multiphase flows research is actually likely to be not due to technical problems with the methods, but due to organizational and educational issue. To solve large and complex problems it is not unreasonable to expect to have to use fully parallel code with efficient adaptive mesh refinement, in addition to the ability to track the fluid interface. Although both parallel codes and AMR codes have been written several times, a new investigator wishing to enter the field faces formidable challenges in writing such codes. Indeed, it is only a minor exaggeration to say that it is probably considerably easier now to invent a new numerical technique than to assemble the pieces needed to do a very large problem. This needs to be dealt with and finding a way where numerical technology can be shared more widely is becoming a critical issue.

Another urgent issue is what to do with all the data that we are now able to produce. So far, DNS of multiphase flows have been used to produce relatively simple statistics, mostly focusing on fully developed statistically steady-states for homogenious flows. The two-fluid model provides the beginning of a theoretical framework where DNS results can be used to help close the equations. It is, however, also clear that much more work needs to be done to develop descriptions suitable to absorbe the DNS data. We have, for example, only rudimentary ideas of what large eddy modeling means in the context of general multiphase flows.

It seems fair to conclude that the future for DNS of multiphase flow looks exciting. Many of the technical difficulties with such simulations have been overcome and there are reasons to be optimistic that future challenges will similarly be overcome. The range of systems that can be studied by DNS is, essentially, limitless and as multiphase systems are an integral part of our very being, the impact of such studies on our ability to understand and control our environment in the broadest sense will be great indeed.

REFERENCES

Agresar, G. and J. J. Linderman and G. Tryggvason and K.G. Powell. An Adaptive, Cartesian, Front-Tracking Method for the Motion, Deformation and Adhesion of Circulating Cells. *J. Comput. Phys*. 43 (1998) 346—380.

Antal, S. P., R.T. Lahey and J.E. Flaherty. Analysis of Phase Distribution in Fully Developed Laminar Bubbly Two-Phase Flow. *Int. J. Multiphase Flow* 17 (1991), 635-652.

Azpitarte, O. E. and G. C. Buscaglia. Analytical and numerical evaluation of two-fluid model solutions for laminar fully developed bubbly two-phase flows. *Chemical Engineering Science* 58 (2003), 3765-3776.

Brackbill, J. U. and D. B. Kothe and C. Zemach. A Continuum Method for Modeling Surface Tension. *J. Comput. Phys*. 100 (1992), 335—354.

Bunner, B. Numerical Simulations of Gas-Liquid Bubbly Flows. Ph. D. Dissertation, The University of Michigan, 2000.

Bunner, B. and G. Tryggvason. Dynamics of Homogeneous Bubbly Flows: Part 1. Rise Velocity and Microstructure of the Bubbles. *J. Fluid Mech.* 466 (*2002),* 17-52.

Bunner, B. and G. Tryggvason Dynamics of Homogeneous Bubbly Flows. Part 2, Fluctuations of the Bubbles and the Liquid. *J. Fluid Mech* 466 (*2002),* 53-84.

Bunner, B. and G. Tryggvason. Effect of bubble deformation on the properties of bubbly flows *J. Fluid Mech* 495 (2003), 77-118.

Choi, H. G. and D. D. Joseph. Fluidization by lift of 300 circular particles in plane Poiseuille flow by direct numerical simulation. *J. Fluid Mech*. 438 (2001), 101—128

de Vries, A. W. G., A. Biesheuvel, and L. van Wijngaarden. Notes on the path and wake of a gas bubble rising in pure water. *Int. J. Multiphase Flow*. 28 (2002), 1823—1835.

Ellingsen, K. and F. Risso. On the rise of an ellipsoidal bubble in water: oscillatory paths and liquid-induced velocity. *J. Fluid Mech*. 440 (2001), 235—268.

Ervin, E. A. and G. Tryggvason. The Rise of Bubbles in a Vertical Shear Flow. *ASME J. Fluid Engineering*, 119 (1997), 443—449.

Esmaeeli, A. and E. Ervin and G. Tryggvason. Numerical Simulations of rising bubbles. In Bubble Dynamics and Interface Phenomena. Editors: J. R. Blake and J. M. Boulton-Stone and N. H. Thomas. Kluwer 1994, 247—255.

Esmaeeli, A. and G. Tryggvason, "Direct Numerical Simulations of Bubbly Flows. Part I—Low Reynolds Number Arrays." *J. Fluid Mech*. 377 (1998), 313-345.

Esmaeeli, A. and G. Tryggvason, "Direct Numerical Simulations of Bubbly Flows. Part II—Moderate Reynolds Number Arrays" *J. Fluid Mech*. 385 (1999), 325-358.

Esmaeeli, A. and G. Tryggvason. Computations of explosive boiling in microgravity. *Journal of Scientific Computing*, 19 (2003) 163-182.

Esmaeeli, A. and G. Tryggvason. A Front Tracking Method for Computations of Boiling in Complex Geometries. *Int. J. Multiphase Flow.* 30 (2004) 1037–1050.

Esmaeeli, A. and G. Tryggvason. Computations of Film Boiling. Part I, Numerical Method. *Int. J. of Heat and Mass Transfer* 47 (2004), 5451-5461.

Esmaeeli, A. and G. Tryggvason. Computations of Film Boiling. Part II, Multi Mode Film Boiling. *Int. J. of Heat and Mass Transfer* 47 (2004), 5463-5476.

Fedkiw, R., T. Aslam, B. Merriman, and S. Osher. A Non-Oscillatory Eulerian Approach to Interfaces in Multimaterial Flows (The Ghost Fluid Method). *J. Comput. Phys*. 152 (1999), 457-492.

Fedkiw, R. Coupling an Eulerian Fluid Calculation to a Lagrangian Solid Calculation with the Ghost Fluid Method. *J. Comput. Phys*. 175 (2002), 200-224.

Feng, J. and H. H. Hu and D. D. Joseph. Direct Simulation of Initial Value Problems for the Motion of Solid Bodies in a Newtonian Fluid, Part 1. Sedimation. *J. Fluid Mech*. 261 (1994), 95--134

Feng, J. and H. H. Hu and D. D. Joseph.Direct Simulation of Initial Value Problems for the Motion of Solid Bodies in a Newtonian Fluid, Part 2. Couette and Poiseuilli Flows. *J. Fluid Mech*. 277 (1995), 271—301.

Fortes, A., D.D. Joseph and T. Lundgren. Nonlinear mechanics of fluidization of beds of spherical particles. *J. Fluid Mech*. 177 (1987), 467-483.

Ferrante, A. and S.E. Elghobashi. On the physical mechanisms of drag reduction in a spatially developing turbulent boundary layer laden with microbubbles. *J. Fluid Mech*. 503 (2004), 345-335.

Ferziger, J. H. Interfacial transfer in Tryggvason's method. International Journal for Numerical Methods in Fluids. 41 (2003), 551 – 560.

Goz, M. F. and B. Bunner and M. Sommerfeld and G. Tryggvason. EDSM2000-11151: The Unsteady Dynamics of Two-Dimensional Bubbles in a Regular Array. Proceedings of the ASME FEDSM'00 ASME 2000 Fluids Engineering Division Summer Meeting June 11-15, 2000, Boston, Massachusetts",

Hu, H. H. Direct Simulation of Flows of Solid-Liquid Mixtures. *Int. J. Multiphase Flow*. 22 (1996), 335-352.

Jimenez, J. and P. Moin. The minimal flow unit in near-wall turbulence. *J. Fluid. Mech*. 225 (1991), 213-240.

Johnson, A. A. and T.E. Tezduyar. 3D simulation of fluid-particle interactions with the number of particles reaching 100. *Comput. Methods Appl. Mech. Engng*. 145 (1997), 301—321.

Juric, D. and G. Tryggvason. Computations of Boiling Flows. *Int. J. Multiphase Flow*. 24 (1998), 387-410.

Kanai, A. and H. Miyata. Direct numerical simulation of wall turbulent flows with microbubbles. *Int. J. Num. Meth. Fluids*, 35 (2001), 593—615.

Kato, H., M. Miyanaga, H. Yamaguchi, and M. M. Guin. Frictional drag reduction by injecting bubbly water into turbulent boundary layer and the effect of plate orientation. In *Proceedings of the* 2*nd International Conference on Multiphase Flow* '95, Kyoto, 1995; 31–38, ICMF95.

Kawamura, T. and Y. Kodama. Numerical simulation method to resolve interactions between bubbles and turbulence. *Int. J. of Heat and Fluid Flow*, 23 (2002), 627–-638",

Kobayashi, K., Y. Iida and N. Kanegae. Distribution of local void fraction of air-water two-phase flow in a vertical channel. *Bull. JSME* (1970), 1005-1012.

Kodama, Y., Kakugawa, A., Takahashi, T., Nagaya, S. and Sugiyama, K. Microbubbles: Drag Reduction and Applicability to Ships. In *Twenty-Fourth Symposium on Naval Hydrodynamics* (2003). Naval Studies Board (NSB). Available at: http://books.nap.edu/books/NI000511/html/

Kunugi, T. MARS for Multiphase Calculation. *Computational Fluid Dynamics Journal* 9 (2001), 563-571.

Ladd, A. J. C. Dynamical simulations of sedimenting spheres. *Phys. Fluids* A 5 (1993), 299—310.

Lafaurie, B. and C. Nardone and R. Scardovelli and S. Zaleski and G. Zanetti. Modelling Merging and Fragmentation in Multiphase Flows with SURFER. *J. Comp. Phys*. 113 (1994) 134—147.

Lee, L. and R.J. LeVeque. An immersed interface method for incompressible Navier—Stokes equations. *SIAM Journal on Scientific Computing*, 25:832--856, 2003.

Nakoryakov, V. E., et al. Study of upward bubbly flow at low liquid velocities. *Izv. Sib. Otdel. Akad. Nauk SSSR* (1986), 15-20 (ref in Antal *et al*.)

Merkle, C. L. and S. Deutsch. Drag reduction in Liquid Boundary Layers by Gas Injection. *Progress in Astraunautics and Aeronautics* 123 (1990), 351-412.

Oguz, H. and A. Prosperetti. Dynamics of bubble growth and detachment from a needle. *J. Fluid Mech*., 257:111--145, 1993.

Oshinowo, T. and M. E. Charles. Vertical two-phase flow, Part 1. Flow pattern correlations. *Can. J. Chem. Engng*. 52 (1974), 25-35.

Pan, T. W. and D. D. Joseph and R. Bai and R. Glowinski and V. Sarin. Fluidization of 1204 spheres: simulation and experiment. *J. Fluid Mech*. 451 (2002), 169—191.

Patankar, S. V. Numerical Heat Transfer and Fluid Flow. Taylor & Francis, 1980

Renardy, Y. and M. Renardy. PROST: A Parabolic Reconstruction of Surface Tension for the Volume-of-Fluid Method. *J. Comput. Phys*, 183 (2002), 400-421.

Roma, A. M. and C.S. Peskin and M.J. Berger. An Adaptive Version of the Immersed Boundary Method. *J. Comput. Phys*. 153 (1999), 509—534.

Sangani, A. S. and A.K. Didwania. Dynamic simulations of flows of bubbly liquids at large Reynolds numbers. J. Fluid Mech. 250 (1993), 307—337.

Serizawa, A., I. Kataoka and I. Michiyoshi. Turbulence structure of air-water bubbly flow II Local properties. *Int. J. Multiphase Flow* 2 (1975), 235--246.

Serizawa, A., I. Kataoka and I. Michiyoshi. Turbulence structure of air-water bubbly flow III Transport properties. *Int. J. Multiphase Flow* 2 (1975), 247--259.

Smereka, P. On the motion of bubbles in a periodic box. J. Fluid Mech. 254 (1993) 79--112

Son, G. and V.K. Dhir. Numerical simulation of film boiling near critical pressures with a level set method. *J. Heat Trans*. 120 (1998) 183—192.

Son, G. and N. Ramanujapu and V. K. Dhir. Numerical simulation of bubble merger process on a single nucleation site during pool nucleate boiling. *ASME J. Heat Transfer*, 2002), 51—62.

Shin, S. W. and D. Juric. Computation of Microstructure in Solidfication with Fluid Convection. *Journal of the Mechanical Behavior of Materials*. 11 (2000), 313.

Sussman, M., A.S. Almgren, J.B. Bell, P. Colella, L. Howell, and M. Welcome. An adaptive level set approach for incompressible two-phase flow, *J. Comp. Phys*., 148 (1999), 81-124.

Takada, N. and M. Misawa and A. Tomiyama and S. Fujiwara. Numerical simulation of two- and three-dimensional two-phase fluid motion by lattice Boltzmann method. *Comput. Phys. Communications* 129 (2000), 233—246.

Takada, N. and M. Misawa and A. Tomiyama and S. Hosokawaî. Simulation of bubble motion under gravity by lattice Boltzmann method. *J. Nucl. Sci. and Technology*. 38 (2001), 330—341.

Takahira, H., M. Takahashi, and S. Banerjee. Numerical analysis of three dimensional bubble growth and detachment in a shear flow. In *Proc. of ICMF Int'l Conference on Multiphase Flow*, 2004.

Tomiyama, A. and I. Zun and A. Sou and T. Sakaguchi. Numerical Analysis of bubble motion with the VOF method. *Nucl. Engr. and Design*, 141 (1993), 69—82.

Tryggvason, G., B. Bunner, A. Esmaeeli, D. Juric, N. Al-Rawahi, W. Tauber, J. Han, S. Nas, and Y.-J. Jan. A Front Tracking Method for the Computations of Multiphase Flow. *J. Comput. Physics* 169 (2001), 708–759

Udaykumar, H. S., R. Mittal and W. Shyy. Computation of solid-liquid phase fronts in the sharp interface limit on fixed grids, *J. Comp. Physics*, 153 (1999), 535-574.

Unverdi, S. O. and G. Tryggvason. A front-tracking method for viscous, incompressible, multi-fluid flows, *J. Comput Phys*. 100 (1992), 25-37

Wang, S. K., S. J. Lee, O. C. Jones Jr., and R. T. Lahey Jr. 3-D turbulence structure and phase distribution in bubbly two-phase flows. *Int. J. Multiphase Flow* 13 (1987), 327--343.

Welch, S. W. J. and J. Wilson. A volume of fluid based method for fluid flows with phase change. *J. Comput. Phys*. 160 (2000) 662-682.

Yoon, H. Y., S. Koshizuka and Y. Oka. Direct calculation of bubble growth, departure, and rise in nucleate pool boiling. *Int. Journal of Multiphase Flow*. 27 (2001), 277-298.