The Effect of Bubbles on Near-Wall Vortical Flows

Gretar Tryggvason, Jiacai Lu, Arturo Fernández, and Asghar Esmaeeli Department of Mechanical Engineering Worcester Polytechnic Institute 100 Institute Road. Worcester Worcester, MA 01609

The effect of bubbles on the evolution of vortical flows near a wall are studied by direct numerical simulations, using a finite volume/front tracking technique that accounts fully for the effect of fluid inertia, viscosity, bubble deformability, and surface tension. Two problems have been studied. In one, the interaction of bubbles with a well-defined vortical flow, consisting of a parabolic velocity profile and a pair of counter-rotating straight vortex filaments near a wall, parallel to the flow direction, is followed. For a wide range of injection sites and bubble sizes, as well as for different number of bubbles, the motion of the bubbles into the vortex core leads to a cancellation of the original vorticity with secondary wall vorticity, resulting in a small transient reduction of the wall shear. In the other study, bubbles are injected near the wall in a turbulent channel flow. The evolution of the bubbles and the modification of the flow is followed as the bubbles migrate away from the wall.

1. INTRODUCTION

Experimentally it has been known for quite some time that the injection of microbubbles into a turbulent boundary layer can lead to a significant drag reduction. The earliest study appears to be McCormick and Bhattacharyya (1973) who found that microbubbles generated by electrolysis reduced the drag on a submerged body. Subsequent investigations by Magdavan, Deutsch, and Merkle (1984, 1985) using a flat plate mounted horizontally with bubbles injected below it found drag reduction of up to 80%. See also Merkle and Deutch (1990) for a review of the work at Penn. State. While the early studies were motivated by high-speed naval vessels or projectile, later work by Japanese researchers has focused on commercial vessels, where viscous drag accounts for most of the total drag. For an overview of this work see Kato *et al.* (1995) and Kodama *et al.* (2002). Although theoretical models based on mixing length theories have been proposed (Legner, 1984, for example), the mechanism responsible for the drag reduction remains essentially unknown. While the general assumption seems to be that the bubbles modify the turbulent structure in the buffer layer, it has also been speculated that the drag reduction is simply due to an air film near the wall. Such a film is certainly possible but does not seem to be supported by experimental observation, since a relatively small amount of bubbles injected into a turbulent boundary layer has been shown to reduce drag. The uncertainty about the exact mechanisms does call for a more detailed examination of the problem

It is well known that bubbles can have significant effect on vortical flows. In the simplest case, the bubbles simply reduce the average density of the liquid and can lead to baroclininc vorticity generation on a scale much larger than the bubble size. This is, for example, the case when a bubble cloud rises in an otherwise bubble free liquid. Bubbles rising through liquid can generate considerable amount of turbulence, as discussed by Lance and Bataille (1991). Sridhar and Kartz (1999) have shown that even very few bubbles entrained into a large vortex can affect the vortical structure in a very significant way. It is also known that it is possible to manipulate turbulent boundary layers in various ways to reduce the wall drag. Du and Karniadakis (2000) found, for example, that transverse-traveling waves could generate up to 50% drag reduction and several researchers (see Min *et al.*, 2003, for a recent contribution) have shown that the addition of polymers into a turbulent boundary layer can reduce drag significantly.

Direct numerical simulations of multiphase flow, where the full continuum equations are solved on a computational grid, sufficiently fine to resolve all continuum scales, date back to the origin of computational fluid dynamics (CFD) at Los Alamos in the early and mid sixties. The difficulty of following the deformation of an unsteady fluid interface separating phases of different properties, and limited computer power restricted the complexity of the systems that could be examined until relatively recently. During the last decade, however, major progress has been made and it is now possible to follow the unsteady motion of dispersed systems with O(100) bubbles, drops, and particles over sufficiently long time that meaningful statistical quantities can be computed (for a review see Tryggvason *et al*, 2001).

A number of authors have examined the behavior of bubbles in turbulent flows using numerical simulations. In most cases, the turbulent flow is resolved fully but the bubbles are modeled as point particles. In some cases the bubbles are assumed to be passive with respect to the fluid, but in other cases the influence of the bubbles is included

as a force added to the Navier-Stokes equations (two-way coupling). For simulations using point particles to model the bubbles, see, for example, Wang an Maxey (1993), Spelt and Biesheuvel (1990), Squires and Eaton (1990), Elghobashi and Trusdell (1993), and Climent and Magnaudet (1999). Direct numerical simulations of the motion of many bubbles, where the flow around each bubble is fully resolved, are more recent and have not, in most cases, included fully turbulent flow. See, for example, Esmaeeli and Tryggvason (1996, 1998, 1999) and Bunner and Tryggvason (1999, 2002a,b, 2003). Direct numerical simulations of bubbles in a turbulent wall-bounded flow have been done recently by Kanai and Miyata (2001) and Kawamura and Kodama (2002) who examined how bubbles modified a channel flow. Xu, Maxey, and Karniadakis (2002) have examined the effect of bubbles on the wall friction in a turbulent channel flow using the so-called force-coupling method (FCM) where the effect of a finite size spherical particle is represented approximately. For bubbles initially near the walls, Xu *et al.* found some reduction in the wall friction.

2. NUMERICAL METHOD

The simulations reported here have all been done using a method outlined in Unverdi and Tryggvason (1992) and Tryggvason *et al.* (2001). However, several refinements have been made for the simulations of bubbles in turbulent channel flow conducted here. The method is based on explicit tracking of the bubble surface and a full resolution of the flow field, both inside and outside each bubble. Full resolution of the dynamics of many deformable bubbles is therefore possible. A single set of equations is solved for both the liquid and the gas, and the phase boundary is treated as an imbedded interface by adding the appropriate source terms to the conservation laws. These source terms are in the form of delta-functions localized at the interface and are selected in such a way to satisfy the correct matching conditions at the phase boundary. While this approach dates back to the original MAC method developed at Los Alamos in the early sixties, several recent embodiments have successfully increased the accuracy of the original approach significantly. The front-tracking method used here has been one of the most successful of these new methods.

The "one-fluid" Navier-Stokes equations are (Unverdi and Tryggvason, 1992):

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

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 **x**_f is the position of the front.

The Navier-Stokes equations are solved by a second-order accurate projection method, using centereddifferences on a fixed, staggered grid. In order to keep the boundary between the phases sharp, and to accurately compute the surface tension, the phase boundary is tracked by connected marker points (the "front"). 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 gradient of the density and viscosity becomes a delta function 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), Tryggvason *et al.* (2001), and Fernández, Lu and Tryggvason (2002, 2003). We note that the only other simulations of fully deformable bubbles in turbulent flows by Kanai and Miyata (2001) and Kawamura and Kodama (2002) have been done using a somewhat similar approach.

The method has been implemented in several codes, most recently as a fully parallel code, written in Fortran 90/95, that has been run on various parallel computers, including IBM SP2s at several institutions and on the Blue Horizon at SDSC. This code includes data structures that allow very complex regions of different properties. For the simulations done here, two major changes have been necessary. As the Reynolds number is increased, the resolution requirement increases, particularly at the wall. We have therefore changed the code to accommodate nonuniform grids in the direction normal to the wall. While we have previously done so for simple codes for two-dimensional systems, the implementation in the fully parallel code was more complex. As the Reynolds number increases the

demands on the advection solver also increases and we have implemented a high order upwind scheme to allow us to accurately deal with such systems. Higher order upwind schemes generally require a broader stencil than the centered difference scheme used earlier in the code and implementing the new scheme therefore affects the parallelization as well.

100y+

Figure 1. A schematic of the initial conditions used.

3. INTERACTION OF BUBBLES WITH WELL-DEFINED VORTICAL STRUCTURES Experiments and direct numerical simulations of wall-bounded turbulent flows show that the near wall region is dominated by streamwise vortices. It therefo

region is dominated by streamwise vortices. It therefore seems reasonable to attempt to explain how bubbles modify the turbulent boundary layer by examining how bubbles interact with well-defined vortical structures. The vortical flow that we have chosen to look at here is a periodic row of line vortices, located near a wall and oriented parallel to the flow. The sign of the vorticity alternates so each periodic domain has a pair of counterrotating vortices. We examine a domain that is bounded by a no-slip wall at the bottom, a full-slip wall at the top and has periodic horizontal boundaries. The flow is driven by a prescribed pressure gradient in the streamwise direction. Ideally, we would like to inject the bubbles into a steady-state flow, but as viscosity will diffuse the vortices, we have to settle for a quasy-steady flow where the vortices are decaying slowly. Setting up such a flow is not completely trivial, however. While we could set up an inviscid steady-state flow consisting of a shear and streamwise vortices, adding viscosity prevents

the emergence of a steady state. Putting streamwise vortices into a shear flow has two effects. First of all, by drawing high-speed flow toward the wall and ejecting low speed fluid away from the wall, the vortices change the mean flow profile. Secondly, the no-slip boundary conditions and viscous diffusion leads to the development of secondary streamwise vortices at the wall. Such secondary vortices are, of course, also seen in real turbulent flows. For our purpose we want to inject the bubbles into a flow that has a modified mean velocity and where the secondary vorticity is present. We generate the initial flow field in two steps: First we specify a vorticity distribution in a plan normal to the flow direction, where each vortex is given by a Lamb-Oseen vortex, with Gaussian vorticity distribution:

$$\omega(r) = \pm \frac{\Gamma}{\pi a^2} e^{-r^2/a^2} \qquad (r < r_0)$$

$$\omega(r) = 0 \qquad (r \ge r_0)$$

Here r is the radial distance from the center of each vortex, and a is the characteristic core radius. The distance



Figure 2. The initial vorticity distribution in a plane normal to the flow direction.

between the two vortex cores, the core radius and the distance from the wall are chosen to be comparable to that in the turbulent boundary layers. Under the boundary condition of no-slip wall, the two primary vortices induce two secondary vortices of opposite sign at the wall. The vorticity field is then advanced in time by integrating the Navier-Stokes equations in vorticity-stream function form to allow the secondary vorticity at the wall to form. As the primary vortices are placed symmetrically in the domain, they do not move. Once the vorticity field has reached an approximate steady state, the velocity in the plane is "frozen." The initial streamwise velocity filed is then specified by a parabolic velocity profile, $w(y) = 1 - (1 - y)^2$, in a $1 \times 1 \times 1$ domain. Using the frozen velocity in the normal plane, the Navier-Stokes equations for the streamwise velocity are evolved to steady state. The pressure gradient is selected in such a way that the initial parabolic velocity would be a steady-state. The presence of the vortices increases the drag and as the vortices diffuse, initially the drag decreases slightly as the flow accelerates, but eventually the drag increases again as the



Figure 3. Two frame from a simulation of the interaction of bubbles with well-defined vortical structures.

vortices disappear and the flow returns to the parabolic profile.

While we have done one simulation of the flow without bubbles to confirm that this is what happens, the time scale of this evolution is much longer that it takes the bubbles to disrupt the vortices. Thus, we are primarily concerned with the time when the vortices, and the wall drag, are decreasing slightly with time and the question of how injecting bubbles modifies the vortex strength and the drag. The computational setup is shown schematically in figure 1, where the initial streamwise velocity is shown along with isovorticity surfaces and a few bubbles. The steady state streamwise vorticity is shown in figure 2.

A more detailed description of this study is in preparation and here we will only show a few representative results and summarize the main findings. We have examined the evolution of the drag and other quantities for a wide range of bubble sizes, initial locations, numbers, vortex strengths, and Reynolds numbers. Figure 3 shows two frames from a simulation of two bubbles initially placed near a vortex pair in a domain that is 72.4×72.4×21.7 wall units large. The bubble diameter is 14.5 wall units and the vortices are initially placed 14.5 wall units from the bottom. As the bubbles are rotated around the vortices, they are also drawn into the low-pressure region at the vortex cores. In the left frame the bubbles have been drawn about half way into the vortex core and in the right frame the bubbles are essentially fully captured. In this particular simulation the bubbles are not torn apart, have relatively little impact on the evolution. In figure 4 the path of three bubbles, with different deformability is shown. As the bubbles are drawn into the vortices, they are also advected downstream with the flow and it is clear from the figure that deformation have essentially no effect on the path. Time is implicit in this figure, but plotting the coordinates of the centroid of the bubbles versus time shows the same results.

Figure 5 shows a few results addressing the effect of bubble size. In the top frame one component of the Reynolds stress, <vw>, where v is the streamwise and w is the wall normal velocity components resulting in the transfer of fast moving fluid toward the wall, is plotted versus time. It is clear that adding bubbles result in a reduction



Figure 4. The path of three bubbles with different deformability.



Figure 5. The Reynolds stress (top frame) and the wall drag versus time for three different bubble sizes.

of the Reynolds stress that increases with the size of the bubbles. It is important to note that the bubbles simulated here are relatively small and in the limit of zero size bubbles there would be no effect on the flow. For much larger bubbles, sometimes the opposite effect is seen. The bottom frame shows the drag on the bottom wall as a function of time. As anticipated, the reduction in the Reynolds stresses results in a transient drag reduction as the bubbles are engulfed into the vortices. Once the bubbles have been fully drawn to the center of the vortices, no further effect is observed.

These simulations, and a large number of others, have consistently shown a relatively small transient drag reduction as bubbles are drawn into well-defined vortex structures near the walls. While this effect is sometimes countered by drag increase due to bubbles being "jammed" to the wall by the vortices, we have seen drag reduction for the vast majority of situations that we have simulated. The explanation appears to be that as the bubbles are drawn into the vortices, the disruption that they cause leads to mixing and mutual cancellation of the primary vorticity with the secondary vorticity at the wall. The cancellation leads to a reduction of strength of the primary vortex and the Reynolds stresses and thus a transient reduction of

drag. Once the bubbles have been fully engulfed by the vortices, no further effect is seen. Indeed, putting bubbles initially at the center of the vortex has essentially no effect on the drag. The primary problem with the proposed mechanism is that it generally results in relatively small drag reduction for a short time. However, turbulent flow are considerably more complex than the simple flow used here and if the mechanism proposed here is operational, we would expect it to be more efficient in flows with more complex vortex structures. To examine that possibility we have done a few simulations with two pairs of vortices as well as artificially generated hairpin vortices. In both cases do we see significantly larger drag reduction.

4. BUBBLES IN TURBULENT CHANNEL FLOW



Figure 6. One frame from a simulation of the evolution of 50 bubbles in a turbulent channel flow. The bubbles and the streamwise vorticity are shown.

While the study described above has given us some insight into how bubbles behave in complex vortical flows, real turbulent flows are, of course, much more complex. To address the full problem, we have done several simulations of bubbles in a turbulent channel flow. Because of the high resolution requirement of such simulations, as well as the difficulty in fully resolving flow at turbulent Reynolds numbers, we have started our investigation by conducting those simulations 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 streamwise direction and 2 in the wall normal direction. The channel is bounded by walls at the top and the bottom and has periodic spanwise and streamwise boundaries. By careful numerical studies, Jimenez and Moin (1991) showed that turbulent flow can be sustained in this channel at wall Reynolds numbers of 3000. In terms of wall units, the dimensions of the channel are 424, 212, and 270. The wall Reynolds number is Re⁺=135. As initial conditions we use a



Figure 7. The wall-drag versus time for flow with and without bubbles. Note that that the scale has been amplified and that the total drag reduction is relatively small.

fully turbulent flow computed using a spectral code by Professors M. Maxey and G. Em. Karniadakis at Brown University. In the units used for the simulations, the kinematic viscosity was 3.33×10^4 , resulting in an average channel velocity of 0.667, for an average pressure gradient of 0.00179. The time for a fluid particle moving with the average velocity to go through the channel one is therefore 4.71. The bubbles, located near the wall will, of course, move slower. The computations were done using a grid of 256, 128, and 192 grid points, uniformly spaced in the streamwise and the spanwise direction but unevenly spaced in the wall normal direction. The smallest cell near the wall was 0.415 units thick and the largest one, at the center of the channel was 1.670 units thick. The initial data was computed using 65 by 65 modes, and was interpolated to generate initial data on the finer grid used in our simulations. The turbulent initial data was done specifying a constant volume flux but we have done simulations both using a constant pressure gradient as well as constant flow rate. We have also continued the turbulence simulation without bubbles to confirm that our code preserves the statistics of the flow. Several years ago there was some debate about the use of second order methods for turbulence simulations but our tests, in agreement with other recent work such as Orlandi (2000), confirmed that such methods indeed give results comparable to those produced by higher order spectral codes.

Several simulations have been done for different bubble sizes, numbers, and deformability (changed by varying the surface tension) as well as initial location. While many simulations were carried out for relatively short times, a few runs were done for a longer time. Force Coupling Method calculations by Xu, Maxy, and Karniadakis (2002) have shown that drag reduction is obtained almost immediately, for those cases where drag reduction is observed. For a given pressure gradient the total wall drag plus the total acceleration of the fluid in the channel must balance the



Figure 8. The averaged velocity at an early time for flow with and without bubbles.

imposed pressure gradient and we checked that this was true as the flow evolved. To study the evolution we monitor several averaged quantities as the simulation progress and save a complete dataset at regular time intervals. Figure 6 shows the bubbles and the stramwise velocity in a plane near the bottom wall are plotted in both cases at a relatively early time for one of our simulations. Here we are following 50 bubbles, initially located near the bottom wall. The bubbles have a radius of $a^+ = 13.75$ wall units (0.1 in simulation units) and are put at 37.125 wall units from the walls, initially. Here we have taken the bubbles to be ten times more viscous than the ambient fluid to make them "solid-like." Small air bubbles in water usually have a nearly immobile surface due to contaminants and while the effect is not exactly the same as increasing the viscosity, it is considerably easier to implement that numerically. Initially the bubbles are located in a square array but as the flow evolves and the bubbles move downstream, the turbulence perturb both their location with respect to each other and their distance from the wall.



Figure 9. The velocity fluctuations (left frame) and the turbulent Reynolds stresses (right frame).



Figure 10. The vortical field, with and without bubbles, visualized by plotting the λ_2 field.

For this simulation the flow rate is kept constant by adjusting the driving pressure at each time step. The wall drag is shown versus time in figure 6 and for this particular case we see a slight drag reduction. The reduction is, however, small and transient. For bubbles in the minimum channel we have not seen any significant reduction in the drag and sometimes we see a slight increase, depending on the particular configuration of each simulation. Indeed, the effect of the bubbles on the average flow quantities is usually small. In figure 8 the average velocity is shown, computed at the same time for flow with and without bubbles and in figure 9 (left frame) the velocity fluctuations, in the streamwise, spanwise and the wall-normal direction are plotted. One component of the Reynolds stresses, <u'v'>, is plotted in right frame. The average fluctuations are found by subtracting the velocity averaged over planes parallel to the wall at any given time from the pointwise velocity, and then averaging the fluctuations over planes parallel to the wall. The figure shows that while there is a significant asymmetry in the streamwise fluctuations between the top and the bottom wall, there is little systematic change as the bubbles are added or the flow evolves. A slightly larger difference is seen in the Reynolds stresses, where adding the bubbles has reduced the turbulent stresses slightly. This component is obviously responsible for transfer of momentum to the wall, so we expect those to correlate with changes in the wall drag. We note that simulations with larger bubbles and more deformable bubbles showed essentially the same behavior.

Figure 10 shows one example of a visualization of the flow field. Here, we use the so-called lambda-2 method (Jeong and Hussain, 1995) to identify

vortical structures in the flow. In this approach, which attempts to identify vortices as regions of both vorticity and low pressure, isosurfaces of the second eigenvalue of $S^2 + \Omega^2$ where S is the symmetric part of the deformation vector and Ω is the antisymmetric part. The vortices in the flow without bubbles are shown in the top frame and with bubbles in the bottom frame. It is clear that the bubbles have disrupted the coherence of the vortical flow, in agreement with what was found in the first part of the investigation. Similar conclusions can also been drawn from plots of the vorticity in various cross sectional planes through the domain (not shown).

5. CONCLUSIONS

While the simulations reported here of bubbles in the minimum turbulent channel, as well as other simulations by other researchers, have shown the feasibility of conducting direct numerical simulations of fully resolved bubbles in turbulent flows—and the results allow us to explore the change in averaged turbulent quantities as the bubbles modify the flow—the absence of drag reduction has obviously been unexpected, particularly in view of the findings from the simpler model problem presented in the first part of the paper. The minimum channel is, however, a somewhat peculiar flow, exhibiting strong anisotropy between the top and the bottom wall and while we expect the effect of the bubbles on the flow to be well captured, the modifications of the bubbles on the flow may not be representative for higher Reynolds numbers and larger channels. Indeed, after we obtained our first results the Brown team ran their FCM for exactly the same situation (the minimum channel) and did not find drag reduction. As they found drag reduction for larger channels, we presume that the results indicate that larger channels are needed. Thus, while the small model problem allowed us to propose a mechanics for drag reduction and to explore the effect of the various parameters on the flow evolution, the significance of the turbulence channel flow simulations is mainly that they have demonstrated the feasibility of fully resolved simulations of bubble/turbulent interactions. Simulations of bubbles in a larger channel are in progress.

ACKNOWLEDGMENTS

This work is supported by DARPA as a sub-award to contract MDA972-01-C-0024 to Brown University. Some of the computations were carried our at the WPI center for parallel computing and at the NPACI computers at the University of Michigan and SDSC.

REFERENCES

- Bunner B. and Tryggvason. G. Direct Numerical Simulations of Three-Dimensional Bubbly Flows. *Phys. Fluids*, 11 (1999), 1967-1969.
- Bunner, B. and Tryggvason, G. Dynamics of Homogeneous Bubbly Flows: Part 1. Rise Velocity and Microstructure of the Bubbles. J. Fluid Mech. 466 (2002), 17-52.
- Bunner, B. and Tryggvason, G. Dynamics of Homogeneous Bubbly Flows. Part 2, Fluctuations of the Bubbles and the Liquid. J. Fluid Mech 466 (2002), 53-84.
- Bunner, B. and Tryggvason, G. Effect of Bubble Deformation on the Stability and Properties of Bubbly Flows. J. Fluid Mech. 495 (2003), 77-118.
- Climent E. and Magnaudet J. 1999 Large-scale simulations of bubble-induced convection in a liquid layer. *Phys. Rev. Lett.* 82 (24): 4827-4830.
- Du, Y. and Karniadakis, G. Em. Suppressing Wall Turbulence by Means of a Transverse Traveling Wave. Science. 288 (2000), 1230-1234.
- Elghobashi, S. E. and Truesdell, G. C. Direct simulation of particle dispersion in decaying isotropic turbulence. *J. Fluid Mech.* 242 (1992), 655.
- Esmaeeli, A., and Tryggvason, G., An inverse energy cascade in two-dim- ensional, low Reynolds number bubbly flows. *J. Fluid Mech.* 314 (1996), 315-330.
- Esmaeeli, A. and Tryggvason, G., Direct Numerical Simulations of Bubbly Flows. Part I-Low Reynolds Number Arrays. J. Fluid Mech. 377 (1998), 313-345.
- Esmaeeli, A. and Tryggvason, G., Direct Numerical Simulations of Bubbly Flows. Part II—Moderate Reynolds Number Arrays J. Fluid Mech. 385 (1999), 325-358.
- Fernández, A., Lu, J. and G. Tryggvason, G. FEDSM2002-31217: The effect of bubbles on vortical flows. Proceedings of FEDSM'02. 2002 ASME Fluids Engineering Division Summer Meeting July 14-18, 2002, Montreal, Quebec, Canada
- Fernández, A., Lu, J. and G. Tryggvason, G. FEDSM2003-45641: Bubble Effects on wall shear in vortical flows. Proceedings of FEDSM'03. 4th ASME/JSME Joint Fluids Engineering Conference, July 6-11, 2003, Honolulu, Hawaii, USA
- Jeong, J. and Hussain, F. On the identification of a vortex, Journal of Fluid Mechanics, 285 (1995), 69-94.

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

- Kato H., Miyanaga M., Yamaguchi H, and Guin M.M. Frictional drag reduction by injecting bubbly water into turbulent boundary layer and the effect of plate orientation. In *Proceedings of the 2nd International Conference* on *Multiphase Flow* '95, Kyoto, 1995; 31–38, ICMF95.
- Kanai, A. and Miyata, H., Direct numerical simulations of wall turbulent flows with microbubbles. Int. J. Numer. Meth. Fluids 35 (2001), 593-615.
- Kawamura, T. and Kodama, Y. Numerical simulation method to resolve interactions between bubbles and turbulence. Int'l J. of Heat and Fluid Flow, 23 (2002), 627--638",
- 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/
- Lance, M. and Bataille, J. Turbulence in the liquid phase of a uniform bubbly air-water flow. *J. Fluid Mech.* 222 (1991), 95-118.
- Legner, H.H. A simple model for gas bubble drag reduction. Phys. Fluids, 27 (1984), 2788-2790.
- Madavan N.K., Deutsch S., and Merkle C.L. Measurements of local skin friction in a microbubble-modified turbulent boundary layer. J. Fluid Mech. 156 (1985), 237–256.
- Madavan N.K., Deutsch S., and Merkle C.L. Reduction of turbulent skin friction by microbubbles. *Phys. Fluids* 27 (1984), 356–363.
- McCormick, M.E. and Bhattacharyya, R., Drag Reduction of a Submersible Hull by Electrolysis, *Naval Engineers Journal*, 85 (1973), No.2, pp. 11-16.
- Merkle, C.L. and Deutch, S. Drag reduction in Liquid Boundary Layers by Gas Injection. *Progress in Astraunautics and Aeronautics* 123 (1990), 351-412.
- Min, T., Yul, J. Y., H. Choi, H., and Josephs, D.D. Drag reduction by polymer additives in a turbulent channel flow. J. Fluid Mech. 486 (2003), 213-238.
- Orlandi, P., Fluid Flow Phenomena. A Numerical Toolkit. Kluwer Academic Publishers. 2000.
- Squires, K. D. and Eaton, J. K. Particle response and turbulence modification in isotropic turbulence. *Phys. Fluids* 2 (1990), 1191-1203.
- Spelt, P. D. M. & Biesheuvel, A. On the motion of gas bubbles in homogeneous isotropic turbulence. J. *Fluid Mech.* 336 (1997), 221-244.
- Sridhar G, and Katz J. Effect of entrained bubbles on the structure of vortex rings. J. Fluid Mech. 397 (1999), 171-202
- Tryggvason, G., Bunner, B., Esmaeeli, A., Juric, D., Al-Rawahi, N., Tauber, W., Han, J., Nas, S. and Jan, Y.-J. A front tracking method for the computations of multiphase flow. *J Comput Phys.* 169 (2001), 708-759.
- Unverdi, S.O., and Tryggvason, G., A front-tracking method for viscous, incompressible, multi-fluid flows, J. *Comput Phys.* 100 (1992), 25-37
- Wang, L.-P. and Maxey, M. R. Settling velocity and concentration distribution of heavy particles in homogeneous isotropic turbulence. J. Fluid Mech. 256 (1993), 27-68.
- Xu, J., Maxey, M. R., and Karniadakis, G. Em., Numerical simulation of turbulent drag reduction using microbubbles. J. Fluid Mech. 468 (2000), 271-281.