Evaluation of Bubble-Induced Turbulence Using Direct Numerical Simulation

Jinyong Feng, Igor A. Bolotnov*

Department of Nuclear Engineering, North Carolina State University,

Raleigh, NC 27606, USA

jfeng3@ncsu.edu, igor_bolotnov@ncsu.edu

Abstract

The presented research evaluates the interaction between a single bubble and homogeneous turbulent flow using direct numerical simulation (DNS) approach. The homogeneous single-phase turbulence is numerically generated by passing a uniform flow through grid planes. The turbulence decay rate is compared with experiment-based correlation. The single phase turbulence is then used as an inflow boundary condition for a set of single bubble studies. By estimating the turbulent field around the fully resolved bubble, the effects of bubble deformability, turbulent intensity and relative velocity on the bubble-induced turbulence are investigated. The existence of bubble creates new vortices in the wake region and the enhancement of turbulence is observed in the region behind the bubble. The results show that the magnitude of the turbulence enhancement would increase as the bubble encounters larger liquid turbulent intensity or higher relative velocity. Set of

* Corresponding author. Tel: +1 518 542 8939
E-mail address: igor_bolotnov@ncsu.edu

bubble Weber numbers from 0.34 to 3.39 are used to investigate the effect of bubble deformability. The more deformable bubble is the higher the increase in the magnitude of the turbulence enhancement behind the bubble. This research provides systematic insight on the bubble-induced turbulence (BIT) mechanism and is important for multiphase computational fluid dynamics (M-CFD) closure model development.

Keywords

DNS, deformable bubble, turbulent intensity, homogeneous turbulent flow, bubble-induced turbulence

1. Introduction

Computational fluid dynamics (CFD) approach has already reached a high level of maturity for the single-phase flows, however closure models development for bubbly two-phase turbulent flows require additional input (Rzehak and Krepper, 2013; Lucas et al., 2007; Pang and Wei, 2011). One of the critical issues in the development of two-phase turbulent model is the understanding of mechanisms in which the existence of bubbles alters the turbulence generation in the liquid phase. The effect of bubbles on the liquid is generally called bubble-induced turbulence (BIT). Based on the well-established single-phase models, two-phase turbulence models are obtained using empirically established closure terms that account for bubble-induced turbulence (Rzehak and Krepper, 2013). The bubble distribution in the multiphase computational fluid dynamics (M-CFD) approach is governed by the interfacial forces. The analytical form (Lahey and Drew, 2001) for bubbles' contribution to the turbulence was formulated by the derivation of basic balance equations for turbulent kinetic energy and Reynolds stresses in the gas/liquid flow (Hill et al., 1995; Kataoka and Serizawa, 1989; Pfleger and Becker, 2001). The available experimental capabilities (Hibiki et al., 1998; Lance and Bataille, 1991; Wang K. et al., 1987) only allowed estimating the net change in turbulence level for multiple bubble flows. However, detailed studies on how individual bubbles contribute to the turbulence are limited and difficult to conduct experimentally. Validated numerical approach with flow control allows the systematic studies presented in this research.

Direct numerical simulation (DNS), where all flow scales, from largest turbulence eddies to Kolmogorov scale, are fully resolved, provides a complete picture of the 3D time-

dependent turbulent flow field. Numerical simulation also allows performing parametric studies more easily than experiment since one can control a single parameter (e.g., surface tension or turbulent intensity) and analyze the impact of this parameter on the two-phase flow turbulence. The previous research on the behavior of a deformable bubble mainly focuses on the transverse migration of bubbles (Tomiya et al., 2002; Dijkhuizen et al., 2010) to estimate lift force and influence of void fraction on the velocity fluctuations in bubbly flow (Brunner and Tryggvason, 2002). Note that most of these papers deal with laminar flows, which are rare in practical engineering applications.

As the liquid turbulence interacts with a bubble, the behavior of spherical and deformable bubbles is quite different. Since the vorticity generated at a free surface is proportional to the local curvature, the deformable bubbles generate turbulent vorticity at a higher rate compared to a spherical bubble (Batchelor, 2000). Therefore, the bubble shape influences the distribution of energy exchange between the liquid and gas phases. Stewart (1995) showed that the turbulent kinetic energy enhancement is observed and caused by the wake capture and collision process of deformable bubble. A more detailed examination of the wake structure by Brucker (1999) suggested that this amplification is due to the enlargement of the wake during the collision. Brunner and Tryggvason (2003) confirmed that the turbulent kinetic energy induced by the bubbles in the liquid, with void fraction of 6%, is larger for deformable bubbles than spherical bubbles. Bolotnov and Podowski (2012) showed that DNS approach is a convenient tool in the development of two-phase turbulence model by evaluating and analyzing the mean phase velocities and various properties of turbulence, e.g. turbulent kinetic energy, in both multiple spherical bubbles and single deformable bubble scenarios. These previous studies provide conceptual ideas

about the general trend of the bubble deformability effect on the bubble-induced turbulence. In the presented research, we analyze the magnitude of the bubble-induced turbulence for a single bubble with controlled conditions in a homogeneous turbulence field.

In this paper, we have separately studied the influence of three major parameters on the bubble-induced turbulence: (i) surface tension, (ii) turbulent intensity and (iii) relative velocity. The bubble deformation study would determine the trend when the bubble would augment or suppress liquid turbulent kinetic energy and investigate the relationship between bubble deformability and BIT. The turbulent intensity study would provide insight on the amount of energy transferred between the bubble and the liquid turbulence as a function of turbulence level already present in the flow. The relative velocity study would help us better understand the impact of relative motion between phases on the energy transfer. Previous numerical studies of bubbly flows typically dealt with nearly spherical bubbles (Ilic et al., 2004), often in laminar flows. Realistic flows encountered in industrial applications or in nature often contain various bubble sizes, including deformed bubbles. The main motivation for the presented research is to review the closure parameters and formulations of major bubble-induced turbulent kinetic energy models (Kataoka and Serizawa, 1989; Lahey Jr and Drew, 2000; Pfleger and Becker, 2001), and propose new formulations for the energy transfer between bubble and liquid based on the obtained results.

2. Numerical methods

2.1. PHASTA development and history

The direct numerical simulations are performed with research CFD-solver, PHASTA (Jansen, 1999). PHASTA is a Parallel, Hierarchic, higher-order, Adaptive, Stabilized (finite element method-based (FEM)) Transient Analysis flow solver for both incompressible and compressible flows. The flow solver has been proven to be an effective tool for a multitude of different types of simulations including, Reynolds-Averaged Navier-Stokes (RANS), Large-Eddy Simulation (LES), Detached-Eddy Simulation (DES), and DNS.

Various benchmarks have verified and validated the PHASTA code for the simulation of turbulent flows. Jansen (1999) described the development and validation of PHASTA for LES to compute the turbulence around airfoil. Then in 2001 (Whiting and Jansen, 2001), Jansen validated the stabilized FEM with higher-order hierarchical basis functions by performing a series of classic single-phase CFD problems, including Kovasznay flow, flow over a backward-facing step and lid-driven cavity flow. Trofimova et al. (2009) performed the DNS for the turbulent single-phase channel flows at $Re_\tau = 180$ and 395. By analyzing and validating various turbulent statistics, they demonstrated the accuracy of the stabilized FEM for turbulence simulation. Araya et al. (2011) introduced a dynamic rescaling-recycling method to the code. This capability enables simulating spatially evolving boundary layers under more general pressure gradient situations. The validation test are performed in zero, favorable and adverse pressure gradient flows and the results agree with the expected values.

Nagrath et al. (2006) implemented the level-set interface tracking method (Sethian, 1999; Sussman et al., 1999) to the PHASTA code to significantly extend the simulation

capability from single-phase to two-phase. They validated the approach by simulating the implosion and rebound of an air bubble and the simulation results were qualitatively similar to those observed/predicted in experimental/numerical studies. The two-phase simulations using PHASTA have been reported in a wide range of applications. Galimov conducted the study about interfacial waves (Galimov, 2007) and plunging liquid jet (Galimov et al., 2010) using PHASTA. Rodriguez et al. (2013) described a parallel mesh adaptation method to refine and coarsen regions of the solution domain on the application of annular two-phase flows. They validated their approach by performing a set of simulations ranging from simple canonical test problems, i.e, two dimensional dam break and solitary wave, up to the experimental annular steam-water flow condition with Re_τ of 792 using PHASTA. Bolotnov et al. (2011) performed the two-phase turbulent bubbly channel flow at $Re_\tau = 180$. They determined the bubbles' influence on the turbulence field by analyzing the mean velocity distributions, local void fraction as well the local turbulent kinetic energy and dissipation rate of the liquid phase. Later an additional study to analyze the influence of bubbles on the turbulence anisotropy at $Re_\tau = 180$ and 400 was performed (Bolotnov, 2013). Behafarid et al. (2015) analyzed the dynamics of large deformable bubbles in a vertical circular pipe and a narrow rectangular channel of different inclination angles from 0^0 to 45^0 . The simulation results of the Taylor bubble flow in a vertical pipe are validated against the theoretical model and the results for bubbles flowing along inclined rectangular channels were validated against experimental data (Maneri and Zuber, 1974) as well. Thomas et al. (2015) implemented a Proportional-Integral-Derivative bubble controller to the PHASTA code and evaluated the interfacial force closure under both laminar and

turbulent flow. Good agreement were found between the obtained drag coefficient and experiment-based correlation (Tomiyama et al., 1998). Recent publication about the bubbly flows study in PWR relevant geometries has performed with PHASTA at Re of 29,079 and 80,775 based on channel hydraulic diameter. As a massively parallel turbulent flow solver, PHASTA demonstrates strong scalability at extreme scale. Sahni (2009) reported that PHASTA scales well up to 32,768 cores. Rasquin et al. (2014) extended the application of PHASTA on a realistic wing design with up to 786, 432 cores.

2.2. Governing equations

The spatial and temporal discretization for the Incompressible Navier-Stokes (INS) equations used in PHASTA have been described in (Whiting and Jansen, 2001). The strong form of the INS equations is given by:

$$\text{Continuity} \quad u_{i,i} = 0 \quad (1)$$

$$\text{Momentum} \quad \rho u_{i,t} + \rho u_j u_{i,j} = -p_{,i} + \tau_{ij,j} + f_i \quad (2)$$

where ρ is the density, u_i is the i^{th} component of velocity, p is the static pressure, τ_{ij} is the viscous stress tensor, and f_i is the body force along the i^{th} coordinate, including the surface tension force and gravitational force. The viscous stress tensor for an incompressible flow of a Newtonian fluid is related to the fluid's dynamic viscosity, μ , and the strain rate tensor, S_{ij} , as:

$$\tau_{ij} = 2\mu S_{ij} = \mu(u_{i,j} + u_{j,i}) \quad (3)$$

We utilize the Continuum Surface Force (CSF) model (Brackbill et al., 1992) which is formulated to numerically model the surface tension effects and it is included in f_i .

2.3. Level-Set interface tracking method

To distinguish between liquid phase and gas phase, the level-set method (Sussman et al., 1999; Sussman and Fatemi, 1999; Sussman et al., 1998; Sethian, 1999) is proposed for interface tracking method (ITM) in this paper and involves modeling the interface as the zero level-set of a smooth function, φ , where φ is often called the first scalar and it represents the signed distance from the interface. The interface is therefore defined by $\varphi = 0$. In case of no phase change in the flow the scalar, φ , is advected with a moving fluid according to

$$\frac{D\varphi}{Dt} = \frac{\partial\varphi}{\partial t} + \mathbf{u} \cdot \nabla\varphi = 0 \quad (4)$$

where $\mathbf{u}$ is the flow velocity vector. The liquid phase is indicated by a positive level set.

As a promising alternative to the Volume of Fluid (VOF) and Front-Tracking (FT) method, the level-set interface tracking method is known for mass conservation issues, especially for under-resolved flows. We introduce an approach to ensure the constant bubble volume for adiabatic two-phase cases. The level set advection equation is given by Eq. (4), in which $\mathbf{u}$ is the advection velocity. As long as the expected bubble volume is known, the PID bubble controller, which will be discussed in section 2.4, can be used to compare the current bubble volume with the prescribed value, and then manipulate the advection velocity accordingly to preserve the bubble volume (Fang, 2016). Figure 1 shows the bubble volume evolution with the prescribed bubble volume of $6.545 \times 10^{-8} \text{ m}^3$. The time range shown in Figure 1 is 0.1 s which corresponds to one quarter flow-through, the number of time one Lagrangian particle passing through the domain. The selected case is

the turbulent intensity study case with bubble local turbulent intensity of 2.86%. With the mass control capability, the case maintains good mass conservation and the bubble volume fluctuation magnitude is around 2.0% throughout the entire simulation. Good mass conservations are observed for all the cases mentioned in this paper while the fluctuation magnitude will vary from 0.4% to 2.3% depending on the conditions.

Figure 1. Comparison of bubble volume evolution of a selected case for turbulent intensity study. Solid and dash lines represent the instantaneous void fraction and the expected void fraction, respectively.

Since evaluating the jump in physical properties using a step change across the interface is challenging numerically using a fixed computational mesh, the properties near the interface are defined using a smoothed Heaviside kernel function, H_ε

$$H_\varepsilon(\varphi) = \begin{cases} 0, & \varphi < -\varepsilon \\ \frac{1}{2} \left[1 + \frac{\varphi}{\varepsilon} + \frac{1}{\pi} \sin\left(\frac{\pi\varphi}{\varepsilon}\right) \right], & |\varphi| < \varepsilon \\ 1, & \varphi > \varepsilon \end{cases} \quad (5)$$

where ε is the interface half-thickness. The fluid properties are then defined as

$$\rho(\varphi) = \rho_1 H_\varepsilon(\varphi) + \rho_2 (1 - H_\varepsilon(\varphi)) \quad (6)$$

$$\mu(\varphi) = \mu_1 H_\varepsilon(\varphi) + \mu_2 (1 - H_\varepsilon(\varphi)) \quad (7)$$

Although the solution may be reasonably good in the immediate vicinity of the interface, the distance field may not be correct throughout the domain since the turbulent flow field distorts the level set contours. Thus, the level set has to be corrected with a re-distancing operation (Sussman et al., 1998; Sussman and Fatemi, 1999).

2.4. Proportional-Integral-Derivative bubble controller

The proportional-integral-derivative (PID) bubble controller is implemented to the PHASTA code to balance the interfacial forces, like drag and lift force (Thomas et al., 2015). The control forces can be extracted from the simulation and used to compute lift and drag coefficients as has been previously demonstrated (Thomas et al., 2015).

The PID controller consists of three error feedback mechanisms: the proportional part which represents the current error; the integral part which represents the accumulation of past errors; the derivative part which represents the prediction of future error. The error output for the bubble control algorithm is treated as the distance between the bubble's current position and its initial position. For each simulated time step, the bubble controller assesses the bubble location and velocity and uses those variables as inputs for control loop feedback. The mathematical form of the PID-based bubble controller is given below (Thomas et al., 2014)

$$CF_i^{(n+1)} = \overline{CF_i^{(n)}} + c_2 [CF_i^{(n)} + c_3 dx_i^{(n)} + c_4 dx_i^{2(n)} + c_5 v_i^{(n)}] \quad (8)$$

where $CF_i^{(n)}$ is the i^{th} component of the control force at time n , $\overline{CF_i^{(n)}}$ is historical average of the i^{th} component of control force at time n , $dx_i^{(n)}$ is the i^{th} component of the bubble position difference between time n and time zero, $v_i^{(n)}$ is the i^{th} component of the bubble's average velocity at time n .

The application and implementation of PID bubble controller is an extensive learning process. Through iterative test-trials, we are able to find the appropriate control coefficients to keep the bubble fixed. Note that as long as the bubble well-controlled, specific coefficients used are not expected to influence the behavior of the overall simulation. Table I provides the value of the control coefficients. This single set of control coefficients are valid for all the single bubble homogeneous turbulent flow simulations we discuss in this paper.

Table I. Summary of control coefficients used throughout this paper.

Control coefficients	c_1	c_2	c_3	c_4	c_5
Value	0.2	0.8	5.0×10^6	5.0×10^4	5.0×10^6

3. Numerical simulation setup

In this section, we will describe the numerical simulation setup for both single-phase and two-phase simulations. Single-phase cases are built to obtain the instantaneous velocity history of homogeneous turbulent field while the two-phase simulation cases utilize the recorded single-phase transient velocity information to perform the parametric studies on the bubble-induced turbulence.

3.1. Single-phase simulation setup

In order to separately discuss the effect of each parameter on the bubble-induced turbulence, a stand-alone simulation of single-phase turbulent flow is designed with periodic boundary conditions to obtain desired turbulence level and flow rate. PHASTA code enables us to record the instantaneous three-dimensional velocity components as a function of time at the periodic outflow location (Zhou et al., 2010). After that, we can utilize the recorded instantaneous field as the inflow boundary conditions for the two-phase simulations. This procedure has been successfully developed and tested before (Feng and Bolotnov, 2015).

In order to obtain homogeneous turbulent flow we apply the uniform pressure gradient force to drive the flow through two grid planes, similar to typical experimental setups (Lance and Bataille, 1991). The existence of small spheres would serve as obstacles to create turbulence. The computational domain for the generation of the single-phase homogeneous turbulent flow in our DNS is prescribed by a periodic box with dimensions of $60 \times 25 \times 15 \text{ mm}^3$ shown in Figure 2(a). Figure 2(b) shows the cross sectional view of the domain in the streamwise direction (x direction). Two grid planes are placed at $x = 2 \text{ mm}$ and $x = 3 \text{ mm}$ streamwise direction in order to leave enough distance to develop the homogeneous turbulence. In each grid we have 60 (in 10×6 arrangement) cells. Each cell has a 1 mm diameter sphere as the obstacle. The distance between the two nearest spheres on the same probe plane is 1 mm.

Figure 2. Geometrical Configuration of the homogeneous turbulence generation domain. (a) shows the 3D geometrical configuration of domain, (b) shows the cross sectional view of the domain in x normal direction

Next we describe the meshing approach for the single-phase simulation domain. We utilize several boundary layers in the mesh design on each obstacle sphere surface to capture the turbulent boundary development shown in Figure 3(b). The thickness of the first boundary layer is estimated based on the smallest turbulence length scale, i.e., Kolmogorov scale. The Kolmogorov scale is estimated using the following equation (Pope, 2000):

$$\eta = \left(\frac{\nu^3}{\varepsilon} \right)^{\frac{1}{4}} \quad (9)$$

where ν is the kinetic viscosity and ε is the dissipation rate of turbulent kinetic energy. The dissipation rate is determined based on the energy balance. We apply the pressure gradient on the whole domain to offset the drag force due to the existence of the spherical obstacles. The drag force on the sphere is given in Eq. (10). Since the sphere Reynolds number is 350, the drag coefficient is found to be 0.617 (Clift et al., 2005).

$$F_{drag} = \frac{1}{2} C_D \rho v_r^2 A \quad (10)$$

where C_D is the drag coefficient, v_r is the relative velocity between solid sphere and liquid and A is the sphere projected area on the lateral plane.

Then the pressure force is determined based on the drag force.

$$F_{pressure} = F_{drag} \quad (11)$$

The power applied on the whole domain is calculated as

$$P = F_{pressure} v_r \quad (12)$$

The dissipation rate is finally determined as

$$\varepsilon = \frac{P}{m} \quad (13)$$

where m is the liquid mass in the whole domain. The dissipation rate is calculated to be $3.49 \times 10^{-2} \frac{m^2}{s^3}$.

Based on the calculation from Eq. (9) to (13), the Kolmogorov scale is calculated to be 6.5×10^{-2} mm. The dimensionless wall distance, y^+ , based on the first layer thickness of the obstacle sphere is 0.33. The parameters of the geometrical and mesh configuration of the homogeneous turbulence model are summarized in Table II.

Table II. Geometrical and mesh configuration of the homogeneous turbulence model.

Setup		28 million elements case	129 million elements case
Domain size (mm ³)		60×25×15	
Bulk mesh size (mm)		0.2	0.1
Mesh setup of sphere boundary layer	Mesh resolution on the surface (mm)	5×10^{-2}	
	First layer thickness (mm)	2×10^{-2}	
	Boundary layer total thickness (mm)	5.19×10^{-1}	

Grid refinement study is performed with two mesh setups, 28 and 129 million elements. The statistics of the mesh configuration is given in Table II. Grid resolution studies are not typical for DNS since the individual runs are quite expensive. Since we are considering the

decay of homogeneous turbulence in the wake of obstacles, the mesh resolution requirements downstream the obstacles vary as the turbulence intensity changes. We have performed the mesh study to demonstrate that the choice of mesh resolution is appropriate to resolve all the turbulent scales in the region of interest. The finer mesh case, 129 million elements, has been created using the same domain as the coarser mesh case (with 28 million elements) while the bulk mesh resolution is refined from 0.2 mm to 0.1 mm in all 3 directions. Identical mesh resolution on the sphere surface are chosen for both cases and it's finer than the Kolmogorov length scale. The mesh configuration of the 28 million elements case is shown in Figure 3. Since the second grid plane is shifted by 1.25 mm in the direction normal to the view, we can only observe the first grid plane in the Figure 3(a).

Figure 3. Mesh configuration of the homogeneous turbulence generation domain. (a) shows the mesh configuration of the whole domain, (b) shows the mesh configuration and boundary layer on the obstacle sphere surface.

Based on the force balance between pressure gradient and the drag force exhibited by the obstacles on the flow shown in Eqs. (10) and (11), a constant pressure gradient of $400 \frac{N}{m^3}$ is applied in the domain to drive the flow. Periodic boundary conditions are used in all three directions in order not to restrain the liquid turbulence structures. During the simulation, we put virtual probe plane to measure the instantaneous velocity information at the downstream (Bolotnov, 2013). This measurement technique is similar to the typical

experimental measurement while the existence of virtual probe plane will not influence the development of flow field. The position of the probe plane, $x = 55$ mm, is chosen to make sure that both the velocity profile and the turbulent kinetic energy are homogeneous and are not directly affected by the shape and spanwise location of the turbulence generating grid. The probe plane which is normal to the stream-wise direction, i.e., x direction, contains 125 (nodes in y direction) $\times 75$ (nodes in z direction) probes which are used to extract the instantaneous velocity information at those points over time and allows us to determine the level of turbulent kinetic energy. By measuring the liquid turbulence consecutively using probe plane, the liquid turbulence are averaged over time window of 1.86 s. When the relative error of liquid turbulence for three consecutive time windows are within 2% , we claim that the liquid turbulence reaches a steady state. Figure 4 shows the statistics of single phase turbulent flow for three consecutive time windows and the results show consistency.

Figure 4. Measurement of turbulence statistics for three consecutive windows and each has a time window of 1.86 s. Dot, dash, dash-dot and solid lines represent the time window 1, the time window 2, the time window 3 and the entire time window containing all three windows, respectively.

After the turbulent flow reaches a statistically steady-state conditions, the transient velocity profile of a plane perpendicular to the flow direction is recorded over the time period corresponding to the 28 domain flow-throughs (e.g. the number of time a Lagrangian particle would cross the domain). For the two-phase simulation mentioned later, the total time window is 1.21 s which are within the statistically steady-state condition shown in Figure 4. The diagram is shown in Figure 5.

Figure 5. Left image shows the generation of homogeneous turbulent flow. The dashed line indicates probe plane location. The simulation on the right utilizes the inflow recorded on the left with several probe planes to verify the turbulence decay rate.

3.2. Two-phase simulation setup

The two-phase flow simulations utilize the recorded instantaneous field of single-phase turbulent flow as the inflow boundary conditions. Then we apply natural pressure condition on the outflow surface in streamwise direction (x direction) and periodic boundary condition in lateral direction, i.e., both y and z directions. The computational domain for each two-phase flow simulation is $37.5 \times 25 \times 15 \text{ mm}^3$. Initially a spherical bubble with diameter $D = 5 \text{ mm}$ is positioned in the channel. It's located in the center of the lateral plane and the streamwise position is case dependent.

The PID bubble control algorithm (Thomas et al., 2015) used in our simulation effectively varies the gravity in the domain to maintain the bubble position via buoyancy force. While simulated bubble size is 5 mm, this does not directly correspond to the realistic bubble size in Earth gravity. To obtain the corresponding bubble size in real physical experiment, we compare the values of the dimensionless number, Eu , in both simulated and Earth gravity conditions as shown in the following equations:

$$Eu_e = Eu_s \quad (14)$$

$$\frac{\Delta\rho_e g_e D_e^2}{\sigma_e} = \frac{\Delta\rho_s g_s D_s^2}{\sigma_s} \quad (15)$$

$$D_e = D_s \sqrt{\frac{g_s \sigma_e}{g_e \sigma_s}} \quad (16)$$

where subscript e and s represent the Earth-gravity based experiment and scaled simulation, respectively. Based on the given fluid properties, bubble size and relative velocity, the corresponding bubble diameter in the physical experiment has a range of 0.45~ 2.32 mm as shown in Table V.

The two-phase flow computational domain has 188, 125, and 75 nodes in x, y and z directions respectively, which results in 25 nodes across the bubble diameter. The total mesh size is 1.73 million elements. Previous research conducted in our group (Thomas et al., 2015) concluded that the interfacial forces between bubble and liquid provide appropriate resolution when the bubble is resolved with at least 20 nodes across spherical bubble diameter. With highly deformed bubbles one may require even more points to properly resolve the curvature of the interface. The liquid properties are identical to the single-phase case and are summarized in Table III.

Table III. Fluid properties in the two-phase turbulence case.

	Liquid	Gas
Density ($\frac{kg}{m^3}$)	996.5	1.161
Viscosity ($\frac{kg}{m.s}$)	8.5439×10^{-4}	1.845×10^{-5}

Figure 6. Domain of the two-phase flow simulation. (a) shows the 3D view of the simulation domain and (b) shows the cross sectional view of the mesh configuration.

4. Results and discussion

In the following sections, we will describe the homogeneous turbulent flow generation approach and the decay of turbulence validation based on the experiment-based correlation. The verification and validation of the level-set interface tracking approach effect on the bubble-induced turbulence is presented. Set of parametric studies on the effect of individual parameters on the bubble-induced turbulence are performed and analyzed.

4.1. Computational mesh study

Block-induced turbulence generation algorithm is utilized to generate the homogeneous turbulent flow. Grid resolution study is performed with 28 and 129 million elements setup described in Table II. As shown in the Figure 7, after averaging the numerical data in z direction, the velocity and turbulent kinetic energy (TKE) are plotted over different mesh configurations. Uniform distribution of velocity and TKE are observed as expected for the homogeneous turbulence. The results from the two meshes are consistent (within 0.6 % for mean velocity and within 2.2 % for the TKE in the center region), which indicates the appropriateness of the coarser mesh for further simulations. For the case with 28 million elements setup, the averaged velocity is $0.297 \frac{m}{s}$ and the averaged turbulent kinetic energy is $6.74 \times 10^{-5} \frac{m^2}{s^2}$. These simulations were carried out on Mira supercomputer at Argonne Leadership Computing Facilities (ALCF) using 8,192 processing cores over about 100 wall-clock hours.

Figure 7. Velocity and turbulent kinetic energy profiles for different mesh refinement. Solid and dotted lines represent 28 million and 129 million mesh cases, respectively.

4.2. Decay of Turbulence

Figure 8 shows the 3D view of the turbulence generation in the domain, including the velocity field and Q-criterion contours (Jeong and Hussain, 1995) around the grid planes. As the inflow passes by the grid sphere, vortices are created around the sphere which is shown in the right of Figure 8.

After the turbulence is generated from the two grid planes, the turbulence is homogeneous and it decays with time. The classic paper by Comte-Bellot and Corrsin (1966) provided the measured $\langle u^2 \rangle$ and $\langle v^2 \rangle$ values from the decay of grid turbulence experiment. They suggested that the normal stresses and turbulent kinetic energy k decay as power laws, which, can be written as

$$\frac{k}{U_0^2} = A \left(\frac{x - x_0}{M} \right)^{-n} \quad (17)$$

where x_0 is the virtual origin, M is the spacing between the two grids. Values of the decay exponent n between 1.15 and 1.45 are reported in the literature (Mohamed and LaRue, 1990; Comte-Bellot and Corrsin, 1966; Kolmogorov, 1941; Kang et al., 2003). The value of A varies widely depending on the geometry of the grid and the Reynolds number.

Figure 8. 3D view of the turbulence generation.

The decay of homogeneous turbulent flow in our case is consistent with previous observations which demonstrates the validity of our approach. Figure 9 shows that the decay of turbulence agrees well with the power law. The decay exponent is found to be 1.5, which is close to the experiment-based range, and coefficient A is found to be 0.0455 in our case.

Figure 9. Decay of turbulent kinetic energy as a function of x/M . Solid and dash-dot lines represent raw data and power approximation line, respectively.

4.3. Verification of level-set approach for bubble-induced turbulence problem

The level-set interface tracking method (Sethian, 1999; Sussman et al., 1994) can numerically distinguish the interface between the gas and liquid phases and allows us to capture the bubble motion and shape deformation. The deformable bubble in the two-phase flow simulation can be fully resolved with level-set ITM by putting enough computational points across the bubble diameter to represent the interface curvature. As with any ITM which uses single-fluid approach, the Navier-Stokes solver deals with one fluid which has its properties change across the interface. The level-set method utilizes a smooth transition (Eqs. (5), (6), and (7)) of the fluid properties from liquid to gas across the interface with typical thickness of 2.5 computational cells. However, the existence of this transitional region may influence the development of turbulent boundary layer and affect the bubble-induced turbulence. We have previously verified that the drag force estimates provided by

the level-set interface tracking agree very well with the experimental-based correlation (Thomas et al., 2015) and thus the property transition region does not have significant influence on the drag force. However, it is important to evaluate the effect of this region on the bubble-induced turbulence generation. In this paper we perform an additional study to quantify the effect of the transition on the turbulence generation by a single bubble.

A separate study is performed using single-phase simulations with a solid sphere (representing a typical bubble) in the domain and introducing various artificial interface thicknesses on the sphere surface. While the level set method is not utilized directly in those simulations, we mimic the property smoothing region by varying the liquid properties near the sphere surface as shown in Figure 11. As the liquid passes around the sphere, the existence of the artificial interface on the development of turbulent boundary layer is investigated. The sphere diameter is 5 mm and the simulation domain size is $25 \times 25 \times 15$ mm³. We place 20 elements across sphere diameter and boundary layers on the sphere surface to help capture the development of turbulent boundary layer. Additional refinement region is placed in the wake of the sphere to capture the wake induced turbulence. The case setup parameters are given in Table IV and the simulation domain is shown in Figure 10. The distance between the probe plane and sphere center is measured in the sphere diameters (D).

Table IV. Geometrical and mesh configuration of the level-set method validation cases.

Domain size (mm ³)	25×25×15	
Bulk mesh size (mm)	0.5	
Mesh setup of sphere boundary layer	Mesh resolution on the surface (mm)	0.2
	First layer thickness (mm)	1×10^{-2}
	Boundary layer total thickness (mm)	1.55
Mesh size in the wake region (mm)	0.2	
Distance from the probe plane to the sphere center	Probe plane 1	1 <i>D</i>
	Probe plane 2	1.5 <i>D</i>

Figure 10. Simulation domain for level-set verification cases.

Figure 11. Density evolution across the interface. The width of the transition region are 1 (hollow sphere), 2 (hollow square), 3 (triangle), 4 (square) and 5 (sphere) elements.

Homogeneous turbulent flow with mean velocity of $0.05 \frac{m}{s}$ is utilized as the inflow boundary condition to perform the study. The corresponding Reynolds number based on sphere diameter is 292 which is typical for a bubble Reynolds number in bubble-induced turbulence studies (Ilic et al., 2004; Fang, 2016; Rensen et al., 2005). Two probe planes are placed in the wake region to record the instantaneous 3D velocity information. The distance between the probe planes and the sphere center are 1 and 1.5 sphere diameter, D , which are described in Table IV. Figure 12 compares the averaged TKE profile versus several artificial interface thicknesses. For the wake region which contains the sphere projected area on the probe plane, except the thickest interface thickness which is the half sphere diameter, the TKE results agree well with the case having no properties transition region. For the region away from the sphere, the TKE profile agrees perfectly. Figure 13 compares the velocity profile versus various interface thicknesses. As the interface thickness

increases, the depth of the velocity valley behind the sphere decreases which is caused by the existence of transition region.

Figure 12. TKE profile from probe plane 1 versus different artificial interface thicknesses, 0 (zero interface thickness), $0.025 D$, $0.05 D$, $0.25 D$ and $0.5 D$ where D is the sphere diameter. Left (a) and right (b) represent the plot at probe plane 1 and 2 (described in Table IV), respectively.

Figure 13. Velocity profile versus different artificial interface thicknesses for probe plane 1.

The relative error for the TKE versus interface thickness is calculated as follows

$$Error = \frac{\sum_{i=1}^N (TKE_i^{(0)} - TKE_i^{(n)})}{\sum_{i=1}^N TKE_i^{(0)}} \quad (18)$$

where $TKE_i^{(n)}$ is the averaged TKE at spanwise position i and the interface thickness is n . $TKE_i^{(0)}$ is the TKE data point for the case without interface thickness. Using the Eq.(18), we plot the relative error versus interface thickness as shown in Figure 14. The relative error for the first probe plane is within 3% and the second probe plane is within 5%. One drawback of the level-set method is the introduction of the transition region. This results in the smearing of the flow properties and variables, forcing them to be continuous at the interface regardless of the appropriate jump condition. Our studies reveal that the influence of level-set method on the turbulent boundary layer development is minimal. As the width of the transition region decreases, the results are expected to approach the ideal situation.

Figure 14. Relative error versus artificial interface thickness for two separate locations in the wake of the sphere. The distance between the probe plane and sphere center are 1D (square) and 1.5D (circle). “D” represents the bubble diameter.

4.4. Effect of transverse bubble motion on the drag force

The PID bubble controller was previously implemented (Thomas et al., 2015) into the PHASTA code to control the bubble at fixed position (with respect to all three axes) and extract the interfacial forces in the three directions. Typically when the bubbles are experimentally observed rising in a standing fluid, they develop wake instabilities and the trajectories of those bubbles are not vertical, but include a transverse motion as well (Hibiki and Ishii, 1999). In order to quantify the effect of the transverse motion of the bubble on the drag force, a separate simulation with $v_r = 0.1 \frac{m}{s}$ has been performed in the presented research by removing the lateral PID control of the bubble. This allows the bubble to freely move in the lateral directions and compare the resultant drag force with the bubble controlled in all three directions. Figure 15 provides the averaged streamwise control force with lateral control force turning on and off over three separate averaged windows of 0.217 s. As shown in the Figure 15, the resultant drag forces are very close in both cases. Based on the mean control force over the total time range, i.e. 0.651 s, the drag coefficients are consistent and the relative error is within 2% (Feng and Bolotnov, 2016). Kataoka and Serizawa (1989) claimed that in the two-phase turbulent kinetic energy equation, the bubble-induced turbulence generation term can be expressed as $F_{drag} v_r$ which has a direct relationship to the drag force experienced by the bubble. We can now conclude that fully controlled bubble results presented in our research are representative of the bubble behavior in terms of drag force interaction and bubble-induced turbulence in multiple-bubbly flows at low void fractions (when there is no bubble/bubble interaction).

Figure 15. Comparison of the streamwise (x) direction control force with lateral control force (solid line) and without lateral control force (dash line) for the verification of PID bubble controller. The control forces are averaged over separate windows (square symbols) and over the whole time range (no symbols), respectively.

4.5. Effect of bubble deformation on the BIT

As shown in Figure 5, we place virtual probe plane to record the instantaneous velocity history from the single phase homogeneous turbulent flow and implement the velocity field as an inflow conditions for the two phase flow simulation. Compared to a physical experiment, numerical simulation enables us to investigate the effect of a single parameter on the two-phase flow without affecting other parameters. We conduct the bubble deformation study by only changing the surface tension. Thus, we have six different Weber numbers (0.34, 0.68, 1.35, 2.03, 2.71 and 3.39) to represent different bubble deformation levels. The range of Weber number is carefully chosen to observe the bubble shape

transition from spherical to ellipsoidal. Based on the selected Weber number range, the corresponding bubble size in real physical experiment (at Earth gravity conditions) is given in the Table V. The Weber number is defined as follows

$$We = \frac{\rho v^2 D}{\sigma} \quad (19)$$

where ρ is the liquid density, v is the relative velocity, D is the bubble diameter and σ is the surface tension.

Table V. Summary of the bubble deformability study

Case #	1	2	3	4	5	6
We	0.34	0.68	1.35	2.03	2.71	3.39
Spherical bubble diameter (mm)	0.45	0.64	0.91	1.16	1.66	2.32
Extended bubble diameter (mm)	0.45	0.67	0.95	1.30	2.00	3.04

We have reduced the mean inflow velocity from $0.3 \frac{m}{s}$ to $0.1 \frac{m}{s}$ while keeping the same Reynolds stress in the two-phase simulation to achieve the desired relative velocity between the liquid and the bubble. The corresponding Reynolds bubble for two-phase flow case is 583. After the flow field around bubble reaches a quasi-steady state, i.e. the deformable bubble has a stable bubble shape, as shown in the Figure 16, we put 13 virtual probe planes normal to the streamwise direction with 60×36 probes on each plane to extract the instantaneous velocity information before, across and behind the bubble.

Figure 16. Screenshot of the velocity field around different deformable bubble with level-set contours

The averaging time window is 0.70 s and during this time period the flow passes through the whole domain 1.87 times. Both the liquid turbulent intensity and the turbulence intensity difference compared to single phase simulation versus dimensionless bubble position (x/D) are plotted in Figure 17. Turbulent intensity is a dimensionless number to measure the magnitude of turbulent level and is defined as

$$I = \frac{u'}{U} \quad (20)$$

where u' is the root-mean-square of the turbulent velocity fluctuations and U is the mean velocity. In this case, the mean velocity is 0.1 m/s. The u' can be computed as

$$u' = \sqrt{\frac{1}{3}(u_x'^2 + u_y'^2 + u_z'^2)} = \sqrt{\frac{3}{2}k} \quad (21)$$

Figure 17. Turbulent intensity versus position with different bubble deformation level. The center of the bubble corresponds to the $x/D=0$.

When the turbulent flow reaches the bubble, the liquid transfers some portion of the turbulent kinetic energy through the bubble surface into the gas inside the bubble. The bubble acts as an energy absorber and the energy is dissipated by the recirculation of the gas within the bubble. However, the existence of the bubble creates additional turbulent eddies behind it, and we expect to observe turbulence enhancement in the wake region. For the more deformable bubble cases, more turbulent kinetic energy is generated as the bubble generates more vorticity compared to spherical bubble and the deformable bubble has relatively larger cross-sectional area which leads to the increased drag force. The vorticity of the liquid surrounding the bubbles increases considerably.

The bubble-induced turbulence is analyzed by comparing the difference of liquid turbulent kinetic energy with and without a bubble. Figure 17(a) shows the overall TKE for different deformable bubble cases compared to the single-phase case and Figure 17(b) shows the TKE difference between the different deformable bubble two-phase cases and the single-phase case. Generally, the liquid turbulent kinetic energy experiences a drop when

the liquid turbulent eddies approach the bubble. However, for the high deformable bubble, i.e. $We = 2.71$, the TKE peak occurs at the bubble location which indicates the additional generation counteracts the dissipation of turbulence energy. And in the tail of wake region, we observe that the bubble induced turbulence increases as the bubble becomes more deformable.

To analyze the effect of highly deformable bubble on the bubble-induced turbulence, an additional test with $We = 3.39$ is performed. For the highly deformable bubble, the simulation results show that the bubble behavior under homogeneous turbulent flow has two modes, stable and vibration shape modes. Here we define the bubble diameter time as the number of times one Lagrangian particle pass through the distance equivalent to the bubble diameter. During the simulation time up to 0.75 s (this corresponds to 15 bubble diameter times), the bubble shape is observed to be steady as shown in Figure 18. After that, the bubble starts to vibrate the shape over time which produces additional turbulent eddies in the wake. Figure 19 compares the streamline pattern around the spherical bubble ($We = 0.34$) and highly deformable bubble ($We = 3.39$). The spherical bubble maintains a constant bubble shape and the streamline pattern around the bubble is stable. However, the highly deformable bubble exhibits an unsteady mode which significantly contributes to the turbulent kinetic energy addition as shown in Figure 21. Further investigation reveals that the unsteady mode for the highly deformable bubble follows a certain time period shown in Figure 20. The large turbulent eddies, plotted by Q-criterion, are periodically generated on the bubble surface and the time interval is about 0.16 s (about 3.2 bubble diameter times) and it is identical to the period of bubble vibration mode. The significant

liquid turbulence addition is certainly correlated to the time varying high surface curvature of highly deformable bubble.

Figure 18. Bubble shape evolution over time. Bubble shape remains stable until time $t = 0.82$ s and becomes unstable after that.

Figure 19. Comparison of streamline pattern around spherical and deformable bubble.

Figures on the left and right columns represent the streamline pattern for spherical bubble

($We=0.34$) and deformable bubble ($We=3.39$), respectively.

Figure 20. Turbulent eddies generation on the highly deformable bubble surface. The contour is plotted using Q-criterion with value of 100.

Figure 21. Comparison of turbulent intensity versus position with different bubble deformation level. For the highly deformable bubble, $We=3.39$, the stable period, vibration period and the entire period containing both stable and vibration periods are plotted respectively.

4.6. Effect of turbulent intensity on the BIT

As we know, the homogeneous turbulent flow follows the power decay law. We can separately put single bubble in different streamwise positions (x/D), 2.00, 3.14 and 4.93, to get different turbulence intensities at the bubble location while keeping the relative velocity fixed at 0.1 m/s and using fixed bubble Weber number value of 0.68. Figure 22 shows the turbulent intensity plot with different local bubble turbulent intensities. The bubble positions are shifted to the origin to directly reflect the influence of turbulence intensity on BIT. The liquid flow turbulence is reduced locally at the bubble location. As mentioned before, the gas-liquid interface would absorb the liquid turbulence by deforming their shape which brings about an increase of the surface energy and also an increase of energy dissipation in the vicinity of the interfaces through turbulence eddy-interfacial structure

interactions (Serizawa and Kataoka, 1990). In the meantime, the turbulence energy in the liquid phase is increased by the wake induced by the bubble. The larger bubble local turbulent intensity, like 0.0344 and 0.0286, would experience a deeper energy sink and introduce larger turbulence enhancement behind the bubble.

Figure 22. Turbulent intensity versus position with different bubble local turbulent intensities, 0.0344 (square), 0.0286 (circle) and 0.0234 (triangle). The center of the bubble corresponds to the $x/D=0$.

The turbulent isotropy level with and without bubble are investigated as well. The turbulent kinetic energy k is defined and calculated as

$$k = \frac{1}{2}(u'u' + v'v' + w'w') \quad (22)$$

where u' , v' and w' are the turbulent velocity fluctuations in x , y and z direction.

Figure 23 shows the plot of turbulent isotropy comparison. Figure 23 (a) shows the decay of $u'u'$, $v'v'$ and $w'w'$ components in the stream-wise direction. Through the block-induced turbulence generation method, we are able to generate homogeneous turbulent flow while the three Reynolds stress components are not identical. Although the turbulent flow is homogeneous, it is not isotropic. The $u'u'$ is 1.35 and 1.88 times as large as $v'v'$ and $w'w'$,

respectively, at the first data point. Comte-Bellot and Corrsin's experiment (Comte-Bellot and Corrsin, 1966) shows that the u'^2 is 10% greater than the v'^2 and w'^2 and the symmetry in the ideal experiment dictated that v'^2 and w'^2 are equal. As flow passes through the domain, the $v'v'$ and $w'w'$ components tend to be identical. So the trend of u'^2 , v'^2 and w'^2 development in our study are reasonable and the ratio of u'^2 to v'^2 or w'^2 depends on the experiment setup. The proportion of $u'u'$ to the total kinetic energy increases. Figure 23 (b), Figure 23 (c) and Figure 23 (d) show the kinetic energy difference between the two-phase case and single-phase case for different turbulent intensity scenarios. The streamwise direction Reynolds stress component $u'u'$ contributes to the energy drop when the turbulent eddies interact with the bubble and fluctuations in y and z direction are slightly influenced at the bubble position. Then, the $v'v'$ and $w'w'$ components reach a peak in the front part of the wake region which is around 1 bubble diameter away from the bubble. The peaks in the wake region may come from the redistribution of $u'u'$ other than being transferred to the bubble. After that $v'v'$ and $w'w'$ components gradually drop and the streamwise fluctuation $u'u'$ starts to play a dominant role in the bubble-induced turbulence in the tail of wake region which is more than 1.5 bubble diameters long. Clearly, the analysis of velocity fluctuation components provides insight on the mechanism of the interaction between the bubble and liquid turbulent flow, which has the potential for the application of turbulent bubbly flow.

Figure 23. Turbulent isotropy comparison for different turbulent intensity. Bubble positions are 2.00, 3.14 and 4.93 for different turbulent intensities cases.

4.7. Effect of relative velocity on the BIT

In addition to the bubble deformability and local turbulent intensity, the relative velocity probably plays the most important role on the amount of the energy transfer between gas and liquid. Three values of relative velocities, 0.08 m/s, 0.10 m/s and 0.12 m/s, are chosen for this study. We keep the same bubble deformation level ($We = 0.68, D = 0.64 \text{ mm}$), by changing the surface tension for those three cases given in Table VI. The bubbles are

separately placed at the positions where the turbulent intensities are expected to be the same.

Table VI. Summary of the bubble deformability study

Case #	1	2	3
Relative velocity (m/s)	0.08	0.10	0.12
Surface tension (N/m)	0.0471	0.0735	0.106
We	0.68	0.68	0.68

The bubble positions are shifted to the origin, $\frac{x}{D} = 0$, to directly reflect the influence of relative velocity on the liquid turbulence. Eight probe planes, located at $x/D = -1.5, -1, -0.5, 0, 0.5, 1, 2$ and 3 , are selected and placed in the domain to extract the instantaneous velocity information before, across and behind the bubble. Figure 24(a) shows the total turbulent kinetic energy in the three different relative velocities case compared between single and two-phase single bubble simulations. Figure 24(b) shows the turbulent kinetic energy difference between single and two phase flow results. As the liquid turbulent flow encounters the bubble, part of the energy from the liquid is transferred to the gas by deforming bubble's shape and the existence of stationary bubble also causes the energy dissipation in the vicinity of the interface through turbulence eddy / interface interactions. The magnitude of energy drop at the bubble location increases as the relative velocity increases. In the meantime, the turbulence energy in the liquid phase is increased by the wake induced by the bubble. It also increases with relative velocity. Since the identical turbulent eddies travel with different velocities to interact with the single bubble, higher relative velocities will lead to more turbulence eddy-interfacial structure interactions within the same given time. As shown in Figure 24(b), the ratio of the energy drop peak at the

bubble position for different relative velocities cases to the case with relative velocity of $0.10 \frac{m}{s}$ are 0.83, 1.00 and 1.29. As described in Sato's model (Sato and Sekoguchi, 1975), the bubble-induced turbulence viscosity term is proportional to the relative velocity which results in the higher dissipation rate around the bubble for higher relative velocity.

Figure 24. Turbulent intensity versus position with different relative velocity. Solid and dash lines represent the two-phase and single-phase cases respectively.

5. Conclusion

In the presented research, we evaluate the interaction between a single bubble and turbulent eddies using DNS approach. The primary goal of this research is to investigate the relationship between the selected two-phase flow parameters and bubble-induced turbulence and provide insight on the development of M-CFD closure laws. The grid induced turbulence generation approach is introduced to numerically obtain homogeneous turbulent flow using massively-parallel flow solver code — PHASTA. The decay of turbulence follows the power decay law with coefficient of 1.5 which is close to the experiment-based correlation. We utilize the instantaneous velocity history of single phase

turbulence as transient inflow boundary conditions to conduct parametric studies on the bubble-induced turbulence, focusing in particular on the bubble deformability, turbulent intensity and relative velocity. The instantaneous velocity history upstream, around and downstream the bubble is collected and analyzed to evaluate those effects on the bubble-induced turbulence. The results show that more deformable bubble introduces more turbulent eddies and increases the magnitude of BIT which is prominent with $We = 3.39$. For highly deformable bubble, $We = 3.39$, the bubble shape evolution has two modes, stable mode and vibration mode. During the vibration mode, the bubble-induced turbulence significantly increases the liquid turbulence and the bubble's vibration follows a certain time period. We also observe that both larger turbulent intensity and relative velocity values will increase the magnitude of the BIT in the wake region. The fully resolved turbulence through DNS also enables us to separately analyze the Reynolds stress components $u'u'$, $v'v'$ and $w'w'$. It provides further insight on the energy transfer mechanism. In the presented research, we are able to quantitatively conduct a set of separate effect studies and obtain a preliminary DNS-based database for the BIT model development. The future work would include the study of the wall effect on the energy transfer and the combined effect with the presence of shear turbulent flow.

6. Acknowledgements

The authors would like to acknowledge the support from National Science Foundation (award 13339933 under CBET-Fluid Dynamics Program). Computational resources were supported by U.S. Nuclear Regulatory Commission's Faculty Development Program. An

award of computer time was provided by the ASCR Leadership Computing Challenge (ALCC) program. This research used resources of the Argonne Leadership Computing Facility, which is a DOE Office of Science User Facility supported under Contract DE-AC02-06CH11357. The solution presented herein made use of the Acusim linear algebra solution library provided by Altair Engineering Inc. and meshing and geometric modeling libraries by Simmetrix Inc.

References

- Araya, G., Castillo, L., Meneveau, C., Jansen, K., 2011. A dynamic multi-scale approach for turbulent inflow boundary conditions in spatially developing flows. *J. Fluid Mech.* 670, 581-605.
- Batchelor, G.K., 2000. *An Introduction to Fluid Dynamics*. Cambridge university press.
- Behafarid, F., Jansen, K., Podowski, M., 2015. A study on large bubble motion and liquid film in vertical pipes and inclined narrow channels. *Int. J. Multiphase Flow* 75, 288-299.
- Bolotnov, I.A., 2013. Influence of Bubbles on the Turbulence Anisotropy. *ASME J. Fluids Eng.* 135, 051301.
- Bolotnov, I.A., Podowski, M.Z., 2012. Informing the development of the turbulence models for bubbly gas/liquid flows using interface tracking simulations. *Advances in Thermal Hydraulics* 2012.
- Bolotnov, I.A., Jansen, K.E., Drew, D.A., Oberai, A.A., Lahey, J.T.,R., 2011. Detached Direct Numerical Simulation of Turbulent Two-phase Bubbly Channel Flow. *Int. J. Multiphase Flow* 37, 647-659.

- Brackbill, J.U., Kothe, D.B., Zemach, C., 1992. A continuum method for modeling surface tension. *Journal of computational physics* 100, 335-354.
- Brücker, C., 1999. Structure and dynamics of the wake of bubbles and its relevance for bubble interaction. *Physics of Fluids (1994-present)* 11, 1781-1796.
- Bunner, B., Tryggvason, G., 2003. Effect of bubble deformation on the properties of bubbly flows. *J. Fluid Mech.* 495, 77-118.
- Bunner, B., Tryggvason, G., 2002. Dynamics of homogeneous bubbly flows Part 1. Rise velocity and microstructure of the bubbles. *J. Fluid Mech.* 466, 17-52.
- Clift, R., Grace, J.R., Weber, M.E., 2005. *Bubbles, Drops, and Particles*. Courier Corporation.
- Comte-Bellot, G., Corrsin, S., 1966. The use of a contraction to improve the isotropy of grid-generated turbulence. *J. Fluid Mech.* 25, 657-682.
- Dijkhuizen, W., van Sint Annaland, M., Kuipers, J., 2010. Numerical and experimental investigation of the lift force on single bubbles. *Chemical engineering science* 65, 1274-1287.
- Fang, J., 2016. Development of Advanced Analysis Toolkit for Turbulent Bubbly Flow Simulations.
- Feng, J., Bolotnov, I.A., 2016. Single bubble drag force evaluation in turbulent flow based on DNS results. *ANS Winter Meeting*.
- Feng, J., Bolotnov, I.A., 2015. Bubble-induced turbulence study in homogeneous turbulent flow using DNS approach. 6068th Annual Meeting of the APS Division of Fluid Dynamics.

Galimov Yu., A., Sahni, O., Shephard, M., Drew A, D., Jansen E., K., Lahey T., R., 2010.

Parallel Adaptive Simulation of a Plunging Liquid Jet. *Acta Mathematica Scientia* 30, 522-538.

Galimov, A., 2007. An Analysis of Interfacial Waves and Air Ingestion Mechanisms. ProQuest.

Hibiki, T., Hogsett, S., Ishii, M., 1998. Local measurement of interfacial area, interfacial velocity and liquid turbulence in two-phase flow. *Nucl. Eng. Des.* 184, 287-304.

Hibiki, T., Ishii, M., 1999. Experimental study on interfacial area transport in bubbly two-phase flows. *Int. J. Heat Mass Transfer* 42, 3019-3035.

Hill, D., Wang, D., Gosman, A., Issa, R., 1995. Numerical prediction of two-phase bubbly flow in a pipe. *Proc. Second Int. Conf. Multiph. Flow.*

Ilic, M., Worner, M., Cacuci G, D., 2004. Balance of Liquid-phase Turbulence Kinetic Energy Equation for Bubble-train Flow. *J Nucl Sci Technol* 41, 331-338.

Jansen, K.E., 1999. A stabilized finite element method for computing turbulence. *Comput. Methods Appl. Mech. Eng.* 174, 299-317.

Jeong, J., Hussain, F., 1995. On the identification of a vortex. *J. Fluid Mech.* 285, 69-94.

Kang, H.S., Chester, S., Meneveau, C., 2003. Decaying turbulence in an active-grid-generated flow and comparisons with large-eddy simulation. *J. Fluid Mech.* 480, 129-160.

Kataoka, I., Serizawa, A., 1989. Basic equations of turbulence in gas-liquid two-phase flow. *Int. J. Multiphase Flow* 15, 843-855.

Kolmogorov, A.N., 1941. Dissipation of energy in locally isotropic turbulence. 32, 16-18 *Dokl. Akad. Nauk SSSR.*

- Lahey Jr, R.T., Drew, D.A., 2000. An analysis of two-phase flow and heat transfer using a multidimensional, multi-field, two-fluid computational fluid dynamics (CFD) model. , 5-8Japan/US Seminar on Two-Phase Flow Dynamics.
- Lahey, R.T., Drew, D.A., 2001. The analysis of two-phase flow and heat transfer using a multidimensional, four field, two-fluid model. Nucl. Eng. Des. 204, 29-44.
- Lance, M., Bataille, J., 1991. Turbulence in the liquid phase of a uniform bubbly air–water flow. J. Fluid Mech. 222, 95-118.
- Lucas, D., Krepper, E., Prasser, H., 2007. Use of models for lift, wall and turbulent dispersion forces acting on bubbles for poly-disperse flows. Chemical Engineering Science 62, 4146-4157.
- Maneri, C., Zuber, N., 1974. An experimental study of plane bubbles rising at inclination. Int. J. Multiphase Flow 1, 623-645.
- Mohamed, M.S., LaRue, J.C., 1990. The decay power law in grid-generated turbulence. J. Fluid Mech. 219, 195-214.
- Nagrath, S., Jansen, K., Lahey Jr, R.T., Akhatov, I., 2006. Hydrodynamic simulation of air bubble implosion using a level set approach. Journal of Computational Physics 215, 98-132.
- Pang, M.J., Wei, J.J., 2011. Analysis of drag and lift coefficient expressions of bubbly flow system for low to medium Reynolds number. Nucl. Eng. Des. 241, 2204-2213.
- Pfleger, D., Becker, S., 2001. Modelling and simulation of the dynamic flow behaviour in a bubble column. Chemical Engineering Science 56, 1737-1747.
- Pope, S.B., 2000. Turbulent Flows. Cambridge university press.

- Rasquin, M., Smith, C., Chitale, K., Seol, E.S., Matthews, B.A., Martin, J.L., Sahni, O., Loy, R.M., Shephard, M.S., Jansen, K.E., 2014. Scalable Implicit Flow Solver for Realistic Wing Simulations with Flow Control. *Computing in Science & Engineering* 16, 13-21.
- Rensen, J., Luther, S., Lohse, D., 2005. The effect of bubbles on developed turbulence. *J. Fluid Mech.* 538, 153-187.
- Rodriguez, J., Sahni, O., Lahey, R., Jansen, K., 2013. A parallel adaptive mesh method for the numerical simulation. *Computers and Fluids* 87, 115-131.
- Rzehak, R., Krepper, E., 2013. CFD modeling of bubble-induced turbulence. *Int. J. Multiphase Flow* 55, 138-155.
- Sahni, O., Carothers D, C., Shephard S, M., Jansen E, K., 2009. Strong Scaling analysis of a parallel, unstructured, implicit solver and the influence of the operating system interference. *Scientific Programming* 17, 261-274.
- Sato, Y., Sekoguchi, K., 1975. Liquid velocity distribution in two-phase bubble flow. *Int. J. Multiphase Flow* 2, 79-95.
- Serizawa, A., Kataoka, I., 1990. Turbulence suppression in bubbly two-phase flow. *Nucl. Eng. Des.* 122, 1-16.
- Sethian, A.J., 1999. *Level Set Methods and Fast Marching Methods*. Cambridge University Press.
- Stewart, C., 1995. Bubble interaction in low-viscosity liquids. *Int. J. Multiphase Flow* 21, 1037-1046.

Sussman, M., Almgren, A.S., Bell, J.B., Colella, P., Howell, L.H., Welcome, M.L., 1999.

An adaptive level set approach for incompressible two-phase flows. *Journal of Computational Physics* 148, 81-124.

Sussman, M., Fatemi, E., Smereka, P., Osher, S., 1998. An improved level set method for incompressible two-phase flows. *Comput. Fluids* 27, 663-680.

Sussman, M., Smereka, P., Osher, S., 1994. A level set approach for computing solutions to incompressible two-phase flow. *Journal of Computational physics* 114, 146-159.

Sussman, M., Fatemi, E., 1999. An efficient, interface-preserving level set re-distancing algorithm and its application to interfacial incompressible fluid flow. *Siam J. on Scientific Computing* 20, 1165-1191.

Thomas, A.M., Fang, J., Bolotnov, I.A., 2014. Estimation of Shear-induced Lift Force in Laminar and Turbulent Flows. *International Topical Meeting on Advances in Thermal Hydraulics - 2014 (ATH '14)*.

Thomas, A.M., Fang, J., Feng, J., Bolotnov, I.A., 2015. Estimation of shear-induced lift force in laminar and turbulent flows. *Nucl Technol* 190, 274-291.

Tomiyama, A., Kataoka, I., Zun, I., Sakaguchi, T., 1998. Drag coefficients of single bubbles under normal and micro gravity conditions. *JSME international journal. Series B, fluids and thermal engineering* 41, 472-479.

Tomiyama, A., Tamai, H., Zun, I., Hosokawa, S., 2002. Transverse migration of single bubbles in simple shear flows. *Chemical Engineering Science* 57, 1849-1858.

Trofimova, A.V., Tejada-Martínez, A.E., Jansen, K.E., Lahey Jr, R.T., 2009. Direct numerical simulation of turbulent channel flows using a stabilized finite element method. *Comput. Fluids* 38, 924-938.

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

Whiting, C.H., Jansen, K.E., 2001. A stabilized finite element method for the incompressible Navier-Stokes equations using a hierarchical basis. *Int. J. Numer. Methods Fluids* 35, 93-116.

Zhou, M., Sahni, O., Kim, H.J., Figueroa, C.A., Taylor, C.A., Shephard, M.S., Jansen, K.E., 2010. Cardiovascular flow simulation at extreme scale. *Comput. Mech.* 46, 71-82.