International Journal of Multiphase Flow 93 (2017) 92–107

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

journal homepage: www.elsevier.com/locate/ijmulflow

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

ARTICLE INFO

Article history:
Received 30 August 2016
Revised 8 December 2016
Accepted 3 April 2017
Available online 4 April 2017

Keywords:
DNS
Deformable bubble
Turbulent intensity
Homogeneous turbulent flow
Bubble-induced turbulence

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 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.

© 2017 Elsevier Ltd. All rights reserved.

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 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 (Tomiyama et al., 2002; Dijkhuizen et al., 2010) to estimate lift force and influence of void fraction on the velocity fluctuations in bubbly flow (Bunner 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 vortic-

* Corresponding author.
E-mail addresses: jfeng3@ncsu.edu, fengjinyong2008@gmail.com (J. Feng), igor_bolotnov@ncsu.edu (I.A. Bolotnov).

http://dx.doi.org/10.1016/j.ijmultiphaseflow.2017.04.003
0301-9322/© 2017 Elsevier Ltd. All rights reserved.

ity 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 Brücker (1999) suggested that this amplification is due to the enlargement of the wake during the collision. Bunner 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 phasic 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 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 et al. (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 discretizations 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:

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

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

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 dy-

Download English Version:

https://daneshyari.com/en/article/4994994

Download Persian Version:

https://daneshyari.com/article/4994994

Daneshyari.com