International Journal of Multiphase Flow 81 (2016) 27–45

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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

Direct Numerical Simulations of spherical bubbles in vertical turbulent channel flow. Influence of bubble size and bidispersity

C. Santarelli, J. Fröhlich*

Institute of Fluid Mechanics, Technische Universität Dresden, Dresden 01062, Germany

A R T I C L E   I N F O

Article history:
Received 14 July 2015
Revised 8 January 2016
Accepted 11 January 2016
Available online 18 January 2016

Keywords:
Direct Numerical Simulation
Bubbly flow
Spherical bubbles
Turbulent channel flow
Bidisperse swarm
Budget of turbulent kinetic energy

A B S T R A C T

The paper presents two Direct Numerical Simulations of bubble swarms in a vertical turbulent channel flow and is intended as a complement to the companion paper, Santarelli and Fröhlich (2015). Both swarms addressed here are composed of spherical bubbles in contaminated fluid with a void fraction of 2.14%, as for the reference case in the companion paper. The first simulation reported is done for a monodisperse swarm of larger bubbles while in the second case a bidisperse swarm is considered. The influence of the bubble size and of the bidispersity is investigated by means of flow visualizations and quantitative statistical analysis. Due to the higher Reynolds number, the behavior of the larger bubbles differs from the one of the small bubbles and this impacts the turbulence of the carrier phase. The results show that for the parameter range considered both swarms induce elongated flow structures in the streamwise direction and that the liquid turbulence is enhanced by the bubbles, as quantified by the budget of the turbulence kinetic energy. In both cases the tendency of bubbles to align horizontally is confirmed while the analysis of mixed pairs in the bidisperse swarm allows elucidating the complex interaction of bubbles of different size.

Introduction

Bubbly flows are essential in many industrial applications, in the environmental engineering, in the food industry, etc. These flows can become extremely complex due to the various parameters and phenomena involved, such as the individual motion of single bubbles (Magnaudet and Eames, 2000; Ern et al., 2012), deformation of bubbles (Lu and Tryggvason, 2008; 2013), background turbulence (Hosokawa and Tomiyama, 2004; Lelouvetel et al., 2014), bubble collisions, coalescence and break up (Frank et al., 2008; Liao and Lucas, 2010), as well as swarm effects and formation of large structures (Lance and Bataille, 1991; Panidis and Papailiou, 2000; Santarelli and Fröhlich, 2015), and oscillation on the container scale (Becker et al., 1999; Kantarci et al., 2005). Thanks to increasing computer power and improved algorithms numerical simulations can nowadays contribute to the understanding of such phenomena. However, bubbly flows involve an extremely wide range of length scales which cannot be addressed by a single simulation. Here, the multiscale approach advocated by Kuipers and coworkers (Deen et al., 2004; Lau et al., 2011) constitutes a suitable framework. Individual simulations are then configured to address features concerning a pre-determined range of length and time scales, with small scales represented by certain assumptions or model terms and large scales introduced via the boundary conditions. In an earlier paper of the present authors (Santarelli and Fröhlich, 2015), labeled SF15 in the following, simulations of two bubble swarms in a vertical channel were conducted to address bubble–turbulence interaction, void distribution, swarm effects, cluster formation, etc., hence investigating the range from the bubble scale to about 50 times the bubble scale. This was done for a void fraction of 2.14% and 0.29%, using the same bubble size but a different number of bubbles. These simulations are unique in that they feature a comparatively large domain with several thousand bubbles and realistic density ratio.

The present paper reports on further simulations in the same framework. A swarm with void fraction 2.14% composed by larger but fewer bubbles is considered to elucidate the impact of the bubble size. Second, a bidisperse swarm is simulated with the same void fraction and half of it represented by the smaller bubbles and half of it by the larger bubbles. Considering these topics a brief review is provided now focusing only on the investigations strictly related to the results presented here.

Göz and Sommerfeld (2004) numerically investigated the rise of bidisperse swarms in otherwise quiescent fluid using a triple periodic domain and considered several combinations of void fraction and bubble volume ratio. The liquid turbulence induced by the

* Corresponding author. Tel.: +49 35146334736.
E-mail addresses: claudio.santarelli@tu-dresden.de (C. Santarelli), jochen.froehlich@tu-dresden.de (J. Fröhlich).

http://dx.doi.org/10.1016/j.ijmultiphaseflow.2016.01.004
0301-9322/

bubbles as well as bubble pairing were discussed and it was observed that the presence of the second class of bubbles can modify the behavior of the bubbles when compared to the monodisperse swarms. Roghair et al. (2013) performed Direct Numerical Simulations (DNS) of bidisperse swarms of deformable bubbles in a triply periodic domain to extend the correlation between the rise velocity of the swarm and the void fraction toward bidisperse swarms. This analysis was performed for different swarms, i.e. for different bubble diameters, different void ratios, and different ratios of the number of large to the number of small bubbles. As a result the authors observed that the correlation proposed agrees well with the one derived for monodisperse swarms. Lu and Tryggvason (2013) investigated the effect of one large deformable bubble on the dynamics of a swarm of small bubbles in turbulent channel flow, compared to a swarm consisting only of small bubbles. The large bubble, rising mainly in the channel center, increased the turbulent kinetic energy (TKE) in the core region due to the production of vorticity in the wake region.

As described above, the information gained by DNS can be employed for developing closure relations in the framework of Euler–Euler computations which allow simulating much larger and more realistic configurations. Politano et al. (2003) used a polydisperse approach combined with the standard $k-\epsilon$ model to investigate the influence of the bubble size on the flow features of upward bubbly flows. Similarly, Krepper et al. (2005; 2008) developed a so-called "inhomogeneous multiple size group" model to account for many classes of bubbles in the framework of Eulerian modeling. The same approach was used by Diaz et al. (2008) to investigate the flow in a bubble column and the influence of the number of bubble classes was addressed. A somewhat different approach was also employed in recent years to account for the polydispersity of the bubbles, where the numerical equations for the fluid are coupled with Population Balance Models (Yeoh et al., 2014). These models account for the different properties of the bubble classes, such as size and velocity. The approach implies the solution of a transport equation for each bubble class and can therefore lead to considerable consumption of computer resources. To overcome this problem, the equations can be rewritten to account for the moments of the targeted distributions (e.g. of the bubble size distribution), as proposed by Hulburt and Katz (1964) and later extended by McGraw (1997) where quadrature approximations are introduced to solve the equations. Buffo et al. (2013) and Yuan et al. (2014), for example, employed improved versions of this method to simulate the flow in a rectangular bubble column where bubble polydispersity was accounted for. The aforementioned numerical works are based on models and this brief overview highlights the need to perform trustworthy fully resolved simulations of bubbly flows where the bubble size and the polydispersity of the swarm strongly influence both the fluid and the bubble dynamics in order to furnish reference data for modeling.

In the present work the focus is on the simulation of bubble swarms in turbulent channel flows, and the analysis started in SF15 is extended so that the new results and the comparison to the previous ones provide detailed information on the influence of the bubble size and of the bidispersity on the flow. As in SF15, the bubbles investigated here have a fixed, spherical shape and contaminated fluid is considered. The simulation set presented here can therefore throw light on the complex physical phenomena involved in this type of flows. Additionally, statistical reference data regarding both phases are provided allowing validation of numerical algorithms and development of closure relations for the modeling of turbulent bubbly flows.

The paper is organized as follows. The numerical methodology employed and the configuration investigated are briefly recalled in Sections 2 and 3, respectively. In Section 4 results for a swarm of large bubbles are reported for both the fluid and the disperse phase. Analogously, in Section 5 the simulation of a bidisperse swarm is addressed. In the last section the main conclusions are reported. Additionally, a detailed validation of the numerical code for the chosen discretization and parameter range is provided by means of two reduced problems in the Electronic Annex of the online version of this article.

Numerical approach

The numerical method employed for the present study is based on a phase-resolving Euler–Lagrange approach. The Navier–Stokes equations for an incompressible fluid are solved on a staggered Cartesian grid using a three-step Runge–Kutta method for the advection in time and a semi-implicit Crank–Nicholson scheme for the evaluation of the viscous terms in each step, completed with an equation for the pressure correction. Bubbles are introduced by an Immersed Boundary Method with their spherical shape being described analytically. The coupling to the fluid phase is accomplished at a given number of forcing points. Such points are distributed on the bubble surface to impose the desired boundary condition. This algorithm was extensively described in Kempe and Fröhlich (2012) and was subsequently employed for the simulation of several physical problems (Heitkam et al., 2012; Kempe et al., 2014; Vowinckel et al., 2014). In Schwarz et al. (2015) the procedure was improved to account for very light particles and bubbles introducing a numerical virtual mass force in the bubble equations of motion (EOM). In Schwarz and Fröhlich (2014) the rise of a single bubble in liquid metal was investigated employing the same method that is used here.

In the present work bubbles are considered as spherical objects and a no-slip condition is applied at the phase boundary. This matches with the behavior of small air bubbles rising in contaminated water, for example. A collision model accounts for bubble–bubble and bubble–wall collision events and is based on the introduction of a repulsion force in the EOM, proportional to the distance between the surfaces (Heitkam et al., 2012). The choice of the parameters in the collision model, as described in SF15, ensures that there is always at least one Eulerian point between the two colliding surfaces, both for bubble–bubble and bubble–wall collision events.

Configuration

Apart from the definition of the bubbles the configuration studied here corresponds to the one in SF15 and features the rise of bubble swarms in upward vertical channel flow. The flow is confined between two parallel vertical walls, where a no-slip condition for the fluid velocity is applied, with y being the wall-normal coordinate and H the distance between the walls. The flow is periodic in the streamwise (x) and spanwise (z) direction. The extensions of the channel are $4.43H \times H \times 2.21H$ in streamwise, wall-normal and spanwise direction, respectively. This choice was discussed in SF15 comparing the present domain to simulations of two-phase flows in similar configurations (Uhlmann, 2008; Garcia-Villalba et al., 2012; Lu and Tryggvason, 2013).

The domain was discretized with $1024 \times 232 \times 512$ mesh points in the x-, y- and z-direction, respectively, yielding a total number of around 120 million grid points which is the same mesh as used in SF15. The gravity force is oriented in the negative x-direction and the mass flow rate was kept constant by means of a volume force in the x-direction, f_x, which is constant in space and instantaneously adjusted in time to yield the desired flow rate. This is a procedure commonly applied for channel flows with streamwise periodicity. Hence, the bulk velocity U_b, averaged over liquid and gas, is constant in time and equal for all simulations, yielding the same bulk Reynolds number $Re_b = U_b\, H/\nu$ for all simulations.