Numerical Simulation of Bubble Breakup and Coalescence in a Turbulent Boundary Layer Flow

Xiaosong Zhang1,2, Jianhua Wang1, Decheng Wan1*

1 Computational Marine Hydrodynamics Lab (CMHL), State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai, China
2 CSIC Shanghai Marine Energy Saving Technology Development Co., Ltd, Shanghai, China

ABSTRACT

It has been proved that injecting bubbles into a turbulent boundary layer can reduce the frictional drag on the plate. Many studies have simulated this problem using Euler-Lagrange method to accurately calculate the motion of bubbles. However, bubble coalescence and breakup have not been fully considered in the Eulerian-Lagrangian simulation of bubble drag reduction problem. The main goal of this paper is to develop bubble breakup and coalescence modules in an Euler-Lagrange code. Simulation of bubbles injected into a turbulent boundary layer is performed. And by means of these two modules, the bubble size distribution is analyzed in detail.

KEY WORDS: Bubble breakup; bubble coalescence; turbulent boundary layer; Eulerian-Lagrangian simulation.

INTRODUCTION

Using bubbles to reduce skin frictional resistance of ship or pipe has been a focus for researchers for a long time. These kinds of practical problems can be simplified to injecting bubbles into a turbulent boundary layer. Hassan et al. (2005) injected bubbles into the channel flow by electrolysis of water and obtained obvious drag reduction effect. In their next experiment, Ortiz-Villafuerte and Hassan (2006) investigated a turbulent boundary layer laden with microbubbles and adopted particle tracking velocimetry technique to analyze the effects of bubbles on turbulent coherent structures. Results showed that the velocity fluctuation in the flow direction is strengthened while the velocity fluctuation perpendicular to the plate is weakened. Murai et al. (2006, 2007) conducted similar experiments and measured the Reynolds stress distribution in the channel. It was found that the Reynolds shear stress decreased effectively near the wall surface, then quickly recovered as the flow develops to the core of the channel and increased outside the original boundary layer. The Reynolds shear stress near the wall is an important part of the turbulence friction resistance. Therefore, the reduction of Reynolds shear stress near the wall was regarded as an important drag reduction mechanism. Above experimental studies have shown that only bubbles entering the turbulent boundary layer can contribute to drag reduction. There are obvious fluid velocity fluctuation and strong shear stress in turbulent boundary layer, which can lead to the breakup and coalescence of bubbles. Bubble breakup and coalescence lead to the continuous change of bubble size distribution, which has an important effect on drag reduction. This phenomenon also puts forward higher requirements for the numerical simulation algorithm.

There are two commonly used methods for numerical simulation of bubble drag reduction problems, Euler-Euler method and Euler-Lagrange method. These two methods correspond to macroscopic algorithm and microscopic algorithm, respectively. Kunz et al. (2003, 2007) developed an unstructured Eulerian-Eulerian code and simulated the bubble drag reduction across a wide range of Reynolds numbers. The code was validated with qualitative and quantitative comparisons. Mohanarangam (2009) combined the Euler-Euler method with population balance model (PBM) to predict the bubble coalescence and breakup in the simulation. More recently, Qin et al. (2017) carried out both experimental test and Eulerian-Eulerian simulation of bubble drag reduction in a turbulent boundary layer. Bubble breakup and coalescence were modeled by the combination of PBM. Bubble size distribution was analyzed in detail. On the other hand, Euler-Lagrange method was adopted in many studies to investigate underlying mechanism. Mattson and Mahesh (2011) developed a one-way coupled Euler-Lagrange code, in which the fluid is solved by DNS method. Bubbles were found to migrate away from the plate, which was regarded as the main reason for drag reduction failure in the downstream. Pang et al. (2014) studied the interaction between bubbles and liquid turbulence with the help of a two-way coupled Euler-Lagrange code. Asiagbe et al. (2017) was the first to use LES model to simulate turbulent flow in the numerical simulation of bubble drag reduction problems. Their results showed that LES model can also resolve the velocity fluctuations reasonably with less computational resources in contrast with DNS method. More recently, Rawat et al. (2019) improved the two-way coupled algorithm to enhance numerical stability when bubbles are close to the wall.

As a usual engineering method in the past, the Euler–Euler model combined with the population balance equation was widely used to predict the bubble size distribution with the help of statistical models.
However, it is worth noting that bubble breakup and coalescence were not considered in the previous Eulerian-Lagrangian simulation for bubble drag reduction. In contrast with Euler-Euler model, the Eulerian-Lagrangian simulation has the advantage to accurately solve the position and velocity of each bubble, and hence the breakup and coalescence of bubbles can be more easily and accurately simulated. Therefore, the main goal of the present work is to predict bubble breakup and coalescence in the Eulerian-Lagrangian bubble drag reduction simulation. The computational code is developed based on the open-source platform OpenFOAM to form a solver dealing with bubbly flow. Bubble breakup and coalescence are implemented in the code as independent modules. A breakup model based on the constraint of a critical Weber value is adopted for bubble breakup simulation. Bubble coalescence is simulated by the film drainage model.

The paper is organized as follows. The numerical method is presented first. Governing equations of our Euler-Lagrange code and the two-way coupled algorithm are introduced. Besides, the developments of bubble breakup module and bubble coalescence module are explained in detail. Next, the computational domain and conditions for the turbulent boundary layer are presented. Numerical results with different daughter bubble size distribution and the flow characteristics are discussed in the simulation part. A brief conclusion is given in the end.

NUMERICAL METHOD

Governing equations for liquid phase solving

In the present study, turbulent boundary layer flow field is solved using Large Eddy Simulation (LES) method. The filtered continuity and momentum equations can be written as:

$$\frac{\partial \bar{\alpha}}{\partial t} + \frac{\partial \bar{\alpha} \bar{u}_i}{\partial x_i} = 0$$

(1)

$$\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = - \frac{1}{\rho} \frac{\partial P}{\partial x_i} - \frac{\partial}{\partial x_i} \left( \tau_{ij} \right) + \alpha g + \frac{f_{L}}{\rho}$$

(2)

where $u_i$ is fluid velocity in three direction ($i = x, y, z$), $\alpha$ is the fluid volume fraction, $\rho$ is the fluid density, $p$ is pressure, $\tau_{ij}$ is viscous stress. The overbar identifies filtered quantities. $f_{L}$ represents the coupled forces from bubbles to the liquid. The sub-grid scale stress tensor $\tau_{ij}$ is required to be closed. The SGS stress is modelled using Wall Adapting Local Eddy-Viscosity Model (WALE) (Nicoud, 1999), in which the turbulent kinematic eddy-viscosity is defined as:

$$\nu_t = \left( C_n \Delta \right)^2 \left( \frac{S_x}{\bar{S}_x} \right)^{\nu_2} + \left( \frac{S_y}{\bar{S}_y} \right)^{\nu_2}$$

(3)

where the filter width $\Delta$ is calculated by $\Delta = V^{1/3}$. $C_n$ is a constant number, which is set to be 0.325 in the present study. $S_x$ is the strain-rate tensor.

Governing equations for bubble tracking

Bubbles are assumed to be non-deformable spheres under a small We number condition. Each bubble is tracked individually by the kinematic equation based on Newton’s second law as follows:

$$m \frac{dv}{dt} = f_d + f_L + f_r + f_g + f_c$$

$$\frac{5mC_p}{4d} \left( u - v \right) + \frac{mp}{\rho_s} C_L \left( u - v \right) \times (\nabla \times u)$$

(4)

$$+ \frac{mp}{\rho_s} \frac{Du}{Dt} + \rho_s \left( 1 - \frac{\rho_s}{\rho_L} \right) + f_c$$

where $v$ represents the bubble velocity, $m$ is the mass of bubble, $d$ is bubble diameter, $\rho_s$ and $\rho_L$ represent the liquid density and bubble density, respectively. Bubbles are subjected to drag force, lift force, fluid acceleration force, gravity, buoyancy and collision force. These forces are written as independent terms on the right side of Eq.4. Drag force coefficient $C_D$ and lift force $C_L$ coefficient are determined by Tomiyama’s drag model (Tomiyama, 2002) and lift model (Tomiyama, 2002), which have been validated and widely used in various bubble systems. A non-linear collide force model proposed by Heitkam et al. (2017) is adopted to calculate the bubble-bubble collision and bubble-wall collision.

Two way coupling algorithm

In order to improve the accuracy of simulation, it is necessary to calculate the reaction of bubbles to fluid in the cases with high bubble phase fraction. The key problems are the calculation of void fraction distribution and coupled forces distribution from the instantaneous bubble sizes and locations. In the present study, a Gaussian distributed scheme as in reference (Ma, 2015) is adopted to smooth the bubble volume into the grid it should be in.

As shown in Fig.1, if a bubble (green circle in the figure) is larger than a cell, then the void fraction of all surrounding cells in the “affect region” (green box in the figure) will receive the contribution of the bubble. For the cell celli in Fig.1, the void fraction can be calculated by:

$$\alpha_i = \sum_{j} \frac{f_{i,j} V_j}{\sum_{k} (V_k f_{k,i})}$$

(5)

where $f_{i,j}$ is a weight function. A three dimensional Gaussian weight function is adopted to express the shape of bubble. The expression of the function is:
where $|x_{ij}|$ represents the distance between cell $k$ and bubble $j$, and $S$ is the standard deviation.

**Bubble Breakup**

Numerical model of bubble breakup includes two parts. One is the breakup criteria, and the other is the daughter size distribution. First for the breakup criteria, there are many mechanisms and corresponding models for bubble breakup in the previous studies, such as turbulent fluctuation, viscous shear stress and interfacial instability. These models can be written as a criterion of critical Weber number. The model proposed by Hesketh (1987) is adopted in the present work because that the influence of wall was included in their experiment. Hence the breakup criteria in our model is given by:

$$We = \frac{\rho \Delta u^2(d) d}{\sigma} > We_{crit} = 1.1(\frac{\rho}{\rho_b})^{1/3}$$

where $\sigma$ is the surface tension coefficient. $\Delta u^2(d)$ represents the mean square velocity difference over the bubble diameter. Bubble breakup is assumed to be binary in the present model. Thus the daughter bubble size after breakup is determined by the breakup volume fraction $f_{v}$, $f_{s}$ is a random variable, which obeys some specific distributions. Based on previous theoretical analysis and experimental measurements, it can be summarized that there are mainly four types of distributions as shown in Table 1.

There are significant differences between the distributions in Table 1. The U-shaped distribution is the most commonly used in previous studies for bubble flow system such as bubble column. However, the applicability of these distributions in simulating bubble drag reduction problems still needs to be studied, which is an important work in this paper.

Table 1 Different kinds of daughter size distribution

<table>
<thead>
<tr>
<th>Type</th>
<th>Reference</th>
<th>Shapes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Uniform Distribution</td>
<td>Narasimhan and Gupta (1979)</td>
<td><img src="image" alt="Uniform Distribution" /></td>
</tr>
<tr>
<td>Bell-Shape Distribution</td>
<td>Lee et al. (1987)</td>
<td><img src="image" alt="Bell-Shape Distribution" /></td>
</tr>
</tbody>
</table>

In every time step, each bubble is evaluated based on the breakup criteria, the corresponding breakup frequency is $1/\Delta t$. If the criteria is true, then breakup takes place. Diameters of the two daughter bubbles are calculated by the size distribution model first. The daughter bubble with the larger diameter is placed at the original position of the parent bubble, while the smaller daughter bubble is placed at a random position with a specified distance away from the larger bubble. As shown in Eq. 8, $\Delta r$ is a random vector whose magnitude is $0.6 \times (d_e + d_i)$ to avoid overlap with the other daughter bubble.

$$r_i = r_0, \quad r_e = r_0 + \Delta r$$

Another problem worth noting is that the new daughter bubble should not overlap the existing bubbles in the flow field or the boundary of the computational domain. Therefore, it takes several loops here to make sure that the new position is good to make sure that the calculation goes smoothly.

**Bubble Coalescence**

Film drainage model is the most widely accepted bubble coalescence model and has been implemented in the code. The basic concept of film drainage model is that there is a thin liquid film between two bubbles when they contact. The liquid film drains out to a critical thickness while two bubbles stay in contact and the coalescence takes place. The criteria is that the duration of two contact bubbles must be larger than a critical time called “drainage time”. The drainage time proposed by Prince and Blanch (1990) is adopted in the present simulation, which can be written as:

$$t_{drainage} = \frac{d_e^4 \rho}{128 \sigma} \ln(\frac{\theta_f}{\theta_i})$$

where $\theta_i$ and $\theta_f$ is the initial liquid film thickness and final liquid film thickness, respectively. As an air-water system, $\theta_i$ is $10^{-4}$ and $\theta_f$ is $10^{-8}$ in the present simulation. $d_{eq}$ is the equivalent radius, which is defined as:

$$d_{eq} = 2\left(\frac{1}{d_1} + \frac{1}{d_2}\right)^{-1}$$
In many previous simulations using the hard sphere to calculate bubble-bubble interaction, the contact time is modeled by specified model because that the hard sphere model cannot handle the elastic collision process. However, this approximation can be eliminated in the present simulation. Bubble interaction pair list is established and updated in every time step. A variable $I_{\text{contact}}$ is used to record the cumulative contact time, if $I_{\text{contact}}$ is smaller than $I_{\text{traingate}}$, then collision is calculated, otherwise coalescence is calculated.

When coalescence takes place, one of the two bubbles is deleted. Diameter and velocity of the other bubble is updated to keep mass and momentum conservation. The expressions are:

$$d_c = \left( d_a^3 + d_b^3 \right)^{1/3}, \quad U_c = \frac{d_a^3 U_a + d_b^3 U_b}{d_a^3 + d_b^3}$$

(11)

COMPUTATIONAL CONDITIONS

A simple box domain is adopted in the present simulation. Bubbles are injected from a boundary at the upper left corner of the computational domain. Illustration of the computational domain can be seen in Fig.2. Qin et al. (2017) carried out experimental investigation for BDR (Bubble Drag Reduction) in turbulent boundary layer using a flat plate with 1.5m in length. Bubble injector is placed at 0.123m away from the leading edge. The corresponding Reynolds number is $1320 < \text{Re}_\theta < 9800$ using a power law to estimate boundary-layer momentum thickness. Length of the present computational domain is the same as their experiment, while width is 0.3m, which is 1/10 of the experiment plate in order to reduce computational cost. Height of the computational domain is set to ensure that the boundary layer never occupies more than one-third of the domain. Air flow rate $Q=47.10$L/min is chosen to simulated. All bubbles are 1mm when injected into the field, which is the same as the diameter of nozzles in the experiment.

Fig.2 Illustration of the computational domain.

Before injecting bubbles into the flow field, a turbulent boundary layer is generated by precursor channel flow simulation method (Mukha, 2017). The inflow-generation simulation is a channel flow with specific frictional velocity. After the channel flow is fully developed, turbulent flow at each time step is sampled from a slice in the field. These flow slices are input as boundary condition in the boundary layer case. Fig.3 shows flow slices at four downstream locations.

Fig.3 Turbulent flow slices at four downstream locations. (a) x/L=0. (b) x/L=0.33. (c) x/L=0.67. (d) x/L=1.

Simulations are carried out on the HPC cluster center in Computational Marine Hydrodynamics Lab (CMHL), Shanghai Jiao Tong University. Each node consists of 2 CPUs with 20 cores per node and 64GB accessible memory (Intel Xeon E5-2680v2 @2.8 GHz). 40 processors are assigned to calculate the BDR case. It costs approximately 180 hours of clock time to complete one computation. Calculation is relatively fast in the early stage due to the small number of bubbles. As the bubbles are continuously injected and broken into small bubbles, the solving of bubble-bubble interaction will slow down the calculation. At the same time, the adopted two-way coupled process is relatively time-consuming and still needs to be further improved.

RESULTS AND DISCUSSION

Effect of daughter bubble size distribution

Four numerical simulations are performed with different kinds of daughter bubble size distributions, which are: uniform, Bell-shape, M-shape and U shape. Bubble coalescence is considered in all of these cases. Bubble size distribution is collected after the flow field is stable. The comparison between numerical results and experimental results can be seen in Fig.4.

In terms of the overall distribution, the maximum number density for all numerical models and experiment data is around $d=1$mm, which equals to the initial injection bubble diameter. The reason for this phenomenon is the coalescence of bubbles downstream. In particular, the results obtained by using the U-shape model are the most consistent with the experiment in terms of the maximum value, because bubble breakup in the U-shape model tends to produce two daughter bubbles with a large diameter difference. For the distribution of bubbles with diameter smaller than 1mm, the results obtained by using the Bell-shape model showed the shape of the central bulge, the diameter of bubbles is mainly concentrated around 0.5mm. This is due to the low probability of producing small daughter bubbles and high probability for equal volume break-up. Result of uniform model is similar as the results of Bell-shape model under the action of bubble coalescence module. M-shape model performs a little better in the prediction of smaller size bubbles, but the effect is not obvious. Among these daughter bubble size distribution models, only the U-shape distribution is the most representative of the underlying physical phenomena. It is because binary equal size break-up requires more energy than binary unequal size break-up (Lau, 2014). On the other hand, the numerical results using U-shape model agree the best with the experimental results in trend. Therefore, the following analyses are based on the simulation with U-shape model.
Flow Characteristics

The comparison of drag reduction effect and total bubble number in the field between experiment data and simulation results with/without breakup and coalescence models are shown in Table 2. $C_f$ and $C_{f0}$ are the frictional drag coefficients with and without bubble injection, respectively. $C_f$ is calculated by $C_f = 2f/(\rho SU^2)$, where $\rho$ is water density, $S$ is plate surface area and $U$ is water flow velocity. $f$ is obtained by integrating the plate surface frictional shear stress, which is controlled by the mixed viscosity and velocity field.

Numerical results without breakup and coalescence model are obtained from our previous simulation (Zhang et al., 2020). It can be seen from Table 2 that both of the two kinds of numerical results are slightly lower than the experimental measurement, and the present results show a larger error. However, the data do not mean that the proposed algorithm in this paper will deteriorate the results. According to the discussion from the paper providing the experimental data, the measured value is higher than the actual value due to the marginal area on the span-wise of the plate. Therefore, it can be believed that both numerical results are acceptable. On the other hand, it can be seen from Fig.4 that the diameters of most bubbles remain close to the injection diameter, which means that the simulation with uniform bubble diameters can also obtain fine results. If the flow field fluctuation is further increased, such as increasing the water velocity, the bubble size distribution will change significantly, and the advantages of the algorithm with bubble breakup and coalescence models will be further reflected. The total bubble number in the simulation with breakup and coalescence models is nearly twice as that in the simulation without the models, which is caused by the bubble breakup under the action of turbulence. The phenomenon can be seen more clearly from Fig.5.

Table 2. Comparison of drag reduction effect.

<table>
<thead>
<tr>
<th></th>
<th>$C_f / C_{f0}$</th>
<th>Error</th>
<th>Bubble number</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment</td>
<td>0.88</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>(Qin et al., 2017)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E-L Simulation without b&amp;c</td>
<td>0.86</td>
<td>2.7%</td>
<td>$\approx 78k$</td>
</tr>
<tr>
<td>(Zhang et al., 2020)</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>E-L Simulation with b&amp;c (Present work)</td>
<td>0.833</td>
<td>5.4%</td>
<td>$\approx 148k$</td>
</tr>
</tbody>
</table>

Fig.5 shows the evolution of two-phase flow field. Bubbles are shown as spheres with different diameters. Under the action of turbulence, bubbles exhibit complex dynamic behavior, including rotation, oscillation, collision, breakup and coalescence. There is a significant migration feature for the motion of bubbles. As the flow develops downstream, migration movement of the bubbles away from the plate becomes more and more obvious. It has been proved that the local void fraction near the plate is the most important factor of bubble drag reduction. Thus, the migration of bubbles has a significant influence on the effective length of BDR.

Bubbles are all 1mm when they are injected into the flow field. Then with the development of two-phase flow, bubbles generally form a new size distribution. It can also be found in Fig.5 that bubbles with different diameters have different spatial distributions. Next, the bubble size distributions in stream-wise direction and wall-normal direction are analyzed in detail.
The total length of the turbulent boundary layer is divided into five parts to analyze the bubble distribution in stream-wise direction. The change of average bubble diameter is shown in Fig.6. It can be found that the average bubble diameter increases gradually along the downstream direction. Bubbles are smaller near the injector and larger near the outlet, which indicates that breakup is the dominant behavior when bubbles are just injected and coalescence is more frequent downstream.

![Fig.6 Bubble size distribution in stream-wise direction.](Image)

According to the comparison, U-shape model performs better in prediction of bubble size distribution. The evolution of bubbles with this model is presented and obvious bubble migration characteristic can be found. Bubble size distributions in stream-wise direction and wall-normal direction are analyzed in detail. Bubbles with smaller diameter are found to be easier to enter the inner layer of turbulent boundary layer.

Future works will focus on the improvement of computational accuracy and efficiency. It can be seen from the comparison of drag reduction effect that the accuracy of the numerical simulation needs to be further optimized. In addition, current algorithms also need to be optimized to reduce unnecessary computation.

**ACKNOWLEDGEMENTS**

This work is supported by the National Natural Science Foundation of China (51879159), The National Key Research and Development Program of China (2019YFB1704200, 2019YFC0312400), Chang Jiang Scholars Program (T2014099), Shanghai Excellent Academic Leaders Program (17XD1402300), and Innovative Special Project of Numerical Tank of Ministry of Industry and Information Technology of China (2016-2309), to which the authors are most grateful.

**REFERENCES**


Lee, CH, Erickson, LE, Glasgow, LA. (1987). “Dynamics of bubble size...