Numerical investigation of a single intermediate-sized bubble in horizontal turbulent channel flow

Sangwon KIM*, Nobuyuki OSHIMA*, Yuichi MURAI** and Hyun Jin PARK**

*Department of Mechanical and Space Engineering, Hokkaido university
Kita-13, Nishi-8, Kita-ku, Sapporo, Hokkaido 060-8628, Japan
E-mail: swkim5834@eis.hokudai.ac.jp

**Division of Energy and Environmental Systems, Hokkaido university
Kita-13, Nishi-8, Kita-ku, Sapporo, Hokkaido 060-8628, Japan

Received: 8 June 2020; Revised: 16 August 2020; Accepted: 6 October 2020

Abstract
Air lubrication systems have gained considerable popularity as a promising drag reduction technology in recent years. However, numerical simulations of intermediate-sized bubbles are quite challenging because of the numerical diffusion of the conventional method and the high deformability of the bubbles. This hinders the study of the physical mechanisms involved in a variety of phenomena in such types of bubbles, such as the bubble–liquid interaction effect, high bubble deformation, and flow in the liquid film generated above the bubble. In this study, a solver, viz. interIsoFoam of OpenFOAM, which is directly captured by the improved volume of fluid method, was applied to solve the gas–liquid interface problem. We established the numerical procedure by dividing it into three stages and validating the accuracy of the given solver to minimize numerical errors such as smearing the volume fractions. The numerical results for variables such as the bubble shape, the skin friction of the liquid film, and the instantaneous momentum flux display trends similar to those observed in the experiments. The calculated bubble shows a high skin friction in the secondary flow, which corresponds to the distribution of streamwise vortices in the secondary flow.

Keywords: Two-phase flow, Turbulence, Drag reduction, Bubble deformation

1. Introduction

To address the increasing concern regarding global warming, the International Maritime Organization (IMO) regulated the requirements regarding greenhouse gas emissions from ships. To reduce these emissions, air lubrication systems are being used as this is an energy-saving technology developed to adhere to industry requirements (ABS, 2019). Air lubrication systems can be divided into two main categories based on the size of the bubbles.

One category is the microbubble method developed by McCormick and Bhattacharyya (1973), and the other is the air film method, which has been found to be practically viable in the last two decades (e.g., Fukuda et al., 1999, 2000). Meanwhile, intermediate-sized bubbles have recently attracted significant research interest, as bubble deformation plays a key role in the process of drag reduction, as studied by Moriguchi and Kato (2002) and Kitagawa et al. (2005). The intermediate-sized bubbles negatively contribute to the drag reduction performance in the downstream region of the microbubble and air film methods. However, Murai et al. (2006) investigated the drag reduction mechanism for these bubbles (10–50 mm) and discovered that a calm region is generated behind the bubble. This feature is quite different from those of the microbubble and air film methods.

This previous research has proven to be a turning point in studies related to intermediate-sized bubbles and is being considered as a new technique for improving drag reduction. An additional advantage is that supplying intermediate-sized bubbles is much easier than generating a high flow rate of microbubbles or stabilizing an air film over a wide area. Murai et al. (2007) and Oishi and Murai (2014) rigorously investigated and described influential characteristics such as the velocity gradient and $u'v'$ contours related to the drag reduction of a single intermediate-sized bubble. In numerical studies, Lu et al. (2005), Sugiyama et al. (2005), Kawamura (2005), and Spandan et al. (2017) also confirmed the positive
role of deformable bubbles in the process of drag reduction.

Although the deformability of the bubbles is considered to significantly affect the drag reduction in intermediate-sized bubbles, the above studies are confined to small bubbles or microbubbles, while no research has been conducted on intermediate-sized bubbles. In this study, we established a numerical procedure for the injection of intermediate-sized bubbles in turbulent channel flows. Subsequently, the behavior of these bubbles was investigated to verify that the numerical results followed the experimental trends.

![Fig. 1 Characteristics of intermediate-sized bubble. Mainly divided into liquid film, capillary wave and secondary flow, where $\tau_w$ and $\tau_0$ are the local skin friction and the mean skin friction of single-phase flow (Murai et al., 2007).](image1)

![Fig. 2 Schematic representations of the skin friction ratio normalized by that of single-phase flow ($\tau_0$) as the diameter of bubble ($D_b$) normalized by boundary layer thickness ($\delta$), which is modified by bubble injection (Park et al., 2015).](image2)

### 2. Characteristics of intermediate-sized bubbles

The intermediate-sized bubbles have sizes between those of microbubbles and air films. Although the definition of the size of the bubble is not a specific size of bubbles, it ranges from medium to large. The definition of this range depends on the authors, for instance, 10–90 mm by Murai et al. (2006), 2–90 mm by Murai et al. (2007), and 6.7–15.5 mm by Oishi and Murai (2014). Fig. 1 shows an intermediate-sized bubble migrating to the right side with the liquid flow, where $\tau_w/\tau_0$ is the local skin friction ratio. Murai et al. (2007) described the bubble characteristics as follows. (1) The intermediate-sized bubbles considerably reduce the local skin friction ratio during their passage. (2) The skin friction on the liquid film gradually decreases from the front to the rear of the bubbles, regardless of their size. (3) The skin friction after the passage of the bubbles increases and oscillates significantly. Intermediate-sized bubbles also face a limitation in terms of the drag reduction performance because they generally show a negative performance, as shown in Fig. 2. It can be observed that the skin friction ratio decreases when the size of bubbles becomes equal to that of a microbubble or an air film. Meanwhile, the skin friction of some bubbles in the intermediate region exceeds that of the single-phase flow.

The intermediate-sized bubbles are included in this region, which indicates that the inclusion of these bubbles should be avoided. It is important to understand the reason for the increase in the skin friction to obtain a stable drag reduction performance. As depicted in Fig. 1, it can be assumed that the secondary flow causes an increase in the average skin friction. In numerical simulations, the spatial relationship between the local skin friction and the vortical structures of the secondary flow has been investigated.

### 3. Numerical Method

#### 3.1 Governing Equations

In this study, the flow is assumed to be incompressible, and the fluids are assumed to be Newtonian. The governing equations in this study for the continuity and momentum equations can be expressed by Eqs. (1) and (2):

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (1)$$

$$..."
\[
\frac{\partial \rho \mathbf{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) = -\nabla p + \nabla \cdot \mathbf{T} + \rho \mathbf{f} + f_s
\]

(2)

where \(\rho\) is the fluid density, \(t\) is the time, \(\mathbf{u}\) is the fluid velocity, \(p\) is the pressure, \(\mathbf{T}\) is the viscous stress tensor, \(\mathbf{f}\) represents the body forces, and \(f_s\) is the surface tension force. In this study, the body forces include gravity and a pressure gradient in the streamwise direction. In a two-phase flow, the integral form of the continuity equation can be expressed as Eq. (3) by modifying the density term \(\rho(x,t)\) at space \(x\), and time \(t\) as the density field has a discontinuity at the interface. The indicator field in Eq. (4), \(H(x,t,\mathbf{t})\), has been defined to remove these insignificant parameters, and becomes 0 when \(\rho(x,t)\) is \(\rho_A\), and 1 when \(\rho(x,t)\) is \(\rho_B\).

\[
\frac{d}{dt} \int_C \rho(x,t) dV = -\int_{\partial C} \rho(x,t) \mathbf{u}(x,t) \cdot d\mathbf{S}
\]

(3)

\[
H(x,t) = \frac{\rho(x,t) - \rho_A}{\rho_A - \rho_B}
\]

(4)

where \(\rho_A\) and \(\rho_B\) are the densities of fluids A and B, respectively. Consequently, the integral form of the continuity equation can be rewritten as Eq. (5).

\[
\frac{d}{dt} \int_C H(x,t) dV = -\sum_{\partial C} S_j \int_{F_j} H(x,t) \mathbf{u}(x,t) \cdot d\mathbf{S}
\]

(5)

where \(C\), represents each cell, \(B_i\) is the list of all faces \(F_j\) belonging to cell \(i\), and \(S_j\) is used to orient the flux to go out of the cell. The left-hand side of Eq. (5) can be expressed by the volume fraction of fluid A in cell \(i\), as shown in Eq. (6). The right-hand side of Eq. (5) can be updated to consider the total volume fraction transported across face \(j\) during the time step \(\Delta V_i(t, \Delta t)\), as shown in Eq. (7). From these equations, Eq. (5) can be expressed using the volume fraction \(\alpha_i(t)\) at each time step, as shown in Eq. (8), where \(V_i\) is the volume of cell \(i\), and \(t\) and \(t + \Delta t\) are the instantaneous times.

\[
\alpha_i(t) = \frac{1}{V_i} \int_C H(x,t) dV
\]

(6)

\[
\Delta V_i(t, \Delta t) = \int_{t}^{t+\Delta t} \int_{F} H(x,\tau) \mathbf{u}(x,\tau) \cdot d\mathbf{S} d\tau
\]

(7)

\[
\alpha_i(t + \Delta t) = \alpha_i(t) - \frac{1}{V_i} \sum_{\partial C} S_j \Delta V_j(t, \Delta t)
\]

(8)

### 3.2 Numerical Schemes and Models

In this study, the computations were performed using the open-source CFD software OpenFOAM® version 1812. This code is based on cell-center finite-volume discretization, and the key components of the finite-volume method are the spatial interpolation and time integration schemes. Some of the discretization schemes and models are presented to aid the understanding of the forthcoming analysis in this subsection, as described in Table 1. The time discretization scheme specifies how the time derivative \(\partial/\partial t\) is handled in the momentum equations. In this study, the Backward scheme corresponds to a second-order accurate backward-differencing scheme that utilizes the current and two previous time-steps, and it can be expressed by Eq. (9).

\[
\frac{\partial}{\partial t} (\phi) = \frac{1}{\Delta t} \left( \frac{3}{2} \phi - 2 \phi^o + \frac{1}{2} \phi^{oo} \right)
\]

(9)

where \(\phi\) is the field variable and \(\phi^o\) and \(\phi^{oo}\) are the field variables in two previous time-steps. The global time step is \(10^{-5}\) s for satisfying the Courant-Friedrichs-Lewy (CFL) condition that the CFL number is below 1 and the interface CFL number is below 0.4. Divergence schemes specify how the convective terms are handled in Eqs. (1) and (2). In this study, total variation diminishing (TVD) schemes were used to calculate the face value by utilizing combined upwind and
central differencing schemes, as shown in Eq. (10).

\[
\phi_f = (1 - \Gamma)\phi_{UD} + \phi_{CD}
\]  

(10)

where \(\phi_f\) is an approximation of the face-averaged field value, \(\phi_{UD}\) is the upwind estimate of \(\phi_f\), and \(\phi_{CD}\) is the central differencing estimate of \(\phi_f\). \(\Gamma\) is the blending factor, which is a function of variable \(r\) representing the ratio of successive gradients, as shown in Eq. (11).

\[
r = 2 \frac{d \cdot (\nabla \phi)_p}{\phi_N - \phi_P}
\]  

(11)

in the above equation, \(\phi_P\) and \(\phi_N\) are the approximations of the face-averaged field value at the cell center \(P\) and the neighbor cell center \(N\); \(d\) is the vector connecting the cell center \(P\) and the neighbor cell center \(N\). The definition of \(\Gamma\) changes according to the TVD scheme. The momentum flux is handled by the \textit{limitedLinear} scheme, in which the blending factor is calculated using Eq. (12), where \(k\) is the input provided by the user. In this study, \(k\) was 0.1. The mass flux was handled by the \textit{vanLeer} scheme, which calculates the blending factor using Eq. (13).

\[
\Gamma = \max[\min(2r/k,1),0] \text{ for limitedLinear}
\]  

(12)

\[
\Gamma = \frac{(r + |r|)}{(1 + |r|)} \text{ for vanLeer}
\]  

(13)

In the case of the gradient and Laplacian terms, a \textit{Gaussian linear} scheme was used. The turbulence model was large eddy simulation (LES) with the dynamic one-equation eddy-viscosity model as a subgrid-scale model. Pressure-velocity coupling was treated by \textit{Pimple} algorithms that combine the pressure implicit with splitting of operators (PISO) and semi-implicit method for pressure-linked equations (SIMPLE) algorithms.

| Scheme/Model                  | Scheme/Model                  |
|-------------------------------|-------------------------------|
| Time discretization scheme    | \textit{Backward}             |
| Divergence schemes            | Gauss \textit{limitedLinear}, Gauss \textit{vanLeer} |
| Gradient schemes              | Gauss \textit{linear}         |
| Laplacian schemes             | Gauss \textit{linear corrected} |
| Pressure-velocity coupling    | \textit{Pimple Algorithm}      |
| Turbulence model              | Large eddy simulation         |
| Subgrid-scale model           | Dynamic one equation eddy-viscosity model |

3.3 Interface Capturing Method

In this study, the \textit{isoAdvector} method, originally proposed by Roenby et al. (2016), was used. \textit{isoAdvector} is a VOF-based geometric surface reconstruction method capable of capturing extremely sharp interfaces by introducing the concept of a face-interface intersection line moving across a face. In Eq. (8), the \textit{isoAdvector} method requires some assumptions to simplify the \(\Delta V(t,\Delta t)\) term. The first assumption is \(u(x,\tau) \approx u(x,t)\), which regards the velocity field as a constant \(\tau\) during the interval of the time step \([t, t+\Delta t]\). The second assumption is that \(u\) on the face \(F_j\) dotted with the differential face normal vector, \(dS\), can be approximated in terms of the constant volumetric face flux \(\phi_f\) in Eq. (14). The remaining surface integral is the instantaneous area of the face \(j\) submerged in fluid \(A\), which is referred to as \(A_j(t)\) in Eq. (15). Here, \(F_j \cap A(\tau)\) represents the face \(j\) submerged in fluid \(A\). By using this definition, Eq. (7) can now be written as Eq. (16).

\[
\mathbf{u}(x,t) \cdot dS = \frac{\phi_f(t)}{|S_j|} dS
\]  

(14)
\[ A_j(\tau) = \int_{\Omega_j} H(x, \tau) \, dS = \int_{\Omega_j \cap \Omega(t)} \, dS \]  
(15)

\[ \Delta V_j(t, \Delta t) = \frac{\phi_j(t) \cdot \phi_{j+1}(t)}{S_j} \int_{\Omega_j} A_j(\tau) \, d\tau \]  
(16)

here, theisoAdvector method introduces a sub-grid scale model for estimating \( A_j(\tau) \) and the intracellular distribution from the given volume fraction data to capture the local information distribution. As shown in Fig. 3, the procedure for constructing an interface can be summarized as follows: (1) Find the surface cells that contain a volume fraction ranging from \( 0 < \alpha < 1 \), as shown in Fig. 3a. (2) Interpolate the volume fraction of a vertex of the surface cells from all surrounding cells, as shown in Fig. 3b. (3) Calculate the face-interface intersection points between two vertices. Thereafter, as shown in Fig. 3c, the interface is constructed from the intersection points and lines in the surface cell. (4) As shown in Fig. 3d, the cell is split and the volume fraction is redistributed such that the sub-cells occupy either \( \alpha = 0 \) or 1. (5) Use the velocity field data to estimate the interface motion during the time interval.

Fig. 3 Schematic procedure of isoAdvector method. (a) Select surface cells (white) based on the exact interface. (b) Interpolate volume fraction of the vertexes from the centroid values of around cells. (c) Construct the interface based on the intersection points, which are calculated between the two vertexes. (d) Distribute the volume fraction to the sub-cells.

To demonstrate the performance of the interface capturing method, Rudman-shearing tests and Rudman-Zalesak solid rotation test (Rudman M, 1998) were conducted, as shown in Fig. 4. The results obtained using theisoAdvector and the conventional method, which is the multi-dimensional limiter for the explicit solution for interface reconstruction (MULES), were compared. The numerical conditions retained the resolution of the grid (800 × 800, physical dimensions 1 m × 1 m) while maintaining the interface CFL number below 0.4.

In the Rudman-Zalesak solid rotation test, a slotted disk was rotated around the center of the computational domain with two revolutions. It is evident that the shape obtained by MULES is severely distorted on the upper side of the disk,
while the shape obtained by the \textit{isoAdvector} preserves the original shape quite well, with a slight deformation in the high curvature regions. In the Rudman-shearing test, a disk was sheared clockwise to stretch a long filament during the former half period by a spiraling flow. Then, the filament was sheared again, anti-clockwise, until the end of the period to merge into the original disk. This situation is similar to a realistic problem with stretching, shearing, fluid merging, and breakup. As a result, the shape obtained by \textit{MULES} is severely deformed in a zigzag manner from the long filament, and the left and right sides of the disk are scattered into small parts when merging into the original disk. In contrast, using the \textit{isoAdvector} preserves the shape much better. Thus, it is expected that the \textit{isoAdvector} method would be more suitable for further research, such as the direct numerical simulation of intermediate-sized bubbles.

![Rudman-shearing test and Rudman-Zalesak solid rotation test](image)

**Fig. 4** Rudman-shear and Rudman-Zalesak solid rotation test results. The red and blue lines represent outline of initial and final shapes, respectively. The grey contours indicate the shape of half period in the case of Rudman-shear test, whereas, one rotation in the case of Rudman-Zalesak solid rotation test.

### 4. Numerical Conditions

The numerical condition was constructed based on the experiment of Oishi and Murai (2014), as shown in Table 2. In addition, the geometrical configuration of the computational domain is presented in Fig. 5. The dimensions of the domain were 10 mm height ($2h$), where $h$ is the half-height of the channel, 75 mm width and 200 mm length between the two horizontal walls. The length and width of the channel were made shorter than in the experiment by using periodic boundary conditions in the stream and spanwise directions. The mean velocity in the current simulation was slightly modified to maintain the same friction velocity ($u_\tau$), and all dimensionless numbers were calculated from the single-phase flow.

| Channel | Experiment | Present CFD |
|---------|------------|-------------|
| Size [mm] | 6000 × 10 × 100 | 200 × 10 × 75 |
| $U_{\text{mean}}$ [m/s] | 1.0 | 1.06 |
| $Fr = U_{\text{mean}}(2hg)^{0.5}$ | 3.19 | 3.37 |
| $Re = U_{\text{mean}}2h/\nu$ | 9260 | 9783 |
| $Re_\tau = u_\tau h/\nu$ | 260 |
| Fluid | Water [17 °C] |
| $\rho$ [kg/m$^3$] | 998.7 |
| $\nu$ [m$^2$/s] | 1.08 × 10$^{-6}$ |

Table 2 Numerical condition based on experiment.
Table 3 Boundary conditions and solvers in each stage.

|         | Stage 1                | Stage 2                                      | Stage 3                                      |
|---------|------------------------|----------------------------------------------|----------------------------------------------|
| Solver  | **PimpleFoam** →       | **InterIsoFoam** →                           |                                               |
| Inlet   | Cyclic (Inlet ↔ Outlet)| Fixed velocity (1.06m/s, 0, 0) →              | Cyclic (Inlet ↔ Outlet)                       |
| Outlet  |                        | InletOutlet →                                |                                               |
| Side    |                        | Cyclic (Left ↔ Right)                        |                                               |
| Bottom  | NoSlip →               | NoSlip, Codestream (Inject) →                | NoSlip, ConstantAlphaContactAngle (1°)       |
| Top     |                        |                                              |                                              |

In this study, the procedure of the numerical simulation was divided into three stages, which are as follows. (1) Generation of turbulent channel flow, (2) injection of intermediate-sized bubbles, and (3) maintaining turbulent channel flow with the injected intermediate-sized bubble, as shown in Fig. 6. The reason for the division of the total simulation is that the computational domain with periodic conditions is a closed system. In this system, some inlet conditions, such as the injection of bubbles without the outlet condition, can cause continuity error. Table 3 describes the solvers and boundary conditions for each stage. Two solvers distributed with OpenFOAM® version 1812 were employed. **PimpleFoam** is a large time-step transient solver for incompressible flow using the Pimple algorithm. In this study, this solver was used for turbulent channel flow in stage 1. **InterIsoFoam** is a modified version of the solver interFoam, which is a solver for incompressible, immiscible, and isothermal two-phase flows using the VOF method for stages 2 and 3. In **interIsoFoam**, the isoAdvector method is implemented for interface capturing instead of MULES. At the inlet and outlet, periodic conditions, also called cyclic conditions, were applied to the coupling condition between them. This coupling
was treated implicitly using the cell values adjacent to each pair of cyclic boundaries with the same topology. In stage 2, the cyclic condition was switched to FixedVelocity condition temporarily to avoid the continuity error in the closed system. For all simulations, the cyclic condition was used on the side boundaries. At the bottom and top walls, the NoSlip condition was utilized to set the velocity to zero. In stage 2, the Codestream was used to temporarily establish the injection condition on the top. This tool is a dictionary entry that contains the C++ OpenFOAM code that is compiled to generate the entry itself. The ConstantAlphaContactAngle condition was additionally applied in stage 3 to ensure that the bubble was detached from the wall and prevented the dispersion on the walls.

In stage 1, a fully developed boundary layer was prepared in the channel. The flow was driven in the whole region by a pressure gradient calculated to create a mean velocity of 1.06 m/s. The grid resolution parameters used in this study are provided in Table 4. The dimensionless thickness in each direction ($\Delta x^+$, $\Delta y^+$, $\Delta z^+$) is defined as:

$$
\Delta x^+ = \frac{u_{x^+}}{\nu}, \quad y^+ = \frac{u_{y^+}}{\nu}, \quad \Delta z^+ = \frac{u_{z^+}}{\nu}
$$

where $\Delta x$, $\Delta y$, and $\Delta z$ denote the thickness of a mesh-layer in each direction. Meanwhile, $\Delta y$ is the distance between certain points in the normal direction and the wall for the dimensionless distance from the wall, $y^+$. Fig. 7 shows the mean velocity profile and turbulence ratio in each direction and the Reynolds shear stress, which were normalized by the friction velocity. As the mean velocity was modified for the current simulation, the velocity profile below $y^+ = 20$ was well resolved but showed a slight deviation beyond that point. This phenomenon is similar to that of turbulent fluctuations, where $u'$ and $v'$ showed deviations after $y^+ = 20$. From these properties, it can be considered that the current mesh conditions were well resolved in the viscous sublayer and the buffer layer. However, the mesh was not sufficiently refined to resolve it accurately in the log-law region. Despite this, the overall trend of $u'v'$, which is one of the important indicators for estimating intermediate-sized bubble effects, followed the experimental results reasonably well.

In stage 2, the injection condition on the top was implemented by Codestream, as shown in Fig. 5b. A uniform injection velocity was set up for 0.28 m/s to maintain the interface CFL number below 0.4. The angle between the x-axis and the injection direction was set to 45° to safely inject the bubble. In addition, the gravitational acceleration acted downward in the normal direction in order to gain buoyancy. Consequently, the bubble was injected, and its shape was maintained successfully, as shown in Fig. 5c. In stage 3, the bubble successfully floated in the channel. In this stage, volume fractions with the value $\alpha = 0.99$ (referred to as noise fractions in this study) were observed, as shown in the left panel in Fig. 8a. In the case of these noise fractions in front of the bubble, this noise fraction remained in the wake region when the bubble made a lap around the computational domain because of the cyclic condition between the inlet and outlet. When the noise fractions were stuck on the upper wall, such as the red fraction in the left panel in Fig. 8a, the skin friction ratio, which was in the range of 0–1, corresponded to the location of the noise fraction, as shown in the right panel in Fig. 8a. In addition, this resulted in nonphysical values of variables such as pressure and velocity, and was considered to be a specific error for the isoAdvector method. These noise fractions mostly escaped from the rear parts of the bubble in the initial stage before the interface stabilized. Therefore, the SetFields utility was used, which is an auxiliary tool for OpenFOAM and is used to set the values of the fields in specific regions. By setting these volume fractions to $\alpha = 1$, the abnormal skin friction was removed, as shown in the right panel in Fig. 8b. A small number of noise fractions remained, but they had no effect on the skin friction when these fraction values existed throughout the channel without stuck noise on the upper wall, such as the green fraction values in the left panel of Fig. 8b.

### Table 4 Numerical parameters of computational domain.

| Size [mm] | $\Delta x^+$, $\Delta y^+$ and $\Delta z^+$ | No. of cell |
|-----------|---------------------------------------------|-------------|
| Streamwise | 0.494, 38.52, 405 | 84 |
| Spanwise | 0.893, 46.48, 84 | 92 |
| Normal | 0.8%–6.69, 92 | 6,259,680 (= 6.26 M) |

Total no. of cell
Fig. 7 Comparison of turbulent properties between the simulation and the experiment results. (a) Mean velocity. (b) Turbulence intensities. (c) Reynolds shear stress.

Fig. 8 Iso-surface of $\alpha = 0.99$ and contours of the skin friction ratio (a) before using setFields, (b) after using setFields. Iso-surfaces are divided into two categories; near the upper wall ($y^+ < 16$) and main flow layer ($y^+ > 16$).
5. Numerical Results and Discussion

5.1 Comparison with Experimental Results

In this study, the numerical results were verified to check whether the trends were similar to those of the experiments. The first part was compared with the experimental results of parameters such as the bubble shape, instantaneous momentum flux, and skin friction from Oishi and Murai (2014). Table 5 lists the physical parameters of the present bubble. In the table, the diameter is calculated using the bubble volume obtained from the numerical simulation. In both the experiment and the present study, the Weber numbers are similar, with minor differences. The bubble velocity in the current study was 4.7% higher than that in the experiment owing to the modification of the mean velocity. The normalized bubble velocity is similar in both cases, which shows that the simulation result matches the experimental trends. However, the equivalent diameter of the bubble in the simulation was approximately 34% smaller than that in the experiment. Hence, some trends such as the bubble shape, skin friction, and the development of the liquid film are expected to vary.

Table 5 Physical parameters of the present bubble.

| Weber number (We = \( \rho U_b^2 D_{\text{equiv.}}/\sigma \)) | Equivalent Diameter \( (D_{\text{equiv.}}) \) [mm] | Bubble velocity |
|---------------------------------|---------------------------------|-----------------|
| Present CFD                     | 168                             | 9.6             |
|                                 |                                 | 1.11            |
|                                 |                                 | 1.05            |
| Oishi and Murai (2014)          | 175                             | 11.0            |
|                                 |                                 | 1.06            |
|                                 |                                 | 1.06            |

Fig. 9 Trajectories of the bubble centroid in each direction (Red lines) and the selected points \( T_1 - T_4 \). (a) Streamwise direction. Black dotted line is approximated by the velocity of the experiment. (b) Spanwise direction. The amplitude in spanwise direction \( \eta_z \) normalized by the equivalent diameter \( (D_{\text{equiv.}}) \) is presented. (c) Normal direction. The amplitude in normal direction \( \eta_y \) normalized by the equivalent diameter \( (D_{\text{equiv.}}) \) is presented.
Fig. 10 Evolution of the bubble shape compared to the experimental result. The red line represents outline of the experiment; (a) Top view, (b) Side view. Here, $b$ is the height of the bubble, $l_1$ and $l_2$ are stream and spanwise length, and $\Delta_{film}$ is thickness of the liquid film, respectively.

Table 6 Shape factor of the present bubble.

| Time | $l_1$ [mm] | $l_2$ [mm] | $b$ [mm] | $l_1/l_2$ | Diff. | $l_1/b$ | Diff. | $l_2/b$ | Diff. | $\Delta_{film}$ [mm] |
|------|-------------|-------------|-----------|------------|-------|---------|-------|---------|-------|---------------------|
| Present CFD | | | | | | | | | | |
| T1 | 14.25 | 13.09 | 4.72 | 1.09 | 24% | 3.04 | 63% | 2.79 | 32% | 0.479 |
| T2 | 12.43 | 13.64 | 4.16 | 0.91 | 3% | 2.39 | 28% | 2.64 | 25% | 0.590 |
| T3 | 12.84 | 14.26 | 4.86 | 0.90 | 2% | 2.66 | 43% | 2.97 | 41% | 0.561 |
| T4 | 12.58 | 14.05 | 5.10 | 0.89 | 1% | 2.47 | 33% | 2.78 | 32% | 0.591 |
| Oishi and Murai (2014) | - | 11.46 | 12.67 | 5.74 | 0.88 | - | 1.86 | - | 2.11 | - |

Fig. 9 shows the trajectories of the bubble centroid. Here, T1–T4 were selected from the trajectories shown in Fig. 9 to compare the parameters at constant time intervals. The trajectory in the streamwise direction increases steadily and is similar to the approximated trajectory calculated from the bubble velocity in the experiment (see Fig. 9a). When viewed from the trajectory in the spanwise direction, it shows a constant state. The dimensionless amplitude which is normalized by the equivalent diameter from $t = 1.7$ s to $4.2$ s reached approximately 0.1 (see Fig. 9b). In addition, it is observed that the bubble bounces in the normal direction and converges to $y/h = 1.4$. However, the dimensionless amplitude is smaller than that in the spanwise direction (see Fig. 9c). From the results of the trajectory in each direction, it is concluded that the bubble in this simulation assumes a quasi-steady state. Fig. 10 presents the evolution of the bubble shape in T1–T4.
and the fluctuation of the capillary wave of the upper surface is observed to be the same as that in the experiment. The bubble shape of the top view gradually approaches the experimental result, and the bottom of the bubble stretches to the same position as that reached by the bubble in the experiment. However, the trend in the shape of the bubble from the side view differs from that in the experiment, as shown in the side view in Fig. 10. For instance, the front side of the bubble tilts less on the $x$-axis as compared to that in the experiment. In addition, the bubble shape is close to a compressed spheroid in the normal direction than the bubble shape in the experiment, and the direction of the capillary wave seems to be different between experiments and numerical calculations.

This is because the effect of the surface tension enhanced by the isoAdvector method and the coarse mesh resolution in the stream and spanwise directions neglected the extensional deformation and capillary wave. Thus, the investigation of these factors will be carried out in future studies. To compare the evolution of the bubble in $T_1$–$T_4$, the shape factors are described in Table 6. The shape factors that were calculated based on the parameters $l_1$, $l_2$, $b$, and $\Delta \text{film}$ are defined in $T_1$ in Fig. 10. On one hand, $l_1/l_2$ approaches the experimental results, and the difference between the simulation and the experiment is approximately 1–3%, except at $T_1$. On the other hand, $l_1/b$ and $l_2/b$ show larger ratios as compared to the value in the experiment and the differences in both the shape factors reach 25–43% relative to that of $l_1/l_2$. The reason is that $l_1$ and $l_2$ are larger than those in the experiment. In addition, the thickness of the liquid film occupies approximately 11% of the bubble height, while the thickness of the liquid film in the experiment was approximately zero. Thus, it is inferred that the bubble shape in the stream and spanwise direction is affected to a greater degree by its physical parameters such as the Weber number as compared to the bubble shape in the normal direction, even if the volumes of the bubble in the simulation and experiment are different. Fig. 11 shows the profiles of the instantaneous momentum flux $u'v'$ along the iso-surface of a single bubble at the bubble centroid. The quantity $u'v'$ indicates not only the dependence of the liquid behavior around the bubble shape relative to that of single-phase flow, but also the local instantaneous source of the momentum flux. Each $u'$ and $v'$ component is obtained from Eqs. (18) and (19).

\begin{align}
    u'(x, y, z) &= u(x, y, z) - \bar{u}_0(x, y, z) \\
    v'(x, y, z) &= v(x, y, z) - \bar{v}_0(x, y, z) = v(x, y, z)
\end{align}

(18) (19)

where, $u(x,y,z)$ and $v(x,y,z)$ are the instantaneous velocities in the stream and spanwise directions, respectively. $\bar{u}_0(x,y,z)$ and $\bar{v}_0(x,y,z)$ are the mean velocities in the respective directions of the single-phase flow. Here, $\bar{v}_0(x,y,z)$ is considered to be zero in Eq. (14). The instantaneous momentum flux is obtained on the interface as a function of the angle from the top of the bubble. The angle is defined based on the centroid of the bubble and is positive in the counterclockwise direction. In general, the region of the profile can be divided into two categories based on the angle, this is, the upper surface ($0^\circ < \theta < 90^\circ$) and the bottom surface ($90^\circ < \theta < 270^\circ$). The present data in the liquid film region show the momentum flux in the region, which is consistent with the experimental result.

![Fig. 11 Instantaneous momentum flux on bubble surface by the angle, normalized by square of the friction velocity.](image)

[DOI: 10.1299/jfst.2020jfst0020] © 2020 The Japan Society of Mechanical Engineers
However, the tendency of the instantaneous momentum flux $u'v'$ is different from the results seen in Fig. 11, as shown in Fig. 12, in which the region of the negative momentum flux appears over the side of the liquid film at all times. This implies that the local velocity in this region experiences backward acceleration. In contrast, the rear parts of the bubble show a spike in the graph of the momentum flux, as in the experiment. These regions correspond to the capillary wave and are close to the experimental value. Oishi and Murai (2014) explained this positive momentum flux by the induction of a high-speed sweep in the rear part for the migration of a large bubble. In addition, these positive and negative momentum fluxes are considered to be the reason for the bouncing of the bubble with time. Meanwhile, the bottom surface ($135° < \theta < 225°$) was not affected by the momentum flux in the simulation but was affected by it in the experiment. This fluctuation is due to the interfacial curvature that determines the position of the separation point. The bubble of the present data has a flat bottom, while that of the experiment has a high curvature and the front is tilted down in the x-axis, as shown in Fig. 10. Finally, we can observe an increasing momentum flux in the front part of the bottom surface ($235° < \theta < 270°$) during both the experiment and numerical simulation. However, an overestimation of the momentum flux can be observed. This is considered to be a numerical error due to the large size of the mesh in the streamwise direction. Fig. 13 shows the local skin friction ratio of the liquid films. The front side of the bubble is on the right side of the graph, which is based on a sample from time $T_1$-$T_4$.

![Fig. 12 Contours of the instantaneous momentum flux on the iso-surface of the bubble in $T_1$-$T_4$.](image)

![Fig. 13 Profile of local skin friction normalized by that of single-phase flow.](image)
The sampling location corresponds to the centerline of the bubble in the spanwise direction. As described by Murai et al. (2007), the present results show that the skin friction ratio initially increases because the instantaneous momentum flux moves backward from the front part of the bubble. After increasing the skin friction ratio, this ratio starts to decrease in the middle of the liquid film, after which it rapidly decreases to reach the minimum value in the rear part. The rapid decreases in the rear parts occurred even when the capillary wave was neglected. Thus, the positive momentum flux is considered to be the reason for the rapid decrease in the rear part because it is universally distributed in this region, as shown in Fig. 12. This also shows that there is similar thickness development between the wall and the top interface of the bubble. However, the value of the skin friction ratio varied. The skin friction ratios in T₁ and T₂ are greater than one in the whole region, while those in T₃ and T₄ become smaller than those in the rear part of the liquid film. This shows that the shape of the bubble approaches a quasi-steady one, and the flow condition around the bubble continues to develop until T₃. Therefore, it is concluded that the current method of numerical simulation using the OpenFOAM solver has sufficient capability and reproducibility to allow us to commence further studies on intermediate-sized bubbles.

5.2 Investigation of Secondary Flow of Intermediate-sized Bubbles

To investigate the behavior of the secondary flow, Fig. 14 presents the profiles and contours of the skin friction ratio at three locations. The profiles are extracted from the front side of the bubble to the calmed wake region where the skin friction ratio approached one, and the extracted length was 0.075 m. The skin friction of the secondary flow shows the fluctuations that increase more than that of the liquid film and forms a horseshoe shape. This is considered to be the reason for the greater increase in the average skin friction ratio in the present result compared to that in single-phase flow. Therefore, the vortical structure of the secondary flow is visualized to show its development after passing the bubble. Figs. 15 and 16 show the iso-surface of the streamwise vortices around the bubble in T₁–T₄. The vorticity values are ±330 s⁻¹, visualized from the side and bottom view. The counter-rotating vortices develop bilateral symmetry. Simultaneously, the counter-rotating vortices are also developed on the same side, but the size of the pair of vortices is unequal. Subsequently, the separated vortices move to the outer side of the bubble and fragment into small-scale vortices. When viewed from the side, these vortices can be observed to originate either from the upper wall or upper surfaces of the bubble. In addition, it is disclosed that the smaller vortex among the unequal pair of vortices on the same side originates from the upper surfaces of the bubble.

Fig. 17 shows the schematic views of the vortical structures around the bubble based on the behavior of the vortices acquired from Figs. 15 and 16. There are four vortical structures between the bubble and the upper wall, and each neighboring structure rotates in a counter direction (see Fig. 17a). The vortices that originate near the upper wall are wider and thinner than those originating from the upper surface of the bubble. The vortices that originate from the upper surface of the bubble approach the upper wall as it moves away from the bubble (see Fig. 17b). The location where these vortices are in contact with the upper wall changes depending on the bouncing period. These vortical structures are not generated continually but show three sequences. In the first sequence, the vortical structures are generated as crescent shapes and are still connected to the main structures (see Fig. 17c-1).
Fig. 15 Top view of Snapshots on the iso-surface of the streamwise vortices. ($\omega_x = \pm 330 \text{ s}^{-1}$)

Fig. 16 Side view of Snapshots of the iso-surface of the streamwise vortices. ($\omega_x = \pm 330 \text{ s}^{-1}$)

Fig. 17 Schematic view of the vortical structures (a) Rear view. (b) Side view on right side of the bubble. (c) Bottom view as the sequences of the vortices.
Moreover, this sequence corresponds to the region that does not have the horseshoe shape and the region behind the bubble in Fig. 14b. In the next sequence (see Fig. 17c-2), the counter-rotating pillar vortices are separated and accompanied by each other in the wake region. These pillar vortices are tilted on the outer side along the x-axis, similar to the horseshoe shapes, and also show a similar shape on the contours of the skin friction ratio. In the third sequence (see Fig. 17c-3), the pillar vortices became the source of disturbance for the main flow with the same mechanism as in the case of single-phase flow. Thus, the pillar vortices are divided into vortices of small scale. If this does not occur, the angle of the vortices gradually decreases. This causes a continuous high skin friction ratio on the downstream.

The sequences that generate streamwise vortices correspond with the trend of the skin friction ratio in the secondary flow, which is expected. Thus, it is considered that the understanding of each sequence to generate the vortices can elucidate the trend of the skin friction ratio in the secondary flow. To understand the behavior of the vortical structures, their instantaneous velocities originating from the vortical structure were reviewed based on the study by Leweke et al. (2016). The instantaneous velocities of the vortical structures can be evaluated based on the direction of rotation and the strength of the vortices. Fig. 18 shows the vorticity profiles in the vertical direction at the two locations (P1 and P2). Here, y/h = 2.0 and 0.0 represent the upper and bottom walls, respectively. As a result, the vorticity near the upper wall (y/h = 2.0) is twice as large as that originating from the upper surface of the bubble (y/h = 1.8). This difference in vorticity strength is responsible for the angular velocity of both vortical structures. In other words, the vorticity center of the pair is located on the upper wall, and the angular velocity of each pair of the vortical structures reaches to the outer side of the wake region. Based on this description, the behavior of the vortical structures during the first and second sequences can be explained schematically as shown in Fig. 18. First, the vortical structures from the upper surface of the bubble approached the upper wall because of the instantaneous velocity due to the pair of vortices (see Fig. 19a). This behavior results in no interaction between the vortices on the upper wall and those on the upper surface of the bubble because of the long distance between them. Thus, the vortical structures develop into a crescent shape without separation in the first sequence, as shown in Fig. 17c-1. Thereafter, the vortical structures on the upper surface of the bubble converge towards the vortical structures originating from the upper wall.

![Fig. 18 The normalized vorticity profile by the half height of the channel (a) P1 (b) P2. Both the locations are presented on left side of the picture (The bottom view of the bubble).](image1)

![Fig. 19 Schematic representation on the instantaneous velocity of the vortical structures from the rear viewpoint (a) The instantaneous velocity of the vortical structure that originated from the upper surface of the bubble (b) The instantaneous and the angular velocity of the vortical structure of the upper surface of the bubble when approaching to the upper wall.](image2)
The angular velocity induced by the difference in the strength of the vortices moves the vortical structures of the upper surface of the bubble to the outer side, separated from the main structures (see Fig. 19b). In addition, it is expected that this process may be accelerated due to the bouncing of the bubble, as observed in Fig. 9b. Finally, the angles of the vortices in the two sequences in Fig. 17c-2 were reviewed by Park et al. (2014) to understand the high skin friction ratio in this region. The angles of the streamwise vortices near the wall of the single-phase flow are mainly distributed near 0°, and the angles of these structures increase with increasing $y^+$. Based on this notation, the characteristics of the vortical structures originating from the upper surface of the bubble are consistent with those observed in the previous study of Park et al. (2014). However, the present data also show that the angles of the vortices near the wall, which are expected to be near 0° from the $x$-axis, are approximately ±25°, as shown in Fig. 15. This implies that the current size of the bubble turns the flow condition below the buffer layer into a turbulent flow with increasing skin friction ratio in this region.

6. Summary and Conclusions

In this study, we investigated the behavior and properties of an intermediate-sized bubble in a turbulent wall flow in a horizontal channel. The following conclusions were drawn:

(a) The current solver (interIsoFoam, which implements isoAdvect) performs a thorough numerical analysis of an intermediate-sized bubble. It minimizes the numerical error (smearing volume fraction) and displays proper volume conservation. In addition, a numerical procedure is established by dividing the numerical simulation into three stages. The numerical results show trends similar to those of the experiment.

(b) In the current bubble, high skin friction is generated at the secondary flow, which corresponds to the distribution of the streamwise vortices in the secondary flow. Thus, the secondary flow is divided into three sequences describing the shape of the vortices. Additionally, the behavior of the secondary flow can be described by the interaction between the pair of vortices that are generated on the upper wall and the upper surface of the bubble.

(c) The intermediate-sized bubble in this study has a large skin friction. This allows us to identify the reason for the increase in drag, but it is not yet possible to determine whether the phenomenon is consistent without changing the size of the bubbles. In the future, we will vary the size of the bubbles to investigate the drag reduction effect of the intermediate-sized bubbles and perform direct numerical simulations by refining the mesh conditions.

Acknowledgements

This Research and development work were supported by the MEXT Doctoral program for Data-Related InnoVation Expert Hokkaido University (D-DRIVE-HU) program and high-performance computing project (hp 190113) and Fundamental Research Developing Association for Shipbuilding and Offshore (REDAS). We would like to thank Editage (www.editage.com) for English language editing.

References

American Bureau of Shipping, Air Lubrication Technology, 2019, pp.1-15.
Fukuda, K., Tokunaga, J., Nobunaga, T., Nakatani. T., Iwasaki, T., Kunitake, Y., Frictional Drag Reduction with Air Lubricant over Super Water Repellent Surface (2nd Report) -Resistance Tests of Tanker and High Length to beam ratio Ship Models-, Journal of the Society of Naval Architects of Japan, Vol. 1999, Issue 186, 1999.
Fukuda, K., Tokunaga, J., Nobunaga, T., Nakatani, T., Iwasaki, T., Frictional drag reduction with air lubricant over a super-water-repellent surface, Journal of Marine Science and Technology, 5, 2000, pp.123-130.
Kawamura, T., Numerical and experimental investigation the mechanism of the microbubble drag reduction, Proc. 6th Symposium Smart Control of Turbulence, 2005.
Kitagawa, A., Hishida, K. & Kodama, Y. Flow structure of microbubble-laden turbulent channel flow measured by PIV combined with the shadow image technique, Experimental in Fluids, 38, 2005, pp.466–475.
Leweke, T., Le Dizes, S. and H.K. Williamson. C., Dynamics and Instabilities of Vortex Pairs, Annual Review of Fluid
Mechanics. Vol. 48, 2016, pp.507-541.
Lu, J., Fernandez, A., Tryggvason, G., the effect of bubbles on the wall drag in a turbulent channel flow, Physics of Fluids, Vol. 17, Issue 9, 2005.
M. McCormick and R. Bhattacharyya, Drag reduction of a submersible hull by electrolysis, Naval Engineers Journal, Vol. 85, Issue. 2, 1973, pp.11-16.
Murai, Y., Fujii, H., Tasaka, Y., Takeda, Y., Turbulent Bubbly Channel Flow Investigated by Ultrasound Velocity Profiler, Journal of Fluid Science and Technology, Vol. 1, Issue 1, 2006, pp.12-23.
Murai, Y., Fukuda, H., Oishi, Y., Kodama, Y., Yamamoto, F., Skin friction reduction by large air bubbles in a horizontal channel flow, International Journal of Multiphase Flow, Vol. 33, Issue 2, 2007, pp.147-163.
Moriguchi, Y., Kato, H., Influence of microbubble diameter and distribution on frictional resistance reduction, Journal of Marine Science and Technology, 7, 2002, pp.79–85.
Oishi, Y., Murai, Y., Horizontal turbulent channel flow interacted by a single large bubble, Experimental Thermal and Fluid Science, Vol. 55, 2014, pp.128-139.
Park, H.J., Tasaka, Y., Murai, Y., Oishi, Y., Vortical structures swept by a bubble swarm in turbulent boundary layers, Chemical Engineering Science, Vol. 116, 2014, pp.486-496.
Park, H.J., Tasaka, Y., Oishi, Y., Murai, Y., Drag reduction promoted by repetitive bubble injection in turbulent channel flows, International Journal of Multiphase Flow, Vol. 75, 2015, pp.12-25.
Roenby, J., Bredmose, H., Jasak, H., A computational method for sharp interface advection, Royal Society Opne Science, Vol. 3, Issue. 11, 2016
Rudman M, Volume-tracking method for incompressible multifluid flows with large density variations, Int. J. Numerical method in Fluids, Vol. 24, 671-691, 1997
Spandan, V., Verzicco, R., Lohse, D., Deformable ellipsoidal bubbles in Taylor-Couette flow with enhanced Euler-Lagrangian tracking, Phys. Rev. Fluids 2, 104304, Vol. 2, Issue 10, 2017
Sugiyama, K., Kawamura, T., Takagi, S., Matsumoto, Y., Recent Progress of Microbubble Flow Simulation for Elucidating Drag Reduction Mechanism, Proc. of the 6th Symp. on Smart Control of Turbulence, 191-202, 2005.