Applied Ocean Research 78 (2018) 254–266

Contents lists available at ScienceDirect

Applied Ocean Research

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

Numerical investigation on bubble size distribution around an underwater vehicle

Sara Vahaji[c], Li Chen[b], Sherman C.P. Cheung[a], Jiyuan Tu[a,*]

[a] School of Engineering, RMIT University, Plenty Rd., Bundoora, Victoria 3083, Australia
[b] Maritime Division, Defence Science and Technology Organisation, Victoria 3032, Australia
[c] School of Engineering, Deakin University, Geelong, Victoria 3217, Australia

ARTICLE INFO

Keywords:
Population balance
Gas-liquid flow
Coalescence
Breakage
Underwater vehicle
CFD

ABSTRACT

The interaction of bubbles with flow boundaries has been of high interest for marine engineering; for example, when a propeller is cavitating or air is entrained in the wake of a maneuvering ship, the strong interaction with the boundary layer will lead to the formation of a bubbly wake. To be able to develop the best mitigation strategy, a deep understanding of the associated physics is required. Only a few articles published in open literature address this two-phase fluid system. In the present study, a 3D numerical simulation has been performed to model a bubbly two-phase flow around the DARPA SUBOFF, in which exhaust gas is discharged into the flow around the object to provide a platform for investigating the distribution of the bubbles around a curved body. The two-phase flow is modelled using the Eulerian-Eulerian approach coupled with the MUSIG model. The bubble distribution is characterized based on different gas discharge configurations. It has been found that the boundary layer flow has a strong effect on bubble formation process, particularly encourages the bubble fragmentation. As a result, many small bubbles will be trapped at the aft of the vehicle.

1. Introduction

Two-phase gas–liquid flows are prevalent in many industrial applications such as chemical engineering, mineral, pharmaceutical, food processing, and metallurgy. For example, when a large quantity of exhaust gas is discharged into water, it would result in a highly turbulent bubbly plume. This plume will generate acoustic and visual signatures, and the strong interaction with the boundary layer flow will lead to the formation of a bubbly wake. In such flows the local hydrodynamic variables (e.g. bubble size distribution (BSD), void fraction, bubble coalescence and breakage rate and interfacial area concentration) can dynamically evolve and this can make the flow structure very complex. The influence of turbulent boundary layer flow on the bubble formation has an important impact on the outcome of the applications. To be able to develop the best mitigation strategy, a thorough understanding of the associated physics is necessary. Capturing bubble evolution may also be important for understanding other associated phenomena (e.g. flow noise, cavitation inception).

To describe the local structure of a two-phase flow, a number of numerical methods have been developed. The Eulerian-Eulerian approach – two-fluid model – is a promising tool to capture the local hydrodynamics. Nevertheless, it still needs a closure assumption on bubble size distribution (BSD) or the interfacial area concentration (IAC). Some studies introduce a simplified assumption of a single bubble (i.e., mono-size, static distribution). However, this assumption may lead to significant inaccuracy in the model predictions. To overcome this deficiency the dynamic population balance has been used to describe the processes of bubble coalescence and break-up. The dynamic population model is implemented with the Multiple Sized Group (MUSIG) model, in which bubbles are discretised into a series of bubble size classes. The bubble evolution due to coalescence and break-up is described by a scalar equation for each bubble size class.

Flows around underwater vehicles has been extensively investigated (both experimentally and numerically) and there is an extensive body of literature on this subject [see [1–3] and refs therein]. Most of these studies deal with a single-phase flow. For example, the David Taylor Research Center has performed a thorough experimental investigation of turbulent single-phase flows around different underwater vehicles, including DARPA SUBOFF [1–3], used in the present study. Ashok and Smits [4] investigated the turbulent wake of DARPA SUBOFF in pitch and yaw. Marshallsay and Eriksson [5] examined a potential use of Computational Fluid Dynamics simulations (CFD) as a tool for assessing the performance of the DARPA SUBOFF. However, to the best of authors' knowledge there is no numerical or experimental data in open

---

* Corresponding author.
E-mail address: Jiyuan.tu@rmit.edu.au (J. Tu).

https://doi.org/10.1016/j.apor.2018.06.013
Received 8 December 2017; Received in revised form 25 May 2018; Accepted 14 June 2018
0141-1187/

Nomenclature

a	Coalescence rate
$a(Mi,Mj)$	Coalescence rate of i and j bubble class in terms of mas
$A_{\alpha\beta}$	Interfacial area density
B_B, B_C	Mass birth rate due to break-up and coalescence
C_1,C_2, C_3, $C_{C\&T}$	Coalescence model constant
C_D	Drag coefficient
C_f	Coefficient of surface area
C_L	Lift coefficient
C_{w1}, C_{w2}	Wall lubrication coefficients
C_{TD}	Dispersion coefficient
D_H	Maximum bubble horizontal dimension
D_{ij}	Equivalent diameter
D_s	Sauter mean bubble diameter
D_B, D_C	Mass birth rate due to break-up and coalescence
d_β	Mean bubble diameter
f	Size fraction
f_{BV}	Break-up volume fraction, v i/ v j
F_{lg}	Total interfacial force
F_B	Break-up calibration factor
F_C	Coalescence calibration factor
F_{lg}^{drag}	drag force
F_{lg}^{lift}	Lift force
$F_{lg}^{wall\ \ lubrication}$	Wall lubrication force
$F_{lg}^{turbulent\ \ dispersion}$	Turbulent dispersion force
g	Gravitational constant
h_o	Initial film thickness
h_f	Critical film thickness
$h\ (Mi,\ Mj)$	Collision frequency in terms of mass
L	Overall length of the Suboff, 4.356 m
M	Mass scale of gas phase (bubble)
P	Pressure
P_b	Breakage probability
$P_e(e(\lambda))$	Energy distribution function
r	Breakage rate
$r\ (Mi,\ Mj)$	Partial breakage rate in terms of mass for i bubble class breaking into j and (i–j) bubble class
$r(Mi)$	Total breakage rate of i bubble class in terms of mass
$R_{\max}$	Maximum radius of the Suboff, 0.254 m
S_i	Mass transfer rate due to coalescence and break-up
t	Physical time
t_{ij}	Time for two bubbles to coalesce
$\mathbf{u}$	Velocity vector
u_t	Turbulent velocity
V	Volume of bubble

Greek letters

α	Void fraction
$\beta(f_{BV},1)$	Daughter bubble size distribution
ε	Dissipation of turbulent kinetic energy
η_{kli}	Coalescence mass matrix
λ	Size of eddy in inertial sub-range
$\lambda(Mi,Mj)$	Coalescence efficiency in terms of mass
$\lambda_{\min}$	Minimum size of eddy in inertia sub-range defined as 11.4 $(\upsilon^3/\varepsilon)^{1/4}$
μ	Viscosity
ρ	Density
σ	Surface tension
τ_{ij}	Contact time for two bubbles
ξ	Internal space vector of the PBE or size ratio between an eddy and a particle
Γ_{km}	Total mass transfer between gas and liquid phases

Super/Subscripts

e	Effective
i, j, k	Index of gas bubble class
t	Turbulent
g	Gas phase
l	Liquid phase
k,m	Liquid and gas phase

literatures for a two-phase fluid flow around a complex body.

The aim of this study is to investigate the interaction between the bubbles and turbulent boundary layer flow around an underwater vehicle (the DARPA SUBOFF) and to explore the effect of the boundary layer on the bubble size distribution. To ensure that our model on population balance modelling [6–13], capturing the coalescence and breakage processes [14,15], and the influence of the pressure on the bubble size distribution in bubbly flows [16–18] performs correctly, it has been validated against existing experimental results. This validation has been performed in two steps: (i) validation of the single-phase flow around the SUBOFF, (ii) validation of two-phase bubbly flow over flat plate. Then we predict and analyse the void fraction and bubble size distribution for different gas discharge configurations for flow over the SUBOFF model.

2. Mathematical modelling

2.1. Two-fluid model

The ensemble-averaged mass and momentum transport equations for continuous and dispersed phases are modelled using the Eulerian modelling framework. Considering the liquid (α^l) as a continuous phase and bubbles (α^g) as a disperse phase, the numerical simulations are presented based on the two-fluid model Eulerian-Eulerian approach as [6,7,9–11,13,14]

continuity equation,

$$\nabla.\,(\rho^k\,\alpha^k\mathbf{u}^k) = \Gamma_{km}(k,\ m = l,\ g) \tag{1}$$

Momentum equation,

$$\nabla.\,(\rho^k\,\alpha^k\mathbf{u}^k\mathbf{u}^k) = -\alpha^k\nabla P + \alpha^k\rho^k g + \nabla.\,[\alpha^k\mu_e^k(\nabla\mathbf{u}^k + (\nabla\mathbf{u}^k)^T)] + F_{km}(k,\ m = l,\ g) \tag{2}$$

2.1.1. Bubble interfacial forces

According to previous studies, the phase distribution is dominated by the interfacial momentum transfer between two phases. The total interfacial force (F_{km}), appearing in Eq. (2) is formulated based on the appropriate consideration of different interfacial sub-forces acting on

Table 1
Details of the mesh sizes studied.

Mesh #	Number of mesh nodes
Mesh1	132,458
Mesh2	249,438
Mesh3	383,708
Mesh4	483,971