Applied Ocean Research 60 (2016) 84–93

Contents lists available at ScienceDirect

Applied Ocean Research

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

Numerical investigation on the gas entrainment of ventilated partial cavity based on a multiscale modelling approach

M. Xiang[a,*], K. Li[b], J.Y. Tu[c,d], H.W. Zhang[a]

[a] College of Aerospace Science and Engineering, National University of Defense Technology, Changsha 410073, PR China
[b] School of Energy and Environment, Inner Mongolia University of Science and Technology, Baotou 014010, China
[c] Institute of Nuclear and New Energy Technology, Tsinghua University, Beijing 100084, PR China
[d] School of Aerospace, Mechanical and Manufacturing Engineering, RMIT University, Victoria 3083, Australia,

ARTICLE INFO

Article history:
Received 24 March 2016
Received in revised form 22 July 2016
Accepted 14 August 2016
Available online 21 September 2016

Keyword:
Ventilated cavitation
Drag reduction
Gas entrainment
Multiscale simulation

ABSTRACT

Ventilated cavitation which is acknowledged as an efficient drag reduction technology for underwater vehicle is characterised by the very disparate length and time scales, posing great difficulty in the application of this technology. A multiscale numerical approach which integrates a sub-grid air entrainment model into the two-fluid framework is proposed in this paper to resolve the complex flow field created by ventilated cavity. Simulations have been carried out for the partially ventilated cavity underneath flat plate, with special efforts putting on understanding the gas entrainment at the cavity tail and the bubble dispersion process downstream. The flow parameters including the void fraction, the bubble velocity and the bubble size distributions in and downstream of the ventilated cavity are fully investigated. Comparisons between the numerical results with the experimental data are in satisfactory agreement, demonstrating the potential of the proposed methodology. The ventilation rate effect on the cavity shape and bubbly flow parameters are further investigated, obtaining the law of bubble dispersion and the bubble size evolution. This research not only provide a useful method for the investigation on the multiscale multiphase flow, but also give insight on understanding the combined drag reduction mechanism resulted from large-scale cavity and microbubbles.

1. Introduction

Ventilated cavitation is commonly acknowledged as a highly effective drag reduction technique for underwater and floating vehicles such as torpedoes, missile and ships [1]. However, ventilated cavitation is characterised by spatial and temporal multiscale phenomena including (i) macroscale cavity formation; (ii) gas entrapment and breakup and (iii) dispersion of microbubbles as shown in Fig. 1, posing a tremendous challenge in understanding of this two-phase flow and application of this new technique. The drag reduction efficiency of ventilated cavity is not only determined by the macroscale cavity shape, but also affected by the microbubbles downstream through altering of the boundary layer structure. Furthermore, the generated bubbly flow could affect the noise characteristics and the propulsion system performance by changing the flow environment [2]. Therefore it is critical to explore the multiscale characteristics for ventilated cavity.

During the last decades, most of the studies focus on the macroscale cavity shape, revealing various cavity evolution process under different conditions [2]. Along with the development of multiphase experimental technique, several experimental works have been carried out to investigate the gas entrainment characteristics by ventilated cavity. Schauer [3] carried out cavitation experiments in water tunnel, using PIV to measure the bubbly flow parameters downstream of ventilated partial cavity, indicating that the maximum void fraction in the wake region approach 10%. Mäkiharju [4] adopted X-ray system to measure the bubble parameters downstream two-dimensional ventilated cavity, obtaining the gas void fraction and bubble size distributions both inside and downstream cavity. It was proved that 20% energy could be saved by using the drag reduction technology of partial ventilated cavity. However, due to the limitation of measure equipment for multiphase flow, the experimental data is insufficient to reveal the whole cavity flow characteristics.

On the other hand, numerical modelling has fast become an effective tool to complement the deficiencies of existing exper-

---

* Corresponding author.
E-mail addresses: xiangmin333@hotmail.com (M. Xiang), kelitsinghua@hotmail.com (K. Li), jiyuan.tu@rmit.edu.au (J.Y. Tu).

http://dx.doi.org/10.1016/j.apor.2016.08.003
0141-1187/

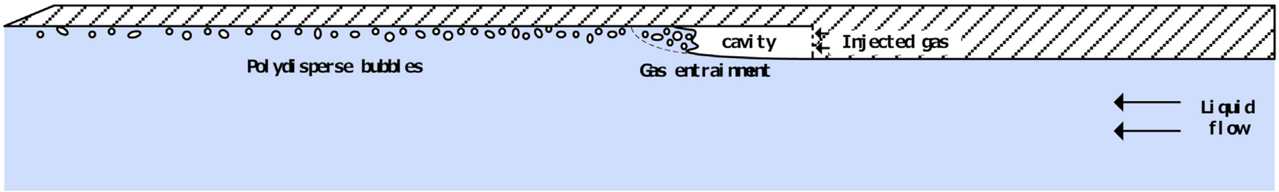

Fig. 1. Schematic diagram of flow structure in typical partially ventilated cavity.

imental techniques. Most numerical works focus on capturing free-surface interface of the ventilated cavity based on homogeneous models, among which the most remarkable work belong to Kunz group [5,6]. However, obvious deficiency is observed at cavity tail region due to the neglect of the gas entrainment process. The complexity of gas entrainment and bubbly flow region in ventilated cavity pose a great challenge for the model development. A few researchers adopted simplified models to calculate the bubbly flow parameters after gas entrainment. Rigby et al. [7] and Thorpe et al. [8] assumed the liquid velocity dowstream cavity was the same as that downstream cylinder. Evans et al. [9] divided the region downstream of cavity into wall jet region and vortex region to calculate the bubble size evolution process. Few researchers have tried to carry out multiscale simulation on gas entrainment process for the other free-surface flow. Laux and Johansen [10] adopted VOF and two-fluid model to predict the entrained bubble parameters generated by plunging jet. Ma et al. [11] established a multiscale model which couples the two-fluid model with the Level Set approach for free-surface bubbly flows. Susann Hansch et al. [12] incorporated the inhomogeneous MUSIG model into two-fluid framework to capture different scales of interfacial structures. In order to stabilize large-scale interface structure, a specific interfacial force is added between the liquid and the continuous gas phase. Kai Yan and Defu Che [13] established a unified framework to resolve multiscale interface structure between gas and liquid. A “volume fraction redistribution” method is adopted for the region containing all three phases. Comparing with the hydraulic jump flows and plunging liquid jet, similar gas entrainment mechanism is observed for ventilated cavitating flow. However, more complex mass and momentum transfer process are involved and little literature has been found on the multiscale investigation of ventilated cavity.

In this paper, a multiscale numerical approach will be proposed to investigate on the multiphase flow generated by ventilated partial cavity. The numerical framework will be developed based on the Eulerian–Eulerian multi-fluid model, coupled with a subgrid gas entrainment model and the compressive VOF model. The population balance approach will also be adopted to capture the bubble size evolution process. Special effort will be put on better understanding the gas entrainment and the bubble dispersion process. The main factors dominating the cavity and bubble evolution dynamics will be analysed to give insight on the drag reduction technology of ventilated cavitation.

2. Mathematical models

2.1. Multi-fluid model framework

The formation of ventilated cavity can be considered as the interaction of three different phases: continuous liquid, continuous gas above the free-surface and disperse bubbles. In this paper, numerical model is established based on the Eulerian–Eulerian multi-fluid framework which solves the ensemble-averaged of mass, momentum and energy conservative equations for every single phase. Interactions between phases are effected via interfacial transfer terms for mass and momentum exchange. Since there is no interfacial heat transfer between the phases in the present study, the energy equation is not needed to be solved.

The continuity equation of all the phases can be written in a generic form as follows:

$$\frac{\partial(\rho_i\alpha_i)}{\partial t} + \nabla\cdot(\rho_i\alpha_i\boldsymbol{u}_i) = m_i \tag{1}$$

where α, ρ and $\boldsymbol{u}$ are the void fraction, density and velocity vector of each phase. The source term m_i refers to the interfacial mass source or sink for the phase i. The Subscripts $i = l$ or g or b denote the liquid, gas and bubble phase respectively. Since the flow is assumed as isothermal, for the liquid phase, there is no interfacial mass transfer due to condensation or evaporation with other phases. For the gas and bubble phase, the mass transfer is induced by gas entrainment which generates bubbles from continuous gas and merging process which combines dispersed bubbles into continuous gas. The merging rate downstream ventilated cavity is neglectable in comparison with the entrainment rate. Therefore the source terms are given as:

$$\mathrm{m}_l = 0,\ \mathrm{m}_b = -\mathrm{m}_g = \mathrm{m}_{ent} \tag{2}$$

Similarly, the momentum equation for all phases can be expressed as:

$$\frac{\partial(\rho_i\alpha_i\boldsymbol{u}_i)}{\partial t} + \nabla\cdot\left(\rho_i\alpha_i\boldsymbol{u}_i\otimes\boldsymbol{u}_i^{\mathrm{T}}\right) = -\alpha_i\nabla P + \rho_i\alpha_i\boldsymbol{g}_i + \nabla\cdot \left[\alpha_i\mu_i{}^{e}\left(\nabla\boldsymbol{u}_i + (\nabla\boldsymbol{u}_i)^{\top}\right)\right] + (\rho_i - \rho_{ref})\boldsymbol{g} + \boldsymbol{F}_i \tag{3}$$

where ρ_l is adopted as the reference density ρ_{ref} to calculate the buoyancy force. $\boldsymbol{F}_i$ represents the interfacial forces contribute to the momentum transfer among phases and $\boldsymbol{g}$ is the gravity acceleration vector. One should be noted that the aforementioned interfacial forces are also strongly governed by the phase distribution as well as the local interfacial area. Details of these interfacial forces are discussed in the following sections.

For the phase distribution all phases, as all three phases are considered as incompressible and steady, the volume conservation equation can be simplified as:

$$\sum\nabla\cdot(\alpha_i\boldsymbol{u}_i) = 0 \tag{4}$$

2.2. Gas entrainment model

Gas entrainment at the cavity tail is due to the complex interaction between surface tension and turbulence. Resulted from different entrainment mechanism, the ventilated cavity presents various shedding styles including re-entrained jet shedding, twin-vortex shedding and unstable vortex shedding. Gas entrainment for partially ventilated cavity shares similar physical characteristics with plunging jet and hydraulic jump [14]. Gas was lost mainly from the cavity closure in the form of ventilated gas sheet or induction trumpet [15,16]. Subsequently, the entrained gas pockets are broken up into small bubbles which are then driven into the fluid. In order to establish a gas entrainment model, a uniform gas entrainment process for partially ventilated cavity can be described in Fig. 2