Ocean Engineering 118 (2016) 58–67

Contents lists available at ScienceDirect

Ocean Engineering

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

Numerical study on the vertical motion of underwater vehicle with air bubbles attached in a gravity field

Y.Z. Xue, B. Cui, B.Y. Ni *

College of Shipbuilding Engineering, Harbin Engineering University, Harbin 150001, China

ARTICLE INFO

Article history:
Received 3 December 2015
Received in revised form
24 February 2016
Accepted 27 March 2016
Available online 7 April 2016

Keywords:
Air bubbles
Underwater vehicle
Drag reduction
Boundary-element method
Froude number

ABSTRACT

Air bubbles may be injected and attached on the surface of a moving underwater vehicle to enhance its speed. Thus the nonlinear interactions between the moving body at a variable speed and the deformable bubble attached are of importance. In this paper, a body with an air bubble attached at a variable speed is simulated using the boundary-element method together with the potential theory. The motion of the body and deformation of the bubble under the action of gravity are considered in a moving coordinate system fixed on the body. Bernoulli equation and auxiliary function method in this moving coordinate system are adopted to calculate the pressure and fluid force on the body. Convergence study with mesh and time step has been undertaken with good convergence obtained. Simulated results have also been compared with other published numerical results and good agreement achieves. On this basis, the influences of different parameters have been further studied, such as Froude number, cavitation number and body acceleration. It is found that bubble shape and jet direction are influenced by the gravity a lot in the range of parameters considered and various jet patterns may generate such as inward jet, oblique jet or upward jet.

1. Introduction

When a vehicle moves forwards in a liquid, such as water, it will encounter much larger drag than it does in the air, because the former has much larger density. Therefore there is always a strong desire to reduce the drag through introducing bubbles or air cavity over the body surface. Especially for the underwater high-speed vehicle, air bubbles may be injected at its forepart to enhance its speed and also reduce the influence of cavitation formed on its motion (Cao et al., 2012).

In terms of the mechanisms, drag reduction through bubbles could be broadly classified as Micro-Bubble Drag Reduction (MBDR or BDR) and Air Layer Drag Reduction (ALDR), which were reviewed by Ceccio (2010). MBDR reduces the frictional resistance by forming the micro bubble-water mixed layer on the body surface. This concept was first demonstrated by McCormick and Bhattacharyya (1973), who studied the drag reduction of an axisymmetric body through hydrogen bubbles generated by electrolyzing water. ALDR reduces the frictional drag by covering parts of the hull surface of the vehicle with an air layer (also known as air cavity, ventilated cavity, or air plenum), which effectively reduces the wetted area of the body surface (Latorre, 1997). Subsequently, a series of experiments, mathematical analyses and numerical simulations have been undertaken to assess the feasibility of these two concepts, and to refine and improve both technologies.

In experiments, numerous tests on a variety of models have been undertaken for drag reduction through bubbles, such as flat plates, axisymmetric bodies and ship hull forms. Some early work includes that by Bogdevich et al. (1977) who employed various methods, including gas injection, air layers and partial cavities, and reported achieving a maximum drag reduction by around 80%. Madavan et al. (1985) measured the local skin friction reduction in a microbubble-modified, zero-pressure-gradient, turbulent boundary layer and the results showed that the integrated skin friction was reduced for all airflow rates and tunnel velocities. Maximum reductions of more than 80% were also observed. Sanders et al. (2006) and Elbing et al. (2008) undertook high Reynolds number experiment to investigate drag reduction for a near-zero streamwise pressure gradient turbulent boundary layer. Murai et al. (2007) investigated the drag reduction using the relatively large bubbles, whose sizes were between those of micro-bubble and air layer methods. Matveev et al. (2009) carried out an air-cavity ship model test with a 56-cm-long stepped-hull in an open surface water channel and identified an optimal trim angle for the largest air-cavity area under different cavity forms.

In mathematical modelling and numerical simulations, the underlying mechanism of bubble drag reduction has always been an active topic of research. For MBDR, research is based mainly on

* Corresponding author.
E-mail address: nibaoyu@hrbeu.edu.cn (B.Y. Ni).

http://dx.doi.org/10.1016/j.oceaneng.2016.03.065
0029-8018/

two approaches: one is the boundary layer theory, which considers the fluid as a homogeneous mixture with reduction in density and modification of the effective viscosity inside the boundary layer (Madavan et al., 1985) and the other is two-phase flow theory (Chu et al., 2010). Many numerical simulations have been based on these two approaches over the past decades and a review can be found in Ceccio (2010). By contrast, fewer investigations have been conducted for the ALDR phenomenon. Matveev (1999) developed a simplified model of a two-dimensional cavity based on the potential flow theory, which was then extended to three-dimensional problems (Matveev, 2007; Matveev et al., 2009). Li et al. (2008) used a two-dimensional air cavity formation for a slender ship and then examined the parameters which directly affected the formation of air cavities, such as cavitation number and ship speed. Choi et al. (2005) and Choi and Chahine (2010) studied the deformation of the air layer beneath a ship hull and considered its effects on wave making resistance with the three-dimensional boundary-element method (BEM). Ni et al. (2013) studied the motion and deformation of a ring bubble along a spheroid surface under constant incoming flow, by using the axisymmetric BEM. Different initial bubble pressure relative to ambient pressure was considered to search for the stable shape of the sliding bubble. However, the gravity was ignored in their model for the simplicity.

In the study of bubble drag reduction on a moving body, it seems that there has been little work which takes full account of the nonlinear interactions between the moving body at a variable speed and the deformable bubble, involving the gravity effect which may play an important role in the bubble evolution (Zhang et al., 2015b). Especially for the high-speed vehicle, most previous work within the framework of BEM is for the steady motion of a body with steady cavity attached (Leng and Lu, 2002; Fu and Li, 2002; Liu et al., 2004). Seldom work has considered the unsteady motion of a moving body with air bubbles attached, with fully nonlinear kinematic and dynamic boundary conditions satisfied on the interface. On the other hand, these are the key issues for this kind of technology to be used successfully in drag reduction. Therefore, it is highly relevant to undertake the research in this area. This will ultimately lead to a better way to generate and control the body with air bubbles injected, and improve the effectiveness of the drag reduction as well as the loads of underwater vehicle during its water exit (Ni et al., 2015b).

2. Mathematical model and numerical method

2.1. Velocity potential flow and Green's function

We consider the vertical motion of a body with an air bubble attached. The body shape here is taken as a cylinder with a spherical forepart, whose diameter is D and total length is L, as shown in Fig. 1. The moving velocity of the body is a prescribed $W(t)$ and the initial submergence of the body is large enough to ignore the influence of free surface. A moving Cartesian coordinate system $O-xyz$ as well as a cylindrical coordinate system $O-r\theta z$ is defined, in which the origin O is located at the centre of body bottom and z axis points vertically upwards and passes through the symmetry axis of the body. When the body moves along z axis, the flow is then axisymmetric. The fluid is assumed to be incompressible and inviscid and the flow to be irrotational. Thus the fluid flow can be described in terms of velocity potential Φ, which satisfies the Laplace's equation:

$$\nabla^2\Phi = 0 \tag{1}$$

in the fluid domain. The impermeable boundary condition on the

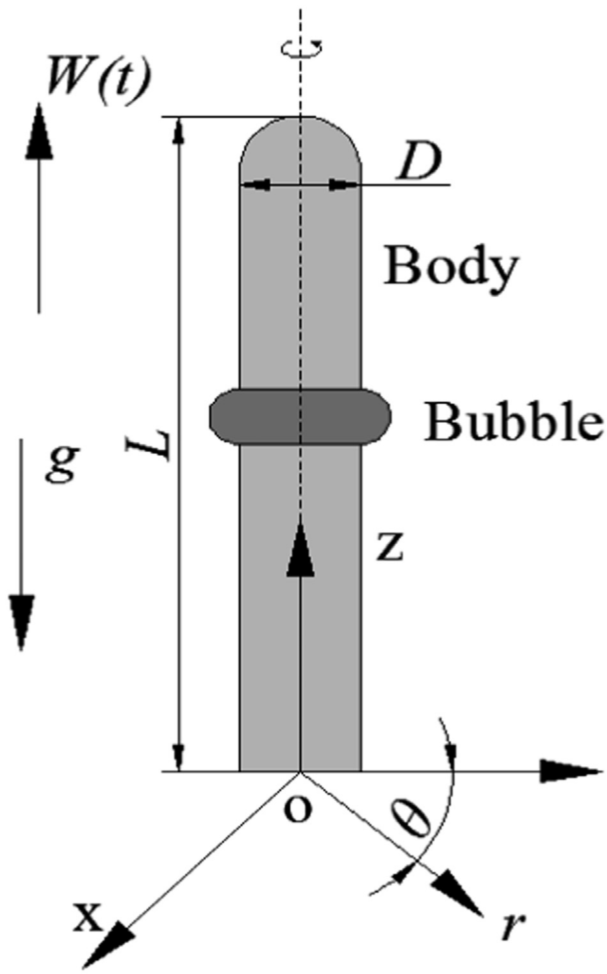

Fig. 1. Sketch of the problem with moving Cartesian and polar coordinate systems.

wetted part of the rigid body surface s_w is given by

$$\frac{\partial\Phi}{\partial n} = W(t)\cdot n_z \tag{2}$$

where n_z is the z component of normal vector $\boldsymbol{n} = (n_r, n_z)$ on the body surface pointing out of the fluid domain.

Furthermore, it is assumed that z_0, r_0, θ_0 is the polar system fixed in the space and it coincides with z, r, θ when the body starts to move. This means $z_0 = z + \int_0^t W(t)dt$, $r_0 = r$, $\theta_0 = \theta$. As a result, in the Lagrangian framework, the fully nonlinear kinematic and dynamic boundary conditions on the bubble surface can be written as

$$\frac{Dr}{Dt} = \frac{\partial\Phi}{\partial r}, \quad \frac{Dz}{Dt} + W(t) = \frac{\partial\Phi}{\partial z} \tag{3}$$

$$\frac{D\Phi}{Dt} = \frac{P_\infty - P_l}{\rho} + \frac{1}{2}|\nabla\Phi|^2 - g\left[z + \int_0^t W(t)dt\right] \tag{4}$$

where D/Dt is the substantial derivative following a fluid particle, ρ is the density of the liquid, g is the acceleration due to gravity, P_∞ is the ambient pressure at infinity in the plane of $z_0 = 0$, and P_l is the fluid pressure on the bubble surface. Taking surface tension into account, one can link P_l with the pressure P_b inside the bubble by the Young–Laplace equation:

$$P_l = P_b - \tilde{\sigma}\kappa \tag{5}$$