ARTICLE IN PRESS

International Journal of Naval Architecture and Ocean Engineering xxx (2017) 1–15

Contents lists available at ScienceDirect

International Journal of Naval Architecture and Ocean Engineering

journal homepage: http://www.journals.elsevier.com/international-journal-of-naval-architecture-and-ocean-engineering/

Numerical investigation of a novel device for bubble generation to reduce ship drag

Jun Zhang [a], Shuo Yang [a], Jing Liu [b, *]

[a] College of Aerospace Engineering, Nanjing University of Aeronautics and Astronautics, Nanjing 20016, China
[b] Energy Research Institute @NTU, Nanyang Technological University, Singapore

ARTICLE INFO

Article history:
Received 10 April 2017
Received in revised form
5 September 2017
Accepted 24 October 2017
Available online xxx

Keywords:
WAIP
Resistance reduction
Hull of ship
OpenFOAM
VOF

ABSTRACT

For a sailing ship, the frictional resistance exerted on the hull of ship is due to viscous effect of the fluid flow, which is proportional to the wetted area of the hull and moving speed of ship. This resistance can be reduced through air bubble lubrication to the hull. The traditional way of introducing air to the wetted hull consumes extra energy to retain stability of air layer or bubbles. It leads to lower reduction rate of the net frictional resistance. In the present paper, a novel air bubble lubrication technique proposed by Kumagai et al. (2014), the Winged Air Induction Pipe (WAIP) device with opening hole on the upper surface of the hydrofoil is numerically investigated. This device is able to naturally introduce air to be sandwiched between the wetted hull and water. Propulsion system efficiency can be therefore increased by employing the WAIP device to reduce frictional drag. In order to maximize the device performance and explore the underlying physics, parametric study is carried out numerically. Effects of submerged depth of the hydrofoil and properties of the opening holes on the upper surface of the hydrofoil are investigated. The results show that more holes are favourable to reduce frictional drag. 62.85% can be achieved by applying 4 number of holes.

© 2017 Society of Naval Architects of Korea. Production and hosting by Elsevier B.V. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

1. Introduction

For ship travelling in the water, total resistance on the submerged hull includes wave making resistance and viscous resistance. Wave making resistance is caused by consumed energy used to generate wave because of the motion of ship. The frictional resistance is one part of viscous resistance besides pressure difference and resistance from eddies formation and separation. It is caused from viscosity of water and the motion of ship.

As a ship speeds up or at high Reynolds number (*Re*), the flow becomes more turbulent and the thickness of the boundary layer increases. Thus the shear stress acting on the wetted surface of the hull becomes higher which causes an increase in frictional resistance (Molland et al., 2011). Magnitude of frictional resistance is determined by viscosity of water, wet area of the hull and speed of ship. About 60% propulsion power provided by burning quantities of fuel is possible to be used to overcome the frictional resistance (Butterworth et al., 2015). Gas emissions such as SO_x and NO_x from burning fossil fuels causes air pollution and greenhouse effect. Scientists and naval architects have explored numerous practical techniques in order to reduce frictional resistance and thereby improve burned fuel efficiency. Air lubrication method has been proved to be much effective through model test (Sinha and Architechture, 2016). Bubble-mixed flow is formed in the near-wall region of a turbulent boundary layer by injecting air to the wetted surface of the hull (Elbing et al., 2008; Mäkiharju et al., 2012; Murai et al., 2007). Theoretically, reduction rate of frictional resistance should be proportional to the distribution area of air bubbles in the turbulent boundary layer. Therefore, it is vital important to retain the continuity of bubbles in a long distance of the near-wall region in the downstream after the injecting point (Elbing et al., 2008). Formation of bubbles can be achieved through electrolysis of water (McCORMICK and Bhattacharyya, 1973) or using compressor and porous media (Bogdevich et al., 1977). However, these methods consume extra energy, thus their advantages are lost in term of net reduction rate of drag.

A novel device Winged Air Induction Pipe (WAIP) was invented by Kumagai et al. (2010, 2015). In this technique, a device is installed on the surface of the wetted hull as shown in Fig. 1(a). It includes hydrofoil, air chamber, and air induction pipe (Fig. 1 (b)).

* Corresponding author.
E-mail address: ljing2005@gmail.com (J. Liu).
Peer review under responsibility of Society of Naval Architects of Korea.

https://doi.org/10.1016/j.ijnaoe.2017.10.009
2092-6782/© 2017 Society of Naval Architects of Korea. Production and hosting by Elsevier B.V. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).

Please cite this article in press as: Zhang, J., et al., Numerical investigation of a novel device for bubble generation to reduce ship drag, International Journal of Naval Architecture and Ocean Engineering (2017), https://doi.org/10.1016/j.ijnaoe.2017.10.009

ARTICLE IN PRESS

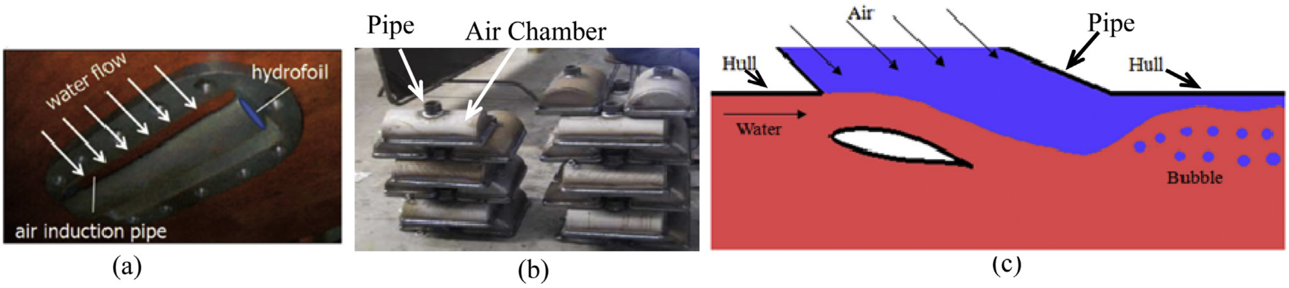

Fig. 1. (a) The WAIP device installed on the surface of the hull, (b) side view of the device air chamber and pipe, and (c) cross-sectional view of the WAIP device and its working principle (Kumagai et al., 2015; Takahashi, 2012).

As ship is moving, the negative pressure over the upper surface of the hydrofoil is formed, and the pressure is lower than the hydrostatic pressure in the depth of WAIP device (Kumagai et al., 2015). So the air is naturally sucked and flowing through the pipe to the wetted hull surface. As presented in Fig. 1(c), plenty of bubbles are formed in the downstream of the hydrofoil owning to Kelvin Helmholtz instability which happens at the interface of multiphase flows with different flow speeds in the turbulence condition. For the lower draft ship, the device needn't consume extra energy and air compressor is not necessary because the WAIP device can utilize the kinetic energy of the moving ship to capture air. In the sea trial conducted by Murai, et al., 10% fuel was saved by installing this kind of device (Murai et al., 2010). Recently, experimental investigation has been done on the WAIP device applied to the actual ships (Kumagai et al., 2010). Drag reduction rate is about 10–15%, which also depends on the ship size, number of WAIP devices, and involving compressor or not.

However, the WAIP device as shown in Fig. 1 requires opening holes on the surface of the hull for installation of air chamber. Air compressor has to be applied in the condition of deeper draft to pump air to the bottom of the hull. In case of too high pressure caused by compressor, air can wrap the hydrofoil, and leads to even high drag on ship. A novel air lubrication technique is proposed by Kumagai et al. (2014) to overcome the drawbacks of the traditional WAIP device. The opening holes are on the upper surface of the hydrofoil instead of the hull surface as shown in Fig. 2. The air can be introduced to the upper surface of the hydrofoil through an air induction pipe as arranged in the experiment by Kumagai et al. (2014). Another option could be that the hydrofoil acts as air chamber with both ends in spanwise exposed to the air. The detailed techniques won't be addressed here since they are not a main concern of this study. This device is applicable to the available ship without modification to the hull. To understand drag reduction mechanism and maximize its performance, parametric study is performed with validated numerical method in this study. This preliminary investigation could provide a clue to better understand underling mechanism of drag reduction and the feasibility of the present WAIP device.

The paper is organized as follows: Section 2 briefly introduces the numerical method used to predict multiphase flow and capture evolution of the free surface. The Reynolds Averaged Navier-Stokes equations (RANS) are solved on the unstructured grid to obtain approximation of the multiphase flow. The Volume of Fluid (VOF) method is used to capture evolution of the free surface. The open-source CFD toolbox OpenFOAM is used to discretize derivatives in the governing equations and to obtain the numerical solution of the multiphase flow and free surface. Section 3 describes validation of the present numerical method and predicts the performance of the WAIP device on its reduction rate of the frictional resistance. Effects of submerged depth of the hydrofoil, diameter of the hole, position of the hole, and number of holes on the formed air layer and drag reduction are investigated numerically. Section 4 summarizes the conclusion of this study.

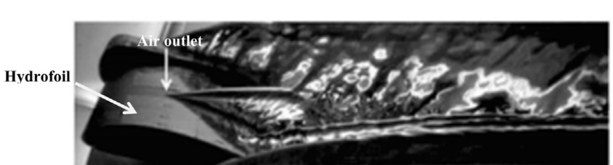

Fig. 2. The hydrofoil with opening hole on the upper surface (Kumagai et al., 2014).

2. Numerical method

A number of solvers are available in the fully developed open-source CFD toolbox OpenFOAM for some specific problems of fluid mechanics. The package interFoam is used in the present study to solve the multiphase flow and evaluation of the free surface with VOF method.

2.1. Velocity field and tubulence model

Three-dimensional (3D), transient, viscous, incompressible, and two-phase immiscible fluid flow is numerically solved by discretizing RANS equations,

$$\nabla \cdot \boldsymbol{U} = 0 \tag{1}$$

$$\frac{\partial \rho \boldsymbol{U}}{\partial t} + \nabla \cdot \left(\rho \boldsymbol{U}\boldsymbol{U}^{\boldsymbol{T}}\right) = -\nabla p^{*} + \nabla \cdot (\mu \nabla \boldsymbol{U}) + \nabla \cdot (\rho \tau) + \boldsymbol{S} \tag{2}$$

where $\boldsymbol{U} = (u_x + u_y + u_z)$ is the velocity vector. t is time. ∇ is vector differential operator. p^* is relative pressure. ρ and μ are fluid properties the density and the dynamic viscosity, respectively. τ is Reynolds stress tensor for turbulence flow. Closure of the turbulence model for τ is k-ω Shear Stress Transport (SST) (Menter, 1993; Menter et al., 2003). The turbulence kinetic energy k and specific dissipation rate ω are estimated from the boundary condition of turbulence quantities turbulence intensity I and length scale l (ANSYS Fluent User guide),

$$k = \frac{3}{2}(U_{avg} I)^2 \tag{3}$$

$$\omega = \frac{k^{1/2}}{C_\mu^{1/4} l} \tag{4}$$

where C_μ is constant and is equal to 0.09, U_{avg} is the averaged velocity. $\boldsymbol{S}$ in Eq. (2) is the source term which includes body force and

Download English Version:

https://daneshyari.com/en/article/8959349

Download Persian Version:

https://daneshyari.com/article/8959349

Daneshyari.com