Available online at www.sciencedirect.com
ScienceDirect
Publishing Services by Elsevier
International Journal of Naval Architecture and Ocean Engineering xx (2017) 1—13
http://www.journals.elsevier.com/international-journal-of-naval-architecture-and-ocean-engineering/

Numerical study on the hydrodynamic characteristics of a propeller operating beneath a free surface

Kwang-Jun Paik

Department of Naval Architecture and Ocean Engineering, Inha University, Incheon, South Korea

Received 2 November 2016; revised 16 February 2017; accepted 20 February 2017
Available online ■ ■ ■

Abstract

The results of a numerical study on the performance of a propeller operating near a free surface are presented. The numerical simulations were performed for the various advance coefficients and the submergence depths of the model propeller. The effects of the model propeller size were investigated using two different model propeller sizes for all cases. The wave pattern of the free surface and the flow structure around the propeller as well as the hydrodynamic characteristics of the propeller were investigated through simulation results. The thrust and torque fluctuated and the trajectory of the tip vortex was distorted due to the interaction with the free surface. The wave pattern of the free surface was related to the tip vortex of the propeller. The decreases in thrust and torque at the small model propeller were greater than those of the large model propeller. The reduction rate of the thrust and torque increased with the advance coefficient.

Keywords: Propeller; Free surface; Tip vortex; Wave pattern; Computational fluid dynamics

1. Introduction

Recently, many researchers have been interested in a ship operation in heavy seas from the point of view of ship motion and ship resistance; thus, many studies have been performed using Computational Fluid Dynamics (CFD) (Orihara and Miyata, 2003; Arribas, 2007; Liu et al., 2011; Sadat-Hosseini et al., 2013). As a ship operating in waves experiences very complicated motions and wave conditions, the propeller attached at the stern of the ship can suffer from various inflow conditions and a change of submergence depth. According to the change of the propeller submergence depth, the interaction between the propeller and free surface can introduce some risks, such as air ventilation and surface piercing running. Furthermore, the resistance of a ship operating in waves increases due to ship motion, which is an added resistance, resulting in a decrease in ship speed. When the ship speed decreases at a constant power, the propeller will be operated at a higher load in a lower advance coefficient region (Nakamura and Naito, 1977; Chuang and Steen, 2011; Ueno et al., 2013). When the distance between the propeller and the free surface becomes closer, the thrust and torque of the propeller can be decreased due to the interaction with the free surface resulting in a decrease in propeller efficiency. Therefore, it is important to study the hydrodynamic characteristics of a propeller interacting with a free surface in detail under a range of operating conditions.

The hydrodynamic characteristics of the propeller in deep water without an interaction with the free surface were studied to examine the hydrodynamic performance of the propeller and the flow structure around a propeller (Paik et al., 2007; Felli et al., 2008, 2011; Carrica et al., 2010; Castro et al., 2011; Baek et al., 2015; Paik et al., 2015). On the other hand, the interactions between the propeller and free surface should be investigated deeply to better understand the variations of thrust and torque of a propeller operating beneath a free surface. Because air ventilation can occur when the propeller operates under the water with a high loading, some

---

E-mail address: kwangjun.paik@inha.ac.kr.

Peer review under responsibility of Society of Naval Architects of Korea.

http://dx.doi.org/10.1016/j.ijnaoe.2017.02.006

Please cite this article in press as: Paik, K.-J., Numerical study on the hydrodynamic characteristics of a propeller operating beneath a free surface, International Journal of Naval Architecture and Ocean Engineering (2017), http://dx.doi.org/10.1016/j.ijnaoe.2017.02.006

studies have investigated the mechanism of air ventilation with thrusters and propellers (Kozlowska et al., 2009, 2011). They found that the vortex forming in air acts on the suction side of the propeller making a vortex funnel and it depends on the forward speed of the propeller. Therefore, the air ventilation event cannot be observed according to the advance coefficient, even in small submerged depth. Califano and Steen (2011) simulated the phenomenon of air ventilation for a fully submerged and highly loaded propeller using commercial CFD code. They concluded that the tip vortex plays an important role for the air ventilation of a conventional propeller. Park et al. (2011) proposed an empirical formulation regarding the variations of thrust and torque for partially submerged propellers through model tests and simulated the loss of thrust during one revolution with a half-submerged propeller using CFD.

On the other hand, there are no many reports on the interaction between the propeller and free surface in various advance coefficients and submergence depths. The effects of the free surface around a propeller were investigated using Particle Image Velocimetry (PIV) for various propeller immersion depths by Paik et al. (2008). This paper shows the axial velocity profiles at the upstream and downstream of the propeller as well as the wave profiles of the free surface. Li et al. (2015) studied the effects of the presence of a free surface on an inclined propeller using CFD.

In this study, the hydrodynamic characteristics, such as the thrust and torque of propeller, the wave pattern of free surface, and the evolution of propeller wake were investigated in five advance coefficients, three submergence depths of the propeller, and two model propeller sizes. The study was performed using CFD based on the incompressible Reynolds averaged Navier–Stokes equations (RANS).

2. Mathematical and numerical model

The governing equations for the numerical simulation are the continuity equation and the incompressible RANS. The integral forms of the equations are expressed as follows:

$$\frac{\mathrm{d}}{\mathrm{d}t}\int_{\Omega}\rho \mathrm{d}\Omega+\int_{S}\rho u_i n_i \mathrm{d}S=0$$

$$\frac{\mathrm{d}}{\mathrm{d}t}\int_{\Omega}\rho u_i \mathrm{d}\Omega+\int_{S}\rho u_i u_j n_j \mathrm{d}S=\int_{S}\left(\tau_{ij}n_j-pn_i\right)\mathrm{d}S+\int_{\Omega}\rho b_i \mathrm{d}\Omega$$

where, ρ and p are density and pressure, respectively. u_i is the velocity tensor and b_i is the tensor of body forces. τ_{ij} is the effective stress of the viscosity and turbulence, defined as

$$\tau_{ij}=\mu_e\left[\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right]$$

The solution of the governing equations was obtained using the second order discretization for time and space. The pressure–velocity coupling was implemented using the SIMPLE (Semi-Implicit Method for Pressure-Lined Equation) algorithm. The SST (Shear Stress Transport) $k-\omega$ model proposed by Menter (1994) was applied to a turbulence model. The free surface was simulated using the VOF (Volume Of Fluid) multiphase model based on HRIC (High Resolution Interface Capturing) scheme, which was proposed by Muzaferija et al. (1998), to capture the wave pattern generated by the propeller operation. These numerical models applied in this study were implemented using STAR-CCM+ ver. 10.04.

3. Setup of numerical simulation

3.1. Propeller geometry

A model propeller used in this study was KP505, which was designed by the Korea Research Institute of Ships and Ocean Engineering (KRISO) for the KRISO container ship called KCS. The diameter of the full-scale propeller is 7.9 m and the number of blades is 5. Table 1 and Fig. 1 show the principal particulars and drawing of KP505. Paik et al. (2008) used KP505 to examine the interaction between the propeller and free surface when a propeller operates beneath the free surface.

3.3. Grid system

In this study, the numerical simulation method was verified through a comparison with the experimental results of the model propellers of 5.4 cm and 25.0 cm. The small model propeller of 5.4 cm was tested in a circulating water channel to examine the interaction with the free surface. The large model propeller of 25.0 cm was tested in the towing tank. Therefore, the numerical simulations were performed for both model propellers to determine the effects of the model scale.

The computational domain for the numerical simulations is shown in Fig. 2, which mimics the circulating water channel used in the model test of Paik et al. (2008) to consider the effect of the channel wall and water depth. The computational domain for the large propeller model is the same as that for the small propeller model. The grid around the model propeller was generated by the trimmer mesh scheme using unstructured grids, and the boundary layer on the propeller blade surface was constructed using a prism layer so that the dimensionless wall distance was smaller than 1 ($y^+ < 1$) for the small propeller. The grids around the tip and root of the model propeller have an additional layer to capture the tip and root vortices, as shown in Fig. 3. In the region of free surface, the number of grids was reinforced to capture the wave pattern of the free surface. The block surrounding the model propeller rotates to consider the relative rotating motion of the propeller to the

Table 1
Principal particulars of the model propeller (KP505).

Diameter (mm)	250.0	Scale ratio	31.60
No. of blades	5	Hub ratio	0.180
P/D (mean)	0.950	Ae/Ao	0.800
Section	NACA66	Rotation	RH

Please cite this article in press as: Paik, K.-J., Numerical study on the hydrodynamic characteristics of a propeller operating beneath a free surface, International Journal of Naval Architecture and Ocean Engineering (2017), http://dx.doi.org/10.1016/j.ijnaoe.2017.02.006