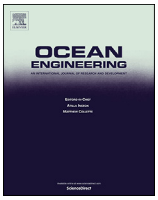
Ocean Engineering 147 (2018) 580–590

Contents lists available at ScienceDirect

Ocean Engineering

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



Numerical prediction analysis of the fluctuating pressure and rudder force of full-scale hull-propeller-rudder system

Cong Sun, Chao Wang [*], Sheng-xia Sun, Xin Chang, Liang Zhang

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

ARTICLE INFO

Keywords:
Full-scale
Propeller
Free surface
Fluctuating pressure
Rudder force
Scale effect

ABSTRACT

In order to analyze the fluctuating pressure and rudder force characteristics of a full-scale hull-propeller-rudder system, a hybrid structured/unstructured grid was combined with the Reynolds-averaged Navier-Stokes (RANS) and volume of fluid (VOF) methods to perform a numerical prediction of model and full-scale performance for the Kriso Container Ship (KCS) case. A numerical simulation for the self-propulsion test of a full-scale ship was first performed to obtain the propeller velocity under the self-propulsion point, the results of which indicated a notable wake fraction and velocity scale effect at the self-propulsion point. Transient two-phase flow calculations were then performed for the model and full-scale KCS hull-propeller-rudder systems. According to the monitoring data, the hull surface fluctuating pressure and rudder force fluctuated periodically over time and the full-scale fluctuating pressure exhibited a higher time-average value as compared to the model-scale pressure. The frequency spectrum curves were also generated by the fast Fourier transform algorithm. The analysis of the frequency spectrum data indicated that the peaks of the fluctuating pressure and rudder force reached their respective maximums at the blade passing frequency (BPF), after which the peak gradually decreased and the full-scale ship reduced more quickly. Furthermore, fluctuating amplitude of full-scale ship was bigger than model-scale.

1. Introduction[1]

Propellers periodically produce time-varying exciting forces in the three-dimensional non-uniform wake field of the stern, including the bearing force transferred to the hull by the shaft system, the fluctuating pressure, and the fluctuating rudder force transferred to the hull surface by the fluid (Yousheng and Guoqiang, 1987). The hull surface fluctuating pressure overpowers these forces when the propeller is cavitating as it is the largest contributor of vibration and noise in the hull. As a result, studies have focused on the generation of variation pattern and prediction methods for this fluctuating pressure. In fact, the amplitude of the propeller-induced hull surface fluctuating pressure has already been categorized as a gauge to measure the success of a propeller design (Xiong, 2002).

Research strategies for the measurement of the propeller-induced hull surface fluctuating pressure generally include theoretical calculations, empirical formulae, and test measurements. In terms of theoretical research, Huse and Wang (1982) generated a systematic study on the effect of the hull and free surface on the fluctuating pressure and obtained an estimation method to determining the free surface factor and solid wall factor, thereby providing an important reference for subsequent theoretical research on fluctuating pressure. van Gent (1994) then established a new idea to examine the effect of the cavitation volume on the propeller blade and its variability on the fluctuating pressure given that the cavitation growth acceleration is the main contributor for the fluctuating pressure within the blade frequency and the low-frequency noise. Empirical formula methods are generally applied to preliminary ship designs given their simple calculations and fairly rough results. Thus far, mature empirical formulae include the Holden method, Takahashi method, Fujino method, and the calculation method described in the Russian Handbook of Ship Structural Mechanics. In addition, the China Classification Society (CCS) also published a CB/Z method to estimate pressure fluctuations, though it has not been widely accepted (Li, 2013). Pressure fluctuation test measurements started early and involved both model and full-scale measurements. Previous literature (Huse, 1972; Friesch J, 1984; Duttweiler and Brennen, 2002; Xiong et al., 2002; Wu et al., 2010) has reported the findings and intensive studies performed on the effects of the propeller cavitation and air content in the test and model wake field correction.

---

[*] Corresponding author.
E-mail address: wangchao806@hrbeu.edu.cn (C. Wang).

https://doi.org/10.1016/j.oceaneng.2017.11.006
Received 18 October 2016; Received in revised form 27 August 2017; Accepted 5 November 2017

Nevertheless, the complex coupling between the propeller and the rudder working at the stern has made it difficult to predict the accurate propeller-induced hull surface fluctuating pressure and rudder force by either theoretical calculation or test measurement. However, the development of the CFD numerical calculation method validated the effectiveness of numerical processes in predicting and analyzing the hull surface. Recently, an increasing amount of studies have numerically calculated the fluctuating pressure, of which the calculation results (Ji et al., 2012; Kehr and Kao, 2011) were well in agreement with the experimental values. Since the development of numerical research and predictions for the propeller-induced hull surface fluctuating pressure, the majority of studies focused on the simulation of non-uniform axial wake flow, with minimal observations on the contribution of the free surface and the propeller-rudder interference. Furthermore, models have been generally categorized as research objects such that few full-scale pressure fluctuation numerical predictions have been reported. Therefore, it is essential to perform full-scale numerical prediction studies on the propeller-induced hull surface fluctuating pressure for a hull-propeller-rudder system taking into account the effect of the free surface.

The present study characterized a KCS ship and KP505 propeller as the study objects. A numerical pool model was established on ANSYS Fluent software. Transient two-phase flow calculations were first generated for the model hull-propeller-rudder system under self-propulsion conditions to obtain the model-scale hull surface fluctuating pressure and rudder force. A full-scale simulation was then performed on the hull-propeller under self-propulsion conditions to determine the self-propulsion point of the ship. Based on the results, the full-scale propeller-induced fluctuating pressure and rudder force on the hall surface were calculated and compared with the calculation results of the model. We then identified the full-scale and model flow fields and wave patterns. A systematic analysis was also performed on the time and frequency domains of the hull surface fluctuating pressure, the rudder force, and their respective scale effects. Given the computational difficulties and the methods, the present study focused on the pressure fluctuations induced by the propeller rotation to conduct a pressure analysis without cavitation. Future research must consider the influence of cavitation on the pressure fluctuations.

2. Mathematical approach

2.1. Control equation and turbulence model

Fluid flow is governed by physical conservation laws. Basic conservation laws include the law of conservation of mass, the law of momentum conservation, and the law of conservation of energy. The present study characterized water as the medium. Water is an incompressible fluid with negligible heat exchange, thereby requiring only the mass and momentum conservation equations for calculations. The detailed formulae are provided in previous literature (Wang, 2004). The present study employed a shear stress transport (SST) turbulence model for the calculation of the propeller hydrodynamic performance. This model effectively integrates the merits of both the k-ε and k-ω models, and well-simulates complex flows in the presence of flow separation and strong adverse pressure gradients.

2.2. Volume of fluid (VOF) model

The VOF method (Karim et al., 2014) calculates the free surface by investigating the fluid-grid volume fraction function of the grid cells and tracing the variation of the fluid rather than the particle movement on the free surface. If the value of the function on each grid of the flow field is known, the movement interface can be traced.

The entire computational domain is defined as Ω; the main phase fluid domain is defined as Ω_1; and the secondary phase fluid domain is defined as Ω_2. The VOF defines the following function:

$$\omega(\vec{x}, t) = \begin{cases} 1, & \vec{x} \in \Omega_1 \\ 0, & \vec{x} \in \Omega_2 \end{cases}. \tag{1}$$

In addition, the velocity field is composed of two non-intersoluble fluids is defined as $\vec{V} = (u, v)$. The functionω then conforms to:

$$\frac{\partial \omega}{\partial t} + u\frac{\partial \omega}{\partial x} + v\frac{\partial \omega}{\partial y} = 0. \tag{2}$$

On each gridI_{ij}, the integer of $\omega(\vec{x}, t)$ on the grid is defined as C_{ij}, thereby defining the VOP function:

$$C_{ij} = \frac{1}{\Delta V_{ij}} \int_{I_{ij}} \omega(\vec{x}, t) dV. \tag{3}$$

The VOF function also conforms to Eq. (2):

$$\frac{\partial C}{\partial t} + u\frac{\partial C}{\partial x} + v\frac{\partial C}{\partial y} = 0. \tag{4}$$

When $C = 0$, the fluids in the grids are all secondary phase fluids; when $C = 1$, the grids are filled with main phase fluids; and when $0 < C < 1$, the grids that contain fluid interfaces become interface grids.

3. Calculation modeling

3.1. Calculation object

The present study characterized a KCS container ship, which was modeled at a reduced scale of 31.6, as the study object (Alejandro et al., 2011). The ship was accompanied by a KP505 propeller. The full-scale parameters are presented in Table 1 (Wang et al., 2016).

3.2. Working conditions

The presented calculation conditions include model-scale and full-scale simulations that did not consider the trim and sinkage. The model-scale calculation conditions included the bare hull resistance and the hull surface fluctuating pressure and rudder force on the integrated hull-propeller-rudder system. The bare hull resistance was calculated by the steady VOF method, whereas the self-propulsion calculations were performed by the transient VOF method. The full- and model-scale calculation conditions exhibited similarities, though the full-scale calculations involved more conditions, such as the hull resistance and propeller thrust under various calculated velocities. Table 2 presents the details of the working conditions.

3.3. Establishment of the computational domain and setting of computational parameters

The flow field computational domain is presented as follows: the inlet is $1L_{PP}$ from the bow; the outlet is $2L_{PP}$ from the stern; and the side and bottom are both $1L_{PP}$ from the hull surface, as illustrated in Fig. 1. The inlet included an air velocity inlet and a water velocity inlet, which were identical. The outlet was set as a pressure outlet using the user-defined function in ANSYS Fluent. The vertical pressure of the outlet varied

Table 1
Main parameters of KCS ship and KP505 propeller.

KCS container ship		KP505 propeller	
LPP (m)	230.0	Diameter (m)	7.9
Draught (m)	10.8	No. of blades	5
Wet surface -rudderless (m^2)	9424	Boss ratio	0.167
Rudder wetted surface (m^2)	115	Pitch ratio (0.7R)	0.997
Reynolds No.	2.39×10^9	Area ratio	0.7