ARTICLE IN PRESS

Ocean Engineering xxx (xxxx) xxx–xxx

Contents lists available at ScienceDirect

Ocean Engineering

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

Numerical simulation of hydrodynamic performance of ice class propeller in blocked flow–using overlapping grids method

Wang Chao*, Sun Sheng-xia, Chang Xin, Ye Li-yu

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

ARTICLE INFO

Keywords:
Ice-class propeller
Ice–propeller interaction
Ice blockage stage
Hydrodynamic load
Overlapping grids method
Numerical simulation

ABSTRACT

One the basis of the technique of computational fluid dynamics (CFD), combined with the overlapping grids method, this paper establishes a numerical simulation method to solve the problem of ice–propeller interaction in a viscous flow field. A numerical simulation was carried out to forecast the hydrodynamic performance of ice-class propeller and flow characteristics in blockage conditions. Trimmed Mesh is used in the entire calculation domain, and the overlapping grid method was used to transfer the information between the propeller rotation calculation domain and the ice calculation domain. The grids in the gap between the ice and propeller were set as partial grid refinement to ensure the accuracy of the flow field in detail. By comparing the CFD result with the model experimental results, errors of hydrodynamic performance results were within 5%. The feasibility of the calculation method can be verified. It appears from the analysis of calculation results that the propeller thrust coefficients and torque coefficients increased sharply at the point at which the gap between the ice and blade was less than 10%R. From the pressure distribution of the blade's surface, it can be shown that the blade screwing in ice interference area resulted in the fluctuating extremum of propeller hydrodynamic coefficient.

1. Introduction

The working environment of an ice-going ship's propulsion device is much more complex than that of a conventional ship because of the multi-operating conditions of an ice-going ship. Especially in ice-breaking sailing conditions, the ice block frequently submerges under the hull and slips to the flow field in front of the propeller; produces extreme loads on the propeller blade; influences the propeller's hydrodynamic performance; and causes serious noise, vibration, and cavitation problems. In studies of ice-class propellers, the mechanism of ice–propeller interaction is always an essential topic and a difficult problem that has been a concern of specialists in the field of ice-going ships. The early stages of research on ice–propeller interaction problems relied mainly on model scale and full-scale propeller tests. Walker (1996) conducted a series of experimental studies on the effects of blockage and cavitation on the hydrodynamic loads associated with noncontact ice–propeller interaction. The tests indicated that blockage of a propeller resulted in increased mean levels of thrust and torque, and the phenomenon of cavitation and oscillation was more serious. Because of some restrictions in the test method, specialists began conducting theoretical research on the ice–propeller interaction problem in the 1990s. Yamaguchi (1993) investigated the performance of the JRPA open propellers used in the ice tank and cavitation tunnel experiments for both uniform and blocked flow using a lifting surface code. Other studies mainly applied the panel method program based on potential flow theory to calculate propeller hydrodynamic load. Bose (1996) applied a three-dimensional and unsteady panel method to predict the performance of the propeller in blockage flow conditions, and the numerical simulation results agreed well with test results (Luznik et al., 1995). Doucet et al. (1998) applied a panel method program called PROPELLA to predict and analyze the axial force fluctuations of several kinds of highly skewed ice-class propellers. Liu et al. (2000, 2002, 2005, 2006, 2007, 2010) applied the panel method program to gradually research the propeller load problem under ice–propeller interaction conditions and also made progress in numerical predictions. Because most of these studies were based on potential flow theory, they could not analyze the problems of effects of fluid viscosity or the boundary layer on the surface of the ice and propeller. In fact, these factors influence the performance of the propeller, the blade surface pressure, the flow field distribution, and the cavitation phenomenon. On the basis of the status of this research, this paper carries out numerical simulation and explores propeller hydrodynamic performance under ice–propeller interaction conditions. It provides a supplement to the ice–propeller interaction problem.

On the basis of the theory of viscous fluid mechanics and the overlapping grid method, this paper simulates a prediction of the

* Corresponding author.
E-mail address: wangchao806@hrbeu.edu.cn (W. Chao).

http://dx.doi.org/10.1016/j.oceaneng.2017.07.028
Received 31 May 2016; Received in revised form 14 June 2017; Accepted 5 July 2017
0029-8018/

Please cite this article as: Wang, C., Ocean Engineering (2017), http://dx.doi.org/10.1016/j.oceaneng.2017.07.028

Nomenclature

D	Diameter of propeller
J	Advance coefficient
ρ	Liquid density
G_k	Turbulence kinetic energy production item
G_ω	Turbulence kinetic energy production item
Y_k	Turbulence dissipative item
Y_ω	Turbulence dissipative item
Y	Distance of unit center to wall
n	Speed of rotation
$K_T = \frac{T}{\rho n^2 D^4}$	Propeller thrust coefficient
$K_Q = \frac{Q}{\rho n^2 D^5}$	Propeller torque coefficient

hydrodynamic performance of the ice-class propeller and pressure characteristics on the blade's surface under ice blockage conditions. First, the thrust coefficients and torque coefficients under open-water working conditions and ice–propeller interaction are calculated, and these results are compared with experimental results. Second, propeller hydrodynamic load and pressure distribution under ice–propeller interaction conditions are calculated. Last, a detailed analysis of results is given.

2. Mathematical model

The computational fluid dynamics (CFD) commercial software STAR-CCM+ (CD-adapco; Melville, NY) was used to solve Reynolds-averaged Navier–Stokes (RANS) equations using the SST model. The work environment of STAR-CCM+ includes importing geometric model, generating grids, solving models, and handling postprocessing. The overlapping grid method was applied to divide computational domain grids using STAR-CCM+. This section introduces Menter's (1993) shear stress transport (SST) $k-\omega$ model and overlapping grid method in detail.

2.1. Turbulence model

This study used the SST $k-\omega$ turbulence model for turbulence closure. The SST $k-\omega$ model is one of the most widely used turbulence models for external aero and hydrodynamics. Menter developed the SST $k-\omega$ model to effectively blend the robust and accurate formulation of the $k-\omega$ model in the near-wall region with the free-stream independence of the $k-\varepsilon$ model in the far field. Reportedly, the model has better computational performance in flows involving separation. The equations (k-omega) of turbulence kinetic energy k and vorticity intensity ω are shown:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}(\Gamma_k \frac{\partial k}{\partial x_j}) + G_k - Y_k + S_k, \text{ and} \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega u_i) = \frac{\partial}{\partial x_j}(\Gamma_\omega \frac{\partial \omega}{\partial x_j}) + G_\omega - Y_\omega + S_\omega, \tag{2}$$

where G_k, G_ω are turbulence kinetic energy production items; Y_k, Y_ω are turbulence dissipative items; and S_k, S_ω are source items.

2.2. Overlapping grid method

Steger et al. (1983) proposed the overlapping grid method. Its distinctive characteristic is that the complex flow field is divided into several simple subregions. Every subregion generates grids separately, and some different subregions have overlapping grids. Information transfer in the flow field is achieved by exchanging data in these overlapping grids between the subregions. This method poses great advantages for dealing with unsteady interaction problems caused by large amplitude mutual movement among multibodies in the flow field.

In the research of ice–propeller interaction, the positions of the ice and the propeller are not in a static state; thus, using a conventional sliding grid for modeling is infeasible when the distance between the ice and blade is minute. Using a dynamic grid also has many shortcomings. The overlapping grid method, however, provides unique advantages in solving this multibody interaction problem. It allows for overlapping, nesting, or overriding among the grid blocks, and it does not need a complicated topology partition. The difficulty of grid generation is reduced, and the relative motion among the grids is free. Thus, an overlapping grid can conveniently simulate the ice–propeller interaction problem under different conditions, and each move does not need to regenerate a grid. Its degree of automation is higher, and it saves machine hours. Therefore, the overlapping grid method is a proper choice for numerical simulation of a propeller load under ice–propeller interaction conditions.

Generally, the establishment of interactive relationship of a subregion grid in an overlapping grid can be summarized in three key steps: defining the grid area, searching the contribution unit, and transmitting data between grids. Defining the grid area is a process of establishing a closed surface and judging the relationship of a space point with the geometric surface (Li Tinghe, 2004). Its main purpose is to separately identify and define each of the subregion grids, and confirm the grid region and boundary that will be used for subsequent CFD calculation before the flow field is calculated. At the same time, needless grids are removed. As an example, Fig. 1 shows the interaction between two objects in the two-dimensional flow field. In the process of calculation, flow field information between the two subregion grids is provided through a data exchange in the deleted boundary of the overlapping grids. Searching the contribution unit refers to a search-specified grid or nodes in the grid space used to identify the grid unit and grid nodes contributing to the grid space. Transmitting data between grids is used to calculate the interpolation point information from the flow field data that are held by contribution points and the corresponding relationship with the interpolation points and to update the change of contribution unit information.

3. Calculation model of ice–propeller interaction

3.1. The establishment of the geometric model

Using the four-blade R–class propeller of the 1200 series loaded on Canadian coast guard icebreakers as the prototype, the research group independently designed propeller Ice-classI as the propeller model that maintains similar performance when compared with the R–class

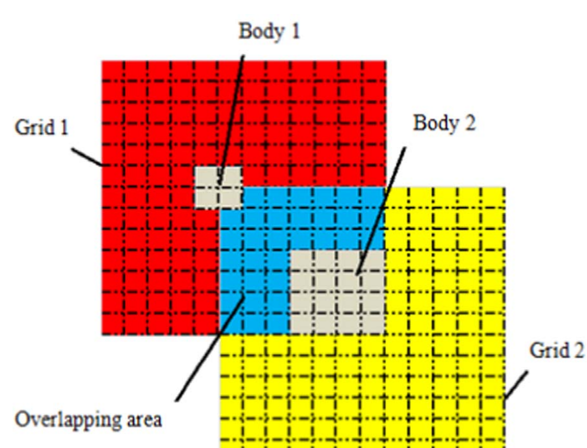

Fig. 1. Overlapping grids of two interactional bodies.