Applied Ocean Research 73 (2018) 141–148

Contents lists available at ScienceDirect

Applied Ocean Research

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

Development of numerical method to simulate flows around a ship at propulsion conditions in regular waves coupling with the ship propulsion plant model

Kunihide Ohashi

National Maritime Research Institute, 6-38-1 Shinkawa, Mitaka, Tokyo, Japan

ARTICLE INFO

Article history:
Received 5 August 2017
Received in revised form 15 January 2018
Accepted 6 February 2018
Available online 21 February 2018

Keywords:
Ship propulsion plant model
RANS
Overset-grid method

ABSTRACT

A numerical method to simulate flows with propeller effects including the response of a ship propulsion plant has been developed. The dynamics of a ship propulsion plant is modeled by the function of the diesel engine control system. Propeller torque which is computed by the propeller model with the interaction to the flow fields is put into the ship propulsion plant model, then, the propeller rotational speed is obtained by solving the equation of the rotational motion of a propeller shaft line. Present method can reproduce the fluctuations of propeller rotational speed and torque in the condition with the regular head waves. The amplitude of fluctuations shows agreement with the measured data. The detailed analysis of the flow fields with ship motions which is difficult to be obtained at experiments is also carried out.

© 2018 Elsevier Ltd. All rights reserved.

1. Introduction

The estimation of a ship performance at an actual sea condition becomes a quite complex problem due to the interactions between many factors. Ship motions are induced by waves on the actual sea, and a ship resistance is increased by waves and ship own motions. The special feature of flows around a ship is strongly viscous flows, and frictional resistance accounts for a large percentage on a total resistance. The viscous flows around a ship hull are also affected by incoming waves and ship motions. Wu et al. [1] analyze the velocity fields around the tanker hull form using PIV measurement results. The bilge vortex which is induced by the flows separation due to the adverse pressure gradient on the ship aft hull surface moves with waves and ship motions. The propulsive performance of a ship varies with incoming waves and ship motions. Nakamura et al. [2] conducted the extensive experiments including propulsive conditions which means the propeller is working just behind of the ship hull. The propeller thrust and torque increase with the propeller rotational speed to adjust the propeller thrust to the ship resistance in waves. The self-propulsion factors which indicate a ship performance vary with the wave length and the wave height. Additionally, the experiment with irregular waves were conducted, the linear superposition method seems useful to predict the mean increases of the propeller thrust and torque in irregular waves. Another important factor is the behavior of a main engine for a ship propulsion. The output of a main engine and shaft rotational speed are affected by waves on a actual sea condition. The interaction between a propeller and wake flows can be observed, and the fluctuation of a propeller torque induce the fluctuation of a shaft rotational speed. The fuel consumption of a main engine relate to the fluctuation of a shaft rotational speed and torque. Kitagawa et al. [3] developed the model testing method including the effect of the response of a main engine, and the fluctuations of the propeller rotational speed and the fuel consumption are obtained.

Recently, Reynolds Averaged Navier–Stokes (RANS) simulations are widely utilized at the design stage of a ship performance, and numerical simulations are gradually applied to more complex problems. The progress of the numerical methods to simulate the flows in waves can be observed in the series of workshops in these fields. In the 2005 workshop [4], the wave diffraction without ship motion conditions are firstly set. The 2010 workshop was held in Gothenburg of Sweden [5]. Several cases with headsea waves including freedom of ship motion are tested, and the resistance coefficient and motions in waves have certain accuracy comparing with the measured data. Sadat-Hosseini et al. [6] summarize the computed results of the tanker hull form with the heading waves including the detailed validation and verification. The effect of the freedom of the ship motion in the simulation and the experiment is revealed, and computed results of the added resistance and ship motions show agreement with the measured data. Additionally, the local flows are compared with PIV measurement results. The latest workshop

---

E-mail address: k-ohashi@nmri.go.jp

https://doi.org/10.1016/j.apor.2018.02.006
0141-1187/© 2018 Elsevier Ltd. All rights reserved.

for ship hydrodynamics was held in 2015 [7], and further complex cases can be found. The ship with motions in regular waves with certain wave angle are selected, and more complex case with which the twin screw vessel which has complex geometry in free running condition with regular waves is also set. The assessment of the last workshop is now undergoing.

A propulsive condition is achieved by the propeller model or rotating the actual propeller grid using the sliding interface technique and the dynamic overset-grid method. Propeller effects are taken into account by body forces of the propeller model, and could the propeller models reduce the computational load drastically comparing with using the actual propeller grid. Wu et al. [8] apply the propeller model based on the wing element theory to the flows around the tanker hull form including the regular head waves and ship motions, and succeed to reproduce the flows which vary with the incoming waves and ship motions comparing with PIV measured results. The fluctuation of the propeller thrust also shows agreement with the experimental data. el Moctor et al. [9] describe the relation between the engine dynamics and the propeller torque during the maneuvering behavior. The engine dynamics is based on the characteristic diagram and the effective mean pressure, the detailed movement of the engine torque and the shaft rotational speed on the engine diagram is plotted.

The object of this work is the development of the numerical method to simulate flows around a hull with regular waves considering the effect of a ship propulsion plant. Unsteady RANS (URANS) solver which can cope with the overset-grid method is coupled with a ship propulsion plant model through a propeller model which compute the body forces to express the propulsion condition. The present method can simulate the response of a ship propulsion plant and the fluctuation of a propeller torque with ship motions in waves. The ship propulsion plant model [10] based on the mathematical equation of diesel engine components is employed. Computational results are validated with the experimental data [3]. Additionally, the detail analysis for the flows around the hull with ship motions is carried out. The relation between the response of the ship propulsion plant and flow fields are examined.

2. Computational method

2.1. Base solver

An in-house structured CFD solver [11] is employed. The governing equation is 3D RANS equation for incompressible flows. Artificial compressibility approach is used for the velocity-pressure coupling. Spatial discretization is based on a finite-volume method. A cell centered layout is adopted in which flow variables are defined at the centroid of each cell and a control volume is a cell itself. Inviscid fluxes are evaluated by the third-order upwind scheme based on the flux-difference splitting of Roe. The evaluation of viscous fluxes is second-order accurate. For unsteady flow simulations, a dual time stepping approach is used in order to recover incompressibility at each time step. It is consisted from the second order two-step backward scheme for the physical time stepping and the first order Euler implicit scheme for the pseudo time. The linear equation system is solved by the symmetric Gauss–Seidel (SGS) method.

For free surface treatment, an interface capturing method with a single phase level set approach is employed.

Incoming Regular headsea waves are generated at the region inside of the computational domain [12]. Body motions are obtained by solving the equations of motion, and motions are taken into account by a moving grid technique with the grid deforming methodology. Grid velocities are contained in the inviscid terms to satisfy the geometrical conservation law. The grid velocities are derived from the volume where an each cell face sweeps. The boundary condition on a body is given as the velocities of the body motion.

In order to avoid the updates of the overset information with the grid deforming, the grid overlapped regions around the body move with the body motion without altering the relative locations. For the region away from the body, the amounts of deformations gradually decrease with the distance from the body. Thus the grids far from the body are fixed. Such the way is adapted to be able to avoid the computational load with using the dynamic overset-grid method.

2.2. Overset-grid method

The weight values for the overset-grid interpolation are determined by an in-house system [13]. The detail of the system can be found on [13], the summary is described.

1. The priority of the computational grid is set.
2. The cells of a lower priority grid and inside a body is identified (called as in-wall cell in here).
3. Receptors cells for which the flow variables have to be interpolated from donor cells are defined. Two layers of cells on a higher priority grid and facing to the outer boundary are set as receptor cells to satisfy the third order discretization of NS solver. Additionally, two cells neighborhood of in-wall cells, the cells of a lower priority grid and inside the domain of a higher priority grid are also set as the receptor cell.
4. The weight values for the overset interpolation are determined by solving the inverse problem based on Ferguson spline interpolation.

Flow variables of the receptor cell are updated when the boundary condition is set. The forces and moments are integrated on the higher priority grid to eliminate the lapped region on body surfaces. At first, the cell face of the lower priority grid is divided into small pieces. Secondly, the small piece is projected to the cell face of the higher priority grid by using the normal vector of the higher priority face. Then the 2D solid angle is computed and the small piece is decided in or out of the higher priority face. Once the small piece is in the higher priority face, the area ratio of the piece is set to zero. Finally, the area ratio is integrated on the lower priority face, then we have the ratio to integrate the forces and moments on lower priority face.

2.3. Propeller model

The propeller model based on the potential theory [14–16] is applied to achieve the propulsion condition. The propeller effect is taken account by the body forces which are computed by the following equations.

$$f_x(r,\theta)=\frac{\Gamma(r,\theta)V_\theta}{r\ ,\ \Delta x}-\frac{\frac{1}{2}C_DNc(r)\sqrt{1+(h(r)/r)^2}V_{ox}V_{o\theta}}{2\pi r\qquad ,\ \Delta x} \tag{1}$$

$$f_t(r,\theta)=\frac{\Gamma(r,\theta)V_x}{r\ ,\ \Delta x}+\frac{\frac{1}{2}C_DNc(r)\sqrt{1+(h(r)/r)^2}V_{o\theta}^2}{2\pi r\qquad ,\ \Delta x} \tag{2}$$

$$f_y=f_t,\ \cos,\ \theta,\quad f_z=f_t,\ \sin,\ \theta \tag{3}$$

where r and θ are the cylindrical coordinate at the propeller plane. The propeller circle is divided into the fan-shaped segments at (r, θ). $\Gamma(r, \theta)$ is the vortex strength at the segment(r, θ), V_x and V_θ are the total inflow velocities, V_{ox} and $V_{o\theta}$ are the circumferential averaged velocities. C_D is a drag coefficient which is given by the empirical formula, N is the blade number of a propeller, $c(r)$ is a chord length at each radial direction, $h(r)$ is the pitch of a free

Download English Version:

https://daneshyari.com/en/article/8059309

Download Persian Version:

https://daneshyari.com/article/8059309

Daneshyari.com