ARTICLE IN PRESS

Ocean Engineering xx (xxxx) xxxx–xxxx

Contents lists available at ScienceDirect

Ocean Engineering

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

Numerical study on stern flow fields of ship hulls with different transom configurations

Trong-Nguyen Duy, Takanori Hino*, Kazuo Suzuki

Department of System Design for Ocean-Space Engineering, Yokohama National University, 79-5 Tokiwadai, Hodogaya, Yokohama, Kanagawa 240-8501, Japan

A R T I C L E I N F O

Keywords:
CFD
Transom configurations
Stern waves
Shape optimization

A B S T R A C T

One of the reasons making the optimal design for a stern shape of a ship hull still difficult even nowadays is the complexity of flow field behind a transom stern. Study on effects of transom configurations on flow fields as well as on the ship resistances is therefore necessary and useful for the transom shape optimization. In this paper, the simulated results of flow fields behind transom stern of a container ship hull with various transom configurations using a CFD method are presented. Transom stern of an original container ship hull is modified in both breadth and height to get new models with various stern shapes. A flow solver which solves the Reynolds Averaged Navier-Stokes (RANS) equations for incompressible flows is employed in the computations of flow fields around ship hulls. The computational results are compared with each other to investigate correlations of the transom shapes with flow fields behind the ship stern and ship resistances. It is considered that the achieved results provide useful information for the practical optimization design of a stern shape of a ship hull.

1. Introduction

In recent years, demands on the sea transportation have been increasing due to the globalization of the world economy. Furthermore, the urgent issues of the environmental protection and recent increase of fuel cost also make the shipbuilding industry be facing with many challenges. The International Marine Organization (IMO) has released and amended in detail the mandatory measures of Energy Efficiency Design Index, Ship Energy Efficiency Management Plan and benchmarks of enhancing energy efficiency to reduce emission of the greenhouse effect gases and to improve ship energy efficiency (IMO, 2012a, 2012b). In these contexts, the optimal hull form design is acknowledged to be one of the most effective solutions to improve ship efficiency as well as to satisfy the environmental protection criteria. The optimal hull form design has a long history with many interests of ship designers and researchers. It has also achieved important results which have been being utilized in the practical designs of real ships such as bulbous bow (Han et al., 2012), Kawasaki Stern End Wedge (Kawasaki Heavy Industry, 2013), ducktail waterline extension model (Wärtsilä, 2009) and so forth. In general, it can be said that the optimal design of a transom stern is more difficult than that of the fore part. Influences of the turbulent flow, effects of transom configurations and hull-propeller interactions make flow field behind the ship stern extremely complicated. Study on characteristics of the stern waves and effects of the transom shapes on flow fields, thus, is necessary and useful for problems of the stern shape optimization. There have been some research works attempted to investigate characteristics of the stern waves, effects of the transom shapes on flow fields behind the ship stern as well as to optimize the stern shape of a ship model.

Baba (1969) pointed out that waves in the field near a hull form have a nonlinear component even in deep water which he called the wave breaking. Regarding the nonlinear component of the stern waves, he measured resistance due to the stern wave breaking of a container ship model having a transom stern. The measured data showed that resistance due to the stern wave breaking may account for as much as 13% of total resistance at a designed speed (Baba, 1973). This result impressed the ship designers with importance of breaking of the stern waves behind the transom stern. Doi et al. (1981) studied characteristics of the nonlinear stern waves and fluid motion in the boundary layer. The experimental results indicated that generation of the nonlinear stern waves is affected by the vertical disturbance velocity of bow waves and transom stern configuration. Yamano et al. (2001) considered that the stern waves are composed of the forward oriented breaking waves and the remaining following waves. Based on the experimental results, they concluded that to reduce ship resistance it is necessary to prevent or attenuate the forward oriented breaking waves which are characterized by high turbulent intensity and often occurs just behind the stern end. Karafiath and Fisher (1987) examined

* Corresponding author.
E-mail address: hino@ynu.ac.jp (T. Hino).

http://dx.doi.org/10.1016/j.oceaneng.2016.10.052
Received 23 April 2015; Received in revised form 17 August 2016; Accepted 28 October 2016
Available online xxxx
0029-8018/

Please cite this article as: Nguyen, T.D., Ocean Engineering (2016), http://dx.doi.org/10.1016/j.oceaneng.2016.10.052

various types of stern wedge to study their effects on powering performance of high speed ships. The examined results showed that in case of high speed patrol boats ($0.5 < Fn < 0.9$), up to 11% of resistance is decreased and the delivered power is reduced up to 14%. Besides, for the destroyers that have the speed range of $0.4 < Fn < 0.45$, ship resistance and delivered power are decreased approximately 5% and 6%, respectively. The experiments also proved that the wedges do not affect delivered power of ships with $0.26 < Fn < 0.3$ and that for the low speed ships ($Fn < 0.26$), the wedges increase delivered power for all designed ships.

Besides, the studies on effects of the transom configurations and appendages on the stern waves and ship resistances have been also carried out. Miyata et al. (1984) showed that effect of a transom part on the stern waves is similar to that of a Stern End Bulb or of a horizontal stern end fin. It reduces the stern wave breaking by suppression of the upward oriented water flows at the stern end which naturally causes increase of pressure resistance and elevation of the free surface. In addition, effects of the stern shapes and of the appended aft-bodies to ship resistances and to the propulsive performance have been taken into account in his study. Relation between the real stern end immersion which is defined as the immersion of transom bottom into the water when ship runs and the stern wave resistance has been examined in the studies by Yamano et al. (2003, 2005). In these studies, effects of ship speed and transom bottom angle which is defined by the inclination of the transom bottom profile from water plane to the stern wave patterns and ship resistances have been also investigated. Doctors and Beck (2005), Doctors (2006) studied the separation of flows past a transom stern and pointed out the correlations between the transom hollow length and resistance due to the stern waves.

Though studies on the stern waves as well as on the stern shape optimization have been attracting many interests of ship designers in recent years, in general, it can be said that these kinds of studies are still limited. Due to limitation of the mathematical models for non-linear flows, the numerical studies on characteristics and generation of the stern waves have been also less than the experimental ones. It is difficult to reproduce the stern waves because of the complexity of phenomena occurring behind the transom stern such as wave interferences, wave breaking, interactions with boundary layer and so forth. However, study on characteristics of the stern waves and effects of the transom shapes on flow fields are indispensable for the stern shape optimization. With the rapid development of robust and computational strategies the current Computational Fluid Dynamic (CFD) methods can provide a good evaluation of ship resistance and accurate predictions of flow field that contains both free surface and turbulent effects. While the computational techniques keep on evolving further, applications of the current CFD methods to full complexity problems of a real world have been entering industrial practices including the hull form design. The present work is an attempt to examine effects of the transom configurations on flow fields behind the ship stern and on ship resistances by using a CFD method. In order to investigate effects of the ship breadth to flow fields, breadth of the KRISO (Korea Research Institute of Ships and Ocean Engineering) Container Ship (KCS) is reduced 2 m only at the stern end plane (Model B-) and is expanded for a whole hull form with ratio of 1.2 (Model B+). In addition, to examine effects of the transom height and state at rest of transom stern on flow field, the initial transom stern of KCS hull form is modified in such a way that the modified transom bottom is flat and on the free surface at rest (Flat model). A Flat immersed model is obtained by a further reduction of the initial transom height so that its bottom profile is flat and transom stern is partly immersed at rest. A Concave model which has a downward stern end profile is created to restrain the stern wave breaking behind the stern end of the initial KCS hull form. A CFD solver is then employed to compute flow fields around the initial KCS and all modified hull forms in the same model scale. Finally, the computed results are compared with each other to investigate effects of transom configurations on flow fields as well as on the hydrodynamic resistance components.

2. Flow solver

The flow solver used in this study is SURF (Solution algorithm for Unstructured RANS with FVM) v6.43 which has been developed at National Maritime Research Institute, Japan (Hino, 1997, 1998, 1999). In SURF code, the governing equations to be solved are the three-dimensional RANS equations for incompressible flows. With the introduction of artificial compressibility into the continuity equation to couple pressure with a velocity field, the final vector form using non-dimensional variables of the flow equations in the Cartesian coordinate system can be written as follows:

$$\frac{\partial \vec{q}}{\partial t} + \frac{\partial(\vec{e} - \vec{e}^{\,v})}{\partial x} + \frac{\partial(\vec{f} - \vec{f}^{\,v})}{\partial y} + \frac{\partial(\vec{g} - \vec{g}^{\,v})}{\partial z} = 0 \tag{1}$$

where flow variable $\vec{q}$, the inviscid fluxes ($\vec{e}$, $\vec{f}$, $\vec{g}$) and viscous fluxes ($\vec{e}^{\,v}$, $\vec{f}^{\,v}$, $\vec{g}^{\,v}$) are defined as:

$$\vec{q} = \begin{bmatrix} p \\ u \\ v \\ w \end{bmatrix}, \quad [\vec{e}, \vec{f}, \vec{g}] = \begin{bmatrix} \beta u & \beta v & \beta w \\ u^2 + p & uv & uw \\ vu & v^2 + p & vw \\ wu & wv & w^2 + p \end{bmatrix},$$

$$[\vec{e}^{\,v}, \vec{f}^{\,v}, \vec{g}^{\,v}] = \begin{bmatrix} 0 & 0 & 0 \\ \tau_{11} & \tau_{12} & \tau_{13} \\ \tau_{21} & \tau_{22} & \tau_{23} \\ \tau_{31} & \tau_{32} & \tau_{33} \end{bmatrix} \tag{2}$$

In which, all variables are made dimensionless using the reference density (ρ), velocity (U) and the length (L). (u, v, w) are the velocity components in the (x, y, z) directions, respectively. β is a parameter of the artificial compressibility. Pressure (p) is modified as follows:

$$p = p^* + \frac{z}{Fn^2} \tag{3}$$

to exclude the gravitational acceleration term in the z-momentum equation. p^* and Fn are the original pressure and Froude number, respectively. Strain rate tensor τ_{ij} is defined as:

$$\tau_{ij} = \left(\frac{1}{Rn} + \nu_t\right)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_i}{\partial x_j}\right), \quad (i,\ j = 1,\ 2,\ 3) \tag{4}$$

where $(x_1, x_2, x_3) = (x, y, z)$ and $(u_1, u_2, u_3) = (u, v, w)$. Reynolds number is defined as $Rn = UL/\nu$ with ν being the kinematic viscosity. ν_t is the non-dimensional kinematic eddy viscosity determined by the turbulence model.

A finite volume approach is adopted for the spatial discretization in which the computational domain is divided into the unstructured polyhedral cells. Cell shapes are hexahedra, tetrahedral, prism or pyramid; face shapes are either triangular or quadrilateral. The cell-centered layout is adopted in which the flow variables ($\vec{q}$) are stored at the centroid of each cell and control volume is a cell itself. The inviscid fluxes are evaluated by using the upwind scheme based on the flux-differencing splitting of Roe (1986). To achieve the second order accuracy in space, the MUSCL approach is used in which the flow variables are reconstructed as a linear polynomial function inside each cell using the cell centroid values. The gradient of velocity on a cell face that is needed for the viscous fluxes calculations is evaluated by using the divergence theorem. The backward Euler scheme is used in the time integration. The symmetric Gauss-Seidel iteration is utilized to solve the linear equations derived from the time linearization of the governing equations.

Free surface conditions which consist of the kinematic and dynamic conditions are implemented in the interface capturing framework. For the dynamic free surface conditions, a single-phase flow approach for the water region is utilized. Flow equations in the water region are