Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2016,28(3):506-518
DOI: 10.1016/S1001-6058(16)60655-8
www.sciencedirect.com/
science/journal/l0016058

Numerical simulations of viscous flow around the obliquely towed KVLCC2M model in deep and shallow water*

Qing-jie MENG (孟庆杰), De-cheng WAN (万德成)
State Key Laboratory of Ocean Engineering, School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai 200240, China
Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai 200240, china,
E-mail: mjie332@163.com

(Received June 24, 2015, Revised November 25, 2015)

Abstract: By solving the unsteady Reynolds averaged Navier–Stokes (RANS) equations in combination with the $k-\omega$ SST turbulence model, the unsteady viscous flow around the obliquely towed tanker KVLCC2M model in both deep and shallow waters is simulated and the hydrodynamic forces, the surface pressure distribution, and the wake field are calculated. The overset grid technology is used to avoid the grid distortion in large drift angle cases. The effects of the free surface are taken into account. At the first stage, the deep water cases with five oblique angles are designed as the benchmark test cases. The predicted wake field, the surface pressure distribution and the hydrodynamic forces acting on the hull agree well with the corresponding experimental data, implying the capability of the present method in the prediction of the viscous flow around the tanker drifting in shallow water. A set of systematic computations with varying water depths and drift angles are then carried out to study the viscous flow around the model drifting in shallow water. The forces and moments, as well as the surface pressure distribution are predicted and analyzed. The most significant changes such as the increased stagnation pressure in the bow, the acceleration of the flow along the ship's sides and in the gap between ship and seabed, the lower hull pressure and finally, the stronger vortices along the bilges and weaker vortices with larger diameters in the wake are noticed.

Key words: drift motionm shallow water, viscous flow, hydrodynamic forces, manoeuvability

Introduction

With the increasing demand for large marine vehicles to transport various products at a relatively low cost, there has been a growing tendency to maneuver larger vessels in confined water areas while the infrastructure does not increase in size or not at the same rate. As a result, it is important to study the effects of a ship maneuvering in shallow or confined water and the difference of the ship behavior in shallow water and open water. When a ship travels in shallow water, maritime disasters such as collision and grounding occur more easily than in open waters. Thus, a better understanding and an accurate analysis of the complex flow around a ship are important for modern ship design. But in shallow water cases, the criterion and the standards of ship maneuverability promulgated by the International Maritime Organization (IMO) cannot guarantee the navigation safety. Therefore, it is necessary to study the ship behavior in shallow water and to establish corresponding standards for shallow water. For these purposes, a series of studies of the ship maneuverability in confined water were carried out worldwide during the last decades[1-6].

Stern et al.[7] provided an overview of recent progress in CFD for naval architecture and ocean engineering and author believed the CFD played a monumental role in the development of ship hydrodynamics over the last ten years[8-11]. Simonsen and Stern[12] studied the effects of drift and rudder angle on forces and moments of the Esso Osaka tanker in

* Project supported by the National Natural Science Foundation of China (Grant Nos. 51379125, 51490675, 11432009, 51579145 and 11272120).
Biography: Qing-jie MENG (1986-), Male, Ph. D.
Corresponding author: De-cheng WAN,
E-mail: dcwan@sjtu.edu.cn

"static rudder" and"pure drift" conditions. Promising results were obtained and the CFD (computational fluid dynamics) is considered as a useful tool for prediction of hydrodynamic forces acting on ships. Using the Fluent CFD code and the improved two equation turbulence model, Wang et al.[13] computed the viscous flow field around a KVLCC2 tanker in static drift motions in shallow water. Their results agree well with experimental data except in very shallow water cases and large drift angle cases, due to the distortion of the boundary layer grid for large drift angles. In addition, the free surface was neglected there. Zou[14] simulated the flow field around a KVLCC2 tanker without appendages maneuvering at varying drift angles and water depths using the CFD solver SHIPFLOW. The effects of the free surface and the sinkage and the trim were neglected. Validation is performed by comparison with model test data.

The objective of this study is to predict and analyze the viscous flow and the hydrodynamic forces of a KVLCC2M tanker maneuvering at various drift angles in both deep and shallow waters in considera-tion of the free surface. First, the capability of the present method for the prediction of the viscous flow around the tanker in drift motions in shallow water is confirmed by comparing the results obtained for deep water cases with five oblique angles with the corre-sponding numerical results by Eça et al.[15] and NMRI (National Maritime Research Institute) experiments by Kume et al.[16], and a good agreement is witnessed. A series of systematic computations with varying water depths and drift angles are then carried out and the predicted forces and moments, as well as the surface pressure distribution are analyzed to reveal the chara-cteristics of the viscous flow around the model drifti-ng in shallow water.

The computation is carried out by an in-house research code based on the finite difference method (FDM), which is developed by our group. The code was proven to be good in simulating the unsteady viscous flow around a ship in confined water[17]. Due to the great inclusiveness of the grid aspect ratio, the FDM is very suitable to solve low speed problems. Refinement grids are used only in the vertical dire-ction to ensure the grid number within an acceptable range as well as the accuracy to capture the free sur-face. On the other hand, the structured grids used for the FDM also reduce the grid number, which results in a higher computational efficiency. Besides, the over-set grid technique[18-23] is introduced to avoid the dete-rioration of the computational accuracy caused by the mesh distortion near the boundary layer region in large drift angle cases. Additionally, the overset grid technique is easy for the local grid refinement[24], which may reduce the grid number and improve the computational efficiency.

1. Mathematical and numerical models

1.1 *Governing equations*

The viscous flow around the ship is assumed incompressible and the numerical problem is descri-bed by the RANS equations coupled with the time-averaged continuity equation in non-dimensional tensor form:

$$\frac{\partial U_i}{\partial t}+U_j\frac{\partial U_i}{\partial x_j}=-\frac{\partial \hat{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left[Re_{eff}^{-1}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]+S_i \tag{1}$$

$$\frac{\partial U_j}{\partial x_j}=0 \tag{2}$$

where $U_i=(u,v,w)$ denote the Reynolds-average velocity components, $x_i=(x,y,z)$ represent the inde-pendent coordinate directions, $\overline{u_iu_j}$ and S_i stands for the Reynolds stresses and a body force due, for insta-nce, to a propeller model, respectively, the piezome-tric pressure $\hat{p}$ and the effective Reynolds number Re_{eff} are:

$$\hat{p}=\frac{p_{abs}}{\rho U_0^{\ 2}}+\frac{z}{Fr^2}+\frac{2k}{3}\,,\quad Re_{eff}^{-1}=Re^{-1}+\gamma_t$$

with k being the turbulence kinetic energy, p_{abs} the absolute pressure, z the local vertical elevation and γ_t the non-dimensional turbulent viscosity obtained from a turbulence model. All the variables and proper-ties are nondimensionalized by the reference velocity U, the ship length L_{pp} and the mass density of the water ρ.

1.2 *Turbulence model*

The SST k-ω turbulence model is chosen to close the RANS equations. As the SST k-ω turbu-lence model employs the k-ω model near walls and the k-ε model away from walls, it gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients.

The turbulence kinetic energy k is computed using a blended k-ε/k-ω model[25]. And in this model, the eddy viscosity ν_t, the turbulence kinetic energy k and the turbulence specific dissipation rate ω can be computed as:

$$\nu_t=\frac{k}{\omega} \tag{3}$$