Numerical Study on Viscous Hydrodynamic Forces Acting on a Berthing Tanker Model

Huaming Wang, Yonghe Xie
School of Naval Architecture and Civil Engineering, Zhejiang Ocean University, Zhoushan, Zhejiang, China

Zaojian Zou, Wei He
School of Naval Architecture, Ocean and Civil Engineering, Shanghai Jiao Tong University, Shanghai, China

ABSTRACT

A berthing ship usually undergoes an unsteady and low-speed lateral motion. Accurate prediction of the viscous hydrodynamic forces on the ship is important to estimate the maneuvering performance of the ship in berthing condition. The flow field around a real ship undergoing berthing motion is very complicated, and the viscous hydrodynamic forces acting on the hull are varying with time during the unsteady berthing process, which is quite different from those for the ship in normal navigation. Especially for real ships, yaw moment and longitudinal force may be remarkable due to the different form at bow and stern part. The focus of the present work is placed on numerical simulation of the viscous flow field and prediction of the unsteady longitudinal and lateral forces and yaw moment acting on a real ship undergoing unsteady berthing motion. A KVLCC2 tanker model is taken as example for the numerical investigation. The unsteady Reynolds-Averaged Navier-Stokes (URANS) equations are solved by using a general purpose CFD code FLUENT with the SST $k$-$\omega$ turbulence model. Detailed flow features at some characteristic time are captured, and the time history of the viscous hydrodynamic forces are presented and analyzed.

KEY WORDS: numerical simulation; berthing motion; viscous flow; hydrodynamic force

INTRODUCTION

Ship berthing is an important phase in the ship’s whole navigation process, during which ship’s safety and manoeuvring performance are concerned. Therefore, it is of great significance for us to study the flow field and hydrodynamic forces on a ship undertaking berthing motion to ensure a safe operation. A berthing ship usually moves laterally at low speed. Due to the characteristics of the ship berthing motion and the geometry of the hull, the flow field and the hydrodynamic performance of the ship in a berthing process is quite different from that in ordinary motion. In the flow field around a berthing ship, the cross flow is dominant and the viscous effect plays an important role in determining the hydrodynamic forces acting on the hull. Violent flow separation and vortices may be produced in the flow field on the lee side of the hull. So the ship may suffer great lateral hydrodynamic force. Especially for real ships, yaw moment and longitudinal force may be remarkable due to the different form at bow and stern part. To gain a clear insight into the ship hydrodynamic performance in berthing motion, it is necessary to investigate the viscous flow field around a real ship in berthing motion and the viscous hydrodynamic forces acting on the hull. Moreover, ship berthing is usually an unsteady process. Accurate prediction of the time history of the hydrodynamic forces according to different berthing speeds may provide guidance to the ship berthing operation.

During the past decades, model test was the main and most effective method in the study of problems in ship hydrodynamics. However, there are some obvious deficiencies for this method, since it is time-consuming and costs a lot of money. Nowadays, the rapid development of the computer science and technology has provided good hardware condition and made the numerical method a powerful tool in prediction of ship hydrodynamic performance. The successful application of the advanced computational fluid dynamics (CFD) technology in the research of ship hydrodynamics has made the numerical simulation of the complicated flow possible. In recent years, a great deal of numerical studies relating to berthing issues has been carried out by using CFD method based on RANS equations. Chen et al. (1997-1998) developed a chimera RANS method to simulate the flow field around the berthing ship in time-domain and some detailed flow features are captured successfully. Toda et al. (2002) applied 3-D CFD technique with finite analytic method (FAM) to evaluate the hydrodynamic force acting on a standard Wigley ship in berthing motion. Recently, the authors (Wang and Zou, 2009) investigated the transient flow field and the lateral hydrodynamic force on an unsteady berthing ship by solving the unsteady Reynolds-Averaged Navier-Stokes equations with finite volume method (FVM). All of the above efforts demonstrated the feasibility and validity of CFD method for the numerical simulation of ship berthing problem.

In this article, a three dimensional unsteady RANS approach based on FVM with a general purpose CFD package FLUENT is applied to simulate the viscous flow field around a real ship in an unsteady berthing motion. SST $k$-$\omega$ turbulence model is employed to enclose the