CFD Study on Hydrodynamic Performance of an escort tug

Yuxuan Duan, Jianxi Yao
School of Naval Architecture, Ocean and Energy Power Engineering, Wuhan University of Technology, Wuhan, China & Southern Marine Science and Engineering Guangdong Laboratory, Zhuhai, China

Song Zhang
School of Transportation and Logistics Engineering, Wuhan University of Technology, Wuhan, China

Zhen Yang*
Southern Marine Science and Engineering Guangdong Laboratory, Zhuhai, China

Jialun Liu
Intelligent Transportation Systems Research Center, Wuhan University of Technology, Wuhan, China
National Engineering Research Center for Water Transport Safety, Wuhan, China

ABSTRACT

Maneuverability is crucial for ship operators to understand the performance of escort tugs and improve seamanship. In this work, an escort tug model is taken as the studied ship. Oblique towing tests (OTT) are numerically simulated via the Reynolds-averaged Navier-Stokes (RANS) method. In addition, the details of the flow field under different working conditions are analyzed. The numerical simulation results of the tug provide a basis for obtaining the hydrodynamic derivatives of the bare hull, which is the basis for the subsequent establishment of the maneuvering motion model.

KEY WORDS: CFD, Captive model test, numerical simulations, RANS, escort tug.

INTRODUCTION

As significant vessels for harbor operations, tugs have a relatively simple hull form, excellent propulsion performance, and great flexibility. It is commonly employed to assist large ships with berthing, departure, escorting, and emergency rescue. Considering that tugs have high requirements for maneuverability, it is necessary to investigate the hydrodynamic performance related to the maneuverability of tugs. Also, understanding the flow around the hull in different states of navigation is important for many designs and operational aspects.

Combining mathematical models and computer simulations to simulate ship motion is an effective way to study ship maneuverability. The hydrodynamic coefficient is one of the essential model parameters in the ship maneuvering motion model. The computational fluid dynamics (CFD) method produces more accurate hydrodynamic coefficients at a cheaper cost. This method has steadily evolved into one of the most widely used ways for calculating hydrodynamic coefficients. Taking the KVLCC2 model as the research object, Feng (Feng, Zou et al. 2015), Liu (Liu 2019) carried out numerical simulation and verification of the oblique towing tests. Zhang (Zhang, Wu et al. 2022) investigated the resistance and moments by carrying out OTT and circular motion tests (CMT). Piaggio (Piaggio, Viviani et al. 2018, Piaggio, Villa et al. 2020, Piaggio, Villa et al. 2020) studied the hydrodynamic force and moment of the escort tug using virtual static and dynamic constraint model tests.

To combine numerical accuracy and time cost, an approach combining numerical simulations with empirical formulas to forecast maneuverability is worth consideration. He (He, Kellett et al. 2016), Zhang (Zhang, Liu et al. 2022) combined the model parameters obtained by numerical calculation with the parameters obtained by empirical formula to establish the corresponding maneuvering motion model and proved the feasibility of this method.

To evaluate the maneuverability of the ship quickly and accurately, the numerical simulations of an ASD (Azimuth Stern Drive) escort tug are carried out via CFD method. Computations are performed by using the RANS solver in commercial software STAR-CCM+, and the governing equations are closed with the Realizable k-ε turbulence model. The Volume of Fluid (VOF) method is used to capture the free surface elevation.

Firstly, this paper calculates the resistance with coarse, medium and fine grid densities, and the grid discretization error is estimated with Grid Convergence Index (GCI) method. Moreover, the calculated hydrodynamic forces and moment are compared with experimental data to validate the numerical method. Then, a series of numerical simulations with different drift angles is conducted to analyze the hydrodynamic performance. In addition, the details of the flow field under different working conditions are analyzed.