Numerical Simulation of Longitudinal Motion of High Speed Craft in Calm Water

Jianmin Liu *, Xiaoping Yu

Marine Engineering Institute, Jimei University, Fujian Xiamen, 361021, China

*Corresponding author e-mail: liujm501@163.com

Abstract. In order to accurately estimate the hydrodynamic performance of high-speed crafts sailing in calm water, a numerical method based on CFD technology was established in this paper. The method included techniques such as cutting body mesh, overlapping mesh, two-phase flow, VOF model, and 6-DOF motion model. The numerical simulation of the craft's heave and pitch coupling motion in hydrostatic water was carried out. The variation of the speed of the high-speed craft with the change of the longitudinal angle and the centre of gravity during the motion was analysed. To prove feasibility and accuracy of numerical simulation method for hydrodynamic performance of longitudinal motion of high-speed crafts, the calculated results were compared with the experimental values of the ship model.

1. Introduction

As the rapidity of ships has received more and more attention, the application research on high-speed ships has become a hot topic. However, due to large changes in the attitude of the high-speed craft during navigation, the hydrodynamic performance prediction methods of conventional ships are no longer applicable. Finding new methods to accurately estimate the hydrodynamic performance of high-speed crafts has become the focus of many scholars.

Research on the hydrodynamics calculation of high-speed crafts had attracted the attention of many scholars as far back as the 1940s. The drag experiment of NACA in the Langley Pool provided experimental basis for its research [1]. Based on the experimental data, a series of experiences or half was obtained. The calculation method of the empirical formula [2, 3] was achieved remarkable results in the prediction of hydrodynamic performance of high-speed crafts. In recent years, research on the hydrodynamic performance of crafts based on CFD technology has become a hot topic. Yusuke Tahara et al. applied the RANS method to simulate and analyze the hydrodynamic performance of the KCS ship in towed state and self-propelled state [4]. NI Chong-ben et al. studied the hydrodynamic performance of high-speed crafts with open degrees of freedom and forecasted ship resistance considering heave and pitching motions of ships. The calculation results reasonably reflected the actual movement of ships. Dong et al. conducted an experimental study on a deep V-type planing craft in the longitudinal movement of a regular wave top wave, and proposed a new method to predict the longitudinal motion of the planing craft at high speed. The accuracy of the method was verified by model experiments [6, 7]. Based on the RANSE VOF solver, Wang et al. performed a numerical simulation analysis of the multi-degree-of-freedom motion of the planing craft which was in a high-speed direct
flight in calm water and motion in head sea in the regular wave. It was considered that this method was feasible and accurate in simulating the motion and resistance of high speed sailing of a planing craft [8].

In this paper, the CFD software STAR-CCM+ is applied to simulate the high-speed craft model by using the cutting body mesh and the overlapping mesh method. The calculated results are compared with the results of the high-speed craft model resistance experiment. It is proved that the CFD method can provide a reliable reference for the calculation of hydrodynamic performance of high-speed crafts accurately and efficiently.

2. Calculation Model

2.1. Model parameters
The calculation model is a high-speed craft model. The high-speed craft model is shown in Figure 1. The main parameters of the high-speed craft model are shown in Table 1.

![Figure 1. Three view drawing of high-speed craft model.](image)

| Dimension | Length L/m | Beam B/m | Draft H/m | Displacement Δ/m |
|-----------|------------|-----------|-----------|-----------------|
| Parameter | 2.3        | 0.6       | 0.35      | 5.25            |

2.2. Calculation Control Domain
The calculation area is: the front of the model takes 2 times the length of the craft; the rear area of the model takes 10 times the length of the craft to facilitate capturing the flow field at the tail of the high-speed craft; the left and right areas of the model take 2.5 times the length of the craft; the area above the free surface takes 2.5 times and the area below takes 5 times, as shown in Figure 2

![Figure 2. The Control Domain.](image)
2.3. Meshing

Using the surface repair function of STAR-CCM+ software, the surface mesh reconstruction of the high-speed craft model is performed to generate a high-quality surface mesh with good triangulation. A cut-off mesh with a boundary layer mesh and an overlapping mesh is generated based on the surface mesh.

An overlapping mesh is used to capture the longitudinal motion pose of the high-speed craft model uses while navigating in calm water. In the overlapping mesh model (Figure 3), the mesh cells are divided into active mesh cells (yellow region parts), passive mesh cells (red region portions), and accepting mesh cells (blue regions), sometimes it is called a ghost grid. Among them, the active grid unit is performed for the discrete equation solving. The passive grid element does not solve the discrete equation and is basically locked. The interpolation grid unit is divided into an acceptor grid unit and a donor grid unit, which are respectively on the boundary of the overlapping grid unit boundary and the background grid unit, and accept interpolation information to perform information transmission between the two parts of the grid. The computational domain grid is a mesh of cut bodies with overlapping mesh regions and with boundary layers, and mesh refinement encryption around the hull and near the free surface to better capture local flow field information. The lattice distribution is shown in Figure 4.

![Figure 3. Overset mesh model.](image1)

![Figure 4. Computational grid.](image2)
2.4. Calculation conditions
In this paper, the numerical simulations of the longitudinal motion of the high-speed craft during hydrostatic navigation are carried out by three different calculations of $Fr=0.75$, $Fr=0.9$ and $Fr=1.05$, and the numerical simulation values are compared with the experimental values of the pool.

3. Control equations and numerical models
For an incompressible viscous flow that satisfies the continuity equation and the RANS equation, its tensor form can be expressed as:

$$
\begin{align*}
\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} (\rho u_i) &= 0 \\
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j) &= -\frac{\partial \rho}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_i}{\partial x_j} - \rho \overline{u_i u_j} \right) + S_i 
\end{align*}
$$

(1)

In the formula, $u_i$ and $u_j$ is the time-average value of the velocity component, $\overline{u_i u_j}$ is the average value of fluctuating velocity, $\mu$ is the dynamic viscosity coefficient of the fluid, $\rho$ is the fluid density, and $S_i$ is the source term.

The improved Realizable $k-\varepsilon$ turbulence model is adopted in the paper. The VOF method is adopted to capture the free liquid surface. The pitch and heave coupling motion in the high-speed craft navigation process is solved by a six-degree-of-freedom model.

4. Analysis of calculation results
In the six-degree-of-freedom movement of the ship, the longitudinal movement is mainly characterized by pitching and heaving. In the numerical simulation process, the solver of two degrees of freedom of pitch and heave is turned on. Firstly, the high-speed craft, which draught value is set by the model experiment, is placed in the flow field. Secondly, the flow field is initialized and the fixed navigation attitude is maintained for a period of time. Lastly, After the flow field is initialized, the hull is released. Given the forward speed of the high-speed craft, the high-speed craft movement is a heave-pitch coupling motion, which tends to stabilize after a period of motion. In this way, the navigational state required for numerical simulation is achieved.

Taking $Fr=0.75$ as an example, the speed craft resistance curve (Fig. 5), is shown in the Figure 5. It can be seen from the figure that it takes 2s time for the high-speed craft to end the large-scale oscillation in the simulation calculation. After the initial short-term oscillation adjustment, the follow-up state of the high-speed craft tends to be stable, and the total resistance of the hull converges to 37.5N. In the initial stage of the high-speed craft navigation, the pitch angle is changed greatly, reaching a maximum value of 5.9° in about 1 s and a steady state in the subsequent navigation at 4.1°.

Figure 5. The curves of high-speed craft resistance.
The calculation results of the high-speed craft resistance and navigation state under different Frude numbers and their comparison with the pool experimental results are shown in Figure 6. It can be seen from the figure that the variation trend of the resistance curve and the trim angle curve of the numerical simulation is basically consistent with the results of the pool test, which is also consistent with the variation law with the increase of the speed.

Figure 6. The comparison of calculated value and experimental value under different Froude number.

5. Flow field around the high speed craft and sailing attitude
When moving longitudinally in calm water the wave chart of the high speed craft model is shown in Figure 7. Figure 7(a), Fig. 7(b), and Fig. 7(c) show the attitude and surrounding flow field of the high-speed craft after dynamic adjustment of the heave and pitch motion by Fr=0.75, Fr=1.0, and Fr=1.2, respectively.

Figure 7. The situation of flow field around High-speed craft under different Froude number.

It can be seen from Fig. 7(a) that when Fr=0.75, the center of gravity of the speed craft model is reduced by about 0.023m, and the hull of the hull is lifted upward by about 4.1°. Because the speed of the hull is small, the tail flow field does not appear obvious. It can be seen from Fig. 7(b) that when Fr=0.9, the hull of the hull is lifted upward by about 5.0°, and the stern splash is more obvious due to the higher speed, and the tail flow field is more obvious. It can be seen from Fig. 7(c) that when the Fr=1.05, the hull of the hull is lifted upward by about 5.2°, and the characteristics of the tail flow of the tail flow field are particularly obvious due to the obvious splattering on the raft and both sides of the hull.

6. Conclusion
Based on the CFD theory, a numerical simulation of the longitudinal motion of a high-speed craft in calm water is presented in this paper. Through the combination of overlap mesh and 6-DOF model, the numerical prediction of the heave and pitch coupling motion of high-speed crafts in hydrostatic
navigation is realized. The simulation results are compared with the experimental results of the pool. The results show that the total hull resistance error and the trim angle error are less than 5%. For the high-speed craft numerical forecast, the engineering accuracy requirements can be met, and the CFD method is feasible and accurate. Regarding the flow field around the hull and the attitude of navigation, the high-speed craft navigation attitude is balanced by dynamic adjustment during the simulation. When the speed of the high-speed craft reaches Fr=1.05, the pitch angle is basically maintained at about 5° after the navigation is stable. At this time, there is a phenomenon of rooster tail flow in the tail of the hull. It is provided a reliable reference for the study of hydrodynamic performance of related high-speed crafts.

Acknowledgments
This work was financially supported by the Natural Science Founds of Fujian Province (No. 2019J01707).

References
[1] Chambliss D. B., Body G. M., The planing characteristics of two V-shaped prismatic surfaces having angles of deadrise of 20 and 40, Washington:Langley Aeronautical Laboratory: NACA. 1953: 28-76.
[2] Savitsky D., Hydrodynamic design of planing hulls. Marine Technology, 1964(1): 71-95.
[3] John M. A. Resistance prediction of planing hulls:state of the art. Marine Technology, 1993,30(4):296-307.
[4] Tahara Y., Wilson R. V., Carrica P. M., RANS simulation of a container ship using a single-phase level-set method with overset grids and the prognosis for extension to a self-propulsion simulator. Journal of Marine Science and Technology Volume, 2006, 11(4): 209-228.
[5] Ni C., Zhu R., Miao G., Fan J., Hull gesture and resistance prediction of high-speed vessels. Journal of Hydrodynamics, 2011, 23(2):234-240.
[6] Dong W., Qiu G., Experimental study on longitudinal motion of deep-V-shaped planning craft, Ship Engineering, 2004, 26(2):14-16. (in Chinese)
[7] Dong W., Liu Z., Wu X., A new theoretical method on longitudinal motion of planning craft in wave, Journal of Ship Mechanics, 2007, 11(1):55-61. (in Chinese)
[8] Wang S., Su Y., Du X., Numerical simulation of planing crafts sailing in calm water and in waves, Journal of South China University of Technology (Natural Science Edition), 2013, 41(3):119-126. (in Chinese)
[9] Wang F., Computational fluid dynamics analysis-CFD software principles and applications, Beijing: Tsinghua University Press, 2004.
[10] Zhang S., Computational fluid dynamics and its application-the principle and application of CFD software, Wuhan: Huazhong University of Science and Technology Press, 2011.
[11] Zhang Z., Zhao F., Numerical calculation of viscous free-surface flow about ship hull, Journal of Ship Mechanics, 2002, 6(6): 10-17.
[12] Liu R., Numerical simulation method and motion boundary tracking, Hefei: University of Science and Technology of China Press, 2001.
[13] Zhao F., Gao C., Xia Q., Overlap grid research on the application of ship CFD, Journal of Ship Mechanics, 2011, 15(4):332-341. (in Chinese)