CFD numerical simulation of a container ship turning motion

WU Zixin1,*, CHEN Weimin2 and HAO Hao3

1Shanghai Ship and Shipping Research Institute, Shanghai, China
2Key Laboratory of Marine Technology Ministry of Communications, PRC
3State Key Laboratory of Navigation and Safety Technology

*Email: wu.zixin@coscoshipping.com

Abstract. The ship turning motion can reflect the turning characteristics of the ship and is closely related to the navigation safety of the ship. In the turning performance test, the trajectory drawn by the center of gravity of the ship is called the turning circle. The turning circle is an important indicator to characterize the turning performance of a ship. At present, the results of ship turning obtained by numerical calculations are still very different from the test ones. In this paper, the free turning motion of a container ship is simulated by a commercial software which is called STARCCM+. In order to reduce the amount of calculation, a body-force propeller model is adopted to replace the real propeller. Through the global solution of the fully viscous flow field, the characteristic parameters and the motion track of the ship turning motion can be gotten. The ship’s motion trajectory and the parameters of the turning motion obtained by numerical prediction are in good agreement with the test values, which fully proves the applicability and reliability of the proposed method in numerical prediction.

1. Introduction

Turning motion is a common mode of motion in ship maneuverability, which reflects the maneuverability of the ship. The final stage of the ship’s turning motion can be approximately regarded as the state of steady turning motion. According to the characteristic parameters of the ship's steady motion, the maneuverability of the ship can be easily analyzed theoretically. Over the years, scholars all over the world have conducted extensive and in-depth experimental researches on the turning performance of ships. However, the cost of the ship model test is high and it is impossible to analyze the changes in the hydrodynamic performance of the ship in detail.

At present, Many scholars intend to use the CFD method to numerically calculate the turning motion of ships. Hyman develops a two-phase flow solver CMFD that considers bubbles, and performed a numerical simulation of the AthenaR/V ship under a given constant radius of turning motion, and quantitatively compared the simulation results of the tail wave with the actual ship observation data. The comparison with test values prove the feasibility of numerical method to simulate the steady turning motion of the ship[1]. Stern summarizes the current methods of numerical simulation of maneuverability using the CFD method on the basis of the international manipulation conference SIMMAN2008, and concludes that the use of the DES method and finer meshes can get more accurate numerical results[2]. Toxopeus solves the unsteady equation for the fully-attached KVLCC2 ship, and uses dynamic grid technology to handle the hull maneuvering motion. The numerically predicted hull lateral force and head turning moment are in good agreement with the
standard ship model test data given by the SIMMAN2008 meeting[3]. Xiang GUO simulates the zig-zag motion of a double propellers double tail fins and double rudders ship based on URANS equation, body force method and overset mesh technology. And a ship model test is performed to verify the reliability of the numerical methods[4]. Pablo uses the body-force method instead of a real propeller to simulate the turning and zig-zag motion of the surface ship MARIN-7967, but the numerical results still have large errors with test ones [5]. By solving the unsteady RANS(Reynolds-averaged Naiver-Stokes) equation, Carrica numerically simulates the maneuvering motion of the DTMB5512 ship model by overset technology. And the numerical results are in good agreement with test ones[6].

Based on the RANS equation and the k-Epsilon turbulence model, this paper combines overset grid technology, 6-DOF(Degree of Freedom) motion model and Body-force propeller model to calculate the turning motion of a container ship. And the model test is performed to verify the accuracy of the numerical calculation.

2. Numerical simulation methodology

2.1. Governing equation
In a fixed coordinate system, the Reynolds average continuity equation and momentum equation of the incompressible fluid are as follows:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \tag{1}
\]

This is the formula of continuity equation, where \( \rho, t, u, v \) and \( \omega \) represent density, time, and velocity component respectively.

\[
\frac{\partial (\rho u)}{\partial t} + \nabla \cdot (\rho u \mathbf{u}) = -\frac{1}{\rho} \frac{\partial p}{\partial x} + u \nabla^2 u + f_x \tag{2}
\]

\[
\frac{\partial (\rho v)}{\partial t} + \nabla \cdot (\rho v \mathbf{u}) = -\frac{1}{\rho} \frac{\partial p}{\partial y} + u \nabla^2 v + f_y \tag{3}
\]

\[
\frac{\partial (\rho w)}{\partial t} + \nabla \cdot (\rho w \mathbf{u}) = -\frac{1}{\rho} \frac{\partial p}{\partial z} + u \nabla^2 w + f_z \tag{4}
\]

This is the formula of momentum equation, where \( p \) and \( f \) represent the pressure acting on the fluid cluster and the sum of the body force acting on the infinite body.

Free surface is treated by VOF (Volume of Fluid), and the k-Epsilon model is selected as the turbulence model. The above equations are all discretized by the finite volume method.

2.2. Body-force model of propeller
The body-force model is used to equate the thrust and torque produced by the propeller. The method ignores the influence of the geometric shape of the propeller blades, and distribute the radius according to the radius change in the cylindrical area defined by the diameter of the propeller disk and the longitudinal thickness of the propeller. The distribution of the force is optimized by Goldstein. And its formulas are as follows:

\[
f_{bh} = A_{bh} \frac{r^* \sqrt{1-r^*}}{r^*(1-r_b^*)+r_b^*} \tag{5}
\]

where \( f_{bh} \) is axial force and \( f_{bh} \) is tangential force; \( f_{hs} = A_{hs} r^* \sqrt{1-r^*} \)

\[
r^* = \frac{r - r_b}{1 - r_b} ; r_b = \frac{R_H}{R_p} ; r = \frac{r}{R_p} \tag{6}
\]

\[
A_h = \frac{105}{8} \pi \Delta (3R_H + 4R_p)(R_p - R_H) \tag{7}
\]
\[ A_g = \frac{105}{8} \frac{Q}{\pi \Delta R_p (3 R_H + 4 R_p)(R_p - R_H)} \]  

(8)

where \( R_p, R_H, r, T \) and \( Q \) represent the radius of the propeller, the radius of hub, radiation radius, thrust and torque of open water propeller.

The propeller in this paper is right-handed, and the open water performance curve of this propeller is shown in figure 1.

\[ \text{Figure 1. Open water performance curve.} \]

2.3. Geometric model and the setting of calculation

The ship is an oil tanker and its basic parameters are shown in table 1, and the model of it is shown in figure 2.

\[ \text{Table 1. Basic parameters of the ship and rudder.} \]

| Parameters                                | Values          |
|-------------------------------------------|-----------------|
| Lpp                                       | 3.763 m         |
| B                                         | 0.719 m         |
| Draft                                     | 0.223 m         |
| Longitudinal position of center of buoyancy | 1.949 m       |
| Height of center of gravity               | 0.323 m         |
| Rolling moment of inertia                 | 24 kg \cdot m^2 |
| Pitch moment of inertia                   | 367 kg \cdot m^2 |
| Initial moment of inertia                 | 398 kg \cdot m^2 |
| Height of rudder                          | 0.162 m         |
| Breadth of rudder                         | 0.1 m           |

\[ \text{Figure 2. Model of ship.} \]

In the numerical calculation, the overlapping grid method is used to divide the grids. The total length of the calculation domain is 5 times the length of the ship. The front end is about 2 times the length of the ship in the positive direction of the X axis. The boundary type is set as the velocity inlet. The rear end is about 3 times the length of the ship in the negative direction of the X axis. The boundary type is set to pressure outlet. The upper boundary is at a distance of 1 time the ship’s length from the origin above the Z axis, the lower boundary is at a distance of 1.5 times the ship’s length from the origin below the Z axis, and the side boundary is about 2 times the length of the ship of the Y axis. The completed grid is shown in the figure 3 and figure 4.
2.4. Numerical simulation results and analysis

This paper calculates the turning motion of 35° under the speed of 1.2m/s and the characteristic parameters of the movement are obtained. Figure 5 and figure 6 respectively show wave pattern and time histories of ship motion.

![Figure 5. Wave pattern for self-propulsion in calm water.](image1)

![Figure 6. Time histories of ship motion.](image2)

Figure 7 shows the wave patterns of the ship model during turning motion. It can be seen that the wave pattern on both sides of the ship model show obvious asymmetry, which is consistent with the actual situation.

![Figure 7. Wave pattern during turning motion.](image3)
Figure 8 shows time histories of sway, surge, roll and pitch. It can be seen from the figure that pitch and roll motion will produce more obvious wave frequency oscillation characteristics, while sway and sway motion show smaller wave frequency motion characteristics. During the whole turning motion, the maximum angle of rolling motion exceeded 16 degrees, which shows that the rolling motion of this ship is very violent.

![Figure 8. Time history curves of ship motion.](image)

3. Ship model experiment and comparison with numerical results

This experiment is carried out in the ship model maneuvering tank of Shanghai Ship and Shipping Research Institute. Figure 9 and figure 10 are the ship model and the experimental process respectively.

![Figure 9. Model of ship.](image)

![Figure 10. Status of ship during turning motion.](image)

Trajectories of the ship model are compared in Figure 11, it can be seen that the numerical results basically match with the test ones. The characteristic parameters during turning motion got by calculation and the test are shown in Table 2. It shows that the numerical simulation results are reliable.
Table 2. Comparison of characteristic parameters.

| Parameters        | Test values | Calculated values | Error |
|-------------------|-------------|-------------------|-------|
| Tactical Diameter | 9.14 m      | 9.38 m            | 2.63% |
| Final Diameter    | 8.05 m      | 8.51 m            | 5.71% |
| Advance           | 11.44 m     | 11.42 m           | -0.17%|

Figure 11. Comparison of trajectory between numerical results with test one.

4. Conclusion
In this paper, CFD method combined with overset grid technology and body-force method are used to simulate the turning motion of a container ship. And characteristic parameters of turning motion are tested in ship model test. Through this research, the following conclusions are obtained.

1) From the calculation results, whether it is the characteristic parameters or the trajectory of the ship's center of gravity during the turning motion, they all match the test well. It proves that the computational method adopted is feasible and precise.

2) Although there is no test result, the motion laws of sway, surge, roll and pitch conform to their typical change form in turning motion. Special equipment will be purchased to measure hull motion and verify the computational results.

3) Because the body-force method ignores the influence of the propeller rotation on the hull, it still causes calculation errors. Real propeller will be adopted to obtain numerical prediction results with higher precision in the future.

References
[1] Hyman M, Moraga F, Drew D. Computation of the unsteady two-phase flow around a maneuvering surface ship[C]. In 26th ONR Symposium on Naval Hydrodynamics, Rome, Italy, 2006.
[2] Simonsen C D, Stern F. Flow pattern around an appended tanker hull form in simple maneuvering conditions[J]. Computer & Fluids, 2005, 34(2): 169-98.
[3] Toxopeus S L, Walre F. van, Hallmann R. Maneuvering and Seakeeping Tests for 5415M [C]. // Proceedings of AVT-189 Specialists’ Meeting Portsdown West, UK, 2011.
[4] Xiang Guo, Ou Yongpeng. Numerical Simulation and Analysis of the Zig-zag Maneuvers of a Ship Model[J]. Journal of Wuhan University of Technology, 2016, 40(1): 184-9.
[5] Pablo M C, Alejandro M C. Turn and zigzag maneuvers of a surface combatant using a URANS approach with dynamic overset grids[J]. Journal of Marine Science and Technology, 2013, 18: 166-81.
[6] Carrica P M. DES simulation of KVLCC1 in turn and zigzag maneuvers with moving propeller and rudder [C]. // Proceedings of the SIMMAN 2008 Workshop on Verification and Validation of Ship Maneuvering Simulation Methods, Lyngby, Denmark: 2008.