Numerical Simulation for Zig-zagTest of “YU-KUN” Based on CFD

Li Qiang, Hong Bi-guang, Kong Qing-wen

Navigation College, Dalian Maritime University, Dalian(116026), Liaoning, China

manumail@163.com

Keywords: Ship; Zig-zagTest; Computational Fluid Dynamics; Numerical Simulation

Abstract. By setting the training ship of Dalian Maritime University, “Yu-Kun” as the research object, numerical simulation was conducted for 10°/10°zig-zagmaneuvering test based on the theory and calculation method of computational fluid mechanics in this paper. According to thecontrastive analysis on thesimulation experiment results and ship test results, the engineering needs can be satisfied by themodel, grid partitioning and numerical computation methods of this paper. Some theoretical and reference values are provided for studies on ship hydrodynamic force under different working conditions, and feasibleresearch methods are offered to numerical simulation of ship motion under different working conditions.

Introduction

In recent years, computational fluid dynamics (CFD) method has been applied to studies on ship hydrodynamic force more and more extensively[1-2]. Based on CFD theory, Reynolds averaged Navier-Stokes (RANS) equation was adopted as control equation and SST k-εturbulence model was combined with wall function to make the control equation closed in this paper. Single-phaselevel set method was adopted for free surface simulation and body force model was used for propellersimulation. Besides, overset gridtechnology was used to realize partitioning of structured grid and 6-DOF motion of the ship. Meanwhile, the equation was discretized by applying finite volume method (FVM)[3], and pressure implicit split-operator (PISO) algorithm was adopted for coupling solution[4]. Finally, numerical simulation was conducted for the viscous flow filed in 10°/10°zig-zagmaneuvering test of “Yu-Kun” of Dalian Maritime University.

Numerical tank

Control equation. The shape of stern is quite complicated and the curvature change is huge, so the surrounding flow field is also complex. Moreover, calculation for the viscous flow filed of ship belongs to high Reynolds number[5], so RANS method is extensively applied in the field of computational fluid mechanics[6]. RANS method was used to study numerical values about zig-zagmaneuvering test of ship in this paper, which could meet the calculation requirements and precision. The tensor forms of continuity equation and momentum equation (RANSequation) of fluid are as follows:

(1)

(2)

In the above formulas, means fluid density; represents time averaged speedcomponent of fluid; denotes fluid pressure; signifies dynamic viscosity of fluid; refers to turbulent fluctuationspeed component corresponding to time averaged speed; is thesource item.

Establishment of computational domain model. On the premise of meeting the test requirements, relative motion coordinate system was adopted in setting of computational domain[9]. Thelength, width and height of computational domain were set as 3.5L, 2.6L and 1.1L. The distance between the inlet and bow was 1L, the distance between the outlet and bow was 1.5L, the distance betweenmid-ship section and left and right boundaries was 1.3L, and the distances between free surface and up and down boundaries were 0.1L and 1L, as shown in Fig. 1

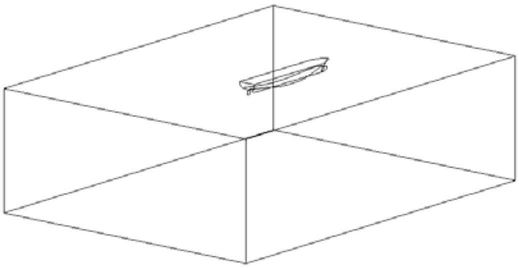


Fig. 1 The computational domain model

Propeller model. In this paper, descriptivebody forcemethod of Hough[8] et al. was used to impose pushing force and torque on the acting zone of propeller. In this method, vortex sources are distributed in paddlesection and wake flow, thus body force meeting boundary conditions of paddle is generated. Thenbody force is imposed on grid in the acting zone of propeller, and solution can be gained in RANSequation.

Brief introduction to “Yu-Kun”. The teaching-training ship of Dalian Maritime University, “Yu-Kun” was set as numerical simulation test object of this paper. Numerous tests have been conducted for “Yu-Kun”on the sea, and thenumerical simulation results of zig-zag test are based on the comparison with zig-zag test data. Main parameters of “Yu-Kun” are presented in Table1:

Table 1 The main parameter of “Yu-Kun”

Main Features	Value	Main Features	Value	Main Features	Value
Length overall(m)	116	Displacement∇(t)	5735.5	Designed draft$d$(m)	5.4
Designed waterline length(m)	106.5	Rudder areaA_R(m^2)	11.8	Propeller diameter D_P(m)	4
Length between perpendiculars(m)	105	Rudder height H_R(m)	4.8	Block coefficientC_p	0.56
Ship width (m)	18	Aspect ratio of rudderλ	1.95	Disk ratioθ	0.7
Molded depth(m)	8.35	Number of rudder blades$Z$	4		

Model and grid

Establishment of hull model and rudder model. When models were established, solid modeling was adopted for both hull and rudder. Hull model and rudder model were established separately. After the models were established, bulk treatment was conducted according to the actualrelative positions of hull and rudder.

Fig. 2 The 3D geometry model of hull and rudder by “Yu-Kun”

The structured hexahedral grid was adopted for partitioning of “Yu-Kun”hull grid and rudder grid. Partitioning of hull grid and rudder grid was treated separately. Thestructured grids shown in Fig. 3 were gained, and the number of hull surfacegrids is 22,540.

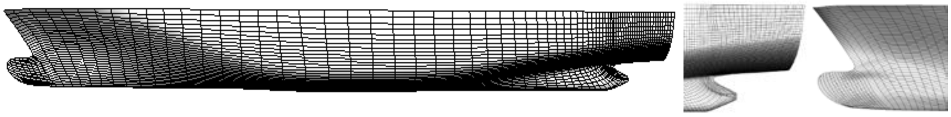


Fig. 3 Girds on hull surface

Structured gridpartitioning of rudder surface is easier when compared with hull, and the number of rudder grids is 2,262. The grids on rudder surface are presented in Fig. 4.

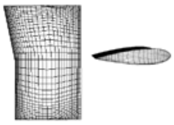


Fig. 4 Grids on rudder surface

Overset grid completed transfer of flow field information through interpolation boundary (pore boundary), so body-fitted grid generation treatment should be conducted for grids on hull and rudder surface. In order to increase the overset grid quality, hull and rudder were made to form O-type body-fitted grids through extruding form. According to the Reynolds number of actual ship, dimensionless distance of grid wall at boundary layer was set as y+=60, dimensionless grid thickness at the first layer of hull surface was set as Δyp=2e-6, the growth rate of grids was set as Ratio=1.25, and there were 45 extrapolation layers. As for body-fitted grids of rudder, there were 41 extrapolation layers, and other parameters were the same with hull settings. The number of O-type body-fitted grids on the hull and rudder is 400,050 and 92,742 respectively, as shown in Fig. 5.

Fig. 5 Body-fitted grids

Structuredhexahedral grid was adopted for grid partitioning of computational domain. In order to better calculate theviscous flow filed around the ship, grid refinement treatment was conducted on bow, stern and free surface, and the number of grids is 727,056, as shown in Fig. 6.

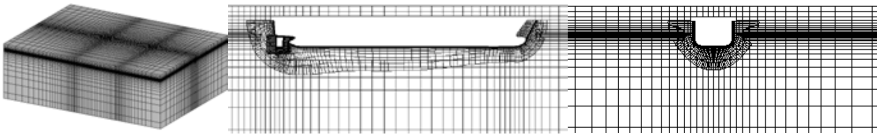


Fig. 6 Computational gridsand Overset grids

After grid generation was completed for hull, rudder and computational domain, oversetting should be conducted for hullbody-fitted grids, rudderbody-fitted grids and computational domain grids.

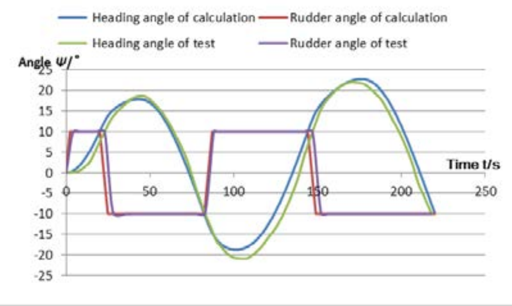
Setting of boundary conditions. The control equation of fluid motion has infinitepossible solutions. For any physical phenomenon, a complete mathematical model should contain control equation and correspondingboundary conditions at the same time. Boundary condition refers to the rule that the solution variable gained at the boundary of solution domain or its first-order derivative changes with position and time. Only by giving reasonableboundary conditions, can the solution of flow field be gained. Therefore, boundary condition is the necessary condition that endows CFD problem with a fixed solution, and all CFD problems have boundary conditions. Boundary conditions of computational domain in this paper are inlet, outlet, wall and open boundaries.

Numerical simulation for zig-zag test

Calculation strategy. Grids after oversetting of hull body-fitted grids, rudderbody-fitted grids and computational domain grids were imported into thesolver, and parameters were set according to the above chapters. Before conducting zig-zag maneuvering test of ship, the ship should reach the speed required by the self-propulsionsimulation at first. Under no-steering situation at self-propulsionsimulation stage, the ship should reach the target speed when propeller maintains a rated speed. At this stage, only surging, heaving and pitching movements were allowed, andthe ship must be in direct movement. As for thecomputational domain, the ship was made to conduct surging, heaving and pitching movements according to literature [10]. Time parameters at self-propulsion stage were as follows: dimensionlesstime stepΔt=0.01 and total time T=15. At self-propulsionsimulation stage of zig-zag test, theship advanced for a distance 9 times more than the ship length and it took 40 hours.

After numerical simulation at self-propulsion stage, the ship speed reached the test value and met the requirements. Afterself-propulsionsimulation stage ended, therudder was controlled according to the rule of zig-zag 10°/10° test, andrudder grids were made to rotate around rudder shaft within -10~10°. In this process, body-fitted coordinate grids of hull and rudder conducted surging, swaying, heaving, rolling, pitching and yawing movements as the whole grid. The setting of computational domain was the same with self-propulsionsimulation stage. Time parameters of zig-zag test were as follows: dimensionlesstime step$\Delta t=0.01$ and total time $T=25$. Zig-zag maneuvering test of ship is a test of evaluating yaw checking ability of the ship, and themaneuvering indexes K and T can be obtained conveniently through the test results. In the process of solving K and T, only results of the first three angles of clearance are required. Therefore, calculation was stopped after working out the first three angles of clearance at steering stage of zig-zag test, and the steering stage took 80 hours.

Comparison and analysis of calculation results . Zig-zag maneuvering test of ship aims to obtain themaneuvering indexes K and T through the relation among heading direction, rudder angle and time, so as to judge the maneuverability of ship. Fig. 7 shows the comparison between numerical calculation results and ship test results of heading direction and rudder angle.



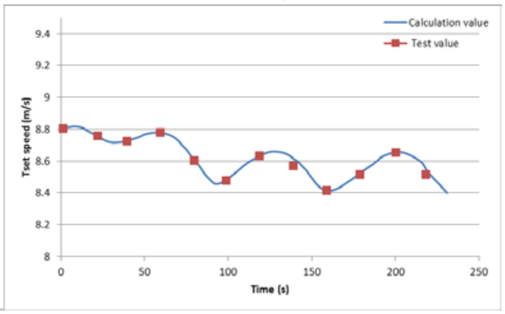
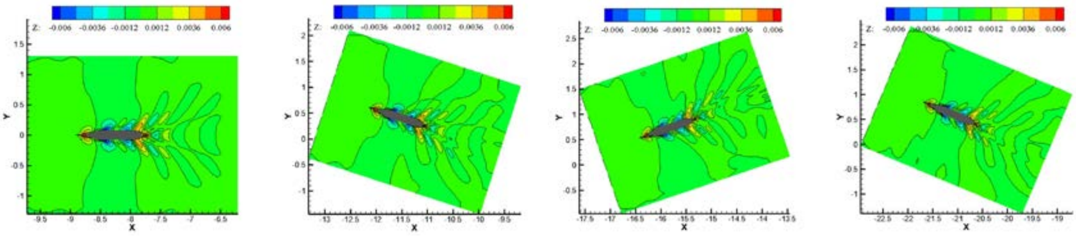


Fig. 7Time histories of the rudder angle and heading angle Fig. 3.2 The variation of the ship speed

According to Fig. 7, theerrors between numerical calculation values and ship test results of angle of clearance, rudder angle and time are lower than 10%, presenting a consistent trend in the overallresults. The precision requirement in engineering is satisfied.However, the angle of clearance gained by numerical calculation is slightly smaller than the actual angle of clearance, and this will be further explored in follow-up error analysis.

In the steering process during zig-zag test of ship, the heading direction was changed and the ship was in oblique motion. Under differentoblique motion states, the ship will undergo different resistances and the propellerefficiencies are also different. As a result, the ship speed changes constantly. The comparison between ship speed in numerical simulation and speed in ship test is presented in Fig. 8.

According to the comparison of ship speed, numerical simulation result and ship test result present the same tendency, and the error is within 10%. The precision requirement in engineering is satisfied. However, in ship test, the ship speed presents a decreasing tendency.



a) Initial position b) Time of the first overshoot angle c) Time of the second overshoot angle d) Time of the third overshoot angle

Fig. 8 Wave patterns

Fig. 8vividly shows wave patterns of the ship in different positions. According to the figure, pressure at bow and stern is the peak value of flow field, and the wave is the most obvious. The wave caused in ship sailing can be seen clearly. The free surface wave patterns are different when the ship is in different positions, which is caused by the ship speed and sailing posture. No comparison is made with the test data, but the wave patterns show that thewave peak and wave valley regions around the hullaccord withthe changing rule.

Conclusion

Calculation of this paper is based on solution of transientRANSequation. SST k-εmodel was adopted as turbulence model, single-phaselevel set method was used for free surface simulation, body force model was adopted as propeller model, and dynamic overset gridtechnology was used to realize 6-DOF motion of the ship. In this way, numerical simulation was conducted for the viscous flow filed in 10°/10°zig-zagmaneuvering test of ship. According to the comparison between numerical results and ship test data, errors of various data are within 10%, which meets the precision requirement in engineering. Through the study of numerical simulation test, the following conclusions are gained:

(1) Selection of turbulence model is of great importance for numerical simulation of ship viscous flow filed, and different turbulence models possess great differences in treating eddy viscosity. Meanwhile, the flow field details are different and the grid quality and arrangement are also different. As it were, selection of turbulence model directly affects the scheme of numerical test. In this paper, SST k-ε model was adopted as turbulence model, and it has calculation characteristics of both near wall region and far wall region. Grids in near wall region are required to be intensive. According to the calculation results, SST k-ε model can catch the details of viscous flow filed around the ship well under the existing computing resource conditions.

(2) Free surface has an important influence on various hydrodynamic performances of hull, so free-surface flow has great significance for the research. Single-phaselevel set method was used to simulate free-surface flow of hull in this paper. According to the numerical simulation results, single-phaselevel set method can accurately simulatefree-surface flow of the ship and clearly catchthe wave of flow field and wave patterns around the hull.

(3) Descriptivebody force method was used as propeller model to replace the actual propeller, and numerical simulation was conducted for theviscous flow filed during zig-zagmaneuvering test of "Yu-Kun" under actual ship size. According to the results, body force method can replacethepropeller in numerical simulation calculation. Efficient and feasible, this method has very high practical value in engineering.

Reference:

[1] Stern F, Toxopeus S, Visonneau M, et al. CFD, potential flow, and system-based simulations of course keeping in calm water and seakeeping in regular waves for 5415M[C]//NATO AVT-189 meeting, Portsdown. 2011.

[2] Carrica P M, Stern F. DES simulations of KVLCC1 in turn and zigzag maneuvers with moving propeller and rudder[C]//Proc. SIMMAN 2008 Workshop on Verification and Validation of Ship Manoeuvering Simulation Methods. 2008.

[3] Goncalves E, Patella R F. Numerical simulation of cavitating flows with homogeneous models[J]. Computers & Fluids, 2009, 38(9): 1682-1696.

[4] Sussman M, Puckett E G. A coupled level set and volume-of-fluid method for computing 3D and axisymmetric incompressible two-phase flows[J]. Journal of Computational Physics, 2000, 162(2): 301-337.

[5] Brewton S, Gowing S, Gorski J. Performance predictions of a waterjet rotor and rotor/stator combination using rans calculations[C]//Proc., 26th Symp. on Naval Hydrodynamics, Rome, Italy. 2006.

[6] Xing T, Carrica P, Stern F. Absolute inertial, relative inertial and non-inertial coordinates for a moving but non-deforming control volume[J].

[7] Pankajakshan R, Remotigue M G, Taylor L K, et al. Validation of control-surface induced submarine maneuvering simulations using UNCLE[C]//24th symposium on naval hydrodynamics, Fukuoka, Japan. 2002.

[8] Carrica PM, Wilson RV, Noack R, et al. Ship motionsusing single-phase level set with dynamic overset grids[J]. Computers &Fluids 2007, 36:1415–1433.

[9] Larsson L, Stern F, Bertram V. Benchmarking of computational fluid dynamics for ship flows: the Gothenburg 2000 workshop[J]. Journal of Ship Research, 2003, 47(1): 63-81.

[10] Carrica P M, Ismail F, Hyman M, et al. Turn and zigzag maneuvers of a surface combatant using a URANS approach with dynamic overset grids[J]. Journal of Marine Science and Technology, 2013,