

ARTICLE

Numerical Simulation of Resistance Field of Hull-Propeller-Rudder Coupling

Hong Xie Baoji Zhang*

College of Ocean Science and Engineering, Shanghai Maritime University, Shanghai, 201306, China

ARTICLE INFO

Article history

Received: 27 October 2020

Accepted: 18 November 2020

Published Online: 31 January 2021

Keywords:

RANS method

Hull-propeller-rudder

Resistance

Numerical simulation

ABSTRACT

Based on the incompressible RANS equation, the KVLCC1 ship's resistance field's numerical simulation is carried out. In this paper, the bare hull (calm water resistance and wave resistance) and hull-propeller-rudder models are studied and compared with the values of the Hydrostatic resistance test. In the hull-propeller-rudder system's performance analysis, the body force method is used to replace the real propeller model. The new calculation domain is set for the hull-propeller-rudder system model and meshed again to obtain the highly reliable numerical simulation results. Finally, the calculation results are analyzed. The research results in this paper can provide technical support for the resistance of similar ship types.

1. Introduction

The commonly used methods for studying the hydrodynamic performance of ships are tank experiments and numerical simulation. Because of the high cost and the long cycle time of the tank experiment, it is difficult for individual researchers. Therefore, most scholars currently use numerical simulation. In recent years, the use of the CFD (Computational Fluid Dynamics) method has been used to study the hydrodynamic performance of ships and has achieved rich research results^[1-2]. There are three types of ship hydrodynamic performance: resistance prediction of the bare hull in still water, performance prediction of ship self-propulsion, and ship-propeller-rudder system performance prediction. Especially, the bare hull's hydrostatic resistance prediction has been studied very maturely, and the error with the experimental value is kept within 3%,

which can meet the requirement of engineering^[3]. Although the performance prediction of ship self-propulsion mainly calculates the propeller's hydrodynamic performance and many researchers^[4], due to the complexity of resistance prediction of the ship-propeller-rudder system, there are few researchers right now. However, the various analyses under the hull, propeller, and rudder interaction are currently a hot topic in ship hydrodynamics.

Due to the mutual coupling, a complicated circumferential flow field will be generated in the waters. On the one hand, the propeller's suction effect will change the flow field at the stern and the pressure on the surface of the hull, thereby increasing the resistance. On the other hand, when the flow field at the stern of the hull changes, it will affect the propeller's thrust. It is very likely to form the cavity, noise, etc. Also, it can change the rudder's side load. Specifically, when the propeller produces an explo-

*Corresponding Author:

Baoji Zhang,

College of Ocean Science and Engineering, Shanghai Maritime University, Shanghai, 201306, China;

Email: zbj1979@163.com

sive wake field for the rudder, its lift will be severely damaged. This paper studies the KVLCC1 ship-propeller-rudder system's resistance performance based on the CFD because of the previous research results. The research results can provide a meaningful reference for the design and calculation of similar ship types.

2. Basic Theory of CFD

The entire flow field uses the continuity equation and the Navier-Stokes equations as the governing equations [5]. It uses the turbulence model, adopts the VOF (Volume of Fluid) method to track the free surface. Besides, the governing equation is discretized by the volume-centered finite difference method. All time-term are used in the Second-order backward difference method; wave generation methods use the boundary velocity method.

Wave equation:

$$\eta = a \cos(kx - \omega_e t) \quad (1)$$

Velocity Field:

$$u(x, y, z) = a \omega_0 e^{kz} \cos(kx - \omega_e t) + U \quad (2)$$

$$v(x, y, z) = 0 \quad (3)$$

$$w(x, y, z) = a \omega_0 e^{kz} \sin(kx - \omega_e t) \quad (4)$$

Where the k is the wavenumber, which depends on the formula: $k = 2\pi/\lambda$; the ω_0 is the Natural frequency of waves. It depends on the recipe: $\omega_0 = \sqrt{2\pi g / \lambda}$.

The establishment of the damping region is necessary for preventing the influence of the reflected wave. The damping model provided by STAR-CCM+ is used to dampen the waves. The length of the damping region is 1-2 times the steady wave-making. In this paper, by adding a damping term [6] at the outlet of the pool to attenuate the wave in the vertical direction, the vertical velocity at the outlet of the numerical wave pool is almost zero to achieve the purpose of wave elimination. The wave dissipation formula is deduced as follows:

$$s_z^d = \rho(f_1 + f_2 |w|) \frac{e^k - 1}{e^1 - 1} w \quad (5)$$

where, $k = \left(\frac{x - x_{sd}}{x_{ed} - x_{sd}} \right)^{n_d}$, x_{sd} is the starting point of the absorbing region, x_{ed} is the outlet boundary of the wave tank, f_1, f_2 and n_d are the parameters of the model, and β is the vertical velocity component.

3. Motion Equation of Ship with 6-DOF

When establishing the ship model's 6-DOF motion equations, it is necessary to establish two reference frames: one is the fixed coordinate system, $O_o X_o Y_o Z_o$,

and the other is the follow-up coordinate system GXYZ fixed on the hull. As shown in Figure 1, the origin of the moving coordinate system is at the center of gravity of the ship, in which the Gx , Gy , and Gz are the intersections of the midship section, the longitudinal section in the center plane, and the waterplane passing through the center of gravity, respectively. Follow-up coordinate system, the X-axis is positive for the bow, the Y-axis is positive for the starboard side and the Z-axis is positive for the downward direction [7]. The STAR - CCM + FDB module activates the heaving and pitching, to complete the ship motion simulation performance.

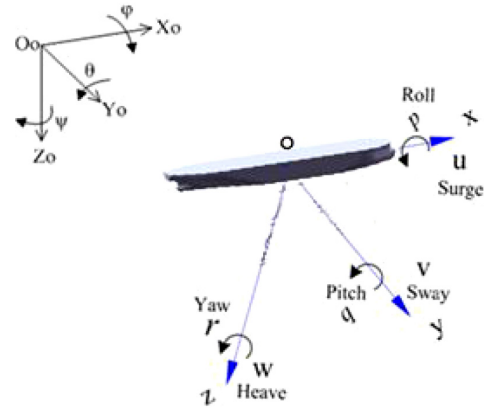


Figure 1. The fixed coordinate system and the follow-up coordinate system

$$\frac{dB}{dt} + \Omega \times B = F \quad (6)$$

$$\frac{dK}{dt} + \Omega \times K + U \times B = M \quad (7)$$

Where the B , Ω , F , K , U , and M , in turn, are the moment of the resultant force, ship's momentum, angular velocity, external force, the moment of momentum, and ship speed, respectively.

4. KVLCC1 Ship Hull

The KVLCC1 ship model is the standard ship type in the international symposium SIMMAN2014. Compared with the ship types such as Wigley and KCS, its surface is more complicated. To compare the numerical simulation results with the experimental values, the scale ratio used in the model is 64.386 (compared with the actual ship model). The 3D model is shown in Figure 2, and the concrete parameters are shown in Table 1.

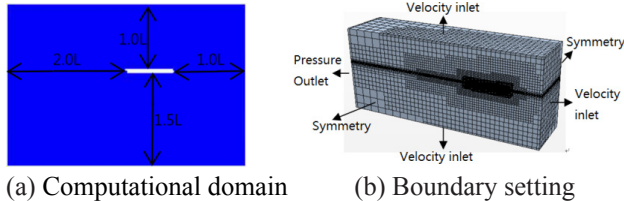


Figure 2. 3D model of KVLCC1 naked hull

Table 1. The main parameters of the KVLCC1 ship model

Parameters	Numerical	Parameters	Numerical
The length between perpendiculars(m)	4.97	Block coefficient, C_b	0.8099
Molded breadth, B(m)	0.901	Froude number, F_n	0.142
Draft, Td(m)	0.323	Speed, Vm(m/s)	0.994
Wet-surface area, S(m ²)	6.589		

The computational domain and boundary of the KVLCC1 ship model are shown in Figure 3. Among them, the inlet is defined as the velocity inlet; the outlet is defined as the pressure outlet. Also, the two sides are set as the plane of symmetry. The other positions are set as the velocity inlet; the hull is a no-slip wall. In order to save computing resources, only half of the hull is simulated.

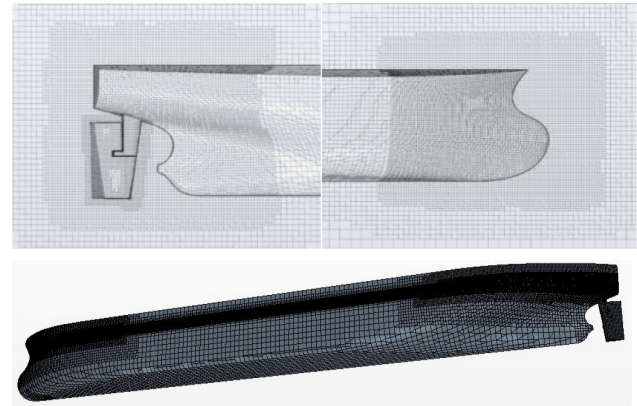
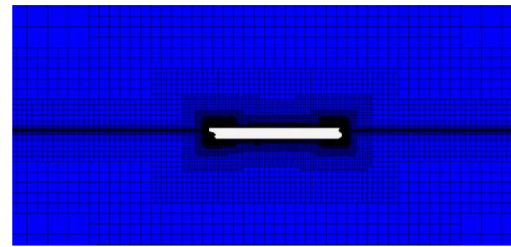
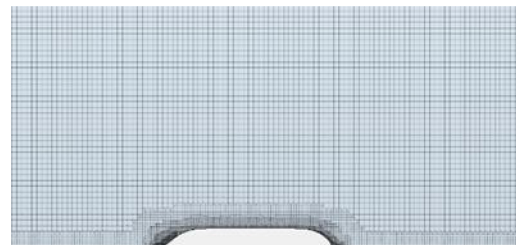
**Figure 3.** The computational domain and boundary conditions

For the computational domain grid, set the raw size to 0.1m, the prism layers number is 6, and the absolute thickness values to be 0.002m. The thickness of the boundary layer grid's height value mainly depends on the Reynolds number. When dividing the grid, the parameter y^+ refers to the first boundary layer grid, which is generally controlled at about 6.25 to 50. It is obtained from the dimensionless local Reynolds number in the near-wall region. The estimating formula is:

$$y^+ = 0.172 \left(\frac{y}{L} \right) \text{Re}^{0.9} \quad (8)$$

Mesh generation is an essential part of numerical simulation. The quantity and quality of the grid will significantly affect the time and results of the numerical simulation calculation. Furthermore, the mesh's quality plays a decisive role in calculation accuracy. To better simulate the hull's motion on the waves, this paper adopts the chimera grid technology that comes with Star-ccm to establish the background (fixed part) grid and overlapping (moving part) grid area, respectively. The background grid is relatively sparse, and the overlapping regions are denser. Overlapping and background grids must be excessive in a particular proportion. The free surface must also

ensure that there are at least 80 grids within a wavelength and at least 20 grids within a wave height. Taking into account computing time and machine location, the final number of grids generated is 2.49 million. Figure 4 shows the mesh generation of the hull and rudder's surface and the mesh refinement of the bow and stern on the ship. From the figures, it can be seen that the grid near the free surface gradually becomes sparse outward from the hull. To capture the free surface more accurately, the water surface also needs to be refined, as shown in Figures 5, 6.

**Figure 4.** Hull meshing**Figure 5.** Free surface and boundary layer meshing**Figure 6.** The grid of water surface

5. Hydrostatic Resistance Calculation

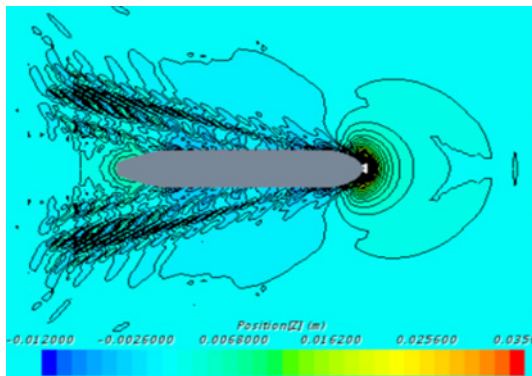
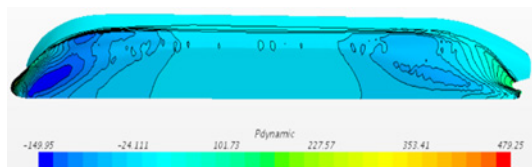
When $F_n=0.142$, the iteration number reaches 1249, so at 5s, the pressure curve stabilizes. The iteration number from 14492 to 17,489, the resistance change rate is less than 5% and stable at 6.612N (only half of the models are selected in the simulation; the entire ship's resistance must be multiplied by 2). Finally, the total number of iterations

Table 2. Comparison of calculation values with experimental values (N)

F_n	Result	Pressure drag	Frictional resistance	Total resistance	Deviation between total resistance and experimental value
0.142	Calculated value	1.023	12.20	13.224	-4.64%
	Experimental value	-	-	13.867	

is 24,735. This simulation uses an eight-core workstation to calculate, which is about wall clock time 12 hours. Compare the calculation results with the experimental results^[8], as shown in Table 2. It can be seen from the table that the calculation result is relatively close to the experimental value, indicating that the reliability of the simulation result is higher.

Figure 7 is the oscillogram of the free surface of the bare hull. It can be seen from the oscillogram that the stabilized waveform shows the shape of the Kelvin wave. Figure 8 shows the pressure distribution on the surface of the bare hull in still water. According to Figure 8, it can be seen from the figure that the pressure in the stagnation area formed at the bow decreases from the maximum. Also, vortices are generated at the bilge and shoulder, which causes the pressure to decrease rapidly and forming two low-pressure regions. There was almost no change in pressure in the mid-hull area. In the middle and rear part of the hull, the pressure first drops until it rises to a certain extent at the stern, mainly due to shoulder waves and vortices' influence.

**Figure 7.** The oscillogram of the free surface of the bare hull**Figure 8.** The pressure profile of the free surface of the bare hull

6. Wave Resistance

The calculation conditions of wave resistance are shown in Table 3. Compared with the drag convergence curve in still water. When the calculation time step is set the same, the time required for the bare hull model's resistance to reach convergence in the first-order regular wave becomes significantly longer. Similarly, when the number of iterations is 2501, that means the physical time is about 10s. At this time, it tends to be stable. Then, when the iteration number is between 5096 and 6019, the value change range is less than 5% and finally stabilizes at 7.86N. The time history curve is shown in Figure 9. Comparing the value with the resistance of the bare hull in still water and the experimental data, as shown in Table 4, it can be found that the resistance in the first-order regular wave is increased by 13.36% compared with the resistance of the bare hull in still water. As can be seen that waves have a significant influence on hull resistance.

The wave added resistance is equal to wave resistance minus the calm water resistance,

$$R_{aw} = R_W - R_T \quad (9)$$

Where the R_{aw} , R_W and R_T are the wave added resistance, the wave resistance and calm water resistance, respectively.

The dimensionless expression of the wave added resistance is:

$$C_{aw} = \frac{R_{aw}}{0.5\rho SV^2} \quad (10)$$

Where the ρ , S and V are the fluid quality density, the hull wet surface and speed, respectively.

Table 3. Calculation conditions

Parameters	Numerical
Froude number, F_n	0.142
Wave steepness, ak	0.0109
Wavelength, $\lambda(m)$	2.485
Wave height, $H(m)$	0.0487
Frequency, $\nu(Hz)$	0.2234

Table 4. Comparison of wave resistance with experimental values (N)

F_n	Results	Hydrostatic resistance	Wave resistance	Deviation between Hydrostatic resistance and Wave resistance
0.142	Calculated value	13.224	15.720	13.36%
	Experimental value	13.867	—	

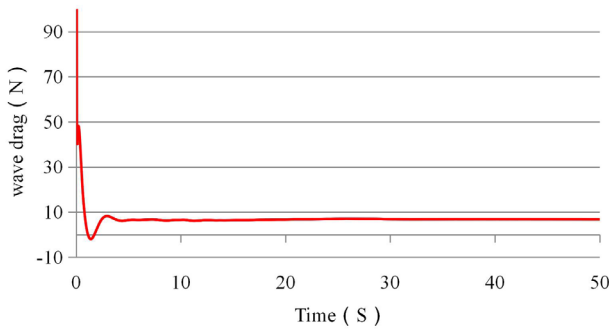
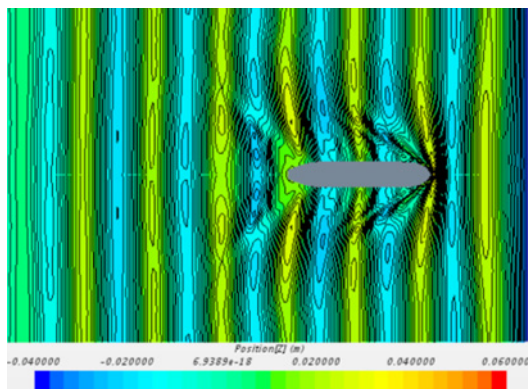
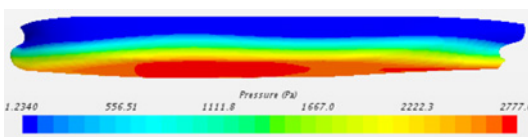
**Figure 9.** The resistance time history curve

Figure 10 shows the oscillogram of the free surface of the bare hull in waves. This oscillogram shows that the incoming current flows from the bow first, so the peak is formed at the bow. Near the middle of the ship, the change of the waveform is relatively stable. At the stern, a ship wave forms and gradually dissipates as the stern moves backward. It can be seen from the oscillogram that under the action of regular waves, the oscillogram of the free surface formed around the hull has slightly changed, which is not very different from the oscillogram in still water.

**Figure 10.** The oscillogram of the free surface of the bare hull in waves**Figure 11.** The pressure profile of the free surface of the bare hull

According to Figure 11, the peak pressure appears near the bow and generally shows a decreasing-increasing-decreasing-increasing trend along the length of the boat. Among them, the pressure attenuates sharply at the bow and creates two low-pressure regions at the bow. Besides, this is the lowest pressure region of the entire hull in the waves. When it is close to the parallel middle body, it rises again. At the rear end of the parallel middle body near the stern, it first reduces the pressure value close to the parallel middle body. Then gradually increase along the stern. In general, speaking, the pressure distribution of the overall ship is not particularly noticeable.

7. Resistance Calculation of Hull-propeller-rudder System

7.1 Calculation Principle of Body Force Method

The VLM (Vortex Lattice Method), based on the potential flow theory, generates the body force that meets the requirements of the edge of the propeller blade^[9]. It assigns the body force to the propeller's grid area, that is, it adds thrust and torque to replace the actual load on the surface of the propeller. The formula of the applied volume force source term is as follows:

$$F = \iiint_V f b dV \quad (11)$$

$$f b_x = \frac{\rho U^2}{L_{pp}} A_x r^* \sqrt{1 - r^*} \quad (12)$$

$$f b_\theta = \frac{\rho U^2}{L_{pp}} A_\theta \frac{r^* \sqrt{1 - r^*}}{(1 - Y_H) r^* + Y_H} \quad (13)$$

$$\text{Where, } r^* = \frac{r_1 - R_H}{R_p - R_H}, Y_H = \frac{R_H}{R_p},$$

$$A_x = \frac{C_t}{\Delta x}, \frac{105}{(4 + 3Y_H)(1 - Y_H)}$$

$$A_\theta = \frac{K_q}{J^2 \Delta x}, \frac{105}{\pi(4 + 3Y_H)(1 - Y_H)}$$

Where r_1 represents the distance from any point in the propeller area to its axis; Y_H is the hub diameter ratio, that

is to say, R_p and R_H are the radii of the propeller and the radius of the hub, respectively; C_t is the dimensionless thrust coefficient; K_Q is the dimensionless torque coefficient.

7.2 Modeling

The Body Force Method (BFM) is used to simulate the propeller. First, a virtual static disk model is created based on the specified propeller curve. The thrust coefficient K_t , torque coefficient K_Q , process ratio J , and open water efficiency η are all derived from open water tests of the KVLCC1 ship's propeller in SIMMAN 2008. Secondly, determine the size and position of the virtual disk, and define the direction of the disk axis. Next, set the inflow velocity plane's properties. Make the velocity plane radius more significant than 10% of the virtual disk radius, and offset the velocity plane to 10% of the virtual disk diameter. Finally, set the virtual disk's rotation rate to be the rotation speed of the propeller in the open water test, which is 8.5rad/s. As shown in Figure 12.

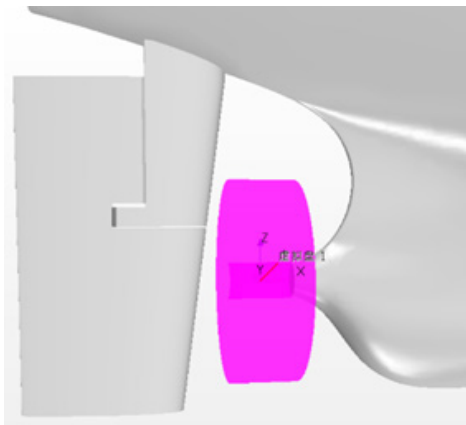


Figure 12. Propeller's action region

7.3 Analysis of Calculation Results

Due to the large number of grids in the entire calculation, it takes about 40 hours in total to use an eight-core workstation. Comparing the results with the hydrostatic test. In Table 5, after considering the propeller rotation, the resistance of the ship-propeller-rudder system has increased by 13.49%. This shows that the propeller still has a significant influence on the hull resistance. Therefore, its

effect needs to be considered when calculating the actual resistance.

Figure 13 shows the oscillogram of the free surface of the ship-propeller-rudder system. From the oscillogram, the free surface presents a clear Kelvin wave, which reflects the reliability of the numerical simulation. The streamline diagram of the surface of the hull and rudder is shown in Figure 14. The streamline diagram shows that when the water flows through the propeller's action region, some streamlines are shifted because of the suction role of the propeller.

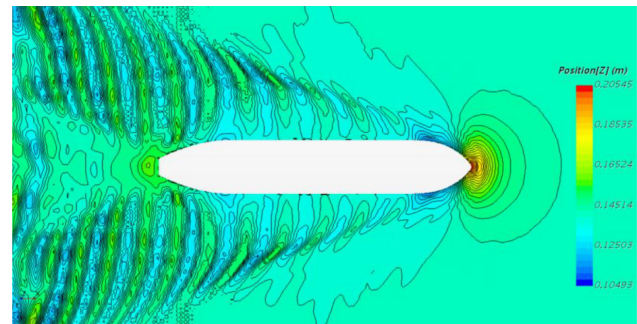


Figure 13. The oscillogram of the free surface of the ship-propeller-rudder system

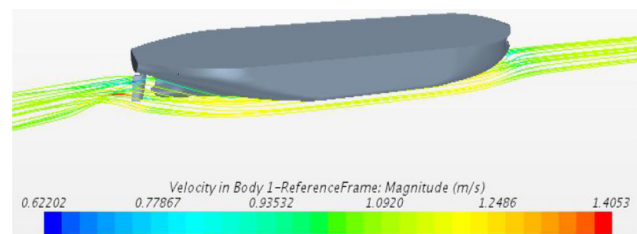


Figure 14. The streamline diagram of the surface of the hull and rudder

To further obtain the force condition of the ship-propeller-rudder system in the fluid, this paper sets four groups of different working conditions for calculation, $Fr=0.142$, as shown in Table 6. Analyze and use the data in Table 6 to draw Figure 15. It can be seen from the curve that the zero of the curves indicates that the net resistance is zero, which means that the resistance of the hull is balanced with the thrust generated by the propeller. If the shipping speed reaches 0.99m/s, the propeller's rotational speed needs to reach 55r/s.

Table 5. Comparison of resistance results (N)

Fn	Research object	Calculated value	Experimental value (Naked hull)	Deviation between the calculated value and experimental value
0.142	hull-propeller-rudder	36.84	31.87	13.49%

Table 6. The net resistance of the ship-propeller-rudder system (N)

Conditions	Rotational speed (r/s)	Propeller thrust	Resistance	Net resistance
1	20	5.5	-51.5	-46
2	40	24.5	-50.5	-26
3	60	59.5	-49.5	10
4	80	99.5	-48.5	51

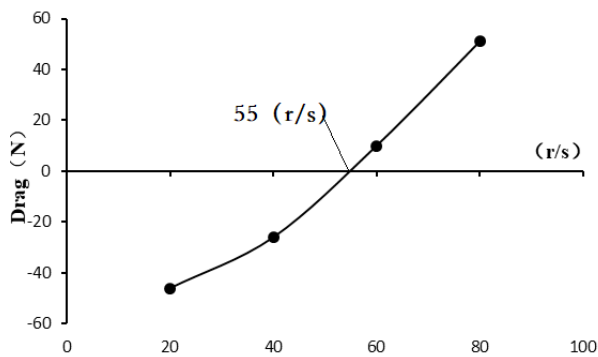
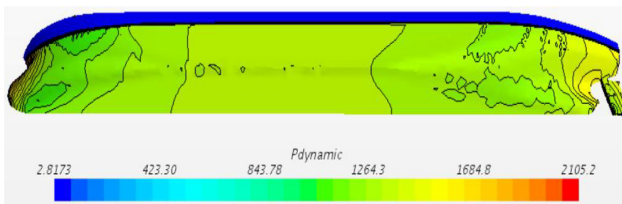
**Figure 15.** The fitting curve of net resistance of the ship-propeller-rudder system**Figure 16.** The pressure profile of the free surface of the ship-propeller-rudder system

Figure 16 shows the pressure distribution on the surface of the entire ship-propeller-rudder system during direct navigation. As shown from the pressure profile, since the rudder angle is not set, the hull surface pressure is symmetrical. There is a high-pressure area at the bow, mainly because it first contacts the inflow and forms a layered decrease along with the entire bulbous bow. Additionally, in the middle and front part, the shoulders on both sides of the hull and the bilge have a small range of low-pressure area, and the pressure in the entire parallel middle body is the same. The pressure at the rear end of the parallel midbody begins to decrease again until the pressure on the stern and the rudder's surface begins to increase gradually, and the pressure at the bottom edge of the rudder gradually decreases. This pressure change leads to the generation of vortices.

8. Conclusion

Based on the CFD theory, the calm water-resistance of the KLVCC1 ship was studied. Firstly, the reliability of the numerical simulation was verified by comparison with experimental values. Then the Stokes wave numerical wave tank was established to simulate the ship's resistance in waves. Finally, based on the body force model, the resistance of the ship-propeller-rudder system is studied. To sum up, the comparison with the static water experimental value proves that rudder and propeller's effect must be considered in the research of resistance through numerical simulation.

References

- [1] Ketut Suastika, Fajar Nugraha. Effects of Parallel-Middle-Body Relative Length and Stern Form on the Wake Fraction and Ship Resistance[J]. *Applied Mechanics & Materials*, 2017, 862: 278-283.
- [2] Andrea Farkas, Nastia Degiuli, Ivana Martić. Assessment of hydrodynamic characteristics of a full-scale ship at different draughts[J]. *Ocean Engineering*, 2018, 156: 135-152.
- [3] Yang Cai Hong, Han Duan Feng, Liu Feng. CFD numerical simulation of resistance prediction for large sea going ships[J]. *Water transportation in China*, 2017, 17(1): 80-81.
- [4] Du Yun Long, Chen Waei Min, Dong Guo Xiang. CFD calculation strategy of static water resistance of typical oil tanker model[J]. *Journal of Jiangsu University of science and Technology (Natural Science)*, 2017, 31(5): 661-665.
- [5] Mingyu K, Olgun H, Osman T. Numerical studies on added resistance and motions of KVLCC2 in head seas for various ship speeds. *Ocean Engineering*, 2017, 140: 466-476.
- [6] Choi, Junwoo, Sung, Bum Yoon. Numerical simulations using momentum source wave-maker applied to RANS equation model[J]. *Coastal Engineering*, 2009, 56(10): 1043-1060.
- [7] Longo J, Stern F. Uncertainty Assessment for Towing Tank Tests with Example for Surface Combatant DTMB Model 5415. *Journal Ship Research*, 2005, 49(1): 55-68.
- [8] Jinbao Wang, Hai Yu, Yuefeng Zhang, et al. Numerical simulation of wake flow field and prediction of resistance performance for low speed and large ship[J]. *Research and development of hydrodynamics*, 2010, 25(5): 648-654.
- [9] Jeon M, Lee S, Lee S. Unsteady aerodynamics of offshore floating wind turbines in platform pitching motion using vortex lattice method. *Renewable Energy*, 2014, 65(5): 207-212.