Numerical simulation and analysis of nominal wake of displacement ship

Xiang Li1,*, Jinshuang Dong1 and Shuai Sun2

1College of civil Engineering and Architecture, Hainan University, Haikou, China
2College of Shipbuilding Engineering, Harbin Engineering University, Harbin, China

*E-mail: lixiang2180@163.com

Abstract. In order to analyze the performance of ship resistance and nominal wake of container ship and bulk carrier, the method of CFD was used to compare the influence factors such as the distribution pattern of grids and turbulence models, and the calculation results were compared with experimental values. The predicted results show that the ship resistance calculated by the model of Volume of Fluid achieves a good agreement with experimental data. An optimal scheme for computing the nominal wake was got by analyzing the results of three grid schemes and two turbulence models. The container ship’s axial nominal wake, tangential nominal wake and radial wake has the same variation trend with the experimental data, so it validates the feasibility of this method set in this paper. Then nominal wake of a bulk carrier took as an example was computed and the “hooked” wake was also got very well. At the same time, the cause of the “hooked” wake was got a good analysis.

1. Introduction

Wake of the Ship is divided into effective wake and nominal wake. Because of the effective wake cannot be measured by test currently, the study of propeller’s performance and wake adapted design is mainly according to the nominal wake. Different types of ship geometry, the wake also has the different characteristic. Several numerical value forecasts on the hydrodynamic performance of the benchmark ship such as container ship KCS, full-formed ship KVLCC2 and ship DTMB5415 were carried out in Goteborg conference of 2000, Tokyo conference of 2005 and the Goteborg conference of 2010, and the research on the wake distribution of the propeller disk was become as a focus [1].

The survey found that numerical method which is applied to forecast model scales of wake field in the world has been made important progress [2-6], and the simulation of ship stern flow field and the total resistance prediction also has made considerable progress in home[7-10]. In this paper, CFD technology was used to make numerical simulation analysis for the wake flow field of a container ship, KCS ship and a bulk carrier. First, CAD software was used to build geometric models, ICEM CFD was used to build up the flow region according to the characteristics of the flow field, and discrete the flow field space by unstructured and structured mixed grids. By comparing the calculation results with experimental data of KCS ship resistance under different speed to verify the feasibility of the numerical method. Then, in order to get the plan which best consistent with the experimental results, the nominal wake field of a container ship was calculated by using of a single-phase flow model, and the influence of grid layout and turbulence model on the calculation results was analyzed. The
reasonable calculation plan was used to simulate and analysis the formation of the wake contours named "hooked" of a bulk carrier.

2. Basic theory
Hydrokinetic governing equations contain a series of differential equations such as continuity equations, Navier-Stokes equations, and energy conservation equations. For liquids such as water in which the propeller rotation is incompressible and the density is constant considering that there is no heat exchange, the continuity equations can be written as

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0$$

(1)

Navier-Stokes equations can be written as:

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}(\mu \frac{\partial u_i}{\partial x_j} - \rho u_i u_j)$$

(2)

where $u_i, u_j$ are time-average velocity ($i, j = 1, 2, 3$), $P$ is the time-average pressure, $\rho$ the density, $\mu$ the viscosity coefficient, and $-\rho u_i' u_j'$ the Reynolds stress item which is unknown. The equations are not closed and a new turbulence equation is added into the system.

3. Establishment of the model
In order to validate the accuracy of the flow field of the ship which is worked out by using the method of this paper, the KCS ship was chose as a study object. Taking the hull resistance as the comparison standard, the consistency of the resistance calculation value and experimental value was checked, and the influence of free face to the accuracy of the calculation results of the displacement ship's hull resistance was investigated. VOF two-phase flow model was adopted when the free face was considered, and single-phase flow model was chosen if ignoring the influence of free face. The RNG $k - \varepsilon$ turbulence model was selected in both conditions.

ICEM CFD software was used to drawn hull geometry model according to the hull lines. Assume that the ship sailing in the infinite water area, the distance between the stem and the water entrance is equal to the overall length, and the distance between the stern and the exit is equal to 3 times the overall length. The depth of the water area is equal to the overall length, and the width of water area is twice the overall length. Because of the curvature of the fore and aft hull changed greatly, the grid of the two regions should be meshed as unstructured, so the whole water area is divided into three parts. Structured grid is selected in hull water area, and unstructured grid stem water area and stern water area. In order to ensure the convergence of solutions, the grid regular degree needs to be more than 0.25. Figure 1 shows the grid map and the grid regular test results of stem water and stern water, from which we can see that the regular degree can satisfy the requirements of computational grid.

![Figure 1](image-url)
In table1, it shows the relative parameters of flow field grid. The range of \( Y^+ \) controlled under 60 is reasonable [11]. Considering that the follow-up work focus on the stern flow field problems, the grid’s size of the stem can be reduced in guarantee under the premise of \( Y^+ \) range to save computation time.

4. Feasibility verification of the numerical methods
The FLUENT software was used to compute the hull resistance respectively when the free-water surface is considered (VOF) or not (single phase flow), and the calculated results were compared and analyzed with experimental data[12] in figure 2. It shows that the VOF calculation results agree well with the experimental values, the error is less than 2\%, which means that using this method to calculate ship resistance with good accuracy. Due to ignore the free-water surface, single phase flow has different simulation status with experiments, and the calculation results deviation slightly larger.

![Figure 2](image)

**Figure 2.** The analyze of resistance calculation value.

Because of the hull wave to the influence of the wake fields mainly due to water points of propeller disk place do circular motion, and this motion is diminished as exponent with the increase of the depth of the water. If the Froude number of the ship is lower, the ship wave’s amplitude is smaller, so the influence of wave on the propeller disk is smaller[13]. And that propeller disk is not below the free surface of general displacement ship, but the stern plate, so hull wave to the influence of the wake is smaller. In table 2, it shows that the convergence time of VOF two-phase flow model is about 10 times of single-phase flow model. From what has been discussed above, single-phase flow model is chose to compute the nominal wake field.

| Fluid Zone | Stem Fluid Zone | Hull Fluid Zone | Stern Fluid Zone |
|------------|----------------|----------------|-----------------|
| Grid Form  | Unstructured grid | Structured grid | Unstructured grid |
| Grid Number | 0.42million | 2.03 million | 1.08 million |
| \( y^+ \) Scope | 48-110 | 20-50 | 20-50 |

### Table 1. Watershed hydrology parameters.

| Calculation Method | Grid Number | Computer | Time Step | Number of iterations per hour | To achieve convergence of iterations | Consumed time (hours) |
|-------------------|-------------|----------|-----------|-------------------------------|-------------------------------------|-----------------------|
| VOF               | 3.53 million | parallel computing in 8 cores CPU | 0.001s | 600 | About 80000 times | 133 |
| Single-phase flow | 3.07 million | parallel computing in 8 cores CPU | - | 800 | About 1100 times | 13.75 |

5. Analyses on influence factors of nominal wake field calculation precision
When using CFD for numerical simulation, the calculation precision is affected directly with many factors, such as whether the grid layout reasonable or not, whether the grid density enough or not and
whether the choose of turbulence model appropriate or not[14-17]. Therefore, it is necessary to analysis the above influence factors, and to get an optimal numerical simulation project.

5.1 Influence of the grid layout form

In this paper, one container ship chose as the research object is used to calculate the nominal wake field, whose main parameters are shown in table 3. The establishment of geometric model and the meshing process is same to the ship KCS. Because of main focus on the flow field data is in the stern, so it needs to put special emphasis on the layout and density of grids in the stern area. According to the characteristics of ship flow field, three kinds of grid layout forms were discussed mainly in this paper, whose main parameters and layout forms were shown in table 4 and figure 3. The regular tetrahedron grid is used to fill the stern domain in scheme 1, and only the grid size closed to the hull surface is smaller. A body of density between the propeller disk and stern is added in scheme 2, which makes the region grid size small. Eight prism boundary layers are added in the stern surface, but the other area grid size same as scheme 1.

![Scheme 1](image1)
![Scheme 2](image2)
![Scheme 3](image3)

**Figure 3.** Three grid plan diagrams.

| Name | Length between perpendiculars | Length of waterline | Molded breadth | Draft | Scale ratio | Computing speed |
|------|--------------------------------|---------------------|---------------|-------|-------------|----------------|
| Symbol | LPP | LWL | B | T | \(\lambda\) | V |
| Value | 149.5m | 153.5m | 22.6m | 8m | 24.7857 | 12.5kn |

| Scheme | Description | Grid Number | \(y^+\) range |
|--------|-------------|-------------|---------------|
| 1      | Added regular tetrahedron grid in the stern domain | 1.08 million | 20-70 |
| 2      | Added a body of density between the propeller disk and stern, divided into tetrahedron grid | 3.03 million | 20-62 |
| 3      | Added 8 Prism boundary layers on the stern surface, the thickness of first layer is 0.0005m, the growth rate is 1.2 | 2.07 million | 20-50 |

In figure 4, it presents the calculation results of axial wake in the three schemes and the experimental values. Through comparing the calculation results with the experimental values, it shows that the grid layout form has great effect on the calculation accuracy. The calculation accuracy is the worst by scheme 1 which can’t simulate the hooked wake distribution. The hooked wake distribution can be simulated by scheme 2, but it’s not obvious and the calculation results of axial wake faction in inner radius are wrapped greatly with the experimental values. The hooked wake distribution appears obviously by scheme 3 whose grid number is less than scheme 2. Except the area around 180 degree in inner radius, the calculation results of axial wake fraction is consistent with experimental values well. Comparing with the other two schemes, the most distinguish is adding the prism boundary layers in scheme 3. Due to add the prism boundary layer, the distribution of value becomes uniformity in the hull surface, which can make the numerical simulation of the flow near the wall more accurately.
Therefore, scheme 3 has the advantage over the other schemes, which will be adapted in follow-up calculation.

Figure 4. Calculation results of different grid schemes and experimental value.

5.2. Influence of turbulence model
The SST k-ω and RNG k-ε turbulence model used widely in the numerical simulation of ship resistance performance are analysed emphatically in this paper. Through comparing the calculated value of axial wake fraction of the two turbulence models respectively with the experimental value on the radial position of r/R=0.7 and r/R=0.8 in figure 5, RNG k-ε turbulence model can simulate the fluctuate of the axial wake fraction in the range of 20° to 50°, better than SST k-ω turbulence model, and the change trend of wake is more consistent with experiment value in the whole propeller disk. Therefore, RNG k-ε turbulence model is chose in the following calculation.

Figure 5. The comparison of calculation results in different turbulence models.
6. Analysis of nominal wake field’s features

6.1. The wake of three directions

Ship’s nominal wake can be divided into axial wake, radial wake and tangential wake. Among them, the axial wake has the dominant influence on propeller performance of hydrodynamic, cavitation, noise and vibration force. So, the axial wake is chosen to instead of the nominal wake in the model test and numerical simulation at the present stage. Then, it will be simple to operate when predicting the performance of propeller. But this method is still insufficient in theory, and the accuracy of the results needs to be discussed. So, in order to provide a complete nominal wake field for reference when computing the propeller’s performance in nominal wake and designing the new propeller according to the nominal wake, the radial wake and tangential wake are also extracted and analysed during analysing the axial wake in the propeller disk.

In figure 6, it shows axial wake fraction distributions in several typical radiiuses, and we can see that CFD calculation results are almost accorded with the experimental values and just have slightly deviation in the inner radius. The cause of error may be due to the curvature of the stern changes is very big, so certain smooth processing need to be adopted when using CFD to set up geometric model which exists tiny difference in the shape compared with the test model. For the axial wake’s inaccurate problem in the inner radius area around 180 degrees, in the follow-up study, the axial wake distribution within the inner radius will be simulated more accurate through further improving the geometric model and grid of the stern domain.

In figure 7, it shows calculation results and experimental values of the velocity vector distribution of radial wake and tangential wake in the whole propeller disk. And figure 8 shows calculation results and experimental values of the circumferential distribution of the radial wake and tangential wake in r/R = 0.7, r/R = 0.8. After analyzing the comparison results, we can obviously find that the experimental value and calculation results maintain good consistency. The radial and tangential velocity at the propeller disk has obvious bilateral symmetry. Near the intersection between propeller disk and the ship longitudinal section, the dominant velocity is radial velocity vector, whose direction is opposite to the specified direction. The tangential velocity component whose value is zero in longitudinal section is very small. The bottom half of the propeller disk has obvious radial velocity component, but it decreases obviously in the upper part of the disk.

Figure 6. Axial wake fraction.
Figure 7. Radial and tangential flow vector distributions.

Figure 8. Calculation results verify of radial and tangential wake fraction.

Figure 9. The axial wake on propeller disk and the vortices of the stern section.
Table 5. Main parameters of a bulk carrier.

| Name             | Length between Perpendiculars | Waterline Length | Molded Breadth | Draft | Scale Ratio | Computing Speed |
|------------------|-------------------------------|------------------|----------------|-------|-------------|-----------------|
| Symbol           | LPP                           | LWL              | B              | T     | λ           | V               |
| Value            | 172m                          | 175.9m           | 28.6m          | 10.2m | 27.1346     | 12kn            |

6.2. "Hooked" wake

Hooked wake can be obtained in the propeller disk of low speed fullships [16]. One bulk carrier is chosen to study the "hooked" wake, whose main parameters are shown in Table 5.

In figure 9, it shows that the axial wake velocity contour map clearly display the "hooked" wake on the propeller disk, which is consistent with actual situation. In order to analyze the generating process of "hooked" wake, the vortices of some stern sections near the propeller disk are worked out. For ship model, the propeller disk locates in the x = 0.136 section, and the stern lines begin to shrink in the section had 0.2 meters away from the propeller disk. From figure 9, it shows that the bilge vortex didn’t generate in the x = 0.336 section, which begins to generate obviously in the x = 0.286 section, and the bilge vortex gradually expanded along with the direction from stem to stern, until to the propeller disk, forms the so-called "hooked" wake. So it is obvious that the "hooked" wake is formed when the bilge vortices generated by the stern lines of low speed full ships shrink acutely enter the propeller disk with water flow.

7. Conclusions

In this paper, CFD technology is used to make numerical simulation analysis for the wake flow field of a container ship, KCS ship and a bulk carrier. Through the comparison with the test results and analysis according to the related theories, some conclusions can be drawn as follows:

(1) The requirement of the grid layout is not very high when calculating the resistance of naked hull separately. The resistance calculation results of VOF multiphase flow model are more consistent with the experimental values than the calculation results of Single-phase flow model.

(2) By comparing the nominal wake field calculation results and test values, it finds that the scheme of adding boundary layers and combining RNG $k-\varepsilon$ turbulence model can obtain the nominal wake distribution more reasonable and accurate.

(3) To single-propeller displacement ships, the radial and tangential velocity at the propeller disk has obvious bilateral symmetry. The main velocity vector near center longitudinal section is radial velocity vector, the tangential velocity is very small, and the bottom half of the propeller disk has more obvious radial velocity component than the upper part of the disk.

(4) The method described in this paper can well simulate the "hooked" wake field of low speed full ships. By the analysis, it shows that the "hooked" wake is formed when the bilge vortices generated by the stern lines of low speed full ships shrink acutely enter the propeller disk with water flow.

Acknowledgements

The research was financially supported by the National Natural Science Foundation of Hainan (Grant No. 519QN189) China.

References

[1] Yu H., Wu Q., Feng X. M., Wang J. B and Cai R. Q.: Numerical simulation on Vicious Wake Field around Various Ships, In: Proceedings of the 23th National Conference on Hydrodynamics and the tenth National congress on Hydrodynamics, Xi An, China, 2011.

[2] REGNSTROM B. and BATHFIELD N.: Drag and wake prediction for ships with appendages using an overlapping grid method. 26th Symposium on Naval Hydrodynamics, Rome, Italy, 2006, pp. 16-20.

[3] Wu C. S., Qiu G. Y., Yan D. J. and Chang L.: Research on the characteristics of wake field for a
ship advancing in regular head short waves, Journal of Ship Mechanics, Vol. 18, No 1-2, 2014, pp. 54-61.

[4] Lübke L. O. and Mach K. P.: LDV measurements in the wake of the propelled KCS model and its use to validate CFD calculations, In: 25th Symposium on Naval Hydrodynamics, Newfoundland, Canada, 2003, pp. 10-14.

[5] Duranted D., Dubbioso G. and Testa C.: Simplified hydrodynamic models for the analysis of marine propellers in a wake-field, Journal of Hydrodynamics (B), Vol. 25, No 6, 2013, pp. 954-965.

[6] Golbraikh E., Eidelman A. and Soloviev A.: On helical behavior of turbulence in the ship wake, Journal of Hydrodynamics (B), Vol. 25, No 1, 2013, pp. 83-90.

[7] Shen H. L. and Su Y. M.: Study of mesh partition methods for numerical simulation of flow field of full form ships, Proceedings of Ship hydrodynamics conference, Hang Zhou, China, 2008, pp. 91-103.

[8] Zhao F., Zhang Z. R. and Chang Y.: Practical application of CFD in wake simulation of a hull model with various appendages, 7th International Conference on Hydrodynamics, Rome, Italy, 2006, pp. 641-646.

[9] Huang S. F., Xu J. and Ma Z.: The research on numerical of resistance predicting for bulk carriers, Proceedings of the 9th National Conference on Hydrodynamics, Cheng Du, China, 2009, pp. 645-651.

[10] Huang W. G., Qiu L. Y. and Jiang Z. F.: A Method to Improve the Precision of Ship Flow prediction Under Restrictive Calculation Conditions, Chinese Journal of Ship Research, Vol. 8, No 1, 2013, pp. 20-25.

[11] Wang F. J. Computational Fluid Dynamics Analysis——CFD theory and Application, Bei Jing, China, Tsinghua University Press, 2004. (In Chinese)

[12] Xu Y, Dong W. C.: Numerical study on wave loads and motions of two ships advancing in waves by using three-dimensional translating-pulsating source, Acta Mechanica Sinica, Vol. 29, No 4, 2013, pp. 494-502.

[13] Xiong Y. and Liu Z. H.: Numerical Prediction on Propulsion Performances of a Container Ship, Journal of WuHan University of technology, Vol. 35, No 4, 2011, pp. 670-674.

[14] Huang J. B., Chen C. Y. and Chen X. P.: Analysis of CFD Computing Strategy for a Container Ship’s Wake Field, Journal of Shanghai Ship and Shipping Research Institute, Vol. 34, No 1, 2011, pp. 8-14.

[15] Bugalski T. and Szantyr J. A.: Numerical analysis of the unsteady propeller performance in the ship wake modified by different wake improvement devices, Polish Maritime Research, Vol. 21, No 3, 2014, pp. 32-39.

[16] Wang J. B., Yu H. and Zang Y. F.: Numerical simulation of viscous wake field and resistance prediction around slow-full ships, Journal Of Hydrodynamics (A), Vol. 25, No 5, 2010, pp. 648-654.

[17] Gang W., Lu D. Q. and Dai S.Q.: Waves induced by a submerged moving dipole in a two-layer fluid of finite depth, Acta Mechanica Sinica, Vol. 21, No 1, pp. 24-31.