Effect of grid and time step for predicting hydrodynamic resistance of blunt body

Lili Wang 1,2,*, Kangning Sun 3, Yufeng Cai 2, Jun Jiao 1,2, Jiaxu Zhang 1,2, Mingzhen Wang 1,2, Xiaohong Han 1,2, Yongze Xu 4

1AVIC Special Vehicle Research Institute, HuBei, China
2Key Aviation Scientific and Technological Laboratory of High Speed Hydrodynamic, HuBei, China
3Chinese Flight Test Establishment, ShanXi, China
4CD-Adapco Software Technique(Shanghai) Co, Ltd, Shanghai, China

*Corresponding author e-mail: 1056289560@qq.com

Abstract. Taking the amphibious vehicle as an example, the hydrodynamic resistance was obtained by using computational fluid dynamics (CFD). This paper investigated the influential factors which contain mesh generation and time step which influences hydrodynamic calculation, and compared the calculation results with the relevant experimental results. This paper analyzed three factors which affecting amphibious vehicle 's resistance calculation, including the first grid thickness of boundary layer and ratio for extending surface mesh to the computational domain. By means of analyzing different factors, a high calculation precision strategy which could serve as reference for amphibious vehicle's hydrodynamic resistance was obtained.

1. Introduction
The calculation accuracy of hydrodynamic drag of amphibious vehicles is of great value for the rapid prediction of hydrodynamic performance of amphibious vehicles and the optimization of appearance [1-3]. While using CFD software for related research, in order to ensure the reliability and credibility of the calculation results, it is necessary to conduct in-depth research on the relevant factors affecting numerical simulation [4-9].

Meshing is one of the key aspects of Computational Fluid Dynamics (CFD). For the numerical simulation of the external flow field of amphibious vehicles, grid quality is very important for the calculation speed, accuracy and convergence of the flow field [10-11]. Regarding the research on the influence of grid on numerical simulation results, most scholars at home and abroad focus on the uncertainty caused by grid, and pay attention to the validation and verification of calculation results when grid parameters are changed. From the perspective of engineering practice, a few researchers propose meshing methods and parameters that are suitable for meeting engineering needs [12-13].

Based on the actual needs of the project, this paper studies the influence of the first layer mesh parameters on the surface (ie y+), boundary layer mesh extension ratio and time step and generates multiple sets of different meshes. Ultimately a meshing method and a time step with a calculation accuracy of 98% was obtained.
2. Calculation object and boundary conditions
The calculation model is an amphibious vehicle. The model has a length of 2m, a width of 0.9m and a height of 0.5m. As shown in Figure 1, the total length of the calculation domain is 5L (L is the length of the vehicle); the speed entrance is 1 times longer to the head of the car; the pressure exit is 4L from the head of the car; the right boundary is 2.5L from the longitudinal section of the car body; and the upper and lower boundaries are separated from the car body 1L length. The calculation model is symmetrical about the mid-longitudinal section, and the model is a hydrostatic direct motion. Therefore, in order to increase the calculation speed, only the half of the amphibious vehicle is mesh-discrete, and the middle longitudinal section is set as a symmetrical boundary.

3. Study on mesh scale of blunt body surface
The model surface mesh can describe the geometry of the model. For a model with complex geometry, if the mesh size is too large, it is likely that the mesh is not close to the body, and the model features cannot be accurately reflected, so that the calculation result is greatly deviated from the actual situation; if the mesh scale is too small, it will result in the number of grids increases greatly, the calculation time becomes longer, and when the number of grids is too large, the rounding error in the calculation process will increase, and the accumulation of rounding errors will also cause deviations in the calculation.

Therefore, it is necessary to select an appropriate mesh size. The mesh size takes into account the calculation accuracy and calculation cost requirements in the case of ensuring the grid body.

For this part of the study, the calculation speed is 4m/s, the grid scale is set to 20mm, which is 1% of the model length.

4. Study of wall distance $y^+$
As shown in figure 2, for a turbulent flow that is sufficiently developed on a solid wall surface, the flow area can be divided into a wall layer (or inner zone) and a core zone (or outer zone) along the normal direction of the wall surface.
In this paper, when the model is 4m/s, the Reynolds number is up to 10 million, and the core area is the complete turbulence area. In the two-equation model, whether it is a standard $k-\varepsilon$ model, a RNG $k-\varepsilon$ model, or a Realizable $k-\varepsilon$ model, it is aimed at fully developed turbulence, and they can only be used to solve the flow in the core region of Figure 2.

In the wall layer, the flow changes greatly. Especially in the viscous bottom layer, the flow is almost laminar, and the turbulent stress hardly works. Therefore, the $k-\varepsilon$ model cannot be used to solve the flow of the wall layer. In this paper, the wall function method is used to solve the problem of solving the wall layer. The wall function method directly relates the physical quantity on the wall to the unknown quantity to be sought in the core area, but it must be used in conjunction with the high Reynolds number $k-\varepsilon$ model.

Therefore, in the calculation of this paper, the core area is a high Reynolds number $k-\varepsilon$ model, and the wall layer is solved by the wall function method.

The wall function method is to arrange the first node in the region where the logarithmic law layer is established. Y+ is an important function in the wall function method. There are many different views on the division of y+. It is suggested in the literature that when $30<y+<500$, the flow is in the logarithmic law layer. It is also suggested in the literature that when $60<y+<300$, the flow is in the logarithmic law layer.

For this part of the study, the calculation speed is 4m/s, and the surface mesh of the car body is 20mm. On this basis, the y+ values are taken as 100, 200, and 250 respectively, and the heights of the corresponding first layer mesh nodes are 0.94mm, 1.89mm, and 2.36mm. The comparison between the calculated values of the hydrodynamic resistance and the experimental values with different y+ values is shown in Table 1.

| Numble | Y+   | Deviation (%) |
|--------|------|---------------|
| 1      | 100  | 3             |
| 2      | 200  | 8.5           |
| 3      | 250  | 13.4          |

From the calculation results in Table 1, it can be seen that the hydrodynamic drag calculated by the fluid domain grid divided by different y+ values has a certain gap. It can be seen that the viscous flow around the car body changes significantly. Comprehensively compare the calculation accuracy and calculation cost, taking y+ to 100 can meet the actual needs of the project. As shown in figure3, it is a flow field diagram when y+ is 100.

![Figure 3. Chart of flow field(y+=100)](image)

5. Study on the extension ratio of boundary layer grid
When the fluid flows through the surface of the model, there is a very thin layer of fluid that adheres to the surface of the model without slipping, and the velocity of the layer of fluid is zero. It can be inferred that in the vicinity of the model, there must be such a fluid: the velocity gradient in the vertical direction of the flow is large, and in this fluid, the effect of viscous force cannot be neglected, so that a layer of fluid is called a boundary layer. Strictly speaking, there is no obvious boundary between the boundary layer area and the mainstream area, and the speed of the mainstream area is usually 0.99U as the outer
edge of the boundary layer. The vertical distance from the outer edge of the boundary layer to the object surface is called the nominal thickness of the boundary layer.

As shown in Figure 4, when dividing the grid, the wall function method does not need to be encrypted at the wall layer, and only the first inner node needs to be arranged in the region where the logarithm law holds, that is, the region where the turbulence is fully developed. When the surface scale of the model surface and the dimensionless parameter y+ of the first layer of the grid height in the fluid domain are determined, the parameters affecting the size of the fluid domain grid are only the boundary layer grid extension ratio R.. For this part of the study, the calculation speed is 4m/s; the car body surface mesh is 20mm; y+ is 100; on this basis, the boundary layer mesh extension ratio is 1.3, 1.2, 1.1. The comparison between the calculated values of the hydrodynamic resistance and the experimental values for different extension ratios is shown in Table 2.

![Figure 4. The meshing method of wall function](image)

### Table 2. Comparison of computational and test resistance of different ratio

| Number | Extension ratio | Deviation (%) |
|--------|----------------|---------------|
| 1      | 1.3            | 3             |
| 2      | 1.2            | -3.8          |
| 3      | 1.1            | -5.6          |

From the calculation results in Table 2, it is known that for the model selected in this paper, the influence of the extension ratio change on the calculation results is not too large, and the hydrodynamic resistance calculated by the fluid domain grids with different extension ratios is fairly near. It can be seen that at 4m/s, the viscous flow around the model is relatively stable, and the flow conditions at different heights from the surface of the model are relatively close. The calculation accuracy and calculation cost are comprehensively compared. The extension ratio is 1.3, which can meet the actual needs of the project. As shown in Figure 3, it is a flow field diagram when y+ is 100 and the extension ratio is 1.3.

### 6. Time step study

The time step is a very important parameter for the solver based on the coupled solution, which controls the entire numerical calculation process. For the calculation of such unsteady problems with a bluff body flow with a free liquid surface, the time step directly affects the convergence speed, calculation accuracy and stability of the numerical simulation. In general, the distance of a grid is moved within a time step. The time step is normally 0.005L/V. L is the model feature length; V is the model speed.

For the research in this part, the calculation speed is 4m/s; the surface mesh of the car body is 20mm; the y+ is 100; the boundary layer mesh extension ratio is 1.3. On this basis, the time step is initially
selected as 0.005s, 0.004s, 0.003s, 0.002s, 0.001s. The comparison between the calculated values of the hydrodynamic resistance and the experimental values at different time steps is shown in Table 3.

| Numble | Time step(s) | Deviation (%) |
|--------|--------------|---------------|
| 1      | 0.005        | 3.0           |
| 2      | 0.004        | 1.1           |
| 3      | 0.003        | -6.2          |
| 4      | 0.002        | -6            |
| 5      | 0.001        | -6            |

It can be seen from the calculation results in Table 3 that the time step has a significant influence on the calculation accuracy of the bluff body resistance, and the maximum error in the result of the dispersion of the time term can reach 10%. When the time step is 0.004s, the calculation accuracy reaches 98%, which has reached the actual precision requirement of the project.

Considering the calculation accuracy and calculation speed, it is more appropriate to select a time step of 0.004s. As shown in figure 5, it is a resistance time chart when $y^+$ is 100, the extension ratio is 1.3, and the time step is 0.004s.

![Figure 5. Curve of resistance with time(t=0.004s)](image)

7. Conclusion
In this paper, commercial CFD software is used to calculate the hydrodynamic drag of bluff body 4m/s. The calculation focuses on the influence of grid on the calculation results. After systematic research, the following conclusions are obtained:

1) The model surface mesh can describe the geometry of the model. The surface mesh size is recommended to be set to 1% of the model length.

2) $y^+$ has a significant influence on the calculation results of bluff body hydrodynamic resistance. Combined with the characteristics of blunt body viscous flow, $y^+$ is 100, which can meet the requirements of actual calculation accuracy.

3) The hydrodynamic drag calculated from the fluid domain grids that are discrete according to different extension ratios is not much different. It can be seen that the viscous flow around the model is relatively stable at 4m/s. Comprehensively compared the calculation accuracy and calculation cost, the extension ratio is 1.3, which can meet the actual needs of the project.

4) The time step has a significant influence on the calculation accuracy of the bluff body resistance. When compared all the factors, the time step is 0.004s, which can meet the actual calculation accuracy.

References
[1] Zhanzhong Han, Guoyu Wang, Weige Yan. Numerical simulation of viscosity resistance around a running amphibian vehicle[J]. Vehicle & Power Technology, 2003,2:6-10.
[2] Li Li, Xiancheng Wang, Shuh, Han, Calculation of wave-making resistance and analysis of flow field for amphibious armored vehicles[J]. Acta Armamentarh, 2010, 31(8): 1102-1105.
[3] Xiangyu Zheng, Xuhua Yu, Fan Zhang, Research on numerical simulation of acceleration performance of amphibious vehicle on water[J]. Computor Simulation, 2012, 29(11):71-74.
[4] Guoying Xu, Jun Wang, Jingtao Zhou. Numerical simulation of the amphibious vehicle's drag
force and attitude based on CFD[J]. Ship Science and Technology, 2006, 28(4):22-25.

[5] Wei Sun, Jianli Han, Xixia Liu. Numerical simulation of amphibious vehicle based on the dynamic mesh model[J]. Ship Science and Technology, 2009, 31(1): 146－150.

[6] Fudong Gao, Lehua Jiang, Cunyun Pan. Numerical calculation on hydrodynamic property for the amphibious vehicle based on computational fluid dynamics[J]. Journal of Mechanical Engineering, 2009, 45(5): 134－139.

[7] Long Cheng. Numerical simulation of flow around an crawler-type amphibious vehicle[J]. Equipment Manufacturing Technology, 2015 (7): 9-18.

[8] Yufeng Cai, Lili Wang, Yu Wang, Hydrodynamical simulation of amphibious vehicle based on CFD and its experimental verification[J], System Simulation Technology, 2018,14(3):183-187.

[9] Lili Wang, Jiaxu Zhang, Tao Liu, Numerical analysis on effect of wave suppression plate on hydrodynamical characteristics of amphibious vehicle[J], System Simulation Technology, 2018,14(2):113-117.

[10] Yunming, Qiu, Rui, Deng,. Discussion of mesh generation method for ship resistance calculation[J], Ship Engineering, 2013,1(35):13-15.

[11] Lili Wang, Xinying Li, Miao Huang, Hydrodynamics research of amphibious aircraft float based on overset mesh[M]. Beijing: Ocean Press, 2017: 1248-1255.

[12] Xiaojun Lv, Qidou Zhou, Jiaxi Duan, Grid parameter and discrete scheme for predicting submarine resistance[J], Journal of Naval University of Engineering, 2014,26(2):40-44.

[13] Liwei Dong, Zhengqi Gu, Shuichang Liu, Effect of meshing on numerical simulation of external flow field around vehicle[J], Automobile Technology, 2012,(1):12-15.