Experiment and simulation on clam water taxiing of seaplane based on CFD

Xiaolong Zheng*, Bin Wu, Mingzhen Wang and Binbin Tang
China Special Vehicle Research Institute, Key Aviation Scientific and Technological Laboratory of High-Speed Hydrodynamic, Jing Men, China

*Corresponding author e-mail: zhengxiaolong1001@126.com

Abstract. In order to verify the feasibility of STAR-CCM+ in handling the clam water taxiing problem of seaplane, capturing the kinetic characteristic of free surface by overlap mesh and Euler VOF method, and compare it with the experiment result. It shows that the simulation results are in good agreement with the experimental values, and the information of the flow field is well consistent with the actual phenomena. From qualitative and quantitative perspectives point of view, it is feasible to use STAR-CCM+ based on overlap mesh method to simulate the calm water taxiing of seaplane, a new method for predicting hydrodynamic performance of seaplane is provided.

1. Introduction
For the seaplane, taxiing take-off on the water surface is a very important part in the whole process of aircraft design. There are many factors affecting the takeoff performance of an aircraft when it taxiing on the water, it may lead to adverse consequences such as "burying bow" and "dolphin movement". At present, the research of taxiing takeoff is mainly carried out by model test and theoretical calculation[1, 2]. With the rapid development of computer technology and the long period and high cost of model test, the advantages of numerical simulation are becoming more and more obvious.

In recent years, many scholars have studied the landing problem of aircraft on water [3, 4], but few have studied the motion simulation and drag performance of taxiing takeoff on water. In this paper, the process of single hull of a seaplane taxiing on the still water has been simulated, STAR-CCM+ is used to realize the translation and rotation of model based on overset mesh, VOF method is used to capture the fine flow field of free surface. The simulation results are compared with the experimental data, and the feasibility and validity of the method for predicting the hydrodynamic performance of the seaplane are verified.

2. Mathematical Model
The simulation of the single hull is carried out in a numerical tank established by computer. The governing equation of viscous numerical tank is composed of continuity equation and momentum equation.

Assuming that the fluid is incompressible, the continuity equation and the RANS equation are:

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

(1)
Among them, $\bar{u}_j$ is time-averaged velocity, $u_j'$ is the fluctuating velocity and $-\rho u_j' u_j'$ is Reynolds stress.

VOF model is used for gas-liquid two-phase flow, and high resolution HRIC method is used for interface tracking. The governing equations of VOF method are as follows:

$$
\frac{\partial \alpha_q}{\partial t} + \nabla \cdot (u_q \alpha_q) = 0
$$

$$
\frac{\partial (\rho \alpha_q \bar{u}_q)}{\partial t} + \nabla \cdot (\rho \alpha_q \bar{u}_q \bar{u}_q) - \alpha_q \nabla p + \nabla \cdot (\mu_q \alpha_q \nabla \bar{u}_q) + \rho_q \alpha_q g + \bar{F}_{D,q} + \bar{F}_{S,q}
$$

Among them, $\rho_q$, $\alpha_q$ and $\mu_q$ represent the density, volume fraction and flow velocity of phase $q$, $\bar{g}$ is gravitational acceleration with a value of 9.81 m/s$^2$. $\bar{F}_{D,q}$ and $\bar{F}_{S,q}$ are inter-phase drag force and surface tension respectively. They are very small for the whole flow field in the course of hull sliding, so their influence can be neglected during the calculation. Among $\bar{u}_c = C_a \left| \nabla \bar{a} \right|$, $\left| \bar{a} \right|$ is the flow velocity, $C_a$ is sharpening factor in the range of 0–1.

3. Computational Model and Experiment Scheme

3.1. Computational Model

For the purpose of studying the hydrodynamic performance of the single hull of a seaplane taxiing on the still water surface, this paper just choose the single hull structure as the calculation model, which does not contain wing and empennage. The total length of the model is 2.24m which is named L, and the initial trim angle is 4.2 degrees.

After introducing the model, establishes the flow field in STAR-CCM+. The distance from bow to the inlet boundary is 1L, and the one from tail to the outlet boundary is 5L. Distance from hull centre to up and bottom boundary are 1L and 2L, the width of the computational domain is 4L. Because of the large movement posture, the fuselage will have large trim and heaving, so this paper uses overlapping grid method to divide the mesh, and establishes a small geometric body to surround the hull as overlapping domain, named it small domain, as shown in Figure1.

![Figure 1. Computational grid model.](image-url)
In terms of motion and grid processing, regional motion is adopted. When the hull moves and rotates under the external forces, there is no relative displacement between hull and the small domain grid, but the small domain moves with the hull, they are a whole. By changing the distribution of flow field parameters in the computational grid, the corresponding relationship between the flow field parameters and the grid after hull displacement is obtained by interpolation. The pattern of regional motion is shown in Figure 2.

![Figure 2. Corresponding relationship between ship displacement and grid](image)

The whole computational domain consists of a small overlapping domain and a large flow field. In order to capture the physical characteristics of the flow near the wall more precisely, the $Y+$ is controlled at 50, and the total number of grids is 865.34 million.

3.2. Boundary Condition
The momentum equation is dispersed by implicit finite volume method and solved in time domain by separating solver. The free surface is captured by VOF method, and the gravity effect is taken into account. The standard atmospheric pressure is used as the reference pressure to initialize the calculation. Turbulence model is Realizable $k$-$\varepsilon$. The second-order upwind model is used to improve the accuracy, the time step is 0.005s when the speed below 6m/s and it will be 0.001s when the speed over 6m/s.

3.3. Experiment Scheme
The Test was completed in the high-speed towing tank of the China Special Vehicle Research Institute, the tank is 510 meters long, 6.5 meters wide and 5 meters deep, the maximum speed of the trailer can be reach 25m/s.

Conventional static water towing test is mainly to measure the resistance, heave and trim angle of the model. Fig.3 shows the installation diagram of the taxiing test of single hull model. A towing point is located at the center of gravity in the longitudinal direction, at the pivot of the model and the towing staff, angle sensor is located to measure the trim angle, displacement sensor installed on the top of the carriage is used to measure the heave at the center of gravity of the model. The towing force is measured by a load cell which is located at the bottom of the towing rod. Measured force in the direction parallel to the water surface. A navigation plate is installed on the bow, and the navigation rods fixed on the seaworthiness instrument are inserted into the navigation films, this is not only prevents the left and right yaw of the model in motion, but also ensure that the test model can move freely around, heave, and trim.
4. Computational Results and Experimental Verification

4.1. Comparisons of calculation results with experimental values

Table 1 shows the comparison of resistance, heave and trim calculated by CFD with experimental values. The resistance is expressed by dimensionless R/G, which is the ratio of resistance to model weight.

| Velocity (m/s) | Experimental values | Calculation results |
|---------------|---------------------|---------------------|
|               | R/G                | Θ(°)       | δ(mm) | R/G | Error (%) | Θ(°) | Error (%) | δ(mm) | Error (%) |
| 2.5           | 0.110              | 7.39       | -7.5  | 0.113 | 3.40      | 7.92  | 7.13      | -8.1  | 8.44      |
| 3.5           | 0.147              | 8.84       | 1.4   | 0.159 | 8.26      | 8.60  | -2.75     | 1.8   | 32.94     |
| 4.5           | 0.212              | 14.57      | 41.7  | 0.211 | -0.84     | 13.75 | -5.65     | 38.50 | -7.73     |
| 5.5           | 0.232              | 14.96      | 75.4  | 0.231 | 0.38      | 14.24 | -4.83     | 69.30 | -8.12     |
| 7             | 0.220              | 13.83      | 102.7 | 0.234 | 6.18      | 13.86 | 0.20      | 94.60 | -7.92     |
| 8             | 0.208              | 13.42      | 110.6 | 0.202 | 2.88      | 13.56 | 1.02      | 101.40| -8.28     |
| 9             | 0.185              | 13.03      | 116.4 | 0.190 | 2.70      | 13.20 | 1.28      | 105.90| -9.02     |

The whole computational domain consists of a small overlapping domain and a large flow field. In order to capture the physical characteristics of the flow near the wall more precisely, the Y+ is controlled at 50, and the total number of grids is 865.34 million.

It can be seen that the resistance and trim angle increase first and then decrease with the increase of velocity, and their values reach the maximum at 5.5m/s. The heave of the model increases with the increase of velocity. Comparing with the test results, it can be concluded that the calculated resistance values are in good agreement with the test values, with an average error of 3.12% and a maximum error of 8.26%. The average error of trim angle is 3.26% and the maximum control is 7.13%. The largest error of heave is 32.94%, but 1.4mm is very small, so this error is in the normal range.
4.2. Wave-Making of Free Surface

Figure 4 shows the free surface wave-making distribution at different velocities. With the increase of speed, the trim angle increases first and then decreases, the peak value appears at 5.5m/s. At the same time, the wave-making around the model and tail waveform also change differently. It can be clearly seen that the width of Kelvin wave in the tails is smaller at high speed. When the speed increases, the phenomenon of "cocktail flow" becomes obvious, and the relative position gradually moves backward with the increase of speed, which is consistent with the actual flow phenomenon in the test process.

Figure 5. Stern flow at different velocities
4.3. Bottom Pressure Distribution

Figure 6. Bottom pressure distribution at different velocities

Figure 6 shows the pressure distribution of hull bottom at different velocities. It can be seen that the distribution of pressure presents a "triangle" shape. The leading edge of the water contact point has the greatest pressure, and the pressure standing line extends backward along this point. And then, the step appears negative pressure. These phenomena are consistent with the flow characteristics of high-speed boats [5, 6]. During the taxiing process, water sprays from the hull bottom to both sides to form a whisker splash, which is also the reason for the formation of pressure standing line, as it shows in figure 7.

It can also find that the position of pressure stagnation point gradually moves backward, because with the increase of speed, the trim angle and heave increase, especially after 5.5m/s, the heave increases greatly, which makes the submerged part of the hull decrease.

5. Conclusion

In this paper, an implicit unsteady computational method based on overlapping grids and $k$-$\varepsilon$ turbulence model is used to simulate the flow around a seaplane when it taxiing on the clam water. The calculated resistance, trim angle and heave are in good agreement with the experimental values, and the flow field cloud chart is highly consistent with the experimental phenomena. It shows that the computational method adopted in this paper is reasonable and effective. This method can be used to preliminarily evaluate the hydrodynamic performance of the seaplane, which greatly reduces the cost of model test and shortens the design cycle. At the same time, this calculation method can provide technical support for the subsequent takeoff and landing simulation of the seaplane in waves.

References

[1] Wu Qingwei, Gao Xiaopeng, Wubin, A method to evaluate the resistance of seaplane sliding in still water[J]. Ship and ocean engineering, 2013,03(42):154-157.

[2] Gu biao, Tang Binbin, Wu Bin, Study on water resistance performance of seaplane[C]. The 17th china international hull exhibition and high performance ship report, Shanghai: Royal institute of Naval Architects, 2012: C09.
[3] Sun Peichen, Gao Xiaopeng, Dong Zusun, Simulation analysis of amphibious aircraft water landing[J]. Ship and ocean engineering, 2014,06:171-174.

[4] Luo Linyin, Yang Shifu, Lv Jihang, Analysis and numeral simulation of water landing response model for amphibian[J]. Journal of machine design, 2013,30(8):86-89.

[5] Wei Zifan, Jing Shengping, Yang Songlin, CFD simulation and comparison analysis of a new type high-speed boat [J]. Chinese journal of ship research, 2016,11(4):22-28.

[6] Sun Yuan, Lu Xiaoping, Li Jingyu, Comparative study on simulation of resistance for planing craft[J]. Chinese journal of ship research, 2019,14(1):27-32.

[7] Dong Zusun. Speedboat dynamics[M]. Huazhong university of science and technology Press,1991.