Research on sliding hydrodynamic performance of flying boat

Xiaoxiang Wang1,2,*, Bin Wu1,2, Xinying Li1,2, Miao Huang1,2

1China Special Vehicle Research Institute, Hubei, China
2Key Aviation Scientific and Technological Laboratory of High-Speed Hydrodynamic, Hubei, China

*Corresponding author e-mail: mr.w.27@qq.com

Abstract. The towing tank test of a scaled flying boat model is designed to obtain its resistance, heaving and pitching characteristics as the speed changes in the sliding progress; then based on the secondary development of Fluent; the global moving mesh and VOF methods are applied in the simulation; the calculation gives the fluid field around the boat, especially the wake flow, and also the pressure distribution of boat step. the results compared to the experiment date prove that using this method to study the hydrodynamic performance of a flying boat is feasible and precise.

1. Introduction
A flying boat is a kind of aircraft which can take off and land on the water surface as well as at the airport on land. When the aircraft is berthing on the water, its hull-type fuselage generates buoyancy to make it float on the water surface without sinking. When it takes off, the propeller engine generates pull force, which makes it slide on the water surface at a considerable speed. When the speed increases continuously, the lift force on the wing overcomes the weight of the fuselage, then the plane takes off. In the process of aircraft water skiing, it will be coupled by the hydrodynamic force of the fuselage and the aerodynamic force of the wing, which together determine the hydrodynamic characteristics of the seaplane when it glides on the water surface, such as resistance characteristics, glide stability, splash and so on. In the early twentieth century, countries represented by Japan, Germany and the Soviet Union devoted considerable efforts to the development of seaplanes from the military needs. A-40 albatrosses, Bie-200, CL-215 and PS-1 were produced. Elmo J. Mottard [1] and Gerard Fridsma [2-3] of the United States also carried out water-skiing research on the take-off of seaplanes. In 1976, China designed the first generation of large seaplane SH-5 by the Institute of Chinese Special Aircraft Research, but the development is relatively backward. There are few publicly published reports on the hydrodynamic performance of seaplane. However, some scholars in the field of ships have made relevant explorations on the hydrodynamic performance of high-speed taxiing bodies, such as Sun Huawei[4] uses CFD to explore the factors affecting the resistance and navigation state of taxiing boats, Dong Wencai [5] of Naval Engineering University has carried out experimental research on the longitudinal motion of deep V-type taxiing boat, which can provide research ideas for hull design of hull-type seaplane.

In this paper, the towing drag test design of an amphibious aircraft is carried out, The total resistance, heave and pitch angle of the model are measured during the process of water skiing to obtain the law of its variation with speed, Then, based on the secondary development of FLUENT, UDF program is compiled to simulate it ,The whole motion grid method and six-degree-of-freedom model are used to simulate the motion of the aircraft, and VOF method is used to track the free surface which directly
reflects the flow field around the hull, especially the stern part, and the pressure distribution at the broken step. The comparison of resistance, pitch and heave with the test results verifies the feasibility of simulating methods based on FULUENT platform.

2. Experimental Design

2.1. Test Facilities
The technical specifications of high-speed hydrodynamic test equipment, high-speed towing test tank and its associated trailers, are as follows:
- Main dimensions of the tank: 510 meters long, 6.5 meters wide, and 5.0 meters deep
- Trailer performance: speed range 0.1-22 m/s, speed stability accuracy is better than 0.2%.
- Test means: test data acquisition, processing and analysis automation

![Towing tank](image)

**Fig 1.** Towing tank

2.2. Design of Test Equipment
There are six degrees of freedom when a real aircraft glides on the water surface. It is impossible to simulate it completely by using scaled tank test, but its attitude, longitudinal stability and rapidity are our most concerned. Therefore, when towing in the tank, it is limited to three degrees of freedom, which are translational motion along the X axis, heave motion along the Z axis and pitch motion around the Y axis. Based on the above considerations, a test device is designed as shown in Figure 2. The whole test device is installed at the bottom of the hydrodynamic test trailer. The motion measurement system consists of a small pulley, a drag gauge, a gyroscope, a position sensor and a course-limited navigation rod. The small pulley can move along the course without sideslip movement, The model can reflect the heave change by connecting the heave rod with the pull-line sensor above. Under the model, the connecting parts are articulated at the center of gravity of the model which ensure pitch motion. A vertical gyroscope is installed in the model to measure its pitch angle. In order to prevent the model from yaw hazard in the course of motion, a heading fixing device is also installed.
2.3. Test State

The design state are selected in the test, The towing speed is 4 m/s~13 m/s, The landing gear is in the retractable state during the test. The flaps deflection angle is 20 degrees, the elevator deflection angle is 0 degrees, and the center of gravity position is 26% MAC. The model parameters are shown in Table 1 below.

| Parameter     | length | width | Step height |
|---------------|--------|-------|-------------|
| Value(m)      | 3.6    | 0.35  | 0.02        |

3. Numerical Calculation

3.1. Global Motion Mesh Method

Traditional dynamic mesh technology[5] uses mesh deformation and reconstruction to realize object motion, and the deformed part can only use unstructured mesh. This method has high computational cost. Secondly, it is easy to produce low-quality mesh during the reconstruction process of pulling and compressing, which affects the effect of water surface capture. The whole moving mesh technology[6] can better avoid this problem. Domains, along with boundaries and all grids, move rigidly with objects, but only considering forces on objects. This process eliminates the process of mesh redrawing. Internal grids do not produce pull or compression deformation like traditional dynamic grids, which can improve the calculation accuracy and save a lot of computing resources.
3.2. Computational Format
The flow field solver is based on unsteady Reynolds time-averaged equation. Enhancement Treatment method is used for wall treatment, SIMPLEC[7] algorithm for pressure-velocity coupling, body force weighted scheme for pressure term, third-order MUSCL scheme for convection term in momentum equation. Convection terms in turbulence equations are discretized by second-order upwind scheme[8], diffusion terms are discretized by second-order central difference scheme, and unsteady terms are discretized by second-order implicit scheme. All boundary conditions are given velocity entrance. The boundary pressure and air distribution are related to the height of water surface. The UDF definition is adopted.

4. Result Analysis
4.1. Comparison of Experiment and Calculation
4m/s, 8 m/s and 13 m/s are selected for simulation analysis according to the three representative velocities including middle, low and high. The time step is 0.001 s and the flow time is 8s. Because of the unsteady motion, all the results are averaged in the last 1 s

| Velocity(m/s) | 4   | 8   | 13  |
|--------------|-----|-----|-----|
| resistance (kg) |      |     |     |
| Cal          | 11.11 | 14.28 | 14.84 |
| exp          | 11.65 | 15.07 | 16.11 |
| erro         | -4.63% | -5.24% | -7.87% |
| pitch (°)    |      |     |     |
| Cal          | 4.05  | 7.79  | 1.70  |
| exp          | 3.83  | 7.64  | 1.51  |
| erro         | 5.74%  | 1.96%  | 12.50%  |
| Heave (mm)   |      |     |     |
| Cal          | 12.2  | 122.5 | 149.2 |
| exp          | 12.9  | 131.1 | 137.7 |
| erro         | -5.42% | -6.56% | 11.50% |

The calculation results show that the resistance under three velocities is less than the test value, the pitch angle is larger than the test value, the absolute error of the three parameters is less than 8% when the velocities are 4 m/s and 8 m/s, and the error is slightly larger when the velocity is 13 m/s. The reason for this error may be that the model will produce a certain splash at high speeds, and the splash resistance ratio is relatively large, while the numerical calculation model processing and mesh division considering the actual computing resources can not capture all the spatters accurately, the other is the blocking effect of the experimental tank and the possibility of numerical calculation.

In general, the errors between the calculated results and the experimental values are acceptable, which can meet the practical application requirements of engineering.

4.2. The Flow Field of Free Surface
In order to visually reflect the characteristics of the free surface, especially the wake of the aircraft, the phase volume fraction nephogram of the water-air interface is given below. In the figure 0 represents air, 1 represents water, and the color between 0 and 1 represents water-air mixture.
The flow field around the fuselage and its tail can be clearly seen in the figure above. At 4m/s, the pitch angle of the aircraft is small. From 0.2 times the length of the fuselage near the bow, obvious upwelling can be observed, and it extends to the fuselage rear body. There are small streams of water in the tail which converge and then disperse. As the speed increases, the pitch angle increases, the upwelling moves back to the vicinity of the breaking step, and the surrounding water surface depression becomes more serious. When the speed increases to 13m/s, the pitch angle of the aircraft decreases to less than 2 degrees, and the rear body is out of water. The gravity of the body is basically provided by the dynamic lift. Only small upwelling caused by breaking step is flowing along both sides of the fuselage.

4.3. Pressure of the Bottom And Step
As one of the typical structural characteristics of high-speed taxiing body, step breakage not only reduces water resistance, but also reduces water adsorption on the fuselage, thus avoiding dolphin movement and improving stability. The pressure changes at the bottom of the hull, especially at the step, are given below, as shown in Fig. 6.

As can be seen from the figure above, with the increase of speed, the hydrodynamic pressure increases significantly, especially at the step. At 4m/s, the head is lifted slightly, the heave value is small, and the bottom of the fuselage is mostly in a wet state. the dynamic pressure is small. At 8m/s, the pitch angle of the fuselage increases sharply. Because of the effect of hydrodynamic lift, the fuselage is obvious. When the speed increases to 13m/s, the aerodynamic force accounts for a larger proportion. The aerodynamic coupling effect reduces the pitch angle of the aircraft. Only a small area near the step is in high-speed water taxiing, and the dynamic pressure reaches 60000Pa.

5. Conclusion
In this paper, the hydrodynamic characteristics of a flying boat and the flow field details such as free surface, wake field and pressure distribution at the bottom of the hull are obtained.
through the design of towing test in the tank and the CFD simulation calculation based on the integral motion grid method. The following conclusions can be drawn:

The simulation results based on FLUENT secondary development platform are compared with the experimental results, and it verifies the feasibility and accuracy of using the integral motion grid method to study the hydrodynamic characteristics of seaplane landing and taxiing.

The flow field around the fuselage and the pressure distribution at the bottom of the fuselage reflect the relevant flow characteristics, which can provide a reference for further optimization of its hydrodynamic performance and structural design.

The drag test only tests the resistance change of the fuselage. How to measure the aerodynamic force and hydrodynamic force separately is of great significance to the analysis and study of the Aero-Hydrodynamic coupling effect. Further research will be carried out in this area in the future.

Acknowledgments
This work was financially supported by Aviation Science Foundation.

References
[1] Elmo J.Mottard. A brief investigation of the effect of waves on the take-off resistance of a seaplane[R], NASA TN D-165, 1959
[2] Gerard Fridsma. A systematic study of the rough water performance of planning boats[R], AD 708694, 1969
[3] Gerard Fridsma. A Systematic study of the rough-water performance of planning boats(Irregular Wave-Part II)[R], AD 728788,1971
[4] Sun Hua-wei. Numerical analysis of planning-hull surface shape on resistance and sailing attitude [D]. Harbin Engineering University.2012
[5] Dong Wen-cai, Yue Guo-qiang. Experimental study on longitudinal motion of deep-V-shaped planning craft [J].Ship Engineering, 2004 26(02)
[6] Guo Bao-dong, Qu Qiu-lin,Liu, Pei-qing. Ditching performance of silent aircraft SAX-40 in hybrid wing-body configuration [J]. Acta Aeronautica Et Astronautica sinica. 2013, 34(11)
[7] Zhang Ming-liang. Three-dimensional simulation of meandering river based on 3D RNG turbulence model[J].Journal of hydromechanics.2008(04)
[8] Wen Zheng,Shi Liang-cheng, Ren Yi-ru.The application tutorial of Fluent computational fluid[M].Tsinghua University Press,2009.