ISSN: 2414-1895 DOI: 10.6919/ICJE.202011_6(11).0033

Hydrofoil Flow Field Analysis of Underwater Glider Using Computational Fluid Dynamics Analysis

Keyi Cao[1], Hongrui Yan[2], Yang Li[3], Chang Liu[4]

[1]College of energy and power engineering, Wuhan university of technology, 430063, China;

[2]Shanghai Starriver Bilingual School, Shanghai, 201100, China;

[3]College of Mechanical and Transportation Engineering, China University of Petroleum (Beijing), Beijing, 102249, China;

[4]Shanghai Experimental School CIE, Shanghai, 200120, China.

Abstract

The aim of the research is to determine the relationship between sweep angle and its nearby fluid field to find a exact ranger where the hydrofoil produces the least disturbance on the surrounding. To approach the expected result, Computational Fluid Dynamics (CFD) is employed to simulate the situation under water. The research also includes the testing of CEF itself to ensure the reliability of the data gained from simulation. Previous research mostly aims to find the best configuration of the underwater glider, hydrofoil to achieve the smallest pressure or resistance. However, few researches are conducted in the purpose of least disturbance about the magnitude of the sweep angle. Due to the complexity of the hydrofoil and underwater situation, this research aims to produce a angle range for least disturbance. Previous data and results all indicate that specific results all occurs around 15, 30, 45, and 60. To simplify to issue and complexity of the situation, CFD simulation sets condition under low speed where Reynold number is low that only laminar flow occurs, as the speed of underwater glider is also very low. The data and results are followed. Disturbance is compounded to determine, including pressure, speed, lift and drag force. Through comparing the computing, the fluid fields around the hydrofoil at different angles, among the selected angles—15 30 45 60—60 degrees yield the least disturbance to the surrounding.

Keywords

Computational Fluid Dynamics, Autonomous underwater vehicle, Myring hull, least disturbance, Sweep angle.

1. Introduction

In the past, human's tool for exploring the ocean is by ships only. With the advancement of technology, the first autonomous underwater vehicle (AUV) was invented in 1990. It was applied in many fields such as scientific data gathering and even military using, with sonar and remote-control devices. Nowadays, various types of underwater glider have been designed. After years of development of underwater glider technology operating in marine environment, many more mature models such as Slocum [1], spray [2], seaglider [3], Seawing [4], petrel [5] have been developed at home and abroad.

An underwater glider includes hull, hydrofoil and other appendages. Many researches discussed the shape optimization of hull before. Extensive researches on CFD-based shape optimization have been carried out to improve productivity, including shape parameterization,[6] development and

ISSN: 2414-1895 DOI: 10.6919/ICJE.202011_6(11).0033

application of multidisciplinary design optimization architectures. [7] They tested many hull models in the sea. However, the study about hydrofoil is limited. For a specific hydrofoil, many factors affect the performance of the glider, such as surface shape, attack angle, sweep forward, etc.

In this study, Computational Fluid Dynamics (CFD) will be used to analyze parameter characteristics about a certain hydrofoil to find the least disturbance sweep forward, with the focus on drag, velocity, pressure, etc.

2. Problem statement

Previous study indicated that with the sweep angle increases, the resistance force will decrease. Therefore, the sweep angle with 15, 30, 45, 60 degrees will be tested separately in this study to find out the relationship between the drag force and the sweep angle.

2.1 Geometric parameter of the hedrofoil

The geometric parameter of this hydrofoil is introduced from NACA (EPPLER 817 HYDROFOIL AIRFOIL), shown in Figure 1. To simplify the calculation, the length and width of the wings will be set equal (Figure 2), which is 0.995m.

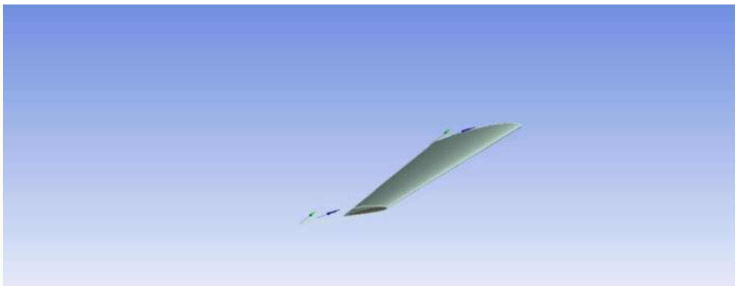

Figure 1. EPPLER 817 hydrofoil airfoil

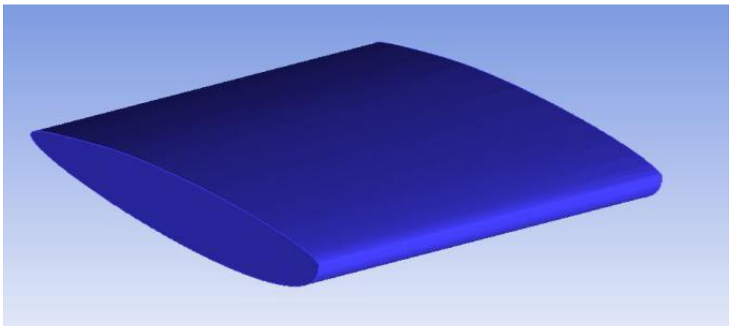

Figure 2. The wings

2.2 Geometric parameter of the hull

The hull of an underwater glider contains three parts, the nose, the middle and the tail. To set these, the Myring hull profile equations can be used to create minimum drag force for a given fineness ratio (l/d). (see Equation 1 & 2). r(x) is the radius of a point; a means length of nose; b is length of body; c stands for length of tail; d is max diameter and n is exponential parameter.

$$r(x)=1/2\ d\{1-[(x-a)/a)]^2\}^{(1/n)} \quad (1)$$

$$r(x)=1/2\ d-[3d/(2c^2)-\tan\theta/c]\,[x-(a+b)]^2+[d/c^3-\tan\theta/c^2]\,[x-(a+b)]^3 \quad (2)$$

In this study, the hull design of the underwater glider has been shown in Figure 3.

ISSN: 2414-1895 | DOI: 10.6919/ICJE.202011_6(11).0033

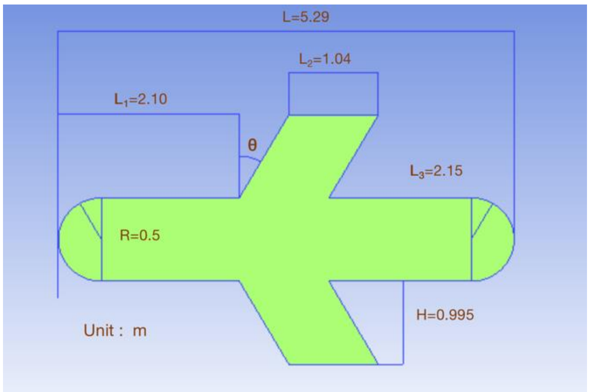

Figure 3. The hull design of the underwater glider

2.3 Methodology

The underwater glider is a small vehicle, with an operating speed about 0.3m/s. In this paper, the sweep angle will be set as 15, 30, 45, 60 degrees. Under each circumstance, the ANSYS fluent will be applied to simulate the flow field, the drag force, and the lift force. The hydrofoil is introduced from NACA data. After setting all the parameters in ANSYS, the calculation process can execute automatically.

3. Validation for ansys

To verify the simulation accuracy, the computational results must be compared with the experiment results.[8] In this case, through changing the attack angle of the glider, the result of drag coefficient can be simulated through CFD. To be specific, the speed of the current is 0.3m/s, and contour for the glider is shown in Figure 4. The exact parameters are displayed in Table 1.

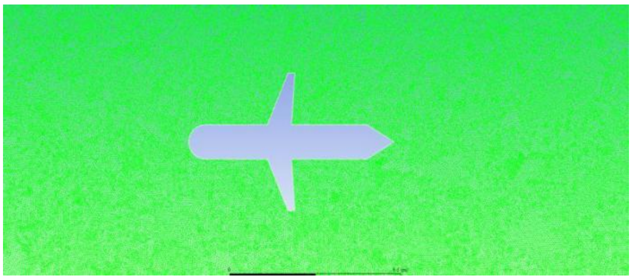

Figure 4. Contour for the glider

Table 1. The setting of parameters

Velocity of water	0.3m/s
Body Length	667mm
Body diameter	100mm
Thickness of the wing	2mm

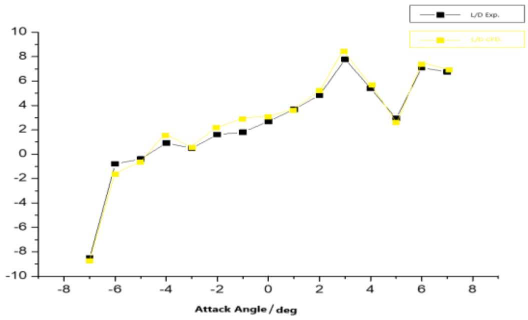

Figure 5. CFD results

ISSN: 2414-1895 DOI: 10.6919/ICJE.202011_6(11).0033

Through changing the attack angle of the wing, different values of Cd can be found. The range for attack angles is from -7° to 7°. Results of the reference and CFD are shown in Figure 5. It can be seen that results are similar to those of previous experiments, and the minor errors can be ignored, which implies the experimental results of this experiment are credible.

4. Convergence

4.1 Mesh generating

3D simulation is the most common way to simulate the flow around a glider. Figure 6 and Figure 7 show the different mesh section of an underwater glider, which are merged for CFD analysis by the mesh generating. The domain size is 20*20*50 (unit: m). It is large enough to ignore the error valve caused by domain field.

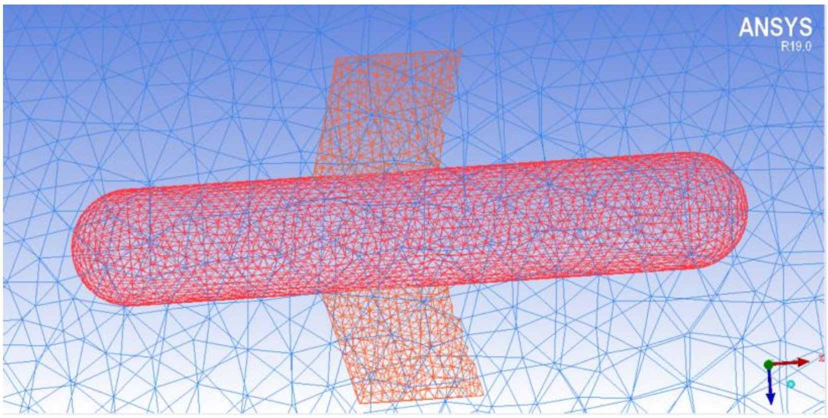

Figure 6. Section of an underwater glider

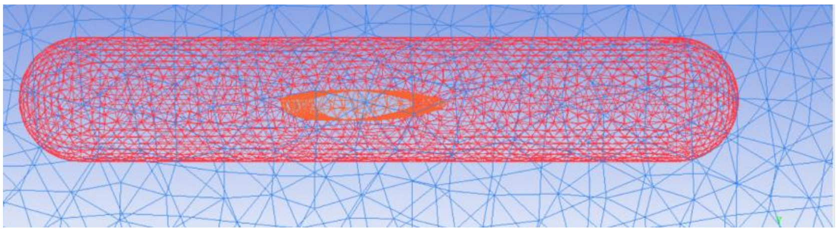

Figure 7. Section of an underwater glider

4.2 Convergence results

The convergence test must be done to ensure that the simulation results are independent to mesh size and grid number. The velocity of water is set to 0.3m/s. The Re number is about 2x10^6. Under different number of grids, the lift force and the drag force will be calculated. Table 2 shows the convergence results.

Table 2. The convergence results

	Grid numbers	Drag force	Lift force
1	106369	0.29016	0.08055
2	800794	0.32819	0.01847
3	817101	0.37475	0.13709
4	902994	0.37474	0.13812

When the grid number is ranging from 100,000 to 800,000, the results changed quickly. However, when the grid number reached to 902,994, the error value is under 0.01. Although when the number of grids reaches 900,000, more time will be spent to simulate each situation. For the accuracy of the final results, the number of grids in future experiments will be set to 902,994.

ISSN: 2414-1895 DOI: 10.6919/ICJE.202011_6(11).0033

5. Results and discussion

5.1 Pressure contour

The pressure distribution of each sweep angle is shown below. The maximum pressure (0.025E3P) occurs in front of the nose section. The minimum pressure occurs around the foil and nozzle.

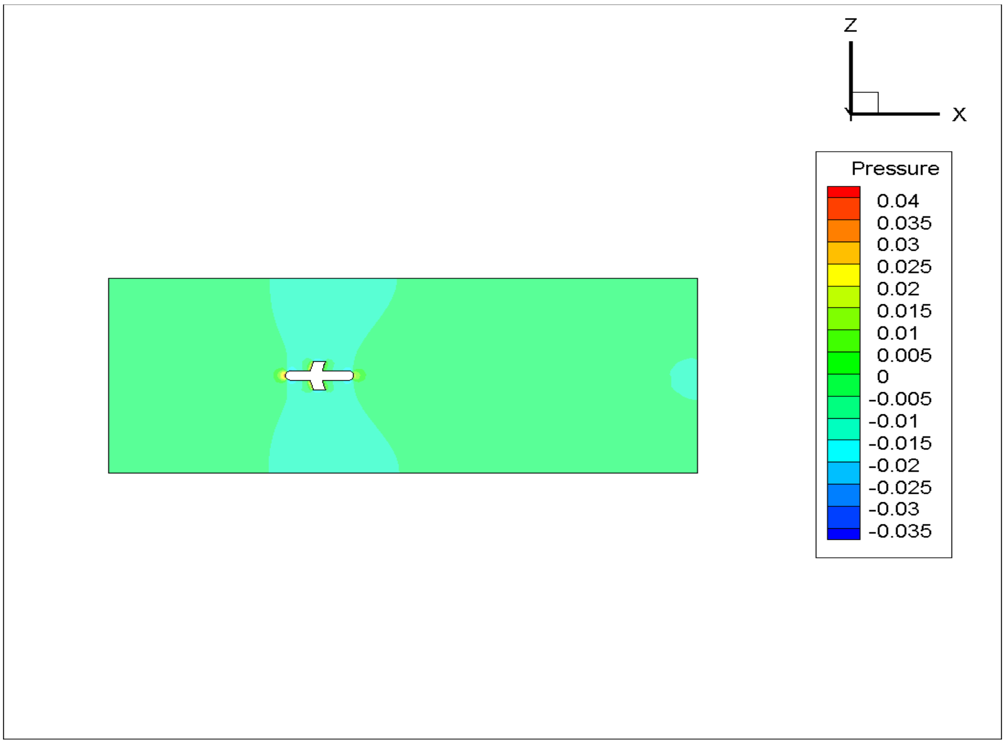

Figure 8. The pressure distribution of 15 degrees sweep angle

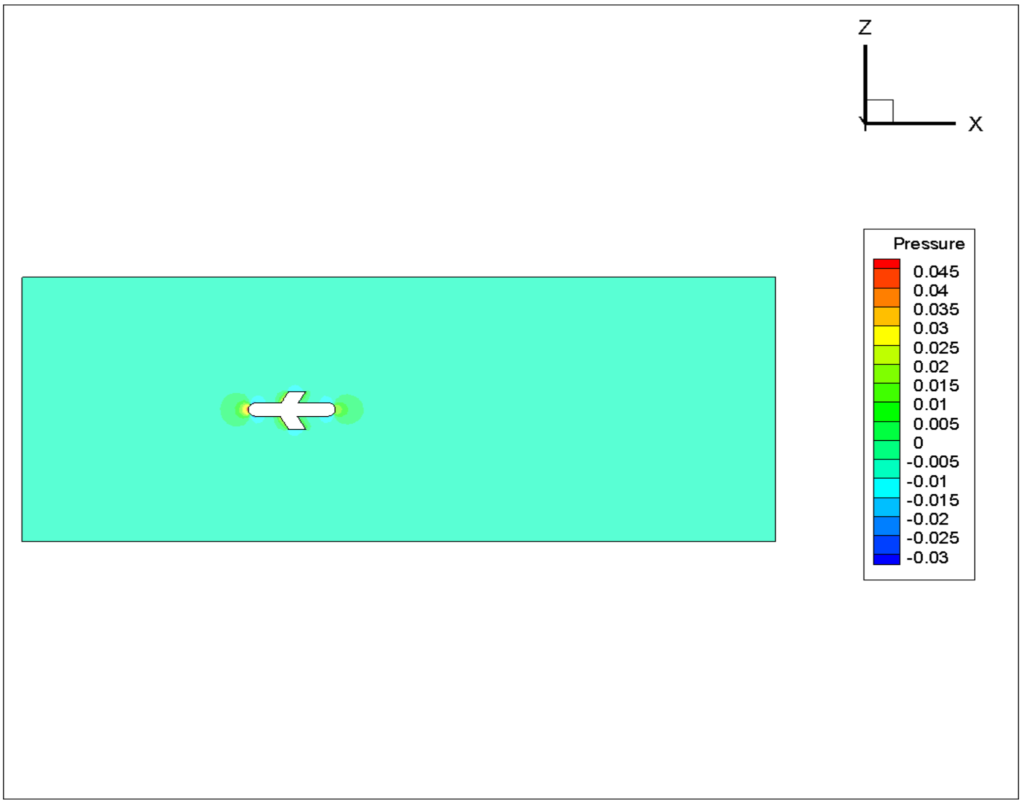

Figure 9. The pressure distribution of 30 degrees sweep angle

ISSN: 2414-1895
DOI: 10.6919/ICJE.202011_6(11).0033

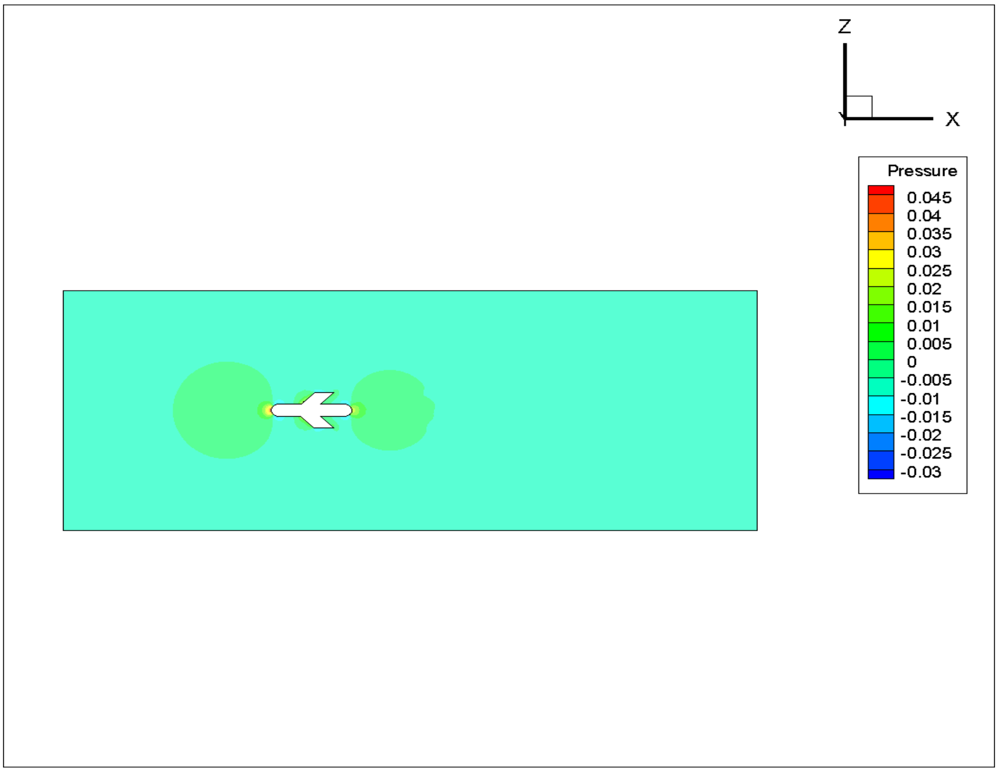

Figure 10. The pressure distribution of 45 degrees sweep angle

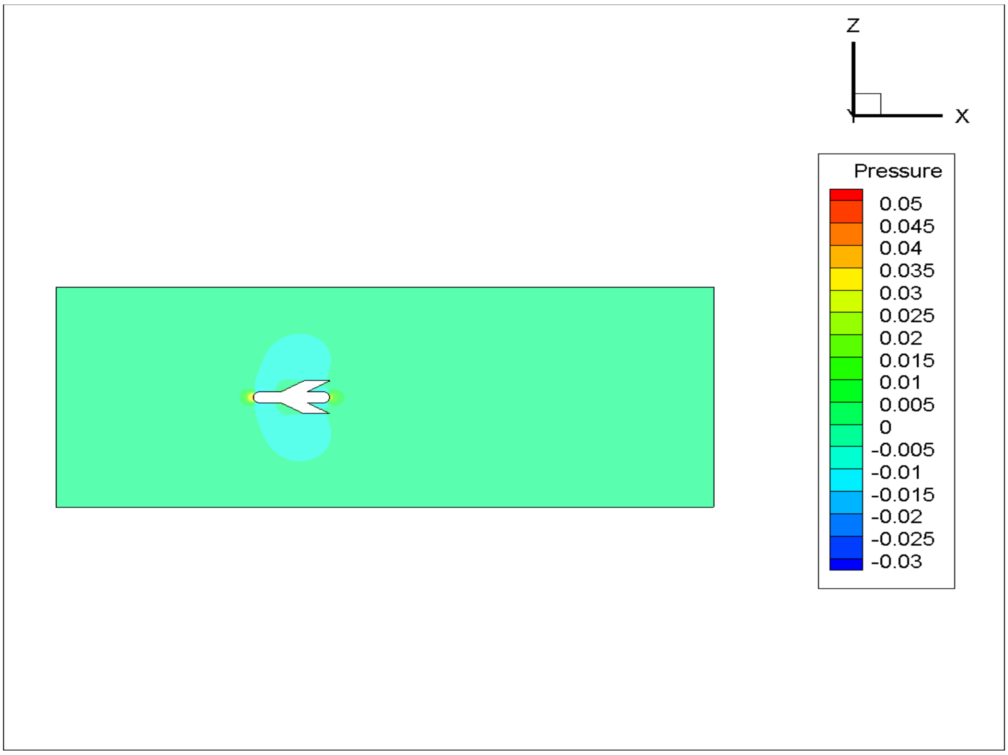

Figure 11. The pressure distribution of 60 degrees sweep angle

5.2 Velocity contour

The water speed distribution is shown as follows. The maximum velocity of water occurs in front of the nose and around the wing. For 45 degrees and 60 degrees, the maximum velocity magnitude around the hydrofoil is lower than other degrees. However, for 45 degrees, the average velocity of water in domain is obviously larger than other degrees. For 60 degrees, the velocity is average distribution.

ISSN: 2414-1895
DOI: 10.6919/ICJE.202011_6(11).0033

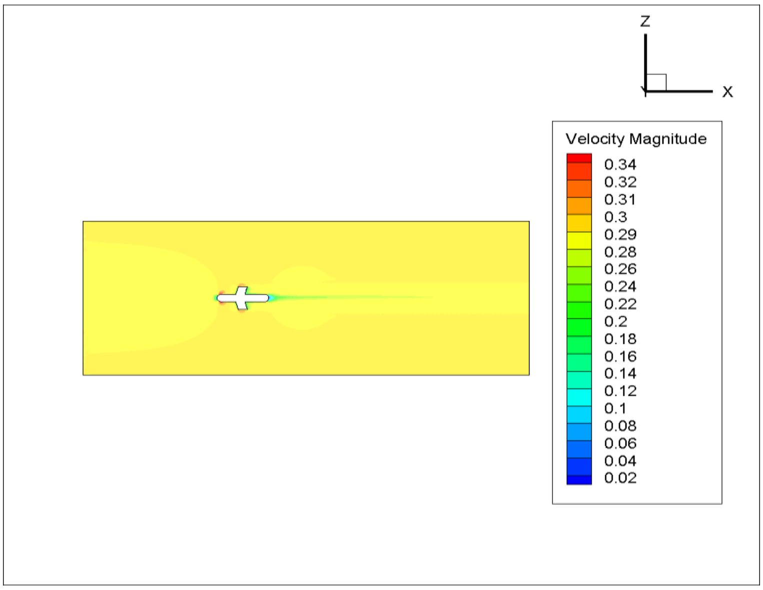

Figure 12. The water speed distribution of 15 degrees sweep angle

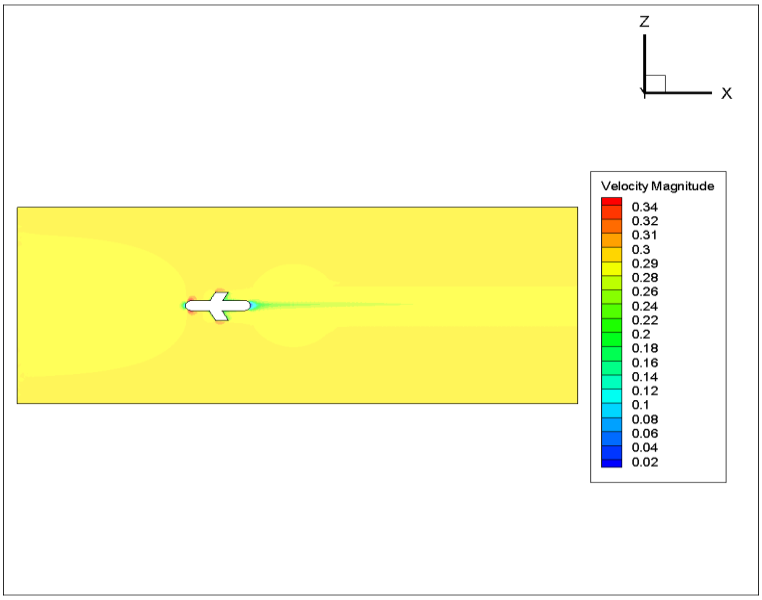

Figure 13. The water speed distribution of 30 degrees sweep angle

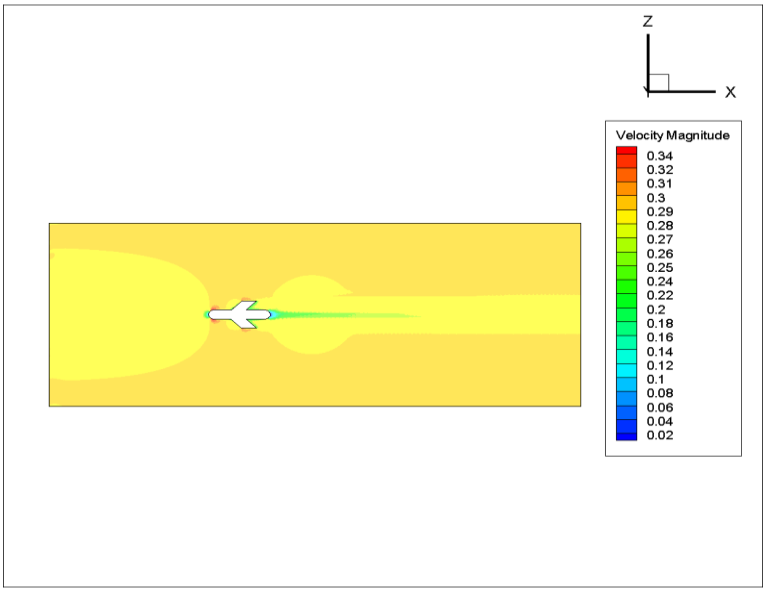

Figure 14. The water speed distribution of 45 degrees sweep angle

Volume 6 Issue 11, 2020
DOI: 10.6919/ICJE.202011_6(11).0033

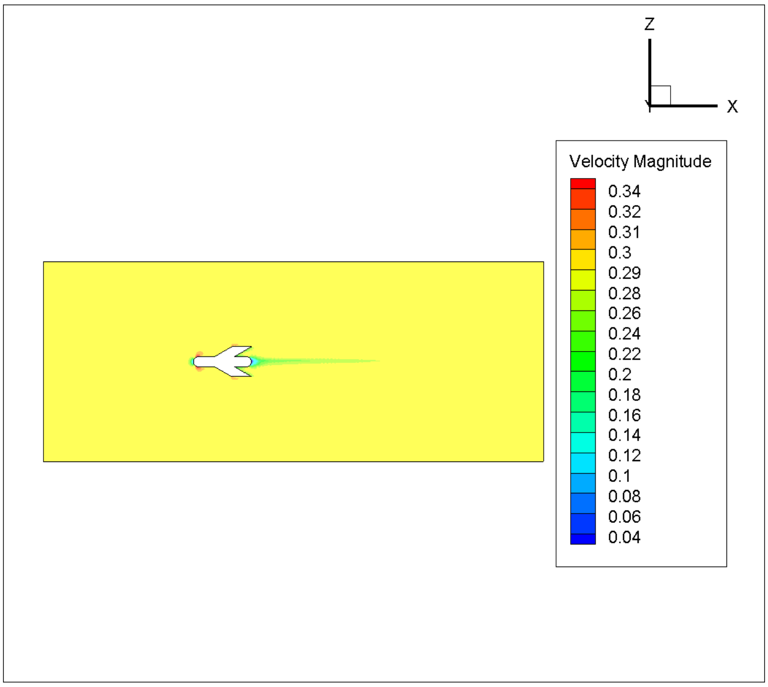

Figure 15. The water speed distribution of 60 degrees sweep angle

5.3 Drag force, life force and L/D

For each case, the Drag force, lift force and L/D are shown in Table 3. The minimum drag force, the largest lift force and the largest L/D occur when the angle is 45 degrees.

Table 3. Drag force, lift force and L/D results

Angle	Drag force	Lift force	L/D
15 degrees	0.29003	0.07246	0.24984
30 degrees	0.27584	0.13506	0.48963
45 degrees	0.26747	0.16956	0.63394
60 degrees	0.27694	0.13256	0.47860

5.4 Discussion

The experimental results show that the lift drag ratio tends to increase with the increase of sweep angle. However, when the sweep angle exceeds 45 degrees, the lift drag ratio decreases, which is not conducive to the change of direction in water. Although the lift drag ratio is the largest at 45 degrees, the velocity gap around it is much larger than that of other models, which means that the turbulence in water is strong. Besides, at a sweep angle of 60 degrees, the flow velocity is obviously average, which is more conducive to the marine ecological environment. From the results of pressure distribution, the influence of fuselage shape on pressure distribution cannot be ignored. In this model, the maximum pressure appears at the fuselage head.

In this experiment, only the size of the sweep was changed. In fact, there are many parameters need to be studied, such as fuselage shape, wing shape, etc. All of these simulations are under idealistic conditions, there are so many uncertainties under water. In these cases, the effect of deformation on the shape of wing is ignored.

ISSN: 2414-1895 DOI: 10.6919/ICJE.202011_6(11).0033

6. Conclusions

Overall, the Computational Fluid Dynamics (CFD) software can provide the simulation results productively and precisely. Although the model shape is complicated, the flow field around an underwater glider can be estimated successfully. In 3D model, the convergence process plays an essential role in the simulation. The grid number will affect the results significantly. Only if the grid number is large enough, the results can be considered precisely. According to the results, the sweep angle will affect the maximum lift force and the minimum drag force. However, the simulation results have its limitations. Because the water velocity changes constantly, the attack angle alters too. The real situation is much more complicated. Many factors like the shape of the fuselage were not involved in the experiment.

References

[1] Stommel H. The SLOCUM Mission[J]. Oceanography, 1989, 2(1): 22-25.

[2] Sherman J, Davis R E, Owens W B, et al. The Autonomous Underwater Glider “Spray”[J]. IEEE Journal of Oceanic Engineering, 2001, 26(4): 437-446.

[3] Eriksen C C, Osse T J, Light R D, et al. Seaglider: A long-range Autonomous Underwater Vehicle for Oceanographic Research[J]. IEEE Journal of Ocean- ic Engineering, 2001, 26(4): 424-436.

[4] Zhang S, Yu J, Zhang A, et al. Spiraling Motion of Underwater Gliders: Modeling, Analysis, and Experimental Results[J]. Ocean Engineering, 2013, 60: 1-13.

[5] Wang S, Sun X, Wang Y, et al. Dynamic Modeling and Motion Simulation for a Winged Hybrid-driven Underwater Glider[J]. China Ocean Engineering, 2011, 25(1): 97-112.

[6] Joung, T.H., Sammut, K., He, F., Lee, S.K., 2012. Shape optimization of an autonomous underwater vehicle with a ducted propeller using computational flow fluid study dynamics analysis. Int. J. Naval Architect. Ocean Eng. 4 (1), 44e56.

[7] Zhang, D., Song, B., Wang, P., Chen, X., 2017. Multidisciplinary optimization design of a new underwater vehicle with highly effificient gradient calculation. Struct. Multidiscip. Optim. 55 (4), 1483e1502.

[8] GONG Yu-long, MA Jie, LIU Yan-ji, ZHANG Kai, Flat Wing Designing and Hydrodynamic Analysis for the Laboratory Underwater Glider.