Proceedings of 2022 IEEE
International Conference on Mechatronics and Automation
August 7 - 10, Guilin, China

Aerodynamic Simulation of Aircraft Crusing Characteristics Based on FLUENT

Lijie Duan[1,2] , Liwei Shi[1,2*] , Shuxiang Guo[1,2,3*] , Pengxiao Bao[1,2]

[1] School of Life Science, Beijing Institute of Technology, Beijing 100081, China

[2] Key Laboratory of Convergence Medical Engineering System and Healthcare Technology, Ministry of Industry and Information Technology, School of Life Science, Beijing Institute of Technology, Beijing 100081, China

[3] Faculty of Engineering, Kagawa University, 2217-20 Hayashi-cho, Takamatsu 760-8521, Japan

E-mails: 3220211482@bit.edu.cn; shiliwei@bit.edu.cn; guoshuxiang@bit.edu.cn; 3220201341@bit.edu.cn

* Corresponding author

***Abstract* - The frame structure of the aircraft and the aerodynamic layout of the main components are closely related to the dynamic characteristics of the aircraft and the aerodynamic forces it receives, which determine the aircraft characteristics and performance indicators of the aircraft. In order to study whether the performance index of the control system meets the actual requirements and the influence of the change of the relevant initial conditions on the aerodynamic characteristics of the aircraft. According to the cruising requirements, we independent design of aircraft model, determine the calculation area of the aircraft and use a new mesh generation technique to generate an unstructured polyhedral mesh and also select a suitable calculation model according to the discrete characteristics of the equation and set the relevant initial conditions of the aircraft and at last uses ANSYS FLUENT to carry out the aerodynamic simulation calculation of the aircraft. The data analysis of the simulation results is achieved, the pressure contour, the vorticity contour, velocity vector contour and streamline contour of the aircraft surface are analyzed. Meanwhile, the lift coefficient, the drag coefficient and the optimum speed for high altitude flight of the aircraft are obtained. The study of the changing law can provide the basis and reference for the aerodynamic flight performance of the aircraft.**

Index Terms – Aerodynamic characteristics; Aircraft; Polyhedral meshing; Numerical simulation; ANSYS FLUENT

I. INTRODUCTION

In recent years, with the rapid development of computers, CFD has been widely used in the research and development of the aerospace field. CFD is based on a computer as a platform and uses related discrete mathematical methods to solve the problems related to fluid mechanics. Numerical simulation and research analysis are carried out, and then the law of the influence of airflow movement in the air on the aircraft at different times can be obtained.

There are two kinds of experiments for the analysis of aerodynamic characteristics of the aircraft. One is to study the aerodynamic characteristics by experimenting with physical objects, and the other is to study the aerodynamic characteristics through the method of computer simulation model. Although using the actual aircraft to conduct experiments can avoid errors caused by inaccurate simulation model settings, and can also avoid the imprecise influence of factors such as the simulated simulation environment, but the cost of conducting experiments through physical objects is very expensive. Moreover, it is difficult to control the environment during the experiment. Each experiment cannot be guaranteed to effectively verify the influence of aerodynamic characteristics changes on the aircraft, so the aerodynamic characteristics analysis of the aircraft is usually carried out by computer simulation model experiments. With the great improvement of computer performance, computer simulation technology has also been widely used in the analysis of the aerodynamic characteristics of aircraft. Simulation technology plays an important role in reducing the number of physical experiments of the system, speeding up the process of research and development, strengthening the comprehensive performance indicators of the replacement aircraft system, and saving research and development costs. The limitations of the traditional need for repeated experiments and the gas flow conditions that are difficult to set in the traditional experimental process can be simulated through computer simulation. However, through computer simulation experiments, detailed and accurate relevant information can be obtained only by setting limited assumptions. Based on a large amount of sufficient data in the simulation process, the change trend of the aerodynamic characteristics of the aircraft can be accurately obtained in most cases [1].

At present, the simulation research on the aerodynamic characteristics of aircraft through CFD technology has achieved rich academic achievements at home and abroad. Study the influence of the nacelle and fuselage of large aircraft on the aerodynamic characteristics of the wing [2][3], Studies the aerodynamic characteristics of non-planar wing aircraft [4], Studies the tilt rotor aircraft in the influence of different feature conversion modes on the aerodynamic characteristics of the aircraft [5], Studies the aerodynamic characteristics of the aircraft under the condition of high angle of attack[6], Studies the aerodynamic characteristics of heavy rain on different aircraft during take-off, cruise and landing [7], Studies the structure and aerodynamic coupling technology of the aircraft wing [8], Studies the aerodynamic and stability characteristics of the aircraft [9], Evaluates the influence of the aerodynamic characteristics of the aircraft through calculation and optimize the design of the fuselage [10], Studies the influence of the large aircraft slat rail on the aerodynamic performance of the aircraft [11], Optimizes the fuselage, wings and tail of the aircraft, and aerodynamic characteristics analysis [12], Studies the relationship between the actual sideslip angle and pressure coefficient through simulation [13], Analysis of the influence of design parameters on aircraft aerodynamic characteristics [14].

However, the design and selection of physical models for different types of aircraft, the discretization of control equations with different methods, the meshing of calculation areas with different types of methods, and the setting of boundary conditions may lead to deviations from the real situation. Therefore, it is necessary to master the principle of airflow flow through a large number of simulation experiments, to divide the calculation area grid and to explore and improve the calculation model.

The research is aimed at obtaining the influence on the aerodynamic characteristics of the aircraft during a high altitude cruising at a constant speed. First, independent design of aircraft model, use the SpaceClaim(SCDM) of the commercial software CFD to pre-processing the aircraft, and set the fluid area, use FLUENT MESHING to divide unstructured polyhedron mesh, and determine the boundary conditions. We optimize the divided polyhedral meshes and generate high-quality meshes. The pressure contour, the vorticity contour, velocity vector contour, streamline contour of the aircraft surface, the lift coefficient, the drag coefficient and the optimum speed for high altitude flight of the aircraft are analyzed. The research can provide a basis and reference for the aerodynamic flight performance of the aircraft.

II. MODELING

A. *Establishment of coordinate system*

Aerodynamic characteristics of aircraft are also an important factor affecting the nose shape design. The nose resistance generally accounts for 20%~40% of the zero lift resistance of the entire aircraft. Under a given dynamic pressure, the resistance mainly depends on the shape and infiltration area. Therefore, in order to achieve the best aerodynamic characteristics and stability of the aircraft, first of all, the model makes the nose very sharp and keeps its shape protruding far out in the face air flow, which helps to weaken the strength of shock waves and reduce wave resistance. Second, the slenderness ratio of the nose, the plane is the length of the nose and the nose section of the ratio of maximum diameter, the design is 2.5, the fuselage of the slenderness ratio of 9, can be compared with supersonic aircraft design plans, thus better drag in aircraft cruising at high altitude, guarantee the high order continuously in the airflow direction, the minimum guarantee within the surface curvature, boundary tangent vector continuously.

During the movement of the aircraft, the actual force exerted on the aircraft includes the gravity, the thrust of the aircraft engine and the aerodynamic force the aircraft. The degree of aerodynamic influence on the aircraft during flight is related to the angle of the air flow relative to the aircraft. The angle can be described by the velocity coordinate system and the body coordinate system. The schematic diagram of velocity and body coordinate system is as follows in Fig. 1.

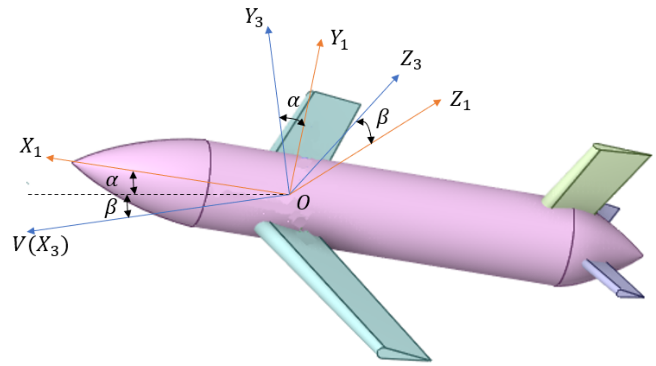

Fig. 1 Velocity and body coordinate system

We establish the velocity coordinate system $O{-}x_3y_3z_3$, taking the centroid of the aircraft as the origin O, the speed vector V of the aircraft overlaps with the Ox_3, which is defined upward as forwards direction, Oy_3 is perpendicular to Ox_3, Oz_3 is perpendicular to surface $x_3(O)y_3$, the velocity coordinate system $O{-}x_3y_3z_3$ is a dynamic seat and related to the velocity vector of the aircraft.

We establish the body coordinate system $O{-}x_1y_1z_1$, take the centroid of the aircraft as the origin O, Ox_1 overlaps with the vertical axis of the aircraft, and define the direction of the head pointing to the aircraft as positive, Oy_1 is perpendicular to Ox_1, Oy_1 is defined as the positive direction upward, and Oz_1 is perpendicular to surface $x_1(O)y_1$.

The connection between the velocity coordinate system and the body coordinate system can be determined by two angles α and β:

Angle of attack α: Project the velocity vector V of the aircraft onto a longitudinally symmetrical plane, The angle between the projection and the longitudinal axis of the aircraft is the angle of attack. If the longitudinal axis is above the projection line, the angle of attack is considered to be positive, otherwise it is negative.

Sideslip angle β: the angle between the aircraft velocity vector V and the longitudinal symmetry plane, if the airflow from the right to the aircraft, then the sideslip angle is positive, otherwise it is negative.

B. *Force analysis*

The aerodynamic force experienced by the aircraft during flight is due to the compressible viscous airflow passing through When the aircraft is on the surface, the pressure difference caused by the uneven pressure distribution on the entire surface of the aircraft and the combined effect of the viscous friction force of the air flow on the surface of the aircraft. The aerodynamic force R exerted on the aircraft during the flight is decomposed into drag X, lift Y and lateral force Z along the speed coordinate system for research. Through relevant experiments, it can be known that the strength of aerodynamic force is proportional to the dynamic pressure q of the air flow and the reference area S of the aircraft. Thus equation(1),

$$\begin{cases} X = C_x q S \\ Y = C_y q S \\ Z = C_z q S \\ q = \frac{1}{2}\rho V^2 \end{cases} \quad (1)$$

Where C_x,C_y and C_z are the drag coefficient, lift coefficient, and lateral force coefficient in turn, $\rho$is the atmospheric density, V is the speed of the aircraft, and S is the reference area of the aircraft. It can be observed from the formula that when the size of the aircraft, the speed of the aircraft and the flying height of the aircraft are known, the analysis of the aerodynamic force of the aircraft in space motion can be converted into the analysis of the corresponding aerodynamic factors.

1) Lift force

The overall lift of the aircraft during flight is the combined effect of the sum of the lift forces received by various parts of the aircraft and the additional lift generated by the mutual interference between each structural units. The formula for calculating the overall lift Y experienced by the aircraft is equation(2),

$$Y = C_y \frac{1}{2}\rho V^2 S \quad (2)$$

When the atmospheric density at the altitude at which the aircraft is flying and the size of the aircraft are known, the lift coefficient C_y is mainly determined by the flight Mach number M of the aircraft, the angle of attack α and the deflection angle of the rudder surface. Thus equation(3),

$$C_y = f(Ma, \alpha, \delta_z) \quad (3)$$

When the angle of attack and the rudder deflection angle are sufficiently small, the lift coefficient can be described as a linear function of α and δ_z. Thus equation(4),

$$C_y = C_{y0} + C_y^{\alpha} a + C_y^{\delta_z} \delta_z \quad (4)$$

Because the shape structure of the aircraft selected in this paper has axial symmetric, with $C_{y0} = 0$, so it can be further expressed as equation(5),

$$C_y = C_y^{\alpha} a + C_y^{\delta_z} \delta_z \quad (5)$$

2) Drag force

Drag is the component of the aerodynamic force acting on the aircraft in the direction of speed. In actual flight motion, the viscosity of air has a significant effect on the resistance of the aircraft, so the role of air viscosity should be fully considered in theoretical calculations. Usually, the air resistance of the aircraft is decomposed into zero liters related to lift Drag X_0 and lift-related induced drag X_i are studied. The air resistance of the aircraft can be expressed as equation(6),

$$X = X_0 + X_i \quad (6)$$

Define the drag coefficient equation(7),

$$C_x = \frac{X}{\frac{1}{2}\rho V^2 S} \quad (7)$$

III. Mesh Generation Technology

The length of the aircraft set in this paper is 72.5m, the width is 65m, and the height is 18.75m. In order to ensure that the calculation results are as close to the real results as possible, and the calculation amount is not too large, an appropriate calculation domain must be set for the simulation model. After repeated experiments, we arrived at a relatively reasonable computational domain. The periphery is a cuboid with a length of 286.25m, a width of 218.75m and a height of 156.25m, in order to simulate the area of airflow around the aircraft. In order to obtain accurate aerodynamic simulation results, it is necessary to deploy dense grids around the aircraft. At the same time, in order to ensure that the far-field boundary conditions are met, the corresponding calculation area must be large enough to avoid the error caused by the reflection of the boundary of relevant value in the space occupied by fluid motion [15].

Meshing plays a vital role in numerical calculation, and the quality of meshing determines the convergence and accuracy of simulation calculation. Generally, the following conditions should be met:

(1) The grid lines must fit the body;
(2) Ensure the regularity of grid line division;
(3) The grid lines should be perpendicular to each other;
(4) The grid lines should be smooth enough;
(5) The grid side lengths should be as equal as possible to ensure accuracy;
(6) Meshing should be fairly fast.

In this paper, according to the layout of the aircraft, the calculation area is divided into a polyhedron grid according to the actual situation. The total number of grids is 2.4 million, the number of nodes is 12.6 million and the number of faces is 16.3 million. This technique has the advantages of high accuracy, high efficiency, good quality convergence and wide Applicability [16].

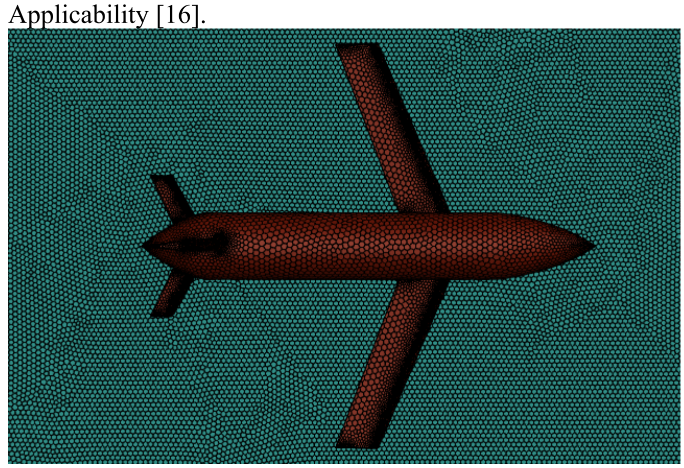

Fig. 2 Aircraft Polyhedron Meshing

It has many adjacent units, so it can calculate the gradient of the control volume more accurately and reasonably predict the local flow distribution. Polyhedral meshes are also less sensitive than tetrahedral meshes to mesh deformations caused by stretching, squeezing. Polyhedral mesh elements can be automatically joined, split, or modified by adding new nodes, elements or faces [17]. The polyhedron meshing of the aircraft is as follows Fig. 2.

IV. Experiment And Result Analysis

A. Initialization and Calculation

During the movement of the aircraft in space, in addition to aerodynamic force, the forces acting on the aircraft mainly include engine thrust and gravity. During the movement of the aircraft, the balance of these forces needs to be maintained. Turbulence is a state of fluid flow. When air flows, turbulence will be formed near the surface of the aircraft due to the difference in the surface structure of the aircraft and the thermal effects of air density differences and changes in air temperature, which will greatly interfere with the aerodynamic force of the aircraft. Therefore, in the process of simulation calculation, it is necessary to select an appropriate turbulence model to simulate the influence of turbulence on the aerodynamics of the aircraft during the actual flight. After comparison, a two-equation (k-omega) turbulence model was selected. This model solves two transport equations related to eddy viscosity. The calculation amount is relatively small, and it has shown good results [18]. We select standard initialization. The calculation time step is 0.01 seconds and the number of time steps is 200 steps.

In this paper, the standard atmosphere is set at an altitude of 7000 meters above sea level. The free flow is 0.53 (M), the atmospheric temperature during flight is 243 (K), the atmospheric pressure is P=41082 (Pa), and the air density ρ =0.5901 (kg/m^3), the acceleration of gravity is 9.78514. The authenticity of the simulation experiment is ensured by setting the scale factor of the grid domain.

B. Aerodynamic simulation and analysis

1) Pressure couture and vorticity contour analysis

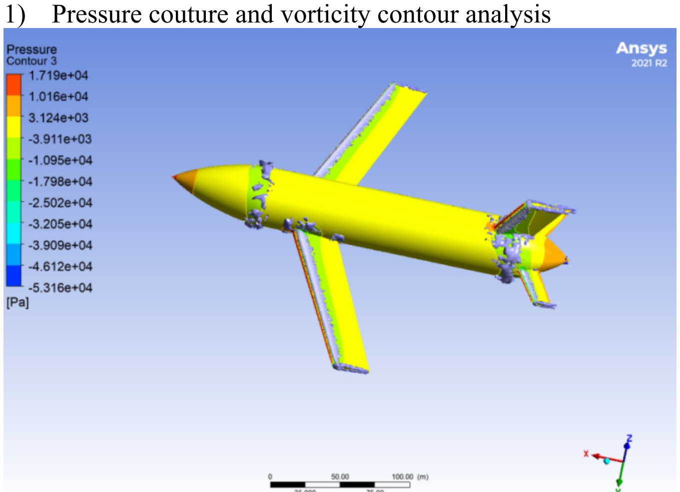

Fig. 3 Pressure Contour and Vorticity Contour of Aircraft

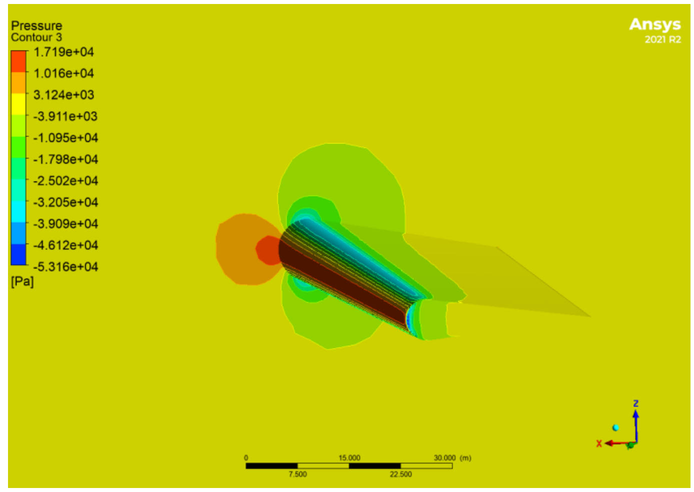

Fig. 4 Pressure Contour of Aircraft Wing

From the pressure contour Fig. 3 and Fig. 4, we can see that the highest pressure occurs in the nose and the airfoil head, because the impact loss of the air flow during high-altitude flight will cause the minimum speed of the head. When the aircraft is flying at a uniform speed at high altitude, and the angle of attack relative to the ground is 0, the average airflow on the lower surface of the wing shows a trend of slow-fast-slow change, which is caused by the enhancement of airflow reflection. Since there is no trailing edge separation vortex, the differential pressure drag is almost zero when the aircraft is flying in parallel at high altitudes. The maximum velocity of the upper and lower surfaces will appear at the place where the degree of arch is the greatest, and the lowest point of pressure also occurs at the place where the degree of arch is the largest, which is caused by the large degree of arch and the squeezing of streamlines [19]. The pressure builds up as the airflow advances toward the trailing edge.

Vorticity is caused by fluid separation. When laminar air rapidly flows over the surface of the wing, due to the viscosity of the fluid, the velocity of the fluid close to the surface will continue to decrease. When the air velocity drops to zero, it reaches the critical state, at which time the fluid separates from the aircraft surface, and then the laminar flow gradually develops into turbulence. From Fig. 3, As the air flows across the surface of the wing, the air on the lower surface of the wing curls up on the tip of the wing to form eddies. The air pressure on the lower surface of the wing is high and the air pressure on the upper surface is low, so the air on the lower surface of the wing moves towards the lower pressure area, creating a vortex at the tip of the wing. At the same time, eddies are generated at the whole front end of the aircraft wing, and the eddies are further reduced from the front end to the back end of the wing until they disappear. The wing wake produces fewer eddies, which makes the fluid flowing through the upper surface faster and lower pressure, so it generates upward lift, so it has good aerodynamic characteristics.

2) Velocity vector diagram analysis

It can be seen from the velocity vector diagram Fig. 5 that the direction of the incoming air is heading towards the aircraft, because the aircraft is flying forward at high speed in the high altitude, so the speed vector of the aircraft is the forward direction. When the aircraft and the air flow meet, the velocity at the tail of the wing decreases significantly, which is beneficial to reducing the impact on the wing and protecting the operation of the aircraft security and stability.

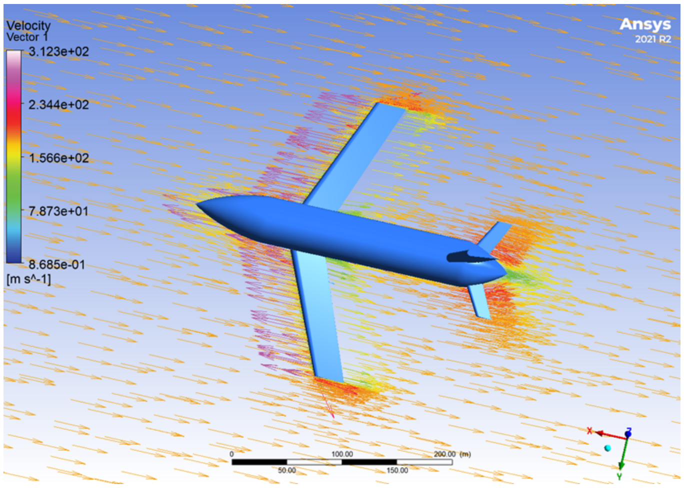

Fig. 5 Velocity Vector Diagram

3) Streamline diagram analysis

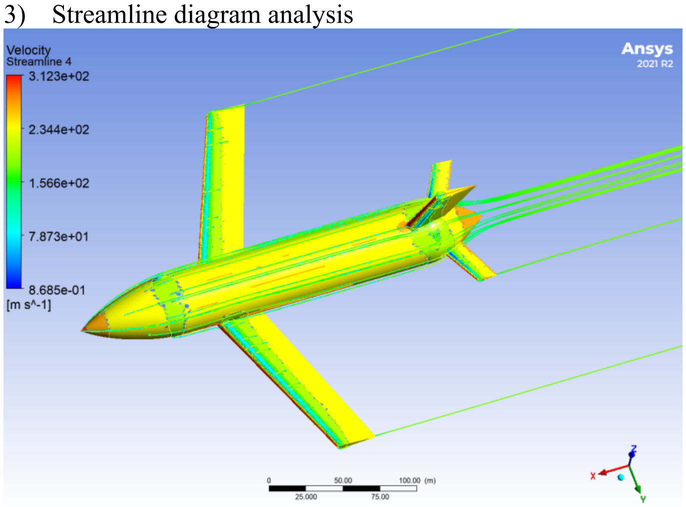

Fig. 6 Flow Line Diagram

It can be seen from the streamline diagram Fig. 6 that the flow of this aircraft in this paper is similar to the "body-fitted" flow around the body, which is an extremely favorable situation for engineering applications.

4) Analysis of lift drag coefficient

Select the time step as 200 steps and the unit time as 0.01s. From Fig. 7 the trend of the curve, because the aircraft is flying at a constant speed at high altitude, the average drag coefficient increases first, and then tends to be stable, when the aircraft speed is about 0.8 the average drag coefficient is 0.14, From Fig. 8 the average lift coefficient is 0.3, while the lift coefficient, the change trend is more violent at the beginning, and then decreases, with a certain periodicity. The magnitude of lift changes significantly faster than drag, and the lift on the upper and lower surfaces provides the thrust required by the aircraft [20].

Through multiple experiments, the lift and drag coefficients of an aircraft flying at different speeds and uniform speeds are shown in Table I. As can be seen from the table, the aerodynamic characteristics of the aircraft are best at speeds of around 0.8 (M).

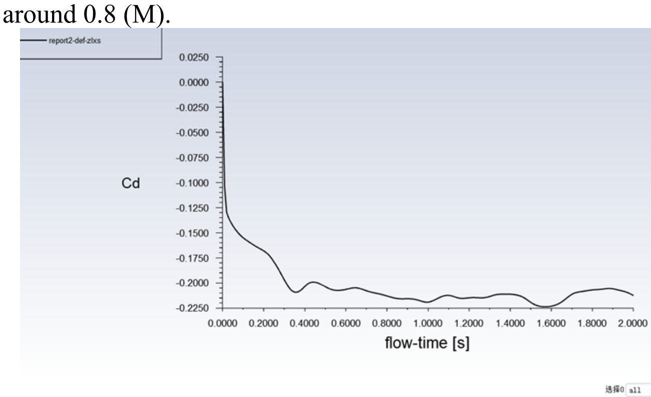

Fig. 7 Aircraft Drag Coefficient Diagram

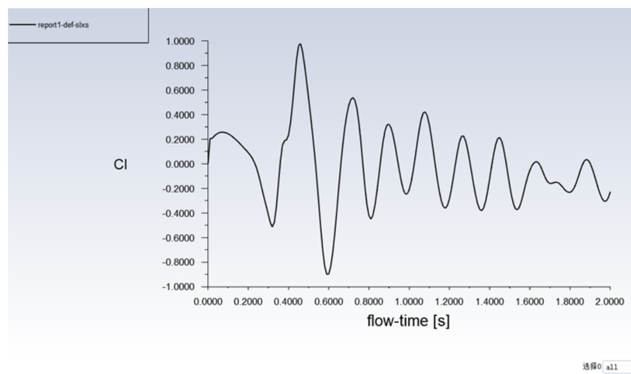

Fig. 8 Aircraft Lift Coefficient Diagram

TABLE I
DRAG-LIFT COEFFICIENTS AT DIFFERENT SPEEDS

Aircraft speed(M)	Drag coefficient	Lift coefficient
0.85	0.16	0.4
0.8	0.14	0.4
0.75	0.17	0.4
0.7	0.19	0.4
0.65	0.22	0.4

V. CONCLUSION

Because the mesh division plays a vital role in the study of the aerodynamic characteristics of the aircraft, the unstructured polyhedron mesh division technique is adopted in this paper. As a new mesh technology, compared with other meshing methods, polyhedral meshes are convenient, quick to generate, have good adaptability to complex shapes and have little dependence on the number of meshes. There are more adjacent elements and high calculation accuracy. Thanks to its unique grid structure, the numerical simulation can converge quickly, and it can directly learn from the mature algorithms of grid technology such as structural and non-structural, which has a great space for development.

In this paper, we use the independent design aircraft physical model set an appropriate fluid calculation domain that can ensure the calculation results are as close to the real results as possible, and the calculation amount is not too large. The calculation area and the aircraft are divided into unstructured polyhedron meshes, and then the aircraft is level flying at high altitude by using CFD technology.

The first, by independently designing the model of the aircraft, the resistance of the aircraft itself to the flight process can be reduced.

For the aircraft model, through aerodynamic simulation calculation and analysis, it can be seen that when the aircraft is cruising at high altitude, the fluid flow through the upper surface is fast and the pressure is low, so the upward lift force is generated. And the upper and lower surfaces of the wing produce pressure difference, so that the air flies to the low-pressure area, produce vortex at the wing tip, which is conducive to the smooth flight of the aircraft.

It can be seen from the velocity vector diagram that when the aircraft and air flow converge, the velocity decreases significantly at the tail of the wing, which is conducive to reducing the impact on the wing.

In addition, the flow of the aircraft in this paper is a close-fitting flight around the body, which is beneficial to engineering applications. The optimal speed of aircraft cruise at high altitude is about 0.8, at present, the average drag coefficient of the aircraft is 0.14, the average lift coefficient is 0.3, there is a certain periodicity, the change of lift force is significantly faster than the resistance, the lift force on the upper and lower surfaces to provide the required thrust for the aircraft.

To sum up, which provides certain theoretical a reference for the later design and optimization of the aircraft and the aerodynamic performance of the aircraft.

Acknowledgment

This work was supported by National Natural Science Foundation of China (61773064, 61503028).

References

[1] K. Dai, Simulation research on aerodynamic characteristics of aircraft based on FLUENT, Eds. Xian: Gongda Press, 2015.

[2] W.H. Wu, Z.L. Fan and D.H. Chen, "Fine optimization design of aerodynamic layout of large aircraft based on adjoint operator," Chinese Journal of Aerodynamics, vol. 30, no. 6, pp. 719-724, 2012.

[3] W.H. Wu, Y. Tao and D.H. Chen, "Aerodynamic layout optimization technology based on adjoint operator and its application in large aircraft Application in Wing Drag Reduction," in Journal of Aerodynamics, vol. 26, no. 7, pp. 1583-1589, 2011.

[4] SURESH C, RAMESH K and PARAMAGURU V, "Aerodynamic performance analysis of a non-planar Cwing using CFD," in Aerospace Science and Technology, vol. 40, no. 1, pp. 56-61, 2015.

[5] P. Li, Q.J. Zhao and Q.X. Zhu, "CFD calculations on the unsteady aerodynamic characteristics of a tiltrotor in a conversion mode," Chinese Journal of Aeronautics, vol. 28, no. 6, pp. 1593-1605, 2015.

[6] BOELENS O J, "CFD analysis of the flow around the X-31 aircraft at high angle of attack," Aerospace Science and Technology, vol. 20, no. 1, pp. 38-51, 2012.

[7] Ismail, M. Wu, Zhenlong, Abu Bakar, Tariq and Sobia. "Aerodynamic Characteristics of Airfoil Cruise Landing and High Lift Configurations in Simulated Rain Environment," Journal of Aerospace Engineering, vol. 28, no. 5, 2015.

[8] OKTAY E, AKAY HU and MERTTOPCUOGLU O. "Parallelized structural topology optimization and CFD coupling for design of aircraft wing structures," Computers and Fluids, vol. 49, no. 1, pp. 141-145, 2011.

[9] BITENCOURT LO, POGORZELSKI G, DE FREITAS RM, et al. "A CFD-based analysis of the 14-Bis aircraft aerodynamics and stability," Journal of Aerospace Technology and Management, vol. 3, no. 2, pp. 137-146, 2011.

[10] NICOLOSI, FABRIZIO, DELLA VECCHIA, PIERLUIGI, CILIBERTI, DANILO, et al. "Fuselage aerodynamic prediction methods," Aerospace science and technology, vol. 5, pp.332-343, 2016.

[11] X. Xu, R.P. Zhang and L. Zhang, "Research on the influence of large aircraft slat slides," Journal of Aerodynamics, vol. 34, no. 3, pp.368-372, 2016.

[12] LEE and JUHE, "Computational analysis of static height stability and aerodynamics of vehicles with a fuselage, wing and tail in ground effect," in Ocean Engineering, 2018, pp. 168.

[13] Y.P. Li and C.L. Jia, "Simulation research on accurate measurement of aircraft sideslip angle static pressure difference," Computer simulation Zhen, vol. 33, no. 10, pp.82-85, 2016.

[14] W.F. Yan, J.H. Wu and Y.L. Zhang, "The influence of key parameters of distributed propulsion on the aerodynamic characteristics of Xiang," Journal of Beihang University, vol. 41, no. 6, pp.1055-1065, 2015.

[15] R. Wang, Z.P. Hu, X. Ren, F.T. Li and W. Wen, "Key Issues in 2.5D Finite Element Modeling—Boundary Conditions, Mesh Division and Computation Domain Selection," Chinese Journal of Vibration Engineering, vol. 34, no. 1, pp.80-88, 2021.

[16] H.G. Zhang, S.Y. Tang, H.R. Yue, et al. "Comparison of Computational Fluid Dynamic Simulation of a Stirred Tank with Polyhedral and Tetrahedral Meshes," Journal of chemistry and chemical engineering, vol. 39, no. 4, pp.311-319, 2022.

[17] H.F. Chen, X.Q. Zhou, Z.B. Feng and S.J. Cao, "Application of polyhedral meshing strategy in indoor environment simulation: Model accuracy and computing time," Indoor and Built Environment, vol. 31, no. 3, pp.719-731, 2022.

[18] Shyam S. Das,Jose C. Páscoa,M. Trancossi and A. Dumas, "Computational Fluid Dynamic Study on a Novel Propulsive System: ACHEON and Its Integration with an Unmanned Aerial Vehicle," Journal of Aerospace Engineering, vol. 29, no. 1, 2015.

[19] M. Cen, L. Li and J. Li, "Analysis of the effect of camber on airfoil performance based on FLUENT," China Rural Water Resources and Hydropower,vol. 9, pp.128-133, 2008.

[20] D.B Li, N. Shu and H.Y. Wang, "Analysis of lift-drag coefficient of two-degree-of-freedom flapping-wing aircraft," Mechanical Engineer, vol. 7, pp.48-51, 2015.

[21] W. Ke and Z. Ze-You, "Numerical Simulation System of Aerocraft Aerodynamic Characteristics for Intelligent Internet of Things," IEEE International Conference on Industrial Application of Artificial Intelligence, Feb. 2020, pp. 334-338.

[22] Q. Wang, M. Zhang, J. Luo and L. Zheng, "Investigation of Icing Effects on Aerodynamic Performance for the High-Lift Configuration of Civil Aircraft," International Conference on Mechanical and Aerospace Engineering, Dec. 2021, pp. 113-118.

[23] Y. Pandya et al., "Aerodynamic characterization of a model aircraft using wind-tunnel testing and numerical simulations," First International Conference on Recent Advances in Aerospace Engineering, Mar. 2017, pp. 1-6.

[24] Z. Wei-Yi, Z. Zhong-Liang, Y. Hai-Yong and W. Xiao-Bin, "The Impact and Mechanism Analysis of Effect of Incoming Flow Velocity on Aerodynamic of Flying Wing Model with Low Aspect Ratio," International Conference on Mechanical Engineering and Automation Science, Jul. 2021, pp. 224-229.