COMPUTER SIMULATION ANALYSIS OF AIRCRAFT CIRCULAR TUBE FLOW FIELD BASED ON FINITE ELEMENT METHOD

Rongrong Sun
Binzhou University
Binzhou, China

Abstract: Flow field characteristics on aircraft surface have an important influence on flight stability. Good flow field characteristics of circular tube can effectively reduce the friction heat generation on the surface. The finite element method is used to simulate the velocity field, pressure field and density field of aircraft circular tube based on FLUENT. The external flow field model of blunt cone of ball head is established, and the reasonable boundary conditions and turbulence models are selected. The results shows that there is an obvious shock wave structure in the flow field, and the physical parameters are clearly stratified, the places with the highest temperature and pressure in the flow field appear near the leading edge of the tube.

Keywords: computer simulation, finite element, FLUENT, boundary condition

I. INTRODUCTION

In the field of hypersonic research, aerodynamic heating affects the development of modern high-speed aircraft to a great extent, and limits the development of aircraft. In high-speed flying aircraft, due to the viscous friction effect of air, a large amount of kinetic energy is transformed into thermal energy, which makes the air temperature on and near the aircraft surface rise rapidly [1]. This physical phenomenon is called aerodynamic heating. If the flying speed of the high-speed aircraft is lower than a certain condition, the heat generated is not large and easy to emit. When the speed exceeds the limit condition, the huge heat generated by aerodynamic heating cannot be dissipated, which causes the surface temperature of the aircraft to rise sharply, which will lead to the structural strength of the aircraft body material weakening, rigidity decreasing, and the body shape being damaged, even out of control.

With the continuous improvement of flight speed and altitude of aircraft, the problem of aerodynamic heating will become more and more serious. The higher the flight speed, the stronger the aerodynamic heating and the higher the temperature of the external environment, it is likely to cause the change of the surface shape of the aircraft, which will affect the structural strength and stiffness of the aircraft. High temperature environment poses a great threat to the internal equipment, which may lead to electrical drift, circuit instability or malfunction of internal electronic devices, and in serious cases will cause ablation and loss. This has a serious impact on the normal flight and precise guidance of aircraft, and may even lead to flight failure. In order to ensure the flight stability and precision of the aircraft, it is necessary to focus on the aerodynamic heating in the design to provide the necessary basis for structural strength design and thermal protection system design [2]. The purpose of this paper is to calculate the distribution of surface physical parameters of aircraft and explore the factors affecting the aero thermal calculation, so as to lay a foundation for the thermal protection design of aircraft.

II. SIMULATION CALCULATION METHOD

A. Overview of CFD

CFD is a systematic calculation and analysis of physical phenomena including fluid flow and heat exchange by means of computer numerical calculation and image display [3]. The basic idea of CFD can be simply summarized as follows: the physical parameters in space and time are replaced by some discrete points, then the mathematical equations of the relationship between variables are established by the prescribed principles and methods, and the approximate values are obtained by solving the equations. CFD can be described as the basic flow equation, and then the numerical simulation is carried out by computer input. The detailed distribution of physical parameters and the mathematical relationship between each parameter and time can be obtained by systematic simulation calculation. In addition, we can get other physical quantities by combining relevant theoretical formulas [4].

There will be some errors in computer simulation calculation. This kind of error is unavoidable. This kind of error includes two parts: truncation error and rounding error. Among them, truncation error is caused by calculation method, which can be reduced by encrypting grid. Rounding errors are determined by the number of digits computed by the computer.

B. Turbulence model

The main methods for simulating turbulent flow and heat transfer can be divided into large eddy simulation, direct simulation and heat transfer simulation. Reynolds average simulation method. Reynolds average simulation method is now widely used in numerical simulation. In this method, unsteady state control equation is used to calculate time averagely, and mathematical model is needed. This is also the basic method used in engineering turbulence calculation at present. Discrete scheme is a kind of interpolation method, which includes discrete control equation, discrete equation and interpolation to obtain physical parameters. First of all, the discrete equation can be established by interpolation. The interpolation method directly affects the results of calculation. The AUSM discrete scheme can keep the shock stability, pressure and density always positive, and the discrete scheme

does not need to be modified by entropy. In this paper, numerical simulations are carried out on the basis of the finite volume method, and the second-order upwind AUSM scheme is used for interpolation calculation.

C. Mesh generation method

The grid is a series of discrete points in the computational domain. When solving the flow problem by numerical method, the first thing to do is to discretize the governing equation. How to discretize in the computational domain is the grid generation technology. At present, the further development of CFD depends on the improvement of computer performance and the deepening of numerical calculation methods. On the other hand, grid generation technology has a great impact on its development process. Under perfect simulation conditions, the grid factor accounts for 80%-90% of the whole simulation project cycle, and the quality of the generated grid directly determines the accuracy and convergence speed of the calculation results.

Generally, structured grids are used to solve complex engineering problems. With the permission of computer configuration, structured grids can control the quality of grids more easily, and ensure the quality of boundary layer grids, which can ensure easy convergence. Unstructured grids have lower requirements on the model and less workload, but the generation period of grids is longer, the quality of grids is difficult to control, and the boundary layer is difficult to control. Grid quality cannot be guaranteed. According to the overall shape of the model, the whole computational domain is divided into several sub-domains at first, then the meshes are generated for each sub-domain separately, and finally the numerical calculation is carried out. The information transfer between sub-domains is transmitted through grid nodes at each boundary. When partitioning blocks, make the shape of the regional model as simple as possible, so as to facilitate mesh generation [5].

D. Definite solution conditions and convergence conditions

The initial condition is to input an initial physical parameter before calculating the flow field. For unsteady flow problems, an initial value of the computational variables needs to be given before calculation, and the change value of the flow field variables can be obtained by calculating the initial value of the computational variables according to the set time step. For the steady flow problem, a physical parameter is also given, and then a steady solution is obtained by iteration calculation. The boundary conditions at the entrance of this paper are set as free infinite boundary conditions. In ANSYS FLUENT, the boundary conditions for pressure lifting are usually used to simulate free flow at infinity, which is also called typical boundary conditions. The density of the boundary condition is calculated by the law of ideal gas when the pressure is far raised. In this paper, the pressure outlet boundary conditions provided by ANSYS FLUENT are used. The variable values of pressure boundary conditions are determined by the calculation results of internal flow, and zero gradient extrapolation is used.

Symmetric surface boundaries are generally used for the physical symmetry of the solved region. Reasonable application of symmetric surface boundary conditions can effectively reduce the workload and greatly improve the efficiency of solution. Under the symmetrical boundary condition, the velocity of the vertical boundary is set to zero, and the values of other physical quantities outside the boundary of the computational domain are equal to those of the nodes within the boundary of the corresponding computational domain.

III. FLOW FIELD ANALYSIS OF AIRCRAFT CIRCULAR TUBE

A. Establishment of model and setting of solving parameters

The model is a regular and symmetrical circular tube structure. Before meshing, the model is simplified to 1/2 circular tube model, which greatly reduces the amount of calculation and saves a lot of time. The ANSYS Mesh software is used to mesh the external flow field of the model, and the total number of meshes is 8,328. The meshing situation is shown in Fig.1. Structured quadrilateral mesh is used to ensure the orthogonality of the element mesh as much as possible. In order to ensure the accuracy of the simulation, the boundary layer of the flow field is refined, that is to say, there are at least 10 layers of grids in the boundary layer of the flow field.

In FLUENT, a double-precision calculator is used to select the coupling implicit method. The single-precision and double-precision are included. Usually, the single-precision calculating speed is higher. When meshing, the near wall is refined, and the physical parameters are extracted from the boundary layer mesh. Therefore, a dual-precision solver is selected here. Because the coupling implicit solver is used here, the initial Courant number is 5. At the beginning of the calculation, Courant is set to a smaller value to start the calculation. According to the fluctuation of the residual, the value of Courant number can be increased slowly.
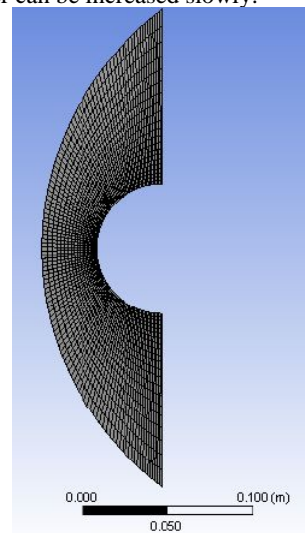

Fig.1 Grid diagram of the outer flow field of a circular tube

B. Verification of calculation results

The flow field analysis results based on CFD are shown in Fig. 2-5. Compared with the wind tunnel test results, the shape and position of the flow field shock wave obtained in this paper are in good agreement with the experimental results. The maximum temperature after wave is about 2258K, the error is 4.19%, and the error is less than 20%. The maximum pressure after shock wave is 3566 Pa. The error is 11.4%, less than 20%, within the acceptable range. The maximum velocity in the flow field is the inlet velocity, the velocity value is 2015 m/s, the error is 8.4%, within the acceptable range. By comparing and analyzing the above results, it can be seen that the simulation results of ANSYS software for this example are reliable.

From the above results, it can be seen that there is an obvious shock wave structure in the flow field, and the physical parameters are clearly stratified. The velocity of the flow field changes most closely to the wall, and the air flows violently. At the same time, the places with the highest temperature and

pressure in the flow field appear near the leading edge of the tube.
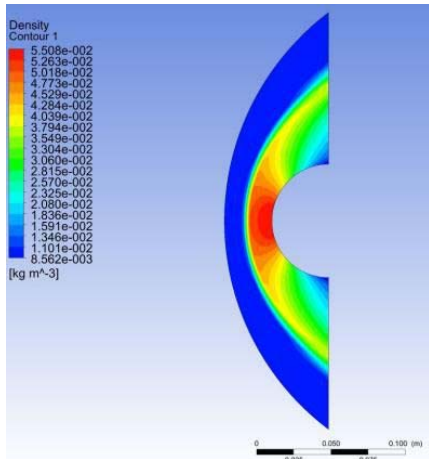

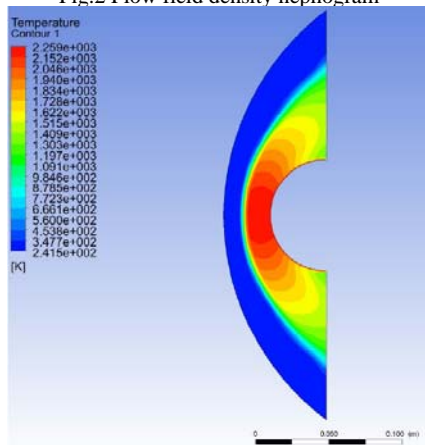Fig.2 Flow field density nephogram

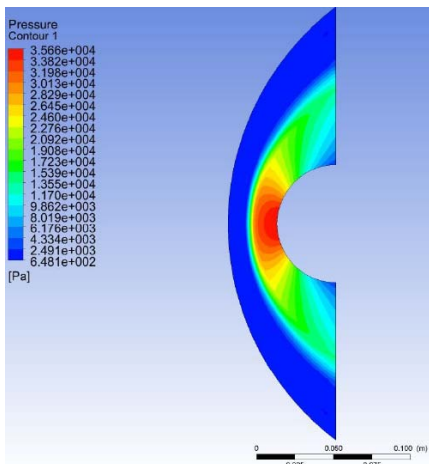Fig.3 Temperature distribution map of outer flow field
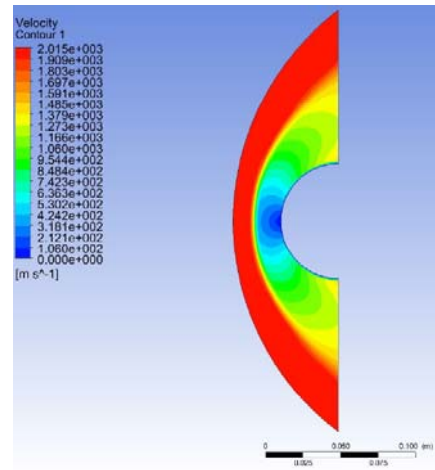

Fig.4 Pressure distribution map of outer flow field

Fig.5 Velocity distribution map of outer flow field

IV. NUMERICAL SIMULATION OF PNEUMATIC HEATING

A. Selection of model and setting of simulation conditions

Ball head blunt cone is a very typical aircraft model, which has the basic characteristics of general aircraft. Whether in the past or now, the study of flow field characteristics of blunt cones with spherical heads is a hot topic in the field of aerothermal research. When flying at high speed, there are complex shock waves in the head of the aircraft. Because of the different flight speed, altitude and attitude, it is more difficult to calculate the characteristics of the external flow field. Because of the simplicity of the two-dimensional pipe model, the description of the flow field is not as intuitive as that of the three-dimensional model, and the intuitive three-dimensional graph is helpful to deepen the understanding of the complex flow field, so the three-dimensional spherical blunt cone is used in this example.

In ANSYS Geometry software, the external flow field model of blunt cone of ball head is established, as shown in the figure. The model of this example is a symmetrical cone. In the simulation, the model is simplified to 1/2 of the 3D cone. In order to divide high quality grids, the flow field is divided into 10 regions.
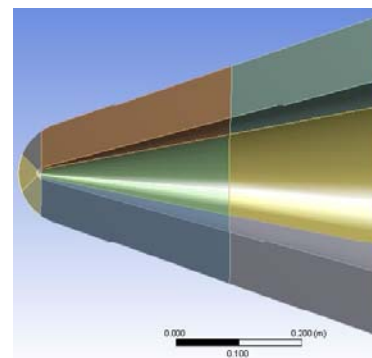

Fig.6 Outer flow field model of blunt cone with spherical head

Using ANSYS Mesh software to mesh the flow field, using structured hexahedron mesh, the model divides 10 regions to process spherical head and surface body-fitted mesh more conveniently. The meshing situation is shown in Fig.7, which avoids the grid distortion caused by one partition. After dividing the region, each region can be maintained well. The

hexahedral grid structure has 8.34 million grid nodes in the flow field area.

B. Analysis of flow field characteristics

The flow field characteristics can be obtained by continuous iteration, in which the temperature and density flow fields are shown in Fig. 7 and Fig.8, respectively. The temperature T at the stagnation point of the flow field is 1169K, with an error of 0.7%. The pressure at the stagnation point of the flow field is P=18590Pa, with an error of 2.1%. The errors of the above two parameters are far less than the expected errors and meet the calculation requirements. The highest velocity in the flow field is 1502 m/s, which is the same as the input velocity under the boundary conditions. Through the above comparison, it is concluded that this chapter is correct and effective to use ANSYS software to calculate the example. The above calculation results prove that ANSYS software can not only simulate the two-dimensional model, but also simulate the aerodynamic heat of the three-dimensional model.

The distribution of physical parameters of the flow field can be clearly seen from the above cloud images of temperature, pressure, density and velocity distribution. The calculated results of the flow field directly show the characteristics of hypersonic flow field. In the flow field, the front edge of the ball head is strongly compressed, resulting in body-fitted shock wave. The air flow parameters change before and after the shock wave. The temperature, pressure and density of the front edge of the ball head (standing point region) increase, and the velocity tends to zero. For the head of the model and other areas, the location of the shock wave can be well described, and the cloud image can truly reflect the physical phenomena of the flow field. Among them, there is a large detached shock wave in front of the blunt cone of the ball head, and the distance between the shock wave and the object surface increases along the flow direction. The above phenomena all conform to the general physical law, and provide a clear and intuitive reference for the study of aerodynamic heat.
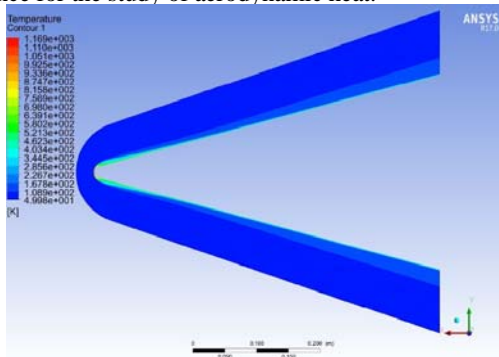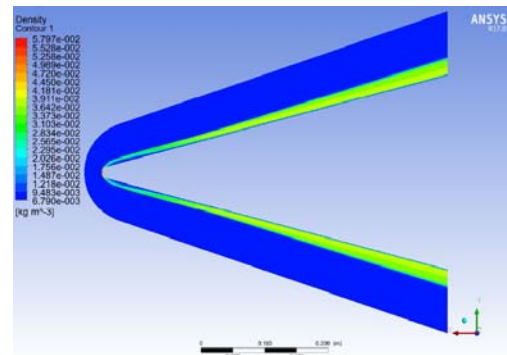

Fig.7 Temperature distribution map of outer flow field

Fig.8 Density distribution map of outer flow field

V. ACKNOWLEDGEMENTS

VI. CONCLUSION

The physical model is described in detail, and the heat transfer of the model is summarized and analyzed. Based on CFD theory and its advantages and disadvantages, the governing equation, turbulence model and discrete scheme of flow field are analyzed in detail. The types and characteristics of grids are summarized. The determination of definite solution conditions is described in detail. Finally, the convergence criteria of physical quantities for each equation are explained. A classic example of aerodynamic heating of two-dimensional pipe flow is used to simulate the calculation. The simulation results of a three-dimensional spherical blunt cone show that the method is also applicable to the three-dimensional model.

VII. REFERENCES

[1] Q. Long. "Reconstruction of blood flow patterns in a human carotid bifurcation: A combined CFD and MRI study," Journal of Magnetic Resonance Imaging, vol.11, pp.299-311, October 2015.

[2] W.G. Habashi, J. Dompierre. "Anisotropic mesh adaptation: towards user‐independent, mesh‐independent and solver‐independent CFD. Part I: general principles," International Journal for Numerical Methods in Fluids, vol.32, pp.206-209, October 2015.

[3] M.S. Gundeti. "Paediatric robotic-assisted laparoscopic augmentation ileocystoplasty and Mitrofanoff appendicovesicostomy (RALIMA): feasibility of and initial experience with the University of Chicago technique," Bju International, vol.33, pp.234-237, August 2016.

[4] L. Zheng. "Modal analysis of threshing drum based on ANSYS,". Agricultural Mechanization Research, vol. 31, pp. 407-411, January 2014.

[5] L. Dahmani. "Crack identification in reinforced concrete beams using ANSYS software," Strength of Materials, vol. 01, pp. 464-468, April 2014.