Research on Solving Ionization Equation of High-Speed Aircraft Based on Euler Monte Carlo Method of Fluid Mechanics

Ming Lu
Department of Mechanical and Electrical Engineering, Chizhou Vocational and Technical College, Anhui 247000, China
210664889@qq.com

Abstract. Taking the aerodynamic calculation of high-speed aircraft as an example, this paper introduces the modelling and application of computational fluid dynamics in aircraft design. The application of computational fluid dynamics in the design of aircraft is accompanied by the development of computer technology and aerodynamics. From the earlier solution of the Euler equation and the NS equation from a macro perspective, to the solution of the NS equation with chemical reactions and ionization that considers the influence of real gases, and from the micro perspective, the Monte Carlo method is used to directly simulate and solve the gas molecule motion, that is, It represents the development of computational dynamics theory and is also the result of the development of computer technology today. This article discusses the application of computational fluid mechanics, including the selection of mathematical models, the selection of calculation methods, and its application to special problems in aircraft design.

1. Introduction
In recent years, with the rapid development of aviation technology, the military field has put forward higher requirements for the flying height and speed of aircraft. The United States, Russia and other major aerospace countries have conducted a lot of research on high-altitude and high-speed aircraft, and have carried out relevant flight tests. This type of aircraft is mostly unmanned. The flight altitude is usually in the stratosphere, and the flight speed is above Mach number Ma3. It has the characteristics of fast speed, long range, strong manoeuvrability, high survivability, etc. It has great strategic significance in the future military field. The accuracy of the aircraft's basic flight parameters (such as speed, altitude, angle of attack, attitude angle, etc.) directly affects flight safety and flight performance. However, any airspeed system will have errors and must be corrected through flight tests. The high-altitude high-speed aircraft has a high-flying height, a fast flying speed, a low external atmospheric pressure, and a strong shock wave effect, which has a great influence on the airspeed measurement of the aircraft, especially the flying speed, which must be corrected.

Since the aeroelastic mechanics of aircrafts are multidisciplinary interdisciplinary subjects, their research issues also involve the difficulties of multiple disciplines such as fluid mechanics, solid mechanics and control theory. Therefore, the aeroelasticity of aircrafts has always been a must for aircraft design workers. Important research problems. In the past aircraft design process, the aircraft designer mainly obtained the aeroelastic characteristics of the aircraft through experiments. Compared
with the steady flow field experiment, the aeroelastic wind tunnel experiment requires the design of special experimental instruments. The experimental process is very complicated and the accuracy is limited. The risk factor of the aircraft flight test is very large. Therefore, it is generally difficult for aircraft design workers to accurately obtain the aeroelastic characteristics of the aircraft, and the aeroelastic characteristics of the aircraft can only be predicted from the perspective of structural mechanics mainly through some engineering calculation methods. For example, the frequency domain method used in most projects is based on linear lift surface theory. The aerodynamic solution can only meet the subsonic velocity and low Mach number of aeroelastic calculations. In the transonic phase of the aircraft, due to the turbulent boundary layer, the separation of the transonic flow, the unstable fluctuation of the shock wave, and the interference of the shock boundary layer, aerodynamics will have complex nonlinear characteristics The nonlinear change of aerodynamic force makes the aerodynamic force solved by traditional engineering algorithm far from the actual situation. For example, flutter, the flutter speed of many aircraft will have a unique "pit" phenomenon in the transonic phase, which greatly increases the possibility of flutter in the transonic phase. In the past, it was difficult for aircraft designers to accurately obtain the transonic speed aeroelastic characteristics of the aircraft at the design stage. Therefore, the aeroelastic characteristics of the subsonic speed of the aircraft can only be estimated, and the flight envelope should be strictly controlled to avoid instability of the transonic speed. When the aircraft encounters aeroelastic problems after going to the sky, it can only be controlled by some empirical methods, such as increasing the wing stiffness and adding spoilers. Such a design method not only has a very low efficiency for the design of the aircraft, but also has insufficient accuracy, and is prone to flight accidents, endangering the safety of the pilot, and even causing the aircraft design to fail. Therefore, there is an urgent need for an accurate and efficient method in the engineering design of aircraft to analyse the transonic aeroelastic characteristics of elastic aircraft and provide a method to effectively overcome the aeroelastic problem. With this as the background, the paper introduces the flow control equations in the three-dimensional rectangular coordinate system, defines the special initial boundary model conditions, and gives the discrete equations based on the finite volume method; at the same time, the three-dimensional SIMPLE algorithm is introduced in detail, and the solver is given. The design method has developed corresponding software [1].

2. Calculation of governing equations of fluid mechanics

2.1. Fluid mechanics control Bernoulli equation

2.1.1. Bernoulli equation on streamline. Bernoulli's principle is often expressed as \( \frac{P + 1}{2} \rho v^2 + \rho gh = C \).

This formula is called Bernoulli's equation. Where \( P \) is the pressure at a certain point in the fluid, \( v \) is the velocity of the fluid at that point, \( \rho \) is the fluid density, \( g \) is the acceleration of gravity, \( h \) is the height at that point, and \( C \) is a constant. It can also be expressed as \( \frac{P + 1}{2} \rho v^2 + \rho gh = \frac{P_i + 1}{2} \rho v_i^2 + \rho g h_i \). It should be noted that because the Bernoulli equation is derived from the conservation of mechanical energy, it is only suitable for ideal fluids with negligible viscosity and incompressibility. The Bernoulli equation on the streamline is shown in Figure 1:
Suitable for ideal fluids (no friction resistance). Each item in the formula represents the difference between the kinetic energy, potential energy and static pressure energy of the unit fluid. If the flow velocity is 0, the basic hydrostatic formula of the equilibrium fluid can be obtained from the Bernoulli equation:

$$z + \frac{p}{\rho g} + \frac{V^2}{2g} = C$$  (1)

2.1.2. Bernoulli equation of total flow. The total flow is the sum of countless elementary flows. By dividing the elementary flow Bernoulli equation along the total flow through the flow cross-sectional area, the Bernoulli equation of the total flow, that is, the total flow energy equation, can be derived.

$$z_1 + \frac{p_1}{\rho g} + \frac{V_1^2}{2g} = z_2 + \frac{p_2}{\rho g} + \frac{V_2^2}{2g}$$  (3)

The kinetic energy correction coefficient $\alpha$ is the ratio of the actual kinetic energy to the kinetic energy calculated at the average speed, and the $\alpha$ value reflects the unevenness of the cross-sectional velocity distribution. Because the dynamic viscosity value of the gas is small and the velocity gradient of the cross-section is small, the actual velocity distribution of the gas flow is relatively uniform, close to the average velocity of the cross-section. Therefore, the kinetic energy correction factor in gas movement is often taken as 1.0. Most of the water flow in the pipe also belongs to this situation. At this time, there is no difference in the form of the Bernoulli equation between the total flow and the streamline [2].

2.2. The Bernoulli equation of the actual total fluid flow

The Bernoulli equation of the actual total flow of fluid:

$$z_1 + \frac{p_1}{\rho g} + \frac{\alpha_1 v_1^2}{2g} = z_2 + \frac{p_2}{\rho g} + \frac{\alpha_2 v_2^2}{2g} + h_f$$  (4)
z represents the potential energy, position height or height head of the fluid per unit weight at a certain point (taken calculation point) on the total flow surface; \( \frac{P}{\rho g} \) represents the pressure of the fluid per unit weight at a point (taken calculation point) on the total flow surface. Energy, pressure tube height or pressure head; \( \frac{1}{2} \frac{av^2}{g} \) represents the average kinetic energy per unit weight of fluid flowing through the flow section, the average velocity height or velocity head; \( h_f \) represents the average mechanical energy loss per unit weight of fluid between the two ends of the total flow.

3. Related equations of governing equations of fluid motion

3.1. Mathematical model of fluid flow

The phenomenon of fluid motion appears in a large number in nature and various engineering fields, and its specific manifestations are various. The dynamic characteristics of any fluid motion are governed by the three most basic laws, namely the conservation of mass, the conservation of momentum and the conservation of energy. The law of conservation of mass: the increase in the mass of the fluid in the microelement per unit time is equal to the net mass flowing into the microelement in the same time interval. The law of conservation of momentum: the rate of increase of fluid momentum in a microelement is equal to the sum of various forces acting on the microelement from the outside. The law of conservation of energy: the rate of increase of energy in the micro-element body is equal to the net heat flux into the micro-element body plus the work done by the volume force and surface force on the micro-element body. These basic laws can be determined by mathematical equations. The energy conservation equation of fluid motion is expressed as follows:

\[
\frac{\partial (\rho u)}{\partial t} + \nabla (\rho u u) = -\rho \nabla p + \nabla (k \nabla T) + \phi + S_f
\]

Where \( \rho \) is the fluid density, \( u \) is the fluid velocity, \( k \) is the fluid thermal conductivity, \( \phi \) is the dissipation function, and \( i \) is the enthalpy per unit mass of gas. In the energy equation of the aircraft, \( \phi \) and \( s \) are both equal to 0.

3.2. Euler method

Now we express the above-mentioned Euler method of describing motion in mathematical formulas. To this end, we must first use a mathematical method to distinguish different fluid particles. Usually, the coordinates of the fluid particles at the initial time are used as a mark to distinguish different fluid particles. At the initial time, the coordinates of the fluid particles are \( (a, b, c) \), which can be curvilinear coordinates or rectangular coordinates \( (x, y, z) \). The important thing is to label the fluid particles without any specific method. We agreed to use a combination of three numbers \( a, b, c \) to distinguish fluid particles. Different \( a, b, c \) represent different particles. Therefore, the motion law of fluid particles can be expressed mathematically as the following vector form, as shown in picture 2.
Figure 2. Euler method of motion

\[ r = r(a, b, c, t) \]  \hspace{1cm} (6)

Where \( r \) is the loss of fluid particles. In a rectangular coordinate system, there is

\[ x = x(a, b, c, t) \]
\[ y = y(a, b, c, t) \]
\[ z = z(a, b, c, t) \]  \hspace{1cm} (7)

The variables \( a, b, c, t \) is called Euler variables. In equation (5), if \( a, b, \) and \( c \) are fixed and \( t \) is changed, then the motion law of a certain fluid particle can be obtained. If \( a, b, \) and \( c \) are changed at a fixed time \( t \), then the position distribution of different fluid particles at the same time is obtained. It should be pointed out that in Euler, the defined area of the lost-path function \( r \) is not a field, because it is not a function of spatial coordinates, but a function of the mass label. Now proceed from equation (4) to find the velocity and acceleration of fluid particles. Assume that the function determined by equation (4) has a second-order continuous partial derivative. Velocity and acceleration are the displacement change rate and velocity change rate per unit time for the same particle, let \( \vec{v}, \dot{\vec{v}} \) be the velocity vector and acceleration vector, then

\[ \vec{v} = \frac{\partial r(a, b, c, t)}{\partial t} \]  \hspace{1cm} (8)
\[ \dot{\vec{v}} = \frac{\partial^2 r(a, b, c, t)}{\partial t^2} \]  \hspace{1cm} (9)

Since \( a, b, \) and \( c \) do not change for the same particle, the above formula writes the partial derivative of time \( t \). In a rectangular coordinate system, the expressions of velocity and acceleration are
\[
\begin{align*}
\frac{\partial x(a,b,c,t)}{\partial t} &= u \\
\frac{\partial y(a,b,c,t)}{\partial t} &= v \\
\frac{\partial z(a,b,c,t)}{\partial t} &= w
\end{align*}
\]

(10)

\[
\begin{align*}
\frac{\partial^2 x(a,b,c,t)}{\partial t^2} &= \ddot{u} \\
\frac{\partial^2 y(a,b,c,t)}{\partial t^2} &= \ddot{v} \\
\frac{\partial^2 z(a,b,c,t)}{\partial t^2} &= \ddot{w}
\end{align*}
\]

(11)

3.3. Monte Carlo method

Now let’s introduce another viewpoint and method to describe fluid motion, namely Monte Carlo method. Different from the Euler method, the Monte Carlo method is different. The focus of the Monte Carlo method is not fluid particles but spatial points. Try to describe the change of fluid motion with time at every point in space. If the fluid motion at each point is already known, the motion of the entire fluid is also clear, so what physical quantity should be used to represent the change in fluid motion at the spatial point? Because different fluid particles will pass through a fixed point in space at different times, it is impossible to observe and record the detailed history of the passing fluid particles before and after standing on the fixed point. In other words, we can’t directly measure the change of the position of each particle with time like the Euler method. Nevertheless, the velocity of the fluid particles passing through the fixed point at different times can be measured, so it is very natural to use the velocity vector to describe the change of the fluid motion on the fixed point. Considering the situation described above, the motion law of fluid particles in the Monte Carlo method can be mathematically expressed as the following vector form. As shown in Figure 3.

![Figure 3. Monte Carlo method](image.png)
\[ v = v(r,t) \] (12)

In the Cartesian coordinate system are:

\[ u = u(x,y,z,t) \]
\[ v = v(x,y,z,t) \]
\[ w = w(x,y,z,t) \] (13)

To fully describe the condition of the moving fluid, you also need to give the state function pressure, density, temperature, etc.

\[ p = p(x,y,z,t) \]
\[ \rho = \rho(x,y,z,t) \]
\[ T = T(x,y,z,t) \] (14)

The variable \( x, y, z, t \) is called the Monte Carlo variable. When \( x, y, z \) is fixed and \( t \) changes, the function in equation (12) represents the law of speed change at a fixed point in space with time. When \( t \) is fixed and \( x, y, z \) changes, it represents the law of speed distribution in space at a certain moment. It should be pointed out that the velocities determined by formula (12) are defined on space points, they are a function of the coordinates \( x, y, z \) of space points, so we are studying fields such as velocity field, pressure field, density field, etc. Therefore, when we use Monte Carlo to describe motion, we can make extensive use of field theory knowledge. If the function in the field does not depend on the loss \( r \), it is called a uniform field; otherwise, it is called a non-uniform field. If the function in the field does not depend on the time \( t \), it is called the steady field, otherwise, it is called the unsteady field [3].

4. Application of computational fluid mechanics to special problems in aviation engineering

The mathematical physical model that CFD should be used in aircraft design is summarized above. In actual aircraft design, according to the design requirements of the aircraft, the use of a reasonable physical model in numerical calculation can make the calculation efficiency and calculation accuracy. You can get satisfactory results. For example, on the issue of high angle of attack, we have used the Euler and N-S equations to calculate an example in the transonic range, which proves that the N-S model is superior to the Euler model, although empirical viscosity correction is also added to the latter. In another problem, that is, when solving the problem of dropping external objects, the hydrodynamic problems involved can be solved by solving the Euler equation and the dynamic equation of the object's motion. Of course, it can also be directly solved by using the NS equation with viscosity, but it is time-consuming [4].

It can be seen that the two general fluid mechanics calculation programs can solve the problem through the use of perfect module design and reasonable calculation modules under different aircraft design requirements. At the same time, the program of computational fluid mechanics is constantly enriched and improved as the theory of fluid mechanics is continuously improved.

The problem of large angle of attack has always been a difficult problem in aircraft aerodynamic calculation. When the angle of attack is large, the vortex downward from the nose may be asymmetric, resulting in significant lateral forces and yaw moments. The flow interacts with the wing or tail. Experiments show that the asymmetric vortex wake is very sensitive to geometric disturbance, geometric shape and Reynolds number, and the largest asymmetric load appears in the boundary layer is neither complete laminar flow nor complete turbulence.

With the tremendous progress in computing conditions and computing technology, and the continuous deep research on the flow mechanism of experiments, the role of computational fluid
mechanics in the calculation of flow at high angles of attack has become greater and greater. Simulating the very important viscous effect in high angle of attack flow cannot be used, so only the NS equation can be used to simulate the compressibility and viscous effect at high angle of attack, including the effect of three-dimensional separation. The flow details of the flow vortex are revealed. The large-angle-of-attack technology projects underway at various NASA research centres in the United States are to develop and verify a N-S equation calculation program suitable for fighter jets. This project includes the test flight of the F-18 high angle of attack research aircraft, wind tunnel blowing test and comparison with numerical calculations. Some of the data below are taken from Ames' results [5].

Figure 4 is a comparison of Thomas's solution using the NS equation and Hummel's results. The calculated separation and reattachment lines are exactly the same as the experimental results. The calculation uses a total of 545025 grid points (129 points in the circumferential direction, vertical 65 points in the direction, 65 points in the longitudinal direction). Different grid densities have a great influence on the calculation results. In the numerical calculation of the flow at high angles of attack, it is necessary to ensure sufficient grid density to simulate the correct flow details.

![Figure 4. Streamline of delta wing surface](image)

Figure 5 shows the comparison between the calculation results of the laminar flow and the pressure distribution of the transition zone and the experimental results. From the calculation results, turbulence has a significant effect on the pressure distribution of the calculation results at the position where x/L can be greater than 0.7. The pressure coefficient obtained by the flow method is relatively high. It can be seen that the application of the turbulence model should be paid attention to in the calculation of large angles of attack. The turbulence model that conforms to the flow can obtain more accurate calculation results [6].
Figure 5. The calculation results (line) and test results (points) of the pressure distribution on the plane's half-wing span

From the calculation results of the surface vortex of the delta wing in Figure 6, it can be seen that the turbulence is very different from the surface vortex detachment position calculated by the laminar flow. The laminar surface vortex detachment position is at a position where x/L is equal to 0.5. Before the development, this is completely consistent with the results observed in the experiment.
5. Conclusion

Computational fluid mechanics is currently in a booming stage in the aerospace field, especially the multiplication of computer speed has led to a leap in computing power. A calculation model that uses the NS equation calculation program as the main body and the corresponding subroutines and parallel operation modules adapts to it. The wider advantage is increasingly accepted by people. It is foreseeable that this model will be extended to all aspects of aircraft design in various fields, thereby accelerating the development process of new designs and models. It is necessary to realize that although the flow can be simulated in this way, the selection of physical models, the discretization of control equations, the generation of grids, and the processing of boundary conditions all inevitably deviate from the actual situation, so the flow mechanism is found through experiments. The research and improvement of grid generation and calculation methods are also necessary auxiliary means to overcome difficulties in the application of computational fluid mechanics. It is worth mentioning that there is a lot of development of foreign commercial software. If you want to choose the software that suits your needs, the relevant personnel must also have the above-mentioned knowledge of pneumatic simulation and programming, and experience in software development and application. It is more convenient to develop your own computational fluid dynamics platform. On the one hand, you can avoid the key technical limitations in commercial software. On the other hand, you can provide a broad space for continuous development of complex flow research. This is the major foreign research institutions. They all have their own software.

References
[1] Kaifeng, H., Gang, L., Lihui, Z., & Zhongjun, M. Research progress on model flight test of powered aircraft with autonomous control system. Journal of Experiments in Fluid Mechanics, 30(2) (2016) 1-7.
[2] Wang, X., Qin, S., Xiang, Y., Wang, F., & Liu, H. Experimental investigation on large aircraft afterbody vortices under the influence of horizontal tail tip vortices. Shiyan Liuti Lixue/Journal of Experiments in Fluid Mechanics, 32(4) (2018) 53-60.

[3] Kuizhi, Y., Liangliang, C., Hu, L., & Yunliang, W. Analysis of jet blast impact of embarked aircraft on deck takeoff zone. Aerospace ence and technology, 45(9) (2015) 60-66.

[4] Wujciow, L., Zurawski, R., & Gorniak, T. Problems in designing fuel installation for small turboprop aircraft. Proceedings of the Institution of Mechanical Engineers, 231(2) (2017) 2228-2238.

[5] Kaparos, P., Papadopoulos, C., & Yakinthos, K. Conceptual design methodology of a box wing aircraft: a novel commercial airliner. Proceedings of the Institution of Mechanical Engineers, 232(14) (2018) 2651-2662.

[6] Zhou, F., Feng, L., Xu, C., Zhao, K., & Han, Z. Determination and verification of critical ice shape for the certification of civil aircraft. Shiyan Liuti Lixue/journal of Experiments in Fluid Mechanics, 30(2) (2016) 8-13.