Analogue Simulation Research on Heating Characteristics of Inner Channel of High-temperature Gas Flow

Zhonghao Sun, Sujun Dong & Xiaoyue Ren
School of Aeronautic Science and Engineering, Beihang University, Beijing, China

Chao Yan & Shenggang Wang
Beijing Research Institute of Mechanical & Electrical Technology, Beijing, China

ABSTRACT: The thermal test of structure generally uses the quartz lamp radiation heating method, but this method is not suitable for heating of the internal surface of a long and narrow irregular channel. This paper puts forward a heating method of insetting the high-temperature gas flow to the channel, and uses CFD numerical simulation method to explore the internal surface temperature rise and temperature distribution conditions of the channel at different temperature and under the impact of Mach number of gas flow and verify the heating capacity of high-temperature gas flow, and also gives out Mach number of the best incoming flow when the total temperature ability of the high-temperature gas tunnel can obtain the highest wall temperature under certain conditions, thus providing a theoretical basis for reasonable determination of the best operating conditions for test.

Keywords: thermal test of structure; high-temperature gas; numerical simulation; heating characteristics

1 INTRODUCTION

The development of modern aerospace technology makes the aircraft have an increasingly higher flight speed. Affected by severe aerodynamic heating, the high-speed aircraft affects the normal operation of its structure and other airborne equipment. For example, high temperature caused by aerodynamic heating may cause abnormal operation of the airborne electronic equipment, change of the mechanical properties of structural materials, reduction in strength and other problems. Therefore, it is very important to solve a series of problems caused by high temperature, carry out the thermal environment simulation test on the ground to simulate the high-temperature environment really withstood by the aircraft structure in the flight process and examine whether the structure can withstand the high-temperature mal-condition [1, 2].

With a great power, a small installation space. This paper has carried out the research on the feasibility of simulation of aerodynamic heat of the head cone type structure based on the heat transfer mode of high-temperature gas shock[5,6]. The results show the high-temperature gas flow generated by the gas generator is characterized by a great power, a high temperature, long working hours, a good transient regulation feature and so on, which is the best choice for the simulation test of a large-size structure, long-term high temperature / large heat flow. Realization of inner-channel heating by using the high-temperature gas flow will be a simple and easy-to-implement method.

Considering that the basic principle of the thermal simulation test of high-temperature gas flow is the convection heating mode between gas flow and wall surface, its heating strength has a great relationship with the geometry of test specimen, temperature of incoming flow, speed and other factors. Therefore, this paper uses Fluent software for certain numerical analysis of the heating characteristics of gas flow in the channel, explores the internal surface temperature rise and temperature distribution conditions of the channel at different temperature and under the impact of Mach number of gas flow, verifies the heating ca-
pacity of high-temperature gas flow, and also gives out Mach number of the best incoming flow when the total temperature ability of the high-temperature gas tunnel can obtain the highest wall temperature under certain conditions, thus providing a theoretical basis for reasonable determination of the best operating conditions for test.

2 DESCRIPTION OF PHYSICAL PROBLEMS

With the research object of the square inner channel, this paper explores the heating characteristics of high-temperature gas flow. The geometric model of the inner channel is shown in Figure 1. The geometric dimensions of the inner and outer cross section are respectively 105mm×64mm and 130mm×83mm, the length is 328mm, the material is C/SiC composite material, the density is 2030kg/m$^3$, and the thermal conductivity coefficient and specific heat changes with the temperature rise.

![Figure 1. Model diagram of test specimen in the square inner channel.](image)

High-temperature gas flow can be generated by using the aviation kerosene combustion chamber; Mach number of gas flow can be controlled by adjusting the air mass flow of the incoming flow; the gas flow temperature can be controlled by adjusting the mass flow rate of aviation kerosene; the maximum total temperature of the gas flow and the maximum Mach number are the main performance parameters of the heating capacity of the combustion chamber.

The physical property parameters of the temperature in the high-temperature gas part are shown in Table 1.

| Temperature | Density $\rho$/kg-m$^3$ | Thermal conductivity coefficient $h$/W(m·K) | Heat capacity $C_p$/kJ/(kg·K) | Viscosity coefficient $\eta$/Pa·s |
|-------------|-------------------------|-------------------------------------------|-------------------------------|-------------------------------|
| 900         | 0.301                   | 0.0763                                    | 1.172                         | 4.67e-05                      |
| 1000        | 0.277                   | 0.0807                                    | 1.185                         | 4.90e-05                      |
| 1100        | 0.257                   | 0.0850                                    | 1.197                         | 5.12e-05                      |
| 1200        | 0.239                   | 0.0915                                    | 1.21                          | 5.35e-05                      |

Internal heating test of high-temperature gas flow uses the convection heat transfer between gas flow and internal surface of test specimen. The test specimen is at a relatively low initial temperature; when the gas flow at the temperature of above test specimen flows, the temperature difference between gas flow and internal surface of test specimen makes the heat being transferred from the fluid to the solid specimen.

The greater the heat flux of heat transfer is, the stronger the heating capacity under such operating condition is.

According to Newton cooling formula, the heat flux of convection heat transfer can be expressed as:

$$q = h(T_\infty - T_w)$$

(1)

Where: $T_\infty$ is the static temperature of gas flow; $T_w$ is wall temperature; $h$ is the convection heat transfer coefficient under such condition.

Therefore, the heating capacity of high-temperature gas flow is mainly determined by the static temperature of incoming flow and convection heat transfer coefficient; for a certain structure, the convection heat transfer coefficient is mainly related to Mach number of incoming flow and increases with the increase of Mach number.

The relation between the total temperature of a certain incoming flow, Mach number and its static temperature is as follows:

$$\frac{T_{total}}{T_\infty} = 1 + \frac{r-1}{2} Ma^2$$

(2)

That is, when the total temperature of gas flow is constant, the greater Mach number is, the smaller the static temperature of gas flow is.

Considering from the formulas (1) and (2), it is found that, when the total temperature of incoming flow is constant, there is a best Mach value, rather than the greater Mach number is, the greater the heat flux of convective heating is. The best Mach value is not only related to the relation between the total temperature and the static temperature of gas flow itself, but also related to the relationship between the convection heat transfer coefficient and Mach number of incoming flow for the specific structure. Both of them determine the best Mach value.

3 COMPUTING METHOD

3.1 Governing equation

The internal heating process of high-temperature gas flow is composed of two parts: a process of convection heat transfer between gas flow and internal surface of solid, and a process of internal heat conduction of solid, and flow is in a turbulent state. Therefore, for the numerical analysis, there is not only a need to solve the fluid infinitesimal mass conservation and
momentum conservation equation, but also solve its energy conservation and turbulence equation, and solid internal heat conduction equation. Among them:

The mass equation is also called as the continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0$$

In the inertia (non-accelerating) coordinate system, the equation set of the momentum conservation equation is expressed as:

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \mathbf{u}) = -\nabla p + \nabla \cdot \mathbf{T} + F,$$

$$\frac{\partial (\rho \mathbf{v})}{\partial t} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \nabla \cdot \mathbf{T} + F,$$

$$\frac{\partial (\rho \mathbf{w})}{\partial t} + \nabla \cdot (\rho \mathbf{w} \mathbf{w}) = -\nabla p + \nabla \cdot \mathbf{T} + F,$$

where: $u$, $v$ and $w$ are respectively the velocity components at the directions of x, y and z; $p$ is the static pressure; $\mathbf{T}$ is the stress tensor; $F$ is the component of body force at the corresponding direction. The stress tensor is given by the following equation:

$$\mathbf{T} = \left[ \left( \frac{\partial \mathbf{u}}{\partial x_j} \delta_{ij} + \frac{\partial \mathbf{u}}{\partial x_i} \right) \right] - \frac{2}{3} \rho \mathbf{u} \mathbf{u} \delta_{ij}$$

Where: $\mu$ is the viscosity coefficient.

The energy equation is:

$$\frac{\partial (\rho T)}{\partial t} + \nabla \cdot (\rho \mathbf{u} T) = \nabla \cdot (k \text{grad} T) + S_\tau$$

Where: $k$ is the heat conductivity coefficient of the fluid; $S_\tau$ is a viscous dissipation term.

The solid internal heat conduction equation is:

$$\frac{\partial (\rho h)}{\partial t} = \frac{\partial}{\partial x_i} \left( k \frac{\partial T}{\partial x_i} \right)$$

3.2 Calculating parameter setting

First, ICEM software is used to complete the frame specimen meshing, and the thickness of near-wall grid is controlled by the fluid-solid coupling wall grid Y+ value in the internal surface, in order to ensure the computational accuracy of the convection heat transfer process. Then, in the calculation model of Fluent software, the energy equation is opened, k-epsilon turbulence model is selected, and the enhanced wall function is checked. With gas as a working medium, its density is set as a non-compressible ideal gas model. The inlet boundary uses the velocity entry conditions to input Mach number, static temperature, turbulence intensity and characteristic size corresponding to the calculated operating conditions.

The outlet boundary uses the pressure outlet conditions, and the outlet pressure is set as an atmospheric pressure. The outer wall of test specimen uses the natural convection model. The convection heat transfer coefficient is 20 W/m·K and the temperature of natural environment is 300K. This paper focuses on exploration of the heating characteristics of gas flow under conditions of two kinds of total temperature at 1,000°C and 1,300°C and different Mach number, as well as under conditions of the same Mach number and different total temperature of incoming flow. The specific operating conditions of 13 kinds of incoming flow are shown in Table 2. The static temperature of incoming flow is determined according to the Equation (2).

| Operating condition | Total temperature /℃ | Mach number | Static temperature /℃ |
|---------------------|-----------------------|-------------|------------------------|
| 1                   | 1000                  | 0.1         | 997.46                 |
| 2                   | 1000                  | 0.2         | 989.90                 |
| 3                   | 1000                  | 0.3         | 977.49                 |
| 4                   | 1000                  | 0.4         | 960.53                 |
| 5                   | 1000                  | 0.5         | 939.38                 |
| 6                   | 1300                  | 0.1         | 1296.86                |
| 7                   | 1300                  | 0.2         | 1287.52                |
| 8                   | 1300                  | 0.3         | 1272.19                |
| 9                   | 1300                  | 0.4         | 1251.22                |
| 10                  | 1300                  | 0.5         | 1225.10                |
| 11                  | 700                   | 0.3         | 682.8                  |
| 12                  | 800                   | 0.3         | 781.03                 |
| 13                  | 900                   | 0.3         | 879.26                 |

4 ANALYSIS OF COMPUTING RESULTS

4.1 Temperature distribution conditions

Figure 2 shows the temperature distribution curves of the internal heating wall of high-temperature gas flow along the gas flow direction under conditions of two kinds of total temperature at 1,000°C and 1,300°C, and different Mach number. The specific data are shown in Table 3. It is observed that:

1) Under the impact of gas flow with two kinds of total temperature and the same Mach number, the temperature distribution curves of the inner wall of test specimen at the gas flow direction are very similar, and the specific temperature difference at the inlet and outlet slightly increases with the increase of total temperature. Therefore, Mach number of incoming flow is a major influencing factor of the temperature distribution of the inner wall of test specimen.

2) The greater the Mach number of gas flow is, the smaller the temperature difference between the inlet and outlet of the internal surface of test specimen is, the better the temperature uniformity is. It is mainly because Mach number of incoming flow is too small, the heat transfer process along the way will make the temperature of gas flow itself
produce greater temperature drop, thus seriously affecting the post-test specimen. Thus, the increase of gas flow velocity is favorable to the temperature uniformity of the inner wall of test specimen.

3) The temperature of inner wall of test specimen gradually decreases from the gas flow inlet to the outlet along the axial direction, and decreases faster within a distance near the outlet. It is mainly caused by more heat losses generated by natural convection between the circular end face and outside at the outlet.

![Temperature distribution curves of internal surface of test specimen at the axial direction under conditions of two kinds of total temperature and different Mach number.](image)

(a) Total temperature of 1,000°C

(b) Total temperature of 1,300°C

Table 3. Temperature distribution conditions of internal surface of test specimen under conditions of two kinds of total temperature of incoming flow and different Mach number.

| Total temperature of incoming flow (°C) | Mach 0.1 | 0.2 | 0.3 | 0.4 | 0.5 |
|----------------------------------------|---------|-----|-----|-----|-----|
| 1000                                   | Inlet temperature | 895 | 919 | 932 | 924 | 904 |
|                                        | Outlet temperature | 783 | 844 | 879 | 882 | 863 |
|                                        | Temperature difference | 112 | 75  | 53  | 42  | 41  |
| 1300                                   | Inlet temperature | 1167 | 1206 | 1213 | 1204 | 1185 |
|                                        | Outlet temperature | 1013 | 1111 | 1140 | 1146 | 1136 |
|                                        | Temperature difference | 154 | 95  | 73  | 58  | 49  |

4.2 Influence of Mach number of incoming flow

Figure 3 shows the temperature variation curves of the internal heating mid-interface of high-temperature gas flow with Mach number of incoming flow under conditions of two kinds of total temperature at 1,000°C and 1,300°C. The specific data are shown in Table 4. It is observed that:

1) Under the condition of the same total temperature, with the increase of Mach number of incoming flow, the temperature of inner wall of midsection of the test specimen presents the distribution characteristic of increasing first and then declining. This is because, as mentioned above, when the total temperature of incoming flow is constant, the increase of Mach number will reduce the static temperature of incoming flow and increase the convection heat transfer coefficient. However, when Mach number is relatively low, the increase of Mach number and gas flow velocity can effectively increase the convection heat transfer coefficient, but has a relatively small impact on the static temperature, so the overall heat transfer effect is strengthened; when Mach number is relatively high, the increase of gas flow velocity weakens the degree of improvement of the heat transfer coefficient, while the static temperature decreases significantly with the increase of Mach number, thus leading to the weakening of the overall heat transfer effect. Thus, when the total temperature of incoming flow is constant, there is a Mach number. If Mach number is greater than or less than the best Mach number, it will weaken the gas flow heating effect.

2) Under two conditions of total temperature of incoming flow, the best Mach number is about 0.3. When total temperature of incoming flow is 1,000°C, the best Mach number is slightly larger, and the change in the temperature of inner wall of test specimen is more obvious in the vicinity of the best Mach number; when total temperature of incoming flow is 1,300°C, the change is more gentle in the vicinity of the best Mach number. That is, when the temperature of incoming flow is relatively low, when the gas flow velocity exceeds the best Mach number, the maximum temperature of the test specimen model will decline rapidly; when the total temperature of gas flow is relatively high, after continuous increase of Mach number, the maximum temperature of the model only declines slightly, with a little change.
4.3 Influence of total temperature of incoming flow

Figure 4 shows the temperature distribution curves of the inner wall of test specimen along the gas flow direction under conditions of different total temperature of incoming flow and Mach number of incoming flow of 0.3. The specific temperature values of the inlet, outlet and midsection are shown in Table 4. It is observed that:

1) The temperature distribution curves of the inner wall of test specimen at the axial direction under conditions of different total temperature of incoming flow and Mach number of 0.3 are approximately parallel distribution, and the temperature uniformity is better.

2) The temperature at the midsection of test specimen increases with the increase of the total temperature of incoming flow, and approximately presents a proportional increase with the temperature difference between the total temperatures of incoming flow. It indicates that, with the increase of temperature of internal surface of test specimen, the required gas flow and its temperature difference continues to increase. That is, in order to obtain a higher heating temperature, there is a need to greatly increase the gas flow temperature.

| Total temperature /°C | Inlet temperature /°C | Outlet temperature /°C | Midsection Temperature /°C | Temperature difference /°C |
|------------------------|-----------------------|------------------------|---------------------------|---------------------------|
| 700                    | 652                   | 616                    | 637                       | 63                        |
| 800                    | 745                   | 704                    | 728                       | 72                        |
| 900                    | 839                   | 792                    | 818                       | 82                        |
| 1000                   | 932                   | 879                    | 908                       | 92                        |

5 CONCLUSION

This paper verifies that the internal heating method of high-temperature gas can meet the needs of thermal simulation test of the inner wall of frame structure by using the numerical simulation method, and summarizes the following three heating characteristics of gas flow, thus providing an important basis for reasonable determination of the operating conditions for thermal test of high-temperature gas flow:

1) The greater Mach number of gas flow is, the smaller the temperature difference between inlet and outlet of the inner wall of the test specimen is, the better the temperature uniformity is;

2) When the total temperature of incoming flow is constant, the best Mach number is about 0.3; if Mach number is less than or greater than the best Mach number, it will decline the temperature of the inner wall of the test specimen;

3) Under the conditions of a certain Mach number, the temperature of the inner wall of the test specimen is similar to linear increase with the gas flow temperature.

REFERENCES

[1] Wang Leshan, Wang Qingsheng. 2000. New development of thermal test technique of structure. *Missile and Space Vehicles*, 29(2): 7-13.

[2] Zhou Hua, Han Ying. 1999. Research progress of hypersonic ground simulation equipment. *Flight Dynamics*, 17(4): 6-10.

[3] Yao Feng, Dong Sujun, Wang Jun. 2010. Research on test methods of heating characteristics of high-temperature gas thermal environment. *Journal of Astronautics*, 31(5): 1446-1451.

[4] Yao Feng, Dong Sujun, Wang Jun. 2010. Simulation research on simulation scheme of high-temperature gas thermal environment. *Journal of Aeronautics and Mechanics*, 25(4): 768-773.

[5] Ho S Y, Paull A. 2006. Coupled thermal structural and vibrational analysis of a hypersonic engine for flight test. *Aerospace Science and Technology*, 10(5): 420-426.

[6] Zhou Guangjiong, Yan Zongyi, Xu Shixiong, Zhang Keben. 2011. *Fluid Mechanics*. Beijing: Higher Education Press.

[7] Ai Qing. 2009. Numerical research on coupled heat transfer of thermal radiation and high-speed flow. Harbin Institute of Technology.