Study on Testing Environment Simulation Method for Thermal Flux Density of Aerothermal

Ju shichao
Shanghai Aircraft Design and Research Institute, COMAC, No.5188 Jinke Road, Shanghai, China
symseer@163.com

Abstract. This paper presents the use of forced convection heat transfer approach to near-space thermal environment of hypersonic aerodynamic ground simulation test method for surface heat flux with a great gradient and temperature gradient of the tip wedge (rn=1.5mm) shape component level thermal test. In the test device test section, two different high subsonic speed airflow were nested within each of the two vents from the exhaust, the impact of the specimen heated to test sharp wedge surface heat flow and temperature distribution in line with a hypersonic flight state aerodynamic heat distribution patterns. Numerical results show that under this scenario, the specimen rear stagnation point heat flux and heat flux can be adjusted two ways to adjust the air flow; test methods within a certain accuracy-related parts for the superb aerodynamic vehicle thermal environment simulation.

1. Introduction
The hypersonic vehicle will suffer severe aerodynamic heating due to its interaction with gases during its flight or return to the atmosphere[1]. Usually when the flight Mach number Ma > 3, the air temperature of the aircraft's head can reach over 400 °C; when the flight speed reaches 8 Mach, the temperature of the aircraft's nose cone can reach 1800 °C, and the temperature of other parts will be above 600 °C [2]; when the speed of Ma = 8 is flying at 27 km, the temperature of the nose cone is about 1800 °C, and the temperature of the wing or the leading edge of the tail is as high as 1450 °C [3]. In order to avoid the damage of aircraft caused by surface high temperature caused by aerodynamic heat, it is necessary to analyze and test the heat-carrying capacity of the structure.

Thermal protection research of hypersonic vehicle thermal protection system can be divided into two types: thermal environment research and thermal protection research. Thermal protection research simulates the thermal environment based on the conclusions of thermal environment research, and then carries out the overall layout of thermal protection system and the selection of thermal protection materials and thermal protection structure[4]. The aerothermal simulation includes a series of radiation heating equipment and airflow heating test equipment. The main equipment are high temperature and high speed wind tunnel, arc heating wind tunnel, gas heating device, high temperature structure wind tunnel and so on. High-temperature and high-speed wind tunnel is used to verify the calculation method of aerothermal environment of aircraft, to provide thermal load control parameters for radiation heating simulation test, to study the heat transfer relationship between surface protrusions of aircraft structure and high-speed airflow, and arc wind tunnel is used to study the high-temperature performance test of small-size typical structural materials. The high temperature structure wind tunnel, which adopts the thermal test method of airflow heating structure, has a wide range of applications. It
mainly studies the aerodynamic shape of aircraft, high temperature structural materials, ramjet performance under simulated high speed flight environment, thermal performance of thermal shield, etc. At present, it is set up in NASA Langley Center.

At present, there are some shortcomings in the domestic ground simulation test system of structural thermal environment, such as the limited size of the specimen and the short working time of the system, and the limited maximum simulation temperature. In order to meet the progress of the thermal protection system and material of hypersonic vehicle, it is necessary to develop the thermal environment simulation test technology and strengthen the test capability of ground equipment in order to ensure the research of hypersonic vehicle, especially its key technology. At the same time, the thermal environment problem challenges the thermal environment simulation test. The development of high temperature, high heat flux gradient and high precision, reliable and stable thermal environment ground simulation test system is of great significance to the progress of thermal protection technology for hypersonic vehicle.

Aiming at the characteristics of aerothermal local high temperature of hypersonic vehicle and the insufficiency of ground test equipment for the thermal environment of traditional structure, this paper presents a thermal environment simulation method for shock heating of specimens by using two different subsonic high-temperature and high-speed airflow ejected from two nested nozzles, and preliminarily verify by means of numerical simulation.

2. Thermal Test Principle of Shape Components
When a hypersonic vehicle is flying in full orbit, the surface heat flow and transient temperature distribution of the whole aircraft can be obtained by using the experimental method, which consumes a lot of money and is not suitable for the initial design stage of an aircraft. Generally, the simple engineering test method is adopted in the design of components stage, which decomposes the whole aircraft into some simple geometric elements and tests each element separately. As shown in Figure 1, a complex aircraft is decomposed into spheres, cones, swept cylinders and wedges. The interference between simple shapes can be corrected by using wind tunnel and flight test data according to the actual situation by using the surface heat flow test method with mature simple shapes.

![Figure 1. Typical decomposition of complex aircraft configuration](image)

2.1. Aerodynamic heat of wedge-shaped parts
The flight thermal environment of hypersonic vehicle is mainly due to the high temperature environment generated on the outer surface of the aircraft by the action of aerodynamic heat, and the heating of the aircraft by the airflow passing through the outer surface of the aircraft has the characteristics of local large heat flux density and large area medium heat flux density. For the typical hypersonic component tip wedge, it can be considered as composed of the head cylinder and the rear plate. For the cylindrical region, the heat flux distribution varies greatly. The front heat flux is the stagnation point heat flux, which is the high heat flux region. For the flat plate region, the heat flux changes little, and the heat flux is smaller than the front stagnation point heat flux, which is the medium heat flux region [5], as shown in Figure 2.
2.2. Metal Heat-proof structure

Metal thermal protection system has many advantages [6], such as high toughness, good impact resistance, high reuse rate and low cost. It has great development potential in large area thermal protection of hypersonic vehicles. Because of the need of high-speed cruise, the front of X-43 is designed very sharp to reduce resistance. For the need of aircraft shape, a large part of the front part of the fuselage is made of solid tungsten material, which simplifies the design of thermal protection in this area. Carbon-carbon leading edge and side strip are used to protect the edge with the highest heating capacity of tungsten wedge head, while relatively short flight time can prevent most of the surface of tungsten wedge head from being heated. Aerodynamic heat of wedge-shaped components has the characteristics of large heat flux gradient, which will cause temperature gradient and uneven thermal expansion in the structure, and thus generate great thermal stress in the front of the structure.

Therefore, in order to avoid the damage due to the surface high temperature caused by aerodynamic heat to the wedge components of aircraft, the heat-carrying capacity of the structure must be analyzed and tested, and the key point is to simulate the corresponding heat flux gradient and temperature gradient.

3. Aerothermal Simulation means of wedge-shaped parts

3.1. High enthalpy gas equipment

High enthalpy gas thermal simulation test equipment is a kind of economical and convenient thermal environment simulation equipment. Experiments have proved that it has enough thermal simulation ability and can simulate the thermal environment of hypersonic vehicle components in a certain precision range. The thermal simulation experiment of hypersonic vehicle is expensive and the experimental ability is insufficient, which meets the urgent need of the development of hypersonic vehicle. The high temperature gas thermal simulation scheme uses the high temperature gas generated by the combustion chamber of the gas to emit and eject the normal temperature and pressure air from the gas nozzle through the high temperature pipeline. The cold and hot air flows through the momentum exchange mixing to blow and heat the sample. The test tail gas after cooling treatment enters the noise reduction tower and discharges into the outdoor atmosphere.

High temperature gas simulates hypersonic aerodynamic heat, which has the following characteristics. The gas outlet velocity is usually subsonic, that is, using subsonic high temperature gas flow to simulate the heating of high speed gas flow. Because of the different nature of the two flow patterns, there are also some differences in the thermal effect of the specimen. As mentioned earlier, hypersonic heating has local large heat flow and large area medium heat flow. The cumulative effect of heating is reflected in the large temperature gradient of the specimen. For gas heating, when the stagnation point heat flux reaches the required value, the temperature speed of gas is often required to be higher, but the heat flux density at the back of the specimen is much higher than that at the high aerodynamic heating value. As a result, the temperature rise at the back of the specimen is very fast,
and the temperature gradient within the specimen cannot meet the requirements. The distribution of heat flux of gas heating with 40 mm nozzle 1800K and $Ma=6$, $H=30Km$, $Tw=300K$ on the surface of the specimen is shown in Figure 3.

Figure 3. Distribution of Heat Flux Density of Gas Heating and Super Aerodynamic Heating on the Surface of the Specimen

3.2. The engineering calculation of convection heating
For approximate calculation of surface heat transfer of wedge-shaped parts with gas impingement tip, the calculation formula of cross-flow cylinder can be selected in the cylinder part, and the calculation formula of laminar heat transfer around the cylinder is adopted.

$$h_{g}d_{f} = K_{s}[1.14 + Re_{d}^{1/2} Pr^{0.4}(1 - \frac{\phi}{90})^{3}]_.$$, $-80^{\circ} < \phi < 80^{\circ}$

$$Re_{d} = \frac{\rho_{g} u_{g} d_{f}}{\mu}$$

The airfoil specimens are 9 degree flat plates formed horizontally behind the circular arc, and the fitting formula of plate heat flow is used.

laminar flow:

$$Nu_{x} = 0.332 Re_{x}^{-1/2} Pr^{1/3}$$

(3)

turbulence flow:

$$Nu_{x} = 0.0296 Re_{x}^{0.8} Pr^{1/3}$$

(4)

The qualitative temperature in the above formula is taken as the reference Eckert temperature $T^*$, and the turbulence correction factor is related to the turbulence. For a smooth plate without pressure gradient, when $Re_{x}=300000-500000$, the boundary layer changes from laminar to turbulent. From the above formula, the required air temperature and velocity can be calculated according to the required simulated heat flux of aerodynamic heat.

3.3. Two-way airflow heating scheme
From the above analysis, it can be seen that it is difficult to simulate a large temperature gradient with a single gas nozzle. For hypersonic aerodynamic heating, including heat conduction and friction heating, both occur in the boundary layer. From the static temperature distribution of hypersonic flow field, it can be seen that the temperature in the boundary layer at stagnation point is much higher than that in the flat area. For gas flow heating, the temperature difference in the boundary layer at stagnation point is not very large compared with that in the flat area.
Therefore, the heat flux field on the whole specimen can be divided into two zones to simulate separately, i.e. high temperature and high speed air flow is used to simulate the high heat flux area of the head, and the medium heat flux in the flat plate area is simulated by the gas in a lower state. A small nozzle with the same size as the head of the specimen is added to the center of the original large nozzle, and the air temperature in the small nozzle is also simulated. At the same time, the original nozzle emits gas with lower temperature.

4. Quasi-static heat flow simulation

4.1. Testing Environment condition

The maximum aerodynamic heating heat flux of hypersonic vehicle usually occurs in the accelerated climbing stage. Because of the fast climbing speed of the vehicle, it can climb to the altitude of 60,000 meters in less than 2 minutes.

4.2. Simulation results and analysis

When the width of the small nozzle is 4 mm and the distance between the small nozzle and the head of the specimen is 8 mm, the gas flow temperature is 1800 K, the velocity is $Ma=0.7$, the temperature of the cold cover flow is 1000 K, the velocity is $Ma=0.4$, and the temperature of the specimen is 300 K, the cold wall heat flow on the surface of the specimen can be obtained. The calculated temperature field is shown in Figure 4.

![Figure 4. temperature field of the Specimen](image)

The heat flux distribution on the surface of the specimen is shown in Figure 5.

![Figure 5. Surface heat flux distribution of double-channel air impingement heating](image)

It can be seen from Figure 5 that the surface heat flux of the specimen can decrease rapidly in the direction of the airflow and tend to be higher than that of the aerodynamic heat by using the method of
small nozzle and cooling hood flow. In the transition zone from the head of the specimen to the plate area, the gas heating rate is higher than that of the high superheat flux. However, the high and super heat flux in the head region is relatively high. Because of the existence of thermal diffusion, the heating effect of the specimen is basically equivalent, and the influence on the temperature distribution behind the specimen will be very small.

5. Transient aerothermal simulation
In order to simulate the transient aerodynamic heat with high temperature gas impact test pieces, it is necessary to know the real aerodynamic heating environment in flight, specifically the aerodynamic heating heat flux density and temperature at each position of the aircraft under flight envelope. Then, according to the obtained heating heat flux density and temperature, the temperature flow of gas is usually selected by numerical calculation method to simulate the thermal environment of the test pieces. Whether the errors between the surface temperature distribution and the true value of a given specimen satisfy the requirements of thermal test or not, the experiment can be carried out, otherwise the gas state needs to be adjusted and the method of numerical calculation revised.

6. Conclusions
In the two-way air impingement heating scheme, two different air streams are ejected from each other's nested nozzles to impinge on the wedge specimen. By setting reasonable air flow parameters, the surface heat flow and temperature distribution of the wedge in the test can conform to the law of aerodynamic heat distribution in a hypersonic flight state. The numerical results show that the heat flux density at the standstill point and at the rear of the specimen can be adjusted by adjusting the two airflow paths. The experimental method can simulate the aerothermal environment of the relevant components of the hypersonic vehicle in a certain precision. The research shows that the method is feasible in engineering and has potential in the thermal simulation of larger-scale components. In transient simulation, the basic process of coordinated control of experimental parameters and the theory of integrated control need to be further explored.

- Hypersonic aerothermal heat flux density at the standstill point and at the rear of the specimen can be adjusted by adjusting the two airflow paths.
- The basic process of coordinated control of experimental parameters and the theory of integrated control need to be further study.

References
[1] Sachin Kumar, Shripad P. Mahulikar Aero-thermal analysis of lifting body configurations in hypersonic flow, Acta Astronautica. 126 (2016) 382–394.
[2] R. Savino, G. Russo, V. D’Oriano, M. Visone, M. Battipede, P. Gili, Performances of a small hypersonic airplane (HyPlane), Acta Astronaut. 115 (2015) 338–348.
[3] John J. Bertin, Russel M. Cummings. Critical Hypersonic Aerothermodynamic Phenomena[J].Annu.Rev.Fluid.Mech,2006.38:129-157.
[4] Rivers H K, Glass D E. Advances in hot-structure development[R]. NASA-2006.
[5] J. Liu, F. Ding, W. Huang, L. Jin, Novel approach for designing a hypersonic gliding–cruising dual waverider vehicle, Acta Astronaut. 102 (2015) 81–88.
[6] Glass D E. Ceramic matrix composite (CMC) thermal protection system (TPS) and hot structure for hypersonic vehicles[R]. AIAA-2008-2682.