Effectiveness analysis of opposing jet thermal protection with hot fuel gas

BinXian Shen*, WeiQiang Liu

Science and Technology on Scramjet Laboratory, College of Aerospace Science and Engineering, National University of Defense Technology, Changsha 410073, China

*Shenbinxian_1603@163.com, liuweiqiang_1103@163.com,

Abstract. The hot fuel gas generating with solid fuel was introduced to an opposing jet thermal protection system (TPS), saving the occupied space and reducing the weight of the gas supply system. In this study, the thermal protection performance of opposing jet TPS with hot fuel gas is investigated. The surface heat flux is initially obtained by solving the Reynolds-average Navier-Stokes (RANS) equations coupled with the Menter’s shear stress transport (SST) model. Then, the obtained heat flux is loaded to the blunt body to solve the temperature distribution. The influence of the hot fuel gas on the opposing jet TPS was analyzed with different flying condition. Results show that the hot fuel gas reduces the insulating capacity of the opposing jet, but the peak heat flux exhibits a prominent reduction in contrast to that without injection. The surface temperature also reduces a lot even the hot fuel gas is adopted than that without injection. To obtain a good cooling efficiency, the jet pressure is enhanced. With a higher jet pressure, the surface heat flux, as well as the surface temperature, is reduced, and the similar cooling efficiency can be obtained compared to that with normal coolant.

1. Introduction

An appropriate thermal protection system (TPS) to protect the material from extreme aerodynamic heating is very important for hypersonic vehicles[1]. The traditional passive thermal protection with insulating and ablative materials can’t acclimate the long-time, continuous aerodynamic heating in hypersonic vehicles[2]. Hence, the novel active thermal protection for thermal protection in hypersonic vehicles interests researchers.

A variety of specific structures or flows are introduced to the nose of vehicles, such as opposing jet[3], forward-facing cavity[4], aero-spike[5]. These structures control the flow characteristics and improve the thermal environment around the nose. Basic research indicates that opposing jet is an accessible means to achieve an effective thermal protection in hypersonic vehicles. The opposing jet, injecting from the stagnation point, pushes the shock wave away from the surface, thereby forming a low-temperature recirculation region which insulates the high-temperature stagnate freestream, realizing a prominent heat flux reduction on nose surface.

Opposing jet was used to protect the vehicles from aerodynamic heating by Warren[6] firstly in the early sixties of past century. Then researchers conducted lots of experimental and numerical work on opposing jet thermal protection. In 2003, Hayashi[7] proved that the opposing jet reduces the surface heat flux in his tunnel experiment with 3.98 Ma. The influence of the jet pressure on the flow motion also has discussed in his research. The influence of the input parameters of coolant and the physical configuration has been discussed. Guo[8] developed a point-collocation non-intrusive polynomial
chaos method to estimate the influences of input parameters on total heat load. Li[9] and Barzegar[10] improved the injector configurations to strengthen the cooling efficiency of opposing jet.

The excellent performance of opposing jet on heat flux reduction has been widely demonstrated, but how to use this active cooling method and how to supply the coolant in really flight always puzzle engineers. A gas generator with solid fuel is introduced to the nose of hypersonic vehicles. The hot fuel gas generated with solid fuel is used as coolant medium in opposing jet thermal protection. In this paper, the thermal protection performance of hot fuel gas is numerically investigated.

2. Fuel gas system
The brief structures of the fuel gas supply system are shown in Fig. 1. A fuel gas generator is installed in the interior of the nose, connecting to the contraction injector. The stored solid fuel is ignited firstly by the igniter when the supply system works. Then the solid fuel burns at a controllable speed and generates a large number of fuel gas. The fuel gas, with a required pressure and speed, injects into the outer flowfield as an opposing jet, pushes the shock wave away. The fuel gas supply system are integrated into the gas generator, which exhibits better performance on economizing occupied space and reducing the weight than the conventional means used in ground experiments.

The solid fuel usually consists of combustible, oxidant, binder, controlled medium, and cooling agent. The applied environment of thermal protection in hypersonic vehicles requires that the combustion characteristics of solid fuel should be low-temperature, controlled, and low-residue. Several solid fuels are listed in Table 1. In this paper, the temperatures of fuel gas are assumed as 900 and 1200K to analysis the influence of jet temperature on the thermal protection performance. Finally, the fuel gas is comprised of nitrogen, carbon dioxide and water vapour.

| Solid fuel       | Combustion temperature | Solid fuel       | Combustion temperature |
|------------------|------------------------|------------------|------------------------|
| 1 GAP/AN         | 1413 K                 | 3 GN/BCN/DHG     | 1278 K                 |
| 2 PAK/KNO3/CuO   | 1310 K                 | 4 ADC/BCN/CuO    | 972 K                  |

3. Numerical Method

3.1 Governing Equations and Discretization
The calculation procedure is divide into two steps. Firstly, the external flow field of hypersonic vehicles is solved. The three-dimensional Reynolds-averaged Navier–Stokes (RANS) equations
coupled with the Menter’s shear stress transport (SST) model are employed as governing equations. The governing equations are solved with a density based (coupled) double precision solver in the Fluent 16.0. The advection upstream splitting method with a first spatially accurate upwind scheme is used for the discretization of inviscid fluxes. The first order upwind scheme is adopted for computing the viscous fluxes.

Then, the internal temperature of the nose body is solved. The heat flux obtained in first step is loaded to the outer surface as input heat flux. The three-dimensional Reynolds-averaged Navier–Stokes (RANS) equations with the realizable steady-state $K$-$\varepsilon$ model are discretized using a second-order upwind interpolation scheme, and the discretized equations are solved using the SIMPLE algorithm in Fluent 16.0.

3.2 Grid and boundary

The 1/6 geometric model is calculated for its rotational periodic, and the grid system is divided by ICEM, which is shown in Fig. 2. The grid independence analysis is analyzed, and the mesh we obtained is credible to simulate the flow field and heat transfer of hypersonic nose.

The freestream and opposing jet flow parameters are show in Table 2 and Fig.2. The freestream condition is in accordance with the real flight at a height of 25 Km and a Mach number of 6 or 8. The opposing jet medium is assumed as compression fuel gas, its pressure are defined with $PR$, which establishing a relationship between the opposing jet and freestream.

$$PR = \frac{P_{0j}}{P_{0\infty}},$$

where $P_{0j}$ and $P_{0\infty}$ refers to the total pressure of the jet and the free stream, respectively.

| Far-field pressure | Pressure inlet | Wall |
|--------------------|----------------|------|
| Air                | fuel gas       | $T_w = 295$ K |
| $Ma_\infty=6 / 8$ Ma | $Ma_j = 1$ | No slip |
| $P_{0\infty}=4.02 / 24.88$ Mpa | $PR = 0.1, 0.2, 0.25 / 0.05, 0.1, 0.15$ |
| $T_{0\infty} = 1812 / 3049$ K | $T_{0j}=300, 900 / 300, 1200$ K |

4. Result and discussions

4.1 Flow field

The distributions of stream line(contours of velocity), the temperature distribution with different $PR$ and jet temperature are shown in Fig.3-5. Fig.3 shows the stream-line with $PR$ being 0.1 and 0.2(with flying condition being 6 Ma). At first, the freestream is stopped by the opposing jet, forming a bow shock wave and an interface. The freestream crosses through the bow shock wave then flows along the interface. In addition, a recirculation region appears inside the interface. Next to the recirculation, the freestream hits against the nose surface and leads to a recompressed shock wave. The difference in Fig.3 shows that the bow shock wave can be pushed farer with a stronger opposing jet. Fig.4 shows the temperature distributions with $PR$ being 0.1 and 0.2 while the jet temperature is 900K. At first, the high-temperature region is pushed farer with the $PR$ increases from 0.1 to 0.2. Then, the temperature gradient in reattachment region is smaller when the $PR$ is 0.2, which benefits on reducing heat transfer. Fig.5 shows the temperature distributions with jet temperature being 300K and 900K while the $PR$ keeps constant. The jet temperature has not evident effect on the location of bow shock wave, but it influence the temperature distribution in reattachment region. A higher jet temperature results in higher temperature and larger temperature gradient, being negative in thermal protection.
4.2 Heat flux

The heat flux distributions with different jet parameters are shown in Fig.6 (with flying condition being 6 Ma) and Fig.7 (with flying condition being 8 Ma). The heat flux with opposing jet and without injection are compared. The opposing jet, as well as the recirculation region can insulate the high-temperature region. Consequently, the heat flux can be considerably reduced in stagnation region. Then, the heat flux increases along the flow direction with the influence of recompression shock wave. Finally, the heat flux reduces with the weak influence of the freestream which in accordance with that without injection. The Fig.6 -7 shows that the heat flux decreases with the increasing PR and increases with the increasing jet temperature. In Fig.6, the peak flux is even higher in reattachment region than that without injection when the PR and jet temperature being 0.1 and 900K respectively. But the thermal protection capacity can be strengthened by enhancing the PR. The results in figures illustrate that the similar or better heat flux reduction with hot gas can be obtained when the PR is stronger enough.
4.3 Temperature distribution
The temperature distributions with different jet parameters are shown in Fig. 8 (with flying condition being 6 Ma) and Fig. 9 (with flying condition being 8 Ma). At first, the temperatures with injection have remarkably reductions compared to that without injection, especially in 8 Ma flying condition. Then the lower temperature is obtained with a higher PR even though the hot fuel gases (900 K in 6 Ma flying condition and 1200 K in 8 Ma flying condition) are used. Finally, the hot fuel gas have better thermal protection performance in higher Mach number flying condition. As we all know, the stagnation temperature increases with the increasing flying Mach number, leading to a larger temperature difference between the hot fuel gas and stagnation temperature which is better on thermal protection.

Fig. 8 Temperature distribution in 6 Ma flying condition
Fig. 9 Temperature distribution in 8 Ma flying condition

5. Conclusion
The hot fuel gas can be used as opposing jet medium to cool the nose of the hypersonic vehicles. The similar even better cooling efficiency of hot fuel gas can be obtained with enhanced PR. In addition, the hot fuel gas exhibits better cooling efficiency in high flying condition.

References
[1] Q. Yang, W. Xie, Z. Peng. (2015) New concepts and trends in development of thermal protection design and analysis technology, Acta Aeronautica Et Astronautica Sinica 36(9): 2981-2991. (in Chinese)
[2] D. Glass. (2011) Physical challenges and limitations confronting the use of UHTCs on hypersonic vehicles, in: 17th AIAA International Space Planes and Hypersonic Systems and Technologies Conference, San Francisco, California. AIAA 2011-2304.
[3] Rui-rui Zhang, Wei Huang, Li Yan, Lang-quan Li, Shi-bin Li, R. Moradi. (2018) Numerical investigation of drag and heat flux reduction mechanism of the pulsed counterflowing jet on a blunt body in supersonic flows, Acta Astronautica 146: 123–133
[4] S. Saravanan, G. Jagadeesh, K. P. J. Reddy. (2009) Investigation of Missile-Shaped Body with Forward-Facing Cavity at Mach 8, Journal of Spacecraft and Rockets 46(3): 577-592.
[5] M.Y.M.Ahmed, N.Qin. (2011)Recent advances in the aerothermodynamics of spiked hypersonic vehicles, Prog. Aerosp. Sci. 47(6): 425–449.
[6] Warren C H E. (1960) An experimental investigation of the effect of ejecting a coolant gas at the nose of a bluff body, Journal of Fluid Mechanics, 8:400-417
[7] K. Hayashi, S. Aso, Y. Tani. (2005) Numerical study of thermal protection system by opposing jet. 43rd, AIAA Aerospace Sciences Meeting and Exhibit. Reno, NV AIAA 2005-188.
[8] Guo J, Lin G, Bu X. (2018) Parametric study on the heat transfer of a blunt body with counterflowing jets in hypersonic flows, Int. J. Heat Mass Transf. 121: 84-96.
[9] S.B. Li, Z. G. Wang, W. Huang. (2016) Effect of the injector configuration for opposing jet on the drag and heat reduction, Aerosp. Sci. Technol. 51: 78-86.

[10] M.B. Gerdroodbary, M. Imani, D.D. Ganji. (2015) Investigation of film cooling on nose cone by a forward facing array of micro-jets in Hypersonic flow, Int. Commun. Heat Mass Tran. 64: 42–49.