Aerospace Science and Technology 48 (2016) 322–327

Contents lists available at ScienceDirect

Aerospace Science and Technology

www.elsevier.com/locate/aescte

Research on thermal protection by opposing jet and transpiration for high speed vehicle

Rong Yisheng [a,b,*], Wei Yuechuan [a,b], Zhan Renjun [a]

[a] Engineering University of the Armed Police Force, Xi'an 710086, PR China
[b] National University of Defense Technology, Changsha 410073, PR China

ARTICLE INFO

Article history:
Received 26 May 2015
Received in revised form 8 November 2015
Accepted 14 November 2015
Available online 17 November 2015

Keywords:
Thermal protect system
Opposing jet
Platelet
Transpiration

ABSTRACT

A thermal protection structure that unites the opposing jet and platelet transpiration is introduced, whose cooling effect is conducted through numerical research method. When opposing jet is applied, the peak heat flux falls significantly at the stagnation point. By introducing platelet transpiration, the temperature also drops dramatically. The new approach combining both opposing jet and platelet transpiration brings out more dramatic temperature reduction. The numerical results show that the new thermal protection structure works efficiently.

1. Introduction

During hypersonic flight, vehicle will encounter serious aerodynamic heating, which burns out the vehicle and leads to mission failure. The heat flux of a vehicle's leading edge with radius 20 mm can reach 2–3 MW/m^2 when it flies at Mach 7 at an altitude of 24 km, and the wall temperature can reach up to 1400 K. To protect the vehicle from the extremely high temperature, it is important to design the thermal protection system (TPS) [1–4]. Van Foreest et al. [5] developed transpiration cooling by using liquid water and the method was proven to be extremely effective. By using a little water, the experimental models were cooled down from temperature over 2000 K in the stagnation point to temperature lower than 300 K. Xiong et al. [6,7] analyzed the effect of a transpiration cooling scheme using porous media. The cooling effect increased along with the raising injection rate, and the increase became smaller and smaller. Platelet can be used to form the structure for transpiration cooling [8], and it is more controllable than porous materials. The coolant can be sent to the place where needed most through special platelet structure. The platelet is widely used for cooling the rocket engine, hypersonic vehicle and re-entry vehicle [9,10].

Due to high heat flux, it needs more coolant near stagnation point. However, it may fail to transpire because of the high pressure there. To protect the tiny stagnation region, opposing jet is a good choice, which also reduces the drag simultaneously [11]. Warren [12] suggested using opposing jet to reduce aerodynamic heating for vehicle and conducted the experiment by ejecting nitrogen and helium gases upstream into a Mach 5.8 freestream of a sphere–cone model. With the development of the aeronautics and astronautics, the advantage of opposing jet becomes more and more attractive. In this century, many scholars kept doing research on this method [13–22]. Aso et al. [13–15] undertook both numerical and experimental studies of opposing jets in a supersonic flow over a sphere–cylinder for thermal protection. The results showed considerable reduction both in drag and heat flux. A high-precision simulation of Navier–Stokes equations was performed by Tian and Yan [16] to study the detailed influences of the free Mach number, jet Mach number and attack angle on the drag coefficient reduction. To study the effect of the intensity of opposing jet reasonably, Rong et al. [17,18] proposed a new parameter R_{PA} by combining the flux with the total pressure ratio. Their study showed that the same shock wave position, reattachment point position, aerodynamic drag, peak heat flux position and total heat load can be obtained with the same R_{PA} with different fluxes and different total pressures, which means the parameter R_{PA} could stand for the intensity of opposing jet and could be used to analyze the influence of opposing jet on flow field and aerodynamic heating. Lu [19–22] combined opposing jet with forward-facing cavity to protect hypersonic vehicles. Their research results showed that the combinatorial system led to the remarkable aerodynamic heating reduction.

In this paper, a thermal protection structure is introduced, which combined the opposing jet with platelet transpiration. The

* Corresponding author at: Engineering University of the Armed Police Force, Xi'an 710086, PR China. Tel.: +86 15029570975.
E-mail address: rong83117@163.com (Y. Rong).

http://dx.doi.org/10.1016/j.ast.2015.11.014

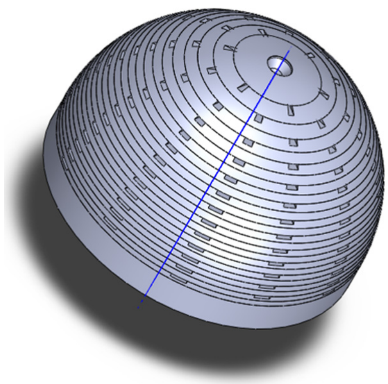

Fig. 1. The nose tip with opposing jet and platelet transpiration.

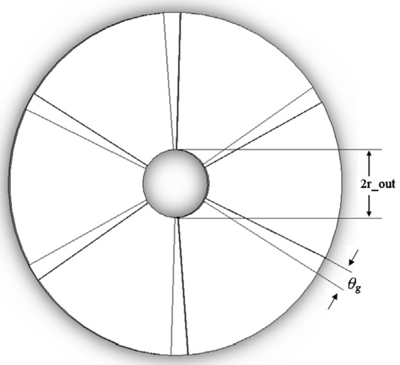

Fig. 2. The structure of a platelet.

opposing jet cools the stagnation region, and then platelet transpiration keeps the whole nose tip cool. A numerical research on cooling effect of the combination structure is conducted. The numerical results show that the thermal protection structure with opposing jet and platelet transpiration cools the nose tip efficiently, and thus validate the thermal protection design is effective.

2. The structure of platelet transpiration cooled nose tip

The structure of the nose tip with opposing jet and platelet transpiration is shown in Figs. 1–3. The platelets are 1 mm thick. On each platelet there are 6 radial grooves which are narrow sectors (see Fig. 2) and the center angle of the sectors is θ_g. The radial grooves are 0.5 mm deep. 24 platelets are stacked together in alternating pattern according to their radial groove configuration precisely to form the nose tip whose radius $r_{nt} = 5$ mm. We mark the bottom platelet the first one and the top platelet the 24th one. Another special material platelet without grooves is put on the top of nose tip as the outlet of opposing jet. The hole space in the center of the platelet is for coolant channel, whose radius is r_{out} for each platelet. For there is a pipe whose radius is r_{in} in the middle of the central channel for opposing jet, the shape of the cross section of the coolant channel is an annulus. The coolant flows from the bottom of the nose tip via the annular interlayer and flows out of the nose tip via the radial grooves (Fig. 3). Because of the symmetry, 1/6 model is analyzed.

3. Wall temperature calculation method

The governing equation for heat transfer inside the nose tip solid wall is

$$\frac{\partial T}{\partial t} = \frac{\lambda}{\rho C_p}\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2}\right) \tag{1}$$

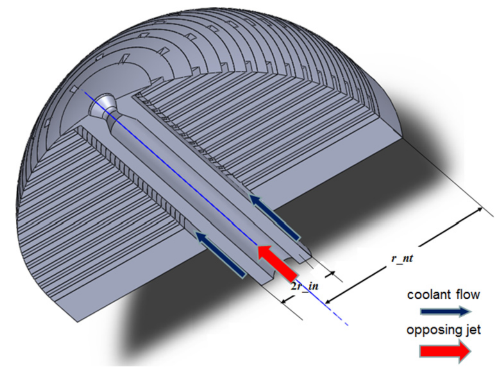

Fig. 3. The nose tip configuration with opposing jet and platelet transpiration.

where T is wall temperature, λ is thermal conductivity, ρ is density and C_p is thermal capacity. The coolant side boundary condition is

$$\lambda_w\left(\frac{\partial T}{\partial n}\right)_w = h_c(T_c - T_w) \tag{2}$$

where h is coolant heat transfer coefficient, subscript c is for coolant and w is for wall. The outflow side boundary condition is

$$\lambda_w\left(\frac{\partial T}{\partial n}\right)_w = q_w. \tag{3}$$

Finite Volume Element [21] is used to calculate the wall temperature. The coolant is also treated as volume elements. Then the energy equation of the kth volume element is

$$Q_k = \sum_{j=1}^{N} A_{kj}(T_j - T_k) \tag{4}$$

where Q_k is the energy of the kth volume element gains or loses and N is the total number of volume elements. During the calculation, several assumptions are made: a) there is convective heat transfer when the coolant flows and no radiation heat transfer; b) latent heat of coolant is ignored; c) the nose tip structure will not be expanded or deformed.

The heat transfer coefficient between wall and coolant is given by equation [9]

$$h_c = 0.027\frac{\lambda_c}{D_h}\,\mathrm{Re}^{0.8}\,\mathrm{Pr}^{0.33}(\mu_c/\mu_w)^{0.14} \tag{5}$$

where D_h is hydraulic diameter, Re is Reynolds number of coolant, Pr is Prandtl number of coolant and μ is dynamic viscosity. And the outer heat flux q_w through the surface of the nose tip is given by CFD method.

4. Flow field calculation for outer heat flux on nose tip

In order to obtain the outer heat flux, CFD method is used to calculate the outflow field. To simplify the calculation, the effect of the coolant at the grooves outlet is ignored.

4.1. Governing equations and numerical scheme

In the present study, axisymmetric Navier–Stokes equations are used as governing equations. AUSMPW scheme [23] with MUSCL interpolation [24] for convective terms and central difference is used for viscous terms. Full implicit LU-SGS method [25] for time integration is used and local time step is used.