International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:6, No:8, 2012

Influence of Cavity Length on Forward-facing Cavity and Opposing Jet Combined Thermal Protection System Cooling Efficiency

Hai-bo Lu, Wei-qiang Liu

Abstract**—**A numerical study on the influence of forward-facing cavity length upon forward-facing cavity and opposing jet combined thermal protection system (TPS) cooling efficiency under hypersonic flow is conducted, by means of which the flow field parameters, heat flux distribution along the outer body surface are obtained. The numerical simulation results are validated by experiments and the cooling effect of the combined TPS with different cavity length is analyzed. The numerical results show that the combined configuration dose well in cooling the nose of the hypersonic vehicle. The deeper the cavity is, the weaker the heat flux is. The recirculation region plays a key role for the reduction of the aerodynamic heating.

Keywords**—**Thermal protection, hypersonic vehicle, aerodynamic heating, forward-facing cavity, opposing jet

I. Introduction

THERE is a severe aerodynamic heating when an aircraft travels at high velocity. The scholars in thermal protection fields are always keep their eyes on the design of high speed vehicles' thermal protection system.

A body containing a forward-facing cavity under a supersonic flow was introduced firstly by Hartmann [1] in 1921, at that time it was used as a new technique for producing sound of high intensity and discrete frequency. In 1959, Burbank [2] reported this idea as a thermal protection technique for the nose-tip of hypersonic vehicles first. Yuceil [3] et al. using an infrared camera indicated that larger-diameter shallow cavities created a stable "cool ring" in the vicinity of the cavity lip, with temperatures locally lower than those of a simple spherical nose. W A Engblom and D B Goldstein[4] research on the distribution of the heat flux and pressure along nose-tip with forward-facing cavity, which *L/D* is 0.75 and 2.(The "L" is the length of cavity and "D" is the diameter of the cavity, shown in Fig.1. Siltond [5] et al. study on the effects of the cavity on ablation onset time and they validated the laminar assumption in CFD. Later, they investigated how to reduce the severe heating and delay the ablation onset in paper [6]. In addition, an experimental parameter study is undertaken to optimize the forward-facing cavity geometry for the most delayed ablation onset. The parameters of cavity length, lip radius, and diameter are independently optimized for a given nose-tip diameter. They found that the best *L*, for a given *Dn*, was four times *D*. The best r was one-fourth the difference between *Dn* and *D*. S Saravanan [7] et al. investigated the effects of a forward-facing cavity on heat transfer and aerodynamic coefficients. Numerical simulation was carried out with steady-state flow assumption and had a good agreement with their tests in hypersonic shock tunnel HST2, at a hypersonic Mach number of 8. The opposing jet was reported as a thermal protection technique for the nose-tip of hypersonic vehicles in early sixties of the 20th century, and the validating experiment was conducted [8]. The effect of total pressure ratio of opposing jet to free stream on the reduction of aerodynamic heating is investigated by Hayashi K et al[9, 10]. The experiment and numerical simulation results showed that as the pressure ratio was increased, the heat flux was decreased at each point. The detailed influences of the free mach number , jet mach number , attack angle on the drag coefficient reduction were studied by high precise simulation of Navier-Stokes equations [11]. In reference [12], three kinds of nose-tip with opposing jet were numerical studied under supersonic (*Ma*=3.98) and hypersonic (*Ma*=8.0) free stream condition. The results show that there is a direct correlation between the nose configurations and the thermal protection effect of the opposing jet, and of all three configurations, the extended nozzle model is found to be the most efficient configuration. In the present study, the forward-facing cavity and opposing jet combined TPS is investigated. The influence of the cavity length on thermal protection efficiency of the TPS is discussed. Due to the combined TPS, remarkable aerodynamic heating reduction in hypersonic flow field is revealed by detailed numerical simulation.

II. Configuration of the Combined TPS

The configuration of the forward-facing cavity and opposing jet thermal protection system is shown in Fig.1, the nozzle exit of the opposing jet is located at the center of the base wall of the cavity, and the diameter is 4mm.

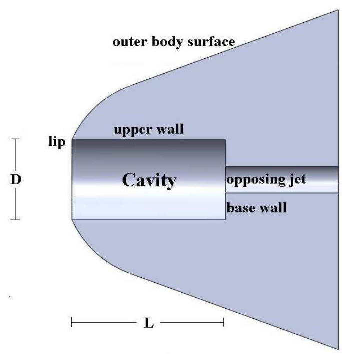


Fig. 1 Schematic of nose-tip with forward-facing cavity

Hai-bo Lu is with the Science and Technology on Scramjet Laboratory, National University of Defense Technology, ChangSha 410073, China (e-mail: lhbboo@sohu.com).

Wei-qiang Liu is with the Science and Technology on Scramjet Laboratory, National University of Defense Technology, ChangSha 410073, China (e-mail: liuweiqiang_1103@163.com).

International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:6, No:8, 2012

III. NUMERICAL METHOD

A. Governing Equations and Discretion

In the present study, the 3-D Navier-Stokes equations with k-ω turbulence model are used as governing equations. The convective terms are approximated using AUSM-DV splitting method, a MUSCL approach with Min-mod limiter is implemented to increase the numerical accuracy [13], and central difference method for the viscous terms. The LU-SSOR scheme is used for the time integration.

B. Grid and Flow Condition

The grid of simulation model (the case with L=32mm, D=8mm) on the symmetry plane and on the wall of the nose-tip is shown in Fig. 2 and Fig.3.

The flow condition is shown in Table I. The fluid medium for opposing jet is assumed as compression air. The wall boundary condition is used as a no-slip one and the wall temperature is assumed isothermal (300K).

TABLE I
FLOW CONDITION

Symbol	Quantity	value
	Free stream	
Ma_∞	Mach number	8
P_0/Pa	total pressure	1939211
$T_{0\infty}$ /K	total temperature	1955.14
	Opposing jet flow	
Ma_{opp}	Mach number	1
PR	ratio of total pressure of free stream to opposing jet	0.2
$T_{0\text{-}opp}$/K	total temperature	300

IV. NUMERICAL AND EXPERIMENTAL RESULTS COMPARISON

In order to validate the numerical results, numerical results of three validating examples (blunt cone, nose-tip with L24mm$D$6mm cavity and L24mm$D$12mm cavity) are compare with the experimental results [7]. Fig. 4 shows the Stanton number (St) along the outer body surface of validating examples. Stanton number based on the free stream condition, is given by the expression

$$St = \frac{q_w}{(T_{aw} - T_w)\rho_\infty c_{p\infty} u_\infty} \tag{1}$$

$$T_{aw} = T_\infty \left\{1 + \sqrt[3]{\mathrm{Pr}}\left[(\gamma-1)/2\right] M_{a\infty}^2\right\} \tag{2}$$

where q_w was the heat flux, T_{aw} was the temperature of the thermal isolation wall, T_w was the temperature of the outer body surface, ρ_∞ was the density of the free stream, $c_{p\infty}$ was the constant-pressure specific heat of the free stream, u_∞ was the velocity of the free stream, Pr was Prandtl number, γ was specific heat ratio.

A good agreement is shown between numerical and experimental results in the figures. Some errors come from the assumption of simulation model, counting error and experimental measurement.

The distribution of temperature on symmetry plane of the three validating examples is shown in Fig. 5, Fig. 6 and Fig. 7. Comparing these figures, the use of forward-facing cavity dose not change the shape of the bow shock but have a great effect on the distribution of temperature behind the shock, especially near the stagnation area.

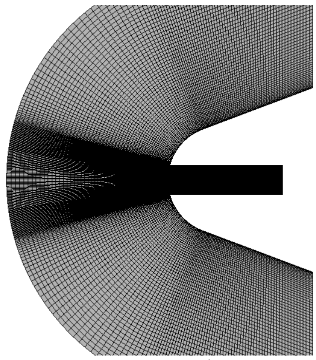


Fig. 2 Symmetry plane grid （L=32mm, D=8mm）

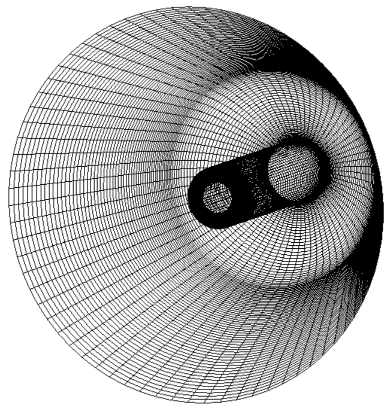


Fig. 3 Nose-tip surface grid (L=32mm, D=8mm)

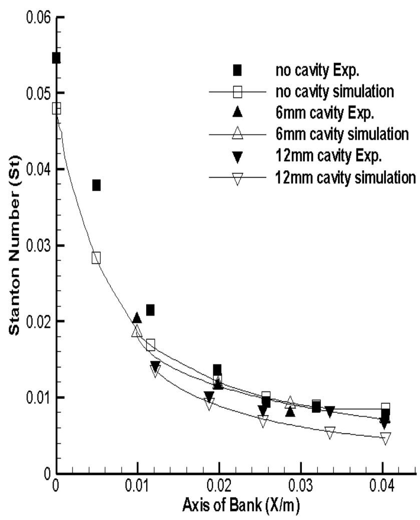


Fig. 4 Stanton number along outer body surface comparison between CFD and experiment [7]

International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:6, No:8, 2012

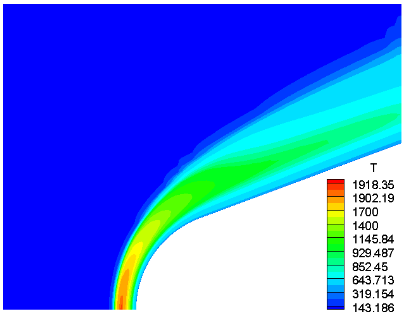


Fig. 5 Temperature/K contours of blunt cone

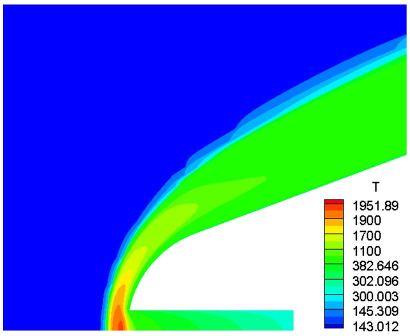


Fig. 6 Temperature/K contours of nose-tip with *D*=6mm cavity

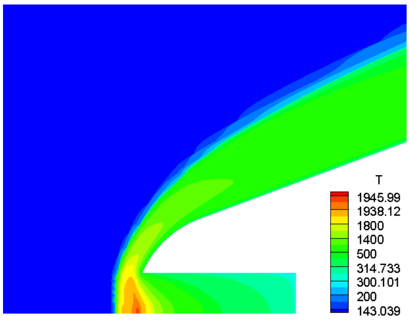


Fig. 7 Temperature/K contours of nose-tip with *D*=12mm cavity

V. Results and Discussion

In order to investigate the influence of the cavity length on thermal protection efficiency of the forward-facing cavity and opposing jet combined TPS, as shown in Table II, calculation cases with different length of cavity were established.

TABLE II
Calculation Cases of Cavity with Different Length

Case	1	2	3	4
L/mm	8	16	24	32
D/mm	8	8	8	8

A. Flow field

The distribution of streamline with Mach number contours, temperature and pressure of the four cases are shown respectively in Fig.8-Fig.11. From these figures, it is evident that after the opposing flow jets out from the nozzle, there is a rapid expansion and a clear reflected wave is formed from the upper wall in the cavity.

Out of the cavity, a Mach disk is formed. Opposing jet meets free stream and forms the interface. The free steam lets the jet layer reattach to body surface, recirculation region is formed around the cavity lip. There is another recirculation region which is located at the corner of the cavity. It is formed by the opposing jet and the shape of the cavity bottom.

In (b) of Fig.8 ~ Fig.11, the highest temperature region is formed behind the bow shock and in front of the cavity.

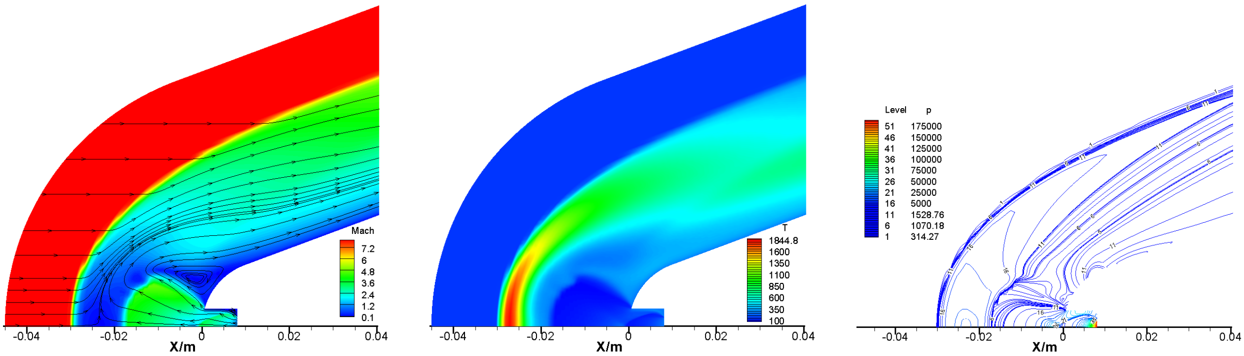
The (C) show that the main pressure alteration is concentrated at the exit of the opposing jet. Table III shows the pressure of the cases at the coordinate origin and the lip cusp. With the cavity length increasing, the pressures are increasing, too. The pressure at the coordinate origin of case 1 is more smaller than others, it is because that, in this case, the coordinate origin is in the severe expansion zone of opposing jet flow (Fig.8).

TABLE III
Pressures at Coordinate Origin and Lip Cusp

Pressure/Pa	1	2	3	4
at the coordinate origin	4461.79	31835.3	36182.1	37992.7
at the lip cusp	7446.79	7602.67	8403.24	9296.87

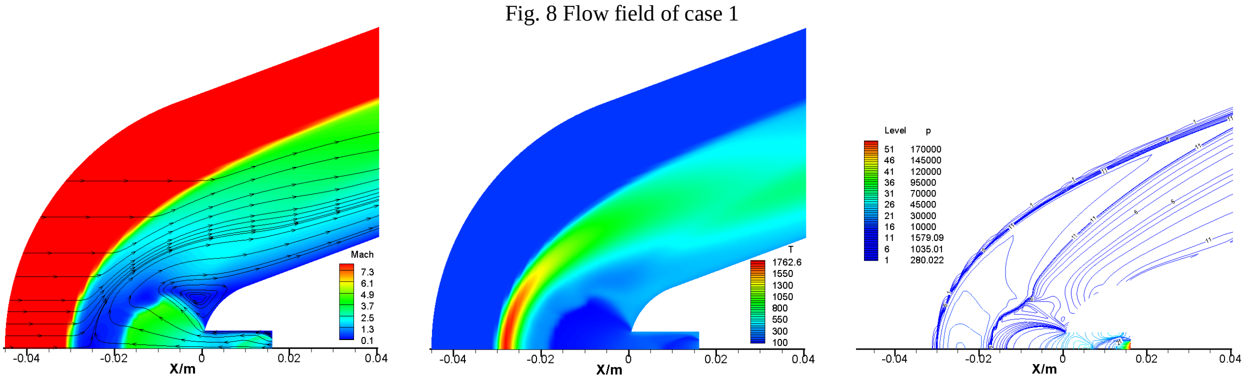
B. Aerodynamic heating

The Stanton number distribution along outer body surface for all cases and nose-tip with *L*24mm *D*6mm or 12mm single forward-facing cavity is shown in Fig.12 (a). Fig.12 (b) shows the Stanton number distribution of four cases. As seen in (a), the combined TPS dose well in cooling the nose tip, the cooling efficiency of combined TPS is much better than the single cavity. the aerodynamic heating is decreased heavily by addition of opposing jet. The expansion of opposing jet flow near the lip even cause the nose-tip heat release there (the Stanton number is negative).



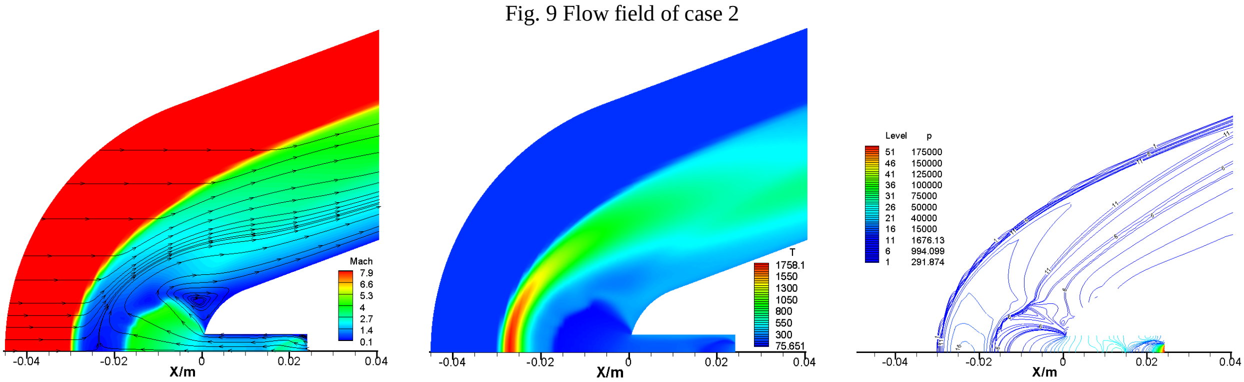
(a) Streamline (*Ma* contours) (b) Temperature/K (c) pressure/Pa

Fig. 8 Flow field of case 1



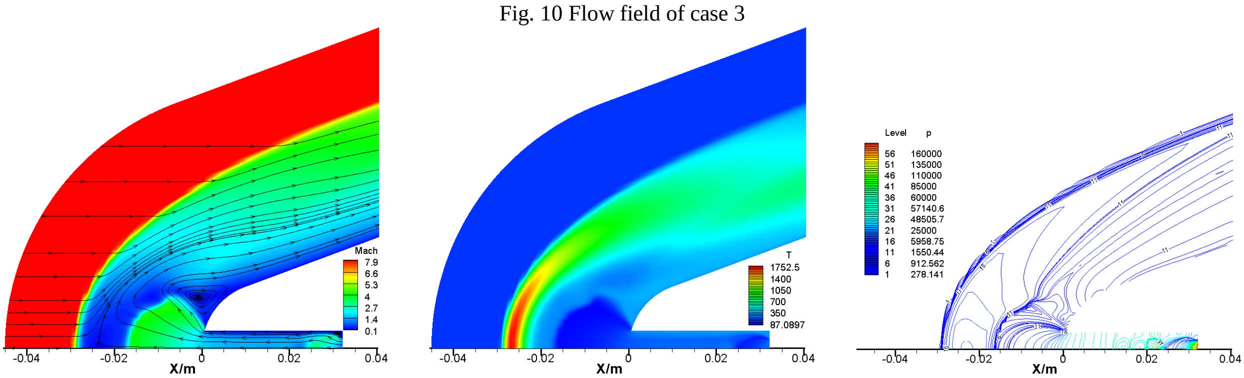
(a) Streamline (*Ma* contours) (b) Temperature/K (c) pressure/Pa

Fig. 9 Flow field of case 2



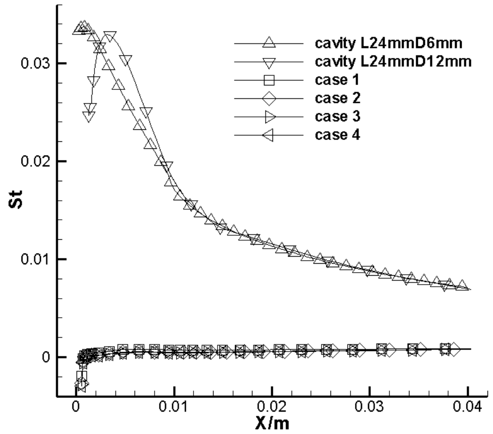
(a) Streamline (*Ma* contours) (b) Temperature/K (c) pressure/Pa

Fig. 10 Flow field of case 3

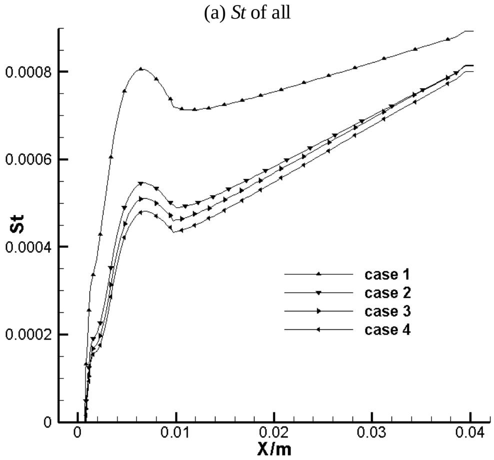


(a) Streamline (*Ma* contours) (b) Temperature/K (c) pressure/Pa

Fig. 11 Flow field of case 4



(a) *St* of all



(b) *St* of cases

Fig. 12 The Stanton number distributions along outer body surface for all cases

As seen in Fig.12 (b), the larger the cavity length is, the more the aerodynamic heating decreasing of nose-tip is. In Table III and (c) of Fig.8 ~ Fig.11, we can see that the combined TPS with larger length cavity has the larger pressure at the mouth of the cavity. This make the opposing jet flow has a stronger circumferential expansion (Fig.8~Fig.11). Then the recirculation region is pushed up, the high temperature free stream is separated further from the nose-tip by the low enthalpy opposing jet flow. The aerodynamic heating is reduced. The recirculation region plays an pivotal role for the heat flux reduction. Along the outer body surface of the nose-tip, the flow changes from a low speed expansion one around the hemisphere into a direct one along the cone. So, there is a change of heat flux tendency at the interface of the hemisphere and cone.

VI. Conclusion

(1) The single forward-facing cavity configuration dose well in cooling the nose of hypersonic vehicle especially at the stagnation point area.
(2) The cooling efficiency of combined TPS is much better than the single cavity at all point. The aerodynamic heating is decreased heavily by addition of opposing jet.
(3) With the opposing jet condition in this paper, the deeper cavity the combined TPS is, the smaller the heat flux along the outer body surface is.
(4) The recirculation region plays an pivotal role for the heat flux reduction.

Aknowledgement

This investigation was supported by a Major Program of National Natural Science Foundation of China (90916018), Research Fund for the Doctoral Program of Higher Education of China (200899980006) and Natural Science Foundation of Hunan Province ,China (09JJ3109).

References

[1] Hartmann J, Troll B, “On a New Method for the Generation of Sound Waves,” *Physics Review*, Vol. 20, No. 6, pp. 719–727, 1922.
[2] Burbank P B, Stallings R L, “Heat-Transfer and Pressure Measurements on a Flat Nose Cylinder at a Mach Number Range of 2.49 to 4.44,” NASA TM X-221, 1959.
[3] Yuceil B, Dolling D S, Wilson D, “A Preliminary Investigation of the Helmholtz Resonator Concept for Heat Flux Reduction,” AIAA 1993-2742, 1993.
[4] W A Engblom, D B Goldstein, “Nose-Tip Surface Heat Reduction Mechanism,” AIAA 96-0354, 1996.
[5] Silton S I, Goldstein D B, “Modeling of Nose Tip Ablation Onset in Unsteady Hypersonic Flow,” AIAA 2000-0204, 2000.
[6] Silton S I, Goldstein D B, “Ablation Onset in Unsteady Hypersonic Flow About Nose Tip with Cavity,” *Journal of Thermophysics and Heat Transfer*, Vol. 14, No. 3, 2000, pp. 421–434.
[7] S Saravanan, G Jagadeesh, K P J Reddy, “Investigation of Missile-Shaped Body with Forward-Facing Cavity at Mach 8,” *Journal of Spacecraft and Rockets*, Vol. 46, No. 3, May–June 2009, pp. 557-591.
[8] C. H. E. Warren, “An Experimental investigation of the effect of ejecting a coolant gas at the nose of a bluff body,” Journal of Fluid Mechanics, 1960, Vol.8, No. 3, P. 400-417.
[9] K. Hayashi, S. Aso, Effect of Pressure Ratio on Aerodynamic Heating Reduction due to Opposing Jet,” AIAA 2003-4041, 2003.
[10] K. Hayashi, S. Aso, Y. Tani, “Numerical Study of Thermal Protection System by Opposing Jet,” AIAA 2005-188, 2005.
[11] T. Tian, C. Yan, “Numerical simulation on opposing jet in hypersonic flow,” Journal of Beijing University of Aeronautics and Astronautics, 2008, Vol. 34, No. 1, P. 9-12.
[12] Isao Tamada, Shigeru Aso, Yasuhiro Tani. “Reducing Aerodynamic Heating by the Opposing Jet in Supersonic and Hypersonic Flows,” AIAA 2010-991, 2010
[13] Azevedo J L F, Heidi K, “Comparison of unstructured grid finite volume methods for cold gas hypersonic flow simulations,” AIAA-98-2629, 1998.