JOURNAL OF SPACECRAFT AND ROCKETS
Vol. 43, No. 1, January–February 2006

Engineering Notes

ENGINEERING NOTES are short manuscripts describing new developments or important results of a preliminary nature. These Notes should not exceed 2500 words (where a figure or table counts as 200 words). Following informal review by the Editors, they may be published within a few months of the date of receipt. Style requirements are the same as for regular contributions (see inside back cover).

Experimental Study on Thermal Protection System by Opposing Jet in Supersonic Flow

Kentaro Hayashi,* Shigeru Aso,† and Yasuhiro Tani‡
Kyushu University, Fukuoka 812-8581, Japan

Introduction

CURRENTLY, developments of reusable launch vehicle (RLV) for a low-cost space transportation system are in progress. In the development of RLV, one of the most important problems is the severe aerodynamic heating at the nose and leading edges of the vehicle. In such supersonic and hypersonic flights, prediction of aerodynamic heating and construction of proper thermal protection system are especially important. Heat-resistant tiles and ablators are currently used for thermal protection systems. However, those thermal protection systems are not reusable.

In the present study, the method using an opposing jet is proposed for fully reusable thermal protection system of RLV. The method can be considered to have almost the same effect of heat reduction at nose region as the method with mechanical spike.[1] The opposing jet works as an aerodynamic spike to move the detached shock wave away from the nose and form a recirculation region, which is quite effective to reduce aerodynamic heating at the nose region.

The schematic diagram of supersonic flowfields with opposing jet injected at the nose of a blunt body is shown in Fig. 1. In the flowfield, the opposing jet forms a Mach disk and contact surface with freestream. The jet layer reattaches to the body surface and forms a recirculation region between the nozzle exit and reattachment point of the jet layer. The recompression shock wave is formed near the reattachment point of the jet layer.

Many studies on opposing jet flow have been conducted in order to reveal the flow mechanism.[2–7] However, most of those studies are related to the stability of flowfield and oscillations of shock waves. Except for Warren,[6] not much study has been conducted to reveal the effects of opposing jet on reduction of aerodynamic heating.

In the present study, geometric ratio of diameters and Mach number are fixed. The flow stability is determined by the total pressure ratio of freestream to opposing jet. We define the total pressure ratio (PR) as follows:

$$PR = p_{0j}/p_{0\infty} \tag{1}$$

where p_{0j} is total pressure of the jet and $p_{0\infty}$ is total pressure of the freestream.

As Finley pointed out,[7] the flowfield is categorized into three conditions such as "stable," "unstable," and "transitional." The condition in which the total pressure ratio is relatively small and oscillation of detached shock wave is observed is called "unstable condition." The condition in which the total pressure ratio is relatively high and no oscillation of detached shock wave and oscillation of recompression shock wave are observed is called "stable condition." There is a transitional condition between unstable and stable conditions.

Warren[6] examined the cooling effect of an opposing jet. However, in his experiments the amount of reduction of aerodynamic heating was quite small. Finley[7] pointed out that the total pressure ratios in Warren's study were too small to form stable flowfields. The condition of the flowfields in Warren's experiments can be categorized as "unstable."

In the present work, we have studied the effect of an opposing jet both in the stable condition and the unstable condition and revealed the effects of opposing jet to the reduction of aerodynamic heating and flow mechanism.

Experimental Apparatus and Procedures

In the present study, a conventional blowdown-type supersonic wind tunnel at Kyushu University is used. The average experimental conditions are shown in Table 1.

The blunt-body model is used in the experiments. The diameter of the blunt body is 50 mm, and the sonic nozzle, the diameter of which is 4 mm and jet Mach number is 1.0, is set at the nose of the model in order to blow an opposing jet against the freestream. Secondary gas of N_2 is injected with specified stagnation pressure p_{0j}. The unit Reynolds number is 4.2×10^7/m. Flow around the model is visualized by the schlieren method.

Table 1 Experimental conditions

Characteristic	Freestream	Opposing jet
Gas	Air	Nitrogen
Mach number	3.98	1.0
Total pressure	1.37 MPa	——
Total temperature	397 K	300 K
Total pressure ratio	——	0.2–0.8

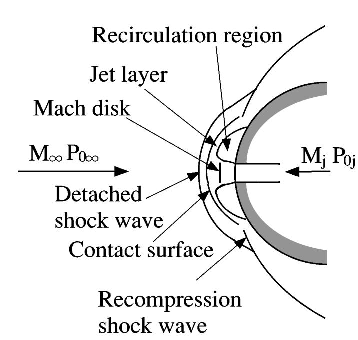
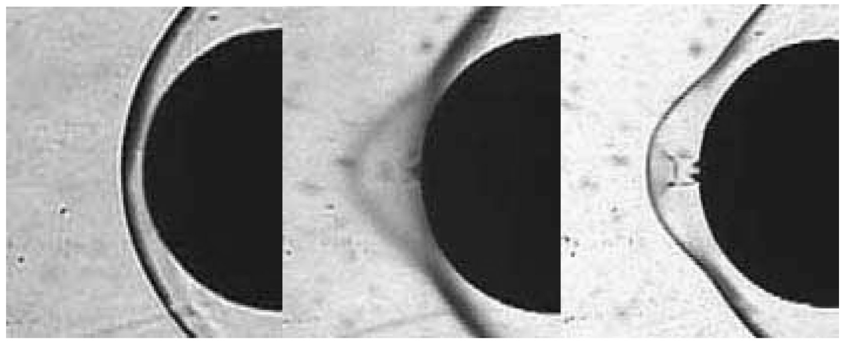


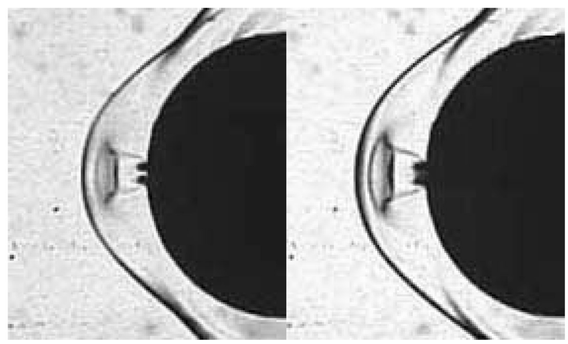
Fig. 1 Flowfield of opposing jet to supersonic mainstream.

Presented as Paper 2003-4041 at the AIAA 33rd Fluid Dynamics Conference, Orlando, FL, 23–26 June 2003; received 3 January 2005; revision received 6 April 2005; accepted for publication 24 May 2005.

*Graduate Student, Department of Aeronautics and Astronautics, 6-10-1, Hakozaki, Higashi-ku. Student Member AIAA.
†Professor, Department of Aeronautics and Astronautics, 6-10-1, Hakozaki, Higashi-ku. Senior Member AIAA.
‡Assistant Professor, Department of Aeronautics and Astronautics, 6-10-1, Hakozaki, Higashi-ku.



a) PR = 0 (no jet) **b) PR = 0.20** **c) PR = 0.40**



d) PR = 0.60 **e) PR = 0.80**

Fig. 2 Schlieren photographs.

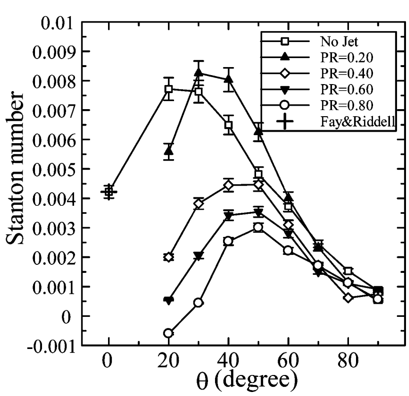


Fig. 3 Distribution of Stanton number.

The measurement of heat flux is based on the calorimeter technique.[8] A substance of calorimeter is copper. The diameter and length of the calorimeter are 2 and 5 mm, respectively. The temperature of the calorimeter is measured by the thermocouples, which are attached to the bottom of the calorimeter. Several calorimeter sensors are set in the blunt body. The measuring points are located from $\theta = 20$ deg through 90-deg × 10-deg increments from the top of the model, where θ is the angle measured from central axis of model.

Experimental Results and Discussion

The results of the flow visualizations are shown in Fig. 2. Because the case of PR = 0.20 (Fig. 2b) is categorized as the unstable condition, the shock wave is invisible because of unsteady oscillation of the bow shock wave. The cases in which PR is larger than 0.40 (Figs. 2c–e), are categorized as the stable condition because the bow shock wave is stationary. In those results, the positions of the bow shock wave, the barrel shock wave, and Mach disk are clearly observed. The bow shock wave moves upstream as PR is increased.

The results of heat-flux distributions in the experiments and theoretical heat flux of stagnation point of blunt body, which is calculated from theory of Fay and Riddell,[9] are shown in Fig. 3.

In the present experiments, Stanton number St is used to compare each heat-flux distributions. Stanton number is defined as

$$St = q_w/(T_{\rm aw} - T_w)\rho_\infty c_{p_\infty} u_\infty \tag{2}$$

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

where q_w is surface heat flux, $T_{\rm aw}$ is an adiabatic wall temperature, T_w is the wall temperature measured in the experiments, ρ_∞ is the freestream density, u_∞ is the freestream velocity, T_∞ is the freestream temperature, M_∞ is the freestream Mach number, Pr_w is the Prandtl number, c_{p_∞} is the specific heat at constant pressure, and γ is the ratio of specific heats. The Prandtl number is set as 0.71.

In the case of no jet, the heat flux obtained in the experiment is equal to theoretical heat flux calculated from Fay and Riddell theory at $\theta = 0$ deg (stagnation point). Considering that Reynolds number is 4.2×10^7/m, the boundary-layer transition can occur between $\theta = 0$ and 20 deg. Therefore a boundary layer becomes a turbulent boundary layer, and heat flux over 20 deg becomes high.

In the case of PR = 0.20, the flowfield is unstable, and the self-induced oscillations of the bow shock wave are observed. Compared with the no-jet case, the heat flux is higher except at $\theta = 20$ deg. The result is similar to that of Warren's experiments.[6] Because the opposing jet does not form a stable flowfield, the flowfield is quite unsteady. The body surface is not perfectly covered with the jet gas, and hot freestream gas sometimes reaches to the wall directly. Hence the heat-flux distribution is higher than that with no jet.

In the case of PR = 0.40, 0.60, and 0.80, where these flows are classified as stable condition, the heat-flux distributions show same tendency.

In the case of PR = 0.40, the remarkable reduction of heat flux is observed. The value of heat flux at $\theta = 20$ deg is 25% of that of the no-jet case. The remarkable reduction of heat flux is observed. Heat flux gradually increases up to $\theta = 50$ deg, then it decreases over 50 deg. The values of heat flux over $\theta = 50$ deg are about 60–80% of that of the no-jet case.

As a whole, the reduction of heat flux is observed in the whole region of the blunt body. Because the body surface is covered with the recirculation region formed by secondary gas, the heat-flux distribution becomes small. The freestream hot gas, which is detached from the body by the opposing jet near the stagnation region, gradually gets closer to the body surface. This causes a gradual increase of heat flux between 20 and 50 deg. As the body surface is covered with the secondary gas, heat-flux distribution over 50 deg is smaller than that of the no-jet case.

As the pressure ratio is increased, the reduction of aerodynamic heating becomes significant. For example, at $\theta = 20$ deg the value of heat flux in the case of PR $= 0.60$ is reduced to 7% of the value of the no-jet case. In the case of PR $= 0.80$, negative heat flux of -8% of the value of the no-jet case is observed. The reason why the heat flux is decreased between 20 and 50 deg at higher PR is that the recirculation region becomes large because of increase of the mass flux of the secondary gas and the distance between the hot freestream and the wall becomes large. Because the coolant gas has the same effect as film cooling, the heat-flux distribution becomes smaller over $\theta = 50$ deg when the pressure ratio is increased.

Reduction of Total Heat Load by Opposing Jet

In the reduction of aerodynamic heating, it is important to reduce the maximum value of heat flux. Also it is important to reduce the total heat load to the body.

The total heat load Q is estimated as an integration of heat-flux distribution over the surface of blunt body as follows:

$$Q = 2\pi R^2 \int_{\theta=20}^{\theta=90} q_w \sin\theta \, \mathrm{d}\theta \tag{4}$$

where R is radius of nose of blunt body, q_w is measured heat flux, and θ is the angle measured from central axis of model. For the evaluation of total heat load, we integrate surface heat flux between $\theta = 20$ and 90 deg because it is impossible to measure heat flux between $\theta = 0$ and 20 deg because of hole of opposing jet. Even if we evaluate total heat load assuming constant heat flux between $\theta = 0$ and 20 deg, both integrations are within an error of 5%. Hence, the evaluation of total heat load Q, based on Eq. (4), is considered accurate enough to discuss the difference of the total heat load because of opposing jet at various PR.

The comparison of total heat loads at various PR is shown in Fig. 4. The total heat load of PR $= 0.20$, which is in unstable condition, is larger than that of PR $= 0$. In the stable cases of PR $= 0.40$, 0.60, and 0.80, the total heat load decreases as the total pressure ratio is increased. The total heat load of PR $= 0.60$, and 0.80 become 58% and 53%, respectively, compared with that of PR $= 0$, which means that the total heat load can be reduced to nearly half.

In the unstable condition the total heat load increases because the body surface is not perfectly covered with the jet gas and hot freestream gas sometimes reaches to the wall directly. It means the present thermal protection system with opposing jet does not work well in unstable condition of lower PR.

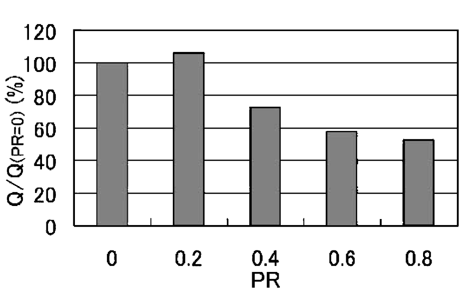


Fig. 4 Total heat load.

In the stable condition of higher PR, the present thermal protection system with the opposing jet shows significant reduction of aerodynamic heat load as the total pressure ratio is increased.

In the stable condition, as the total pressure ratio is increased thermal protection effect becomes higher. However, a decrease of total heat load becomes smaller gradually as the total pressure ratio is increased. It can be caused by following reasons. Significant change of heat-flux distribution is observed below $\theta = 50$ deg, and small change is observed after $\theta = 50$ deg as shown in Fig. 3. The ratio of surface area between 50 and 90 deg occupies about 65% of the total surface area of blunt body. Therefore significant reduction in lower angle of θ does not greatly contribute to the total heat load to the blunt body. The result suggests that an additional method to reduce surface heat flux over $\theta = 50$ deg is necessary to reduce total heat load more.

Conclusions

In the present study, the reduction of aerodynamic heating by opposing jet is experimentally investigated. The major conclusions are summarized as follows:

1) In the stable condition, a significant reduction of surface heat flux has been observed. The reduction of aerodynamic heating caused by the opposing jet is proved to be quite effective at the nose of the blunt body.

2) As the pressure ratio is increased, the heat flux decreases over the model surface, and remarkable reduction of aerodynamic heating is observed in nose region at high PR.

3) In the unstable condition, no reduction of aerodynamic heating is observed. The results show that PR should be large enough to form stable flow for the reduction of aerodynamic heating.

4) The total heat load to the blunt body is also estimated. In the stable condition, as the total pressure is increased, thermal protection effect becomes higher.

References

[1]Motoyama, N., Mihara, K., Miyajima, R., Watanuki, T., and Kubota, H., "Thermal Protection and Drag Reduction with Use of Spike in Hypersonic Flow," AIAA Paper 2001-1828, April 2001.

[2]Laptoff, M., "Wingflow Study of Pressure Drag Reduction at Transonic Speed by Projecting a Jet of Air from the Nose of a Prolate Spheroid of Fineness ratio 6," NACA RM L5109, Oct. 1951.

[3]Love, E. S., "The Effects of a Small Jet of Air Exhausting from the Nose of a Body of Revolution in Supersonic Flow," NACA RM L52I19a, Nov. 1952.

[4]Love, E. S., Grigsby, C. E., Lee, L. P., and Woodling, M. J., "Experimental and Theoretical Studies of Axisymmetric Free Jets," NACA TR 6, 1959.

[5]Charczenko, N., and Hennessy, K. W., "Investigation of a Retrorocket Exhausting from the Nose of a Blunt Body into a Supersonic Free Stream," NASA TN D-751, 1961.

[6]Warren, C. H. E., "An Experimental Investigation of the Effect of Ejecting a Coolant Gas at the Nose of a Bluff Body," *Journal of Fluid Mechanics*, Vol. 8, 1960, pp. 400–417.

[7]Finley, P. J., "The Flow of a Jet from a Body Opposing a Supersonic Free Stream," *Journal of Fluid Mechanics*, Vol. 26, 1966, pp. 337–368.

[8]Schultz, D. L., and Jones, T. V., "Heat-Transfer Measurements in Short-Duration Hypersonic Facilities," AGARDograph 165, Feb. 1973.

[9]Fay, J. A., and Riddell, F. R., "Theory of Stagnation Point Heat Transfer in Dissociated Air," *Journal of the Aeronautical Sciences*, Vol. 25, No. 2, 1958, pp. 73–85.

T. Lin
Associate Editor