Thermal Protection System for Re-Entry Vehicle in High Speed Flows

Yash Dadlani
Department of Aerospace Engineering
IIT Kanpur

CONTENTS

LIST OF FIGURES

LIST OF TABLES

Thermal Protection System for Re-Entry Vehicle in High Speed Flows

***Abstract*—The reduction of drag and aerodynamic heating is the most important design objective for hypersonic vehicles. Sadly, these two goals frequently clash. On the one hand, the design of sharp, narrow forebodies minimizes drag, ensuring greater ranges and more cost-effective flights. They are more susceptible to aerodynamic heating, though. Yet when it comes to aerodynamic heating, blunt forebodies are preferable even if they cause greater drag. Moreover, blunt geometries are favoured over narrow ones in the context of hypersonic vehicles due to practical increased volumetric efficiency, and better accommodation of crew or onboard equipment.**

A blunt object travelling at hypersonic speeds generates a strong bow shock wave in front of its nose, which is what causes the high drag and aerodynamic heating levels. Many efforts have been made to alter the flowfield in front of the vehicle's nose in order to decrease both the drag and the aerodynamic heating. For the RLV's entirely reusable thermal protection system, the opposing jet approach is suggested in the current study. In order to generate a recirculation zone and transfer the detached shock wave away from the nose, the opposing jet acts as an aerodynamic spike. This is a very effective approach to reduce aerodynamic heating in the nose region.

I. Introduction

Reusable launch vehicle (RLV) research for a low-cost space transportation system is now under way. The significant aerodynamic heating at the vehicle's nose and leading edges is one of the biggest issues with the development of RLV. Prediction of aerodynamic heating and the creation of an appropriate thermal protection system is particularly crucial in such supersonic and hypersonic missions. Thermal protection systems now employ heat-resistant tiles and ablators. These heat protection systems, however, cannot be reused. For the RLV thermal protection system, a totally reusable opposing jet approach is suggested. The method can be considered to have almost the same effect of heat reduction at the nose region as the method with a mechanical spike [1]. The opposing jet functions as an aerodynamic spike to create a recirculation zone and move the detached shock wave away from the nose. This method of reducing aerodynamic heating in the nose area is quite successful. A supersonic flowfield is shown schematically in Fig. 1 with an opposing jet injected at the nose of the blunt body. In the flowfield, the opposing jet forms a Mach disc and a contact surface with the freestream. Between the nozzle exit and the point where the jet layer reattaches to the body surface, a recirculation area is created. The reattachment point of the jet layer serves as the location where the recompression shock wave is formed.

Studying the flowfield in front of the nose is also essential. Many investigations have been done on the stability of flowfield and shockwave oscillations. The flow stability is determined by the total pressure ratio of the freestream to the opposing jet [2]. The total pressure ratio is defined as:

$$PR = P_{o}j/P_{o}\infty$$

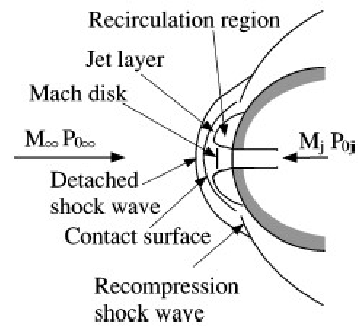

Fig. 1 Flowfield of opposing jet to supersonic mainstream.

Three states of the flowfield are distinguished: "stable," "unstable," and "transitional."[2] Unstable situation is the state in which oscillation of the detached shock wave is seen and the overall pressure ratio is relatively low."Stable condition" is the state in which there is no oscillation of the detached shock wave or oscillation of the recompression shock wave, and the total pressure ratio is often high. Between unstable and stable conditions, there is a transitional state. Warren [3] looked at how a counterflowing jet could cool things down. Yet, in his trials, there wasn't much of a reduction in aerodynamic heating. As noted by Finley[2], the total pressure ratios in [3] were insufficient to create stable flowfields. In the current work, we investigate the impact of an opposing jet in both stable and unstable conditions and will reveal how it affects the flow mechanism and aerodynamic heating.

II. Experimental Method

Table 1 displays the average experimental conditions. The tests make use of the blunt-body model. The model's sonic nozzle, which has a 4 mm diameter, ejects a jet of nitrogen as secondary gas at specified stagnation temperature, P_{oj} (Total Pressure). The nozzle is centered at the model's nose to eject the gas against freestream. The blunt body's diameter is 50 mm. The Reynolds number in units is $4.2*10^7$.

Schlieren Technique is used to visualise flow around the model. For quantifying heat flow, calorimeter is used[4]. Copper is the calorimeter material. The calorimeter has a 2

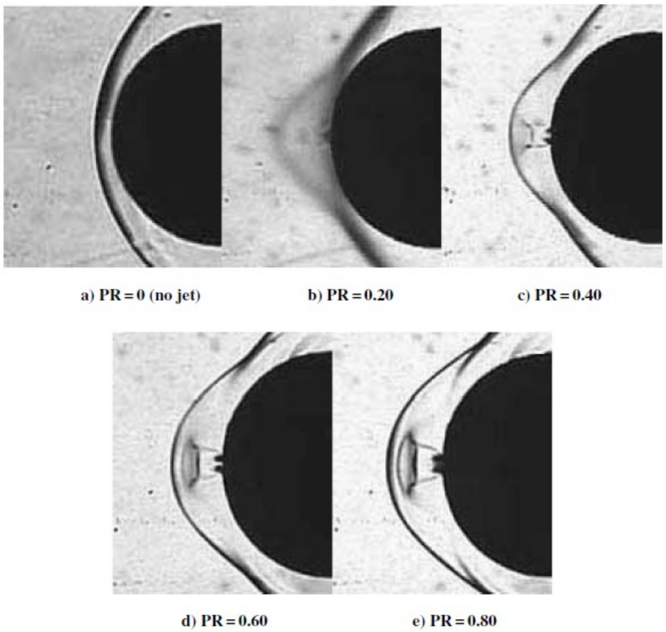

Fig. 2 Schlieren photographs.

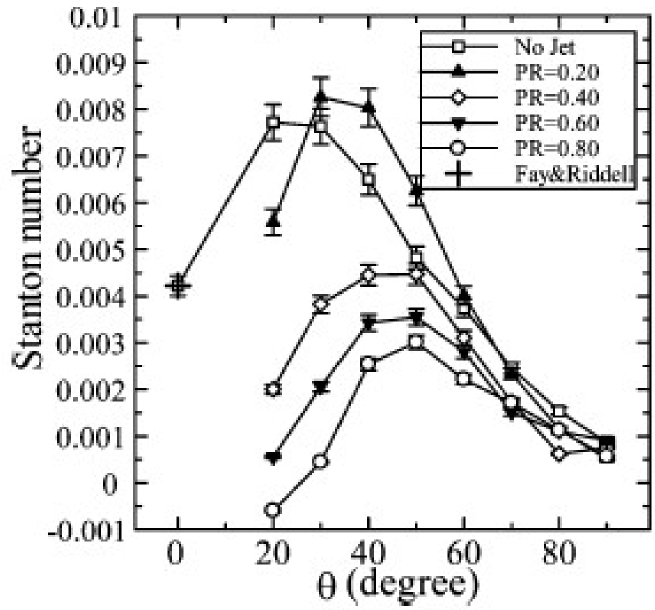

Fig. 3 Distribution of Stanton number.

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

mm diameter and a 5 mm length. The thermocouples, which are affixed to the calorimeter's bottom, are used to gauge its temperature. In the blunt body there are several calorimeter sensors installed. The measurement points are situated from θ = 20 deg to 90 deg, in intervals of 10 deg from the model's top, where θ is the angle measured from the model's central axis.

III. Experimental results

Flow Visualizations are showed in Fig.2 .For PR = 0.2(Fig 2b), the flow is unstable since the shock wave is not clearly distinguished. It is due to self induced oscillation of bow shock wave. The cases in which PR is larger than 0.40 (Figs. 2c–e), are 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, is calculated from Fay and Riddell[5] in Fig[3].

$$\text{St} = \text{q}_w/(T_aw - T_w)\rho_\infty C_p\infty U_\infty$$

$$\text{T}_aw = T_\infty(1 + \sqrt[3]{Pr_w}[(\gamma - 1)/2]M_\infty^2)$$

where q_w is the surface heat flux, T_{aw} is the adiabatic wall temperature, ρ_∞ is the freestream density, U_∞ is the freestream velocity, T_∞ is the freestream temperature, Pr_w is the prandtl no. , $C_{p\infty}$ is the specific heat at constant pressure, M_∞ is the free stream mach no. and γ is the ratio of specific heat. The Pr no. is 0.71.

For case of no jet, the heat flux in experiment is equal to theoretical heat flux from [5] at stagnation point. Since our Re no. is $4*10^7$, the boundary layer transition can occur between θ = 0 and 20 deg. This implies that heat flux over 20 deg becomes high because boundary layer is turbulent.

In the scenario where PR=0.20, the flowfield is unstable, and the shockwave oscillations are observed. With the exception of at θ = 20 deg, the heat flow is larger than in the no-jet situation. Hot freestream gas sometimes directly touches the wall because the body surface is not completely covered by the jet gas. As a result, there is more heat flow than there would be without the jet.

When PR = 0.4, a dramatic decrease in heat flow is seen. At 20 deg, the heat flow value is 25% lower than in the no-jet situation. Heat flux gradually rises up to a temperature of 50 deg, after which it starts to fall. The heat flow values overθ =50 deg are around 60-80% of what they would be in the no-jet condition. The whole area of the blunt body exhibits a decrease in heat flow overall. The heat-flux distribution becomes minimal because the body surface is covered by the recirculation area created by secondary gas. The opposing jet in the stagnation area pushes the freestream hot gas away from the body. However, for 20 deg $< \theta <$ 50 deg, the heat flux gradually increases, because the freestream hot gas gets closer to the surface.

When the PR is raised, the reduction in aerodynamic heating becomes significant. For instance, the heat flow value at 20 degrees with PR=0.60 is just 7% of the value with no jet. At PR=0.80, a negative heat flow that is around 8% of the value of the no-jet scenario is observed. The recirculation area grows larger and the heat flux drops between 20 and 50 deg at higher PR as a result of the rise in the mass flow of the secondary gas and the accompanying increase in the distance between the hot freestream and the wall. As the PR is raised, the heat-flux distribution decreases over θ = 50 deg because the coolant gas has the same effect as film cooling.

IV. Total Heat Load Calculation

It's crucial to lower the maximum value of heat flux in order to decrease aerodynamic heating. Minimizing the body's overall heat load is also necessary. The total heat load Q is estimated as the integration of the measured heat flux for 20 deg $< \theta <$ 90 deg. The integration over 0 deg $< \theta <$ 20 deg is not considered because it is difficult to calculate the heat flux in the stagnation region because the opposing jet emanates from nose. The integration is reasonably accurate with an error of 5 % .

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

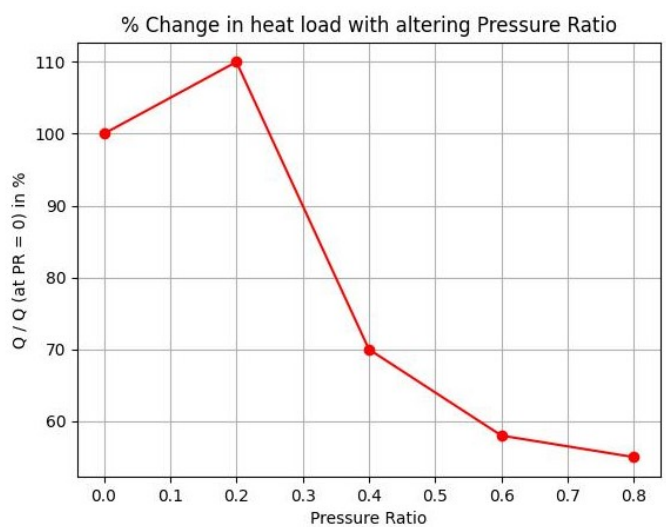

Figure 4 compares the overall heat loads at various PRs. The overall heat load of the unstable PR=0.20 is greater than the stable PR=0. The total heat load reduces when the total pressure ratio is raised in the stable situations of PR=0.40, 0.60, and 0.80. In comparison to PR=0, the total heat load of PR=0.60 and PR=0.80 is lowered by 58% and 53%, respectively, meaning that the total heat load may be cut in half. Because the body surface is occasionally directly exposed to hot freestream gas in the unstable situation (PR = 0.2) and not completely covered by jet gas, the overall heat load increases. It means the present case of opposing jet flow for PR = 0.20 does not work well in reducing the heat load.

The current thermal protection system with the opposing jet exhibits a substantial reduction in aerodynamic heat load under steady conditions of higher PR. As the overall pressure ratio rises under stable conditions, the thermal protection effect also rises. Yet, as the overall pressure ratio is raised, there is a slow decrease in the total heat load.It can be caused by following reasons. Significant change of heat-flux distribution is observed below θ = 50 deg, and small change is observed above = 50 deg as shown in Fig. 3.

More than half of the total surface area of a blunt body is occupied by the between 50 and 90 deg. As a result, a large reduction in the lower angle of θ does not significantly increase the overall heat load on the blunt body. The outcome indicates that in order to decrease total heat load further, a second strategy to lower surface heat flux above

$$\theta$$

= 50 deg is required.

V. Conclusion

The present work investigates experimentally how opposing jets might reduce aerodynamic heating. The following summarises the main conclusions:

1) It has been seen that the surface heat flow has significantly decreased in the steady state. At the blunt body's nose, it has been demonstrated to be fairly efficient in reducing the aerodynamic heating brought on by the opposing jet.
2) The heat flow throughout the model surface reduces as the pressure ratio rises, and at high PR, a substantial drop in aerodynamic heating is shown in the nose region.
3) No decrease in aerodynamic heating is seen in the unstable situation. The findings indicate that PR needs to be sufficiently large to create a stable flow and reduce aerodynamic heating.
4) An estimate of the blunt body's overall heat load is also made. As the overall pressure rises under stable conditions, the thermal protection effect increases.

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] 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.

[3] 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.

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

[5] 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.