Numerical Study on Aerodynamic Heating Reduction by Opposing Jet

Hayashi, Kentaro
Department of Aeronautics and Astronautics : Graduate Student

Aso, Shigeru
Department of Aeronautics and Astronautics : Professor

Tani, Yasuhiro
Department of Aeronautics and Astronautics : Assistant Professor

https://hdl.handle.net/2324/3293

出版情報：九州大学工学紀要. 66 (1), pp.39-54, 2006-03. 九州大学大学院工学研究院
バージョン：

Memoirs of the Faculty of Engineering, Kyushu University, Vol.66, No.1, March 2006

Numerical Study on Aerodynamic Heating Reduction by Opposing Jet

by

Kentaro HAYASHI[*], Shigeru ASO[**] and Yasuhiro TANI[***]

(Received February 20, 2006)

Abstract

A numerical study on a reduction of aerodynamic heating by opposing jet in supersonic and hypersonic flow has been conducted. Flow field around a hemisphere model is calculated in the free stream of Mach number of 4 ands 8. The coolant gas is injected through the nozzle at the nose of the model. In numerical analysis, axisymmetric Navier-Stokes equations are solved with k-ω turbulence model. Flow structures at various conditions are investigated and flow mechanisms of reduction of aerodynamic heating are revealed. Also optimal jet conditions for the reduction of aerodynamic heating are obtained.
According to the investigation of various conditions of opposing jet, important phenomena of flow field and some effective jet conditions are found. It becomes clear that a performance of the thermal protection system by opposing jet is greatly affected by jet conditions.

Keywords: Aerodynamic heating, Jet, Supersonic flow, Numerical analysis

1. Introduction

Currently, developments of RLV (Reusable Launch Vehicle) 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. Mechanical spike and ablator 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 re-circulation region, which is quite effective to reduce aerodynamic heating at the nose region.

[*] Graduate Student, Department of Aeronautics and Astronautics
[**] Professor, Department of Aeronautics and Astronautics
[***] Assistant Professor, Department of Aeronautics and Astronautics

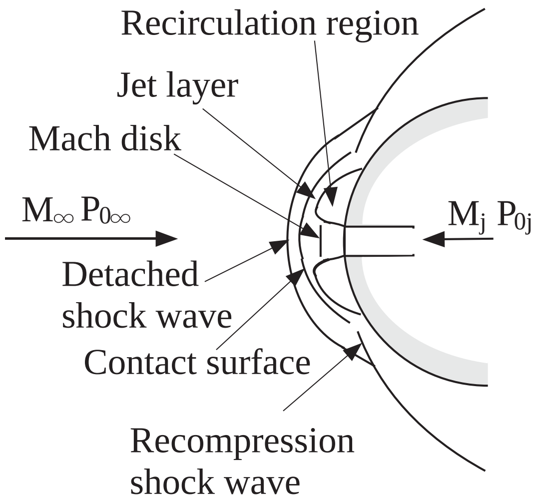


Fig. 1 Flow field of opposing jet to supersonic main stream.

The schematic diagram of supersonic flow fields with opposing jet injected at the nose of a blunt body is shown in **Figure 1**. In the flow field, the opposing jet forms a Mach disk and contact surface with free stream. The jet layer reattaches to the body surface and forms a re-circulation 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 flow field and oscillations of shock waves. Except for Warren[6)], quite little study has been conducted to reveal the effects of opposing jet on reduction of aerodynamic heating.

Governing parameters of the flow field of opposing jet are follows.

- Total pressure ratio between free stream and opposing jet
- Mach number of free stream and opposing jet
- Diameters of blunt body and nozzle exit

The total pressure ratio is defined as PR.

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

where p_{0j} is total pressure of the jet, $p_{0\infty}$ is total pressure of the free stream.

As Finley pointed out[7], the flow field 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 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 flow fields. The condition of the flow fields in Warren's experiments can be categorized as “unstable”.

In the present work, we have studied the effect of 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.

In addition, considering thermophysics of flow field, the temperature of free stream and opposing jet is important parameter of flow field. Therefore total temperature of free stream and opposing jet is added to governing parameters of flow field of opposing jet.

In our previous study[8-9], experimental results show that significant decreases of surface heat flux are obtained and opposing jet is proved to be effective on aerodynamic heating reduction around a stagnation region of the blunt body. However, thermophysics of the flow field is hardly known. Therefore numerical simulations of the flow field of opposing jet have been conducted in the present study. As the flow of the present study will be turbulent, a turbulence model should be considered in the numerical analysis. Thus k-ω turbulence model[10] is applied.

In the present study, we have investigated the effects of governing parameters of flow field and physical mechanics of aerodynamic heating reduction by opposing jet. Total pressure ratio, diameter of nozzle exit, Mach number of nozzle exit and temperature of nozzle exit were changed. Flow conditions are shown in **Table.1 and 2.**

Table 1 Flow conditions in supersonic flow.

Free stream	Gas	Air
	Mach number	3.98
	Total pressure	1.37MPa
	Total temperature	397K
Opposing jet	Gas	Air
	Mach number	1.0, 2.0, 3.0
	Total pressure ratio, PR	0.4 - 0.8
	Total temperature	200,300,400K
Wall	Temperature	295K

Table 2 Flow conditions in hypersonic flow.

Free stream	Gas	Air
	Mach number	8
	Total pressure	4.50MPa
	Total temperature	800K
Opposing jet	Gas	Air
	Mach number	1.5
	Total pressure ratio, PR	0.025 - 0.097
	Total temperature	300K
Wall	Temperature	300K

2. Governing Equations and Numerical Methods

2.1 Governing equations and numerical methods

In the present study, axisymmetric full Navier-Stokes equations are used as governing equations. AUSM-DV scheme[11)] with MUSCL interpolation for convective terms and full implicit LU-ADI factorization method[12)] for time integration are used. Central difference is used for viscous terms.

2.2 Turbulence model

The flow is turbulent around the body because Reynolds number based on the diameter of blunt body is 2.1×10^6. Heat flux distributions of experiments show that the flow is turbulent flow. Hence a turbulence model should be considered in the numerical analysis. k-ω model[10)] which is proposed by Wilcox is used. To avoid excessive production of turbulence energy, we modify C_μ function as an expression by Craft et.al[13)].

2.3 Grid and flow conditions

Figure 2 shows the grid system used in the present study. The number of grid points is 242 in the ξ direction (along the body) and 160 in the η direction (perpendicular to the body). 61 points in the ξ direction are distributed to express the exit of the coolant gas at the nose of the body. The physical properties at nozzle exit are given by use of the isentropic relations for the prescribed total pressure ratio and total temperature of the jet. Flow conditions are shown in **Table.1 and 2.**

2.4 Boundary conditions

The pressure gradient perpendicular to the wall is assumed to be zero and no slip boundary condition is applied on the wall. The wall temperature is assumed to be constant.

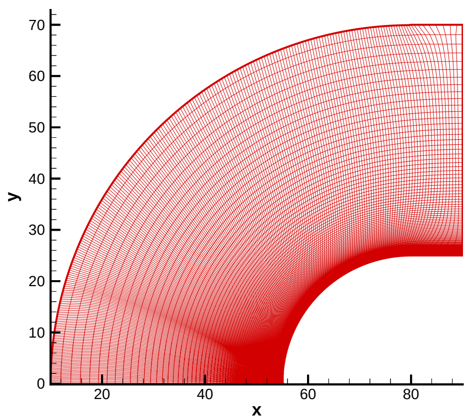
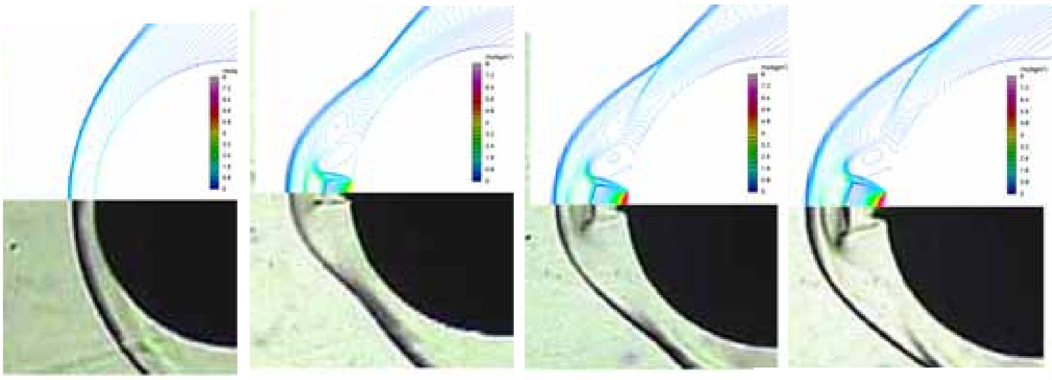


Fig.2 Grid of blunt body (242×160).

3. Aerodynamic Heating Reduction in Supersonic Flow

3.1 Comparisons between experiments and numerical results

Figure 3 shows comparison between density contours and Schlieren photographs. Numerical results show good agreement with experiments. **Figure 4** shows heat flux distribution at each total pressure ratio. It also shows good agreement enough to discuss themophysics of flow field.



No jet PR=0.40 PR=0.60 PR=0.80

Fig. 3 Density contours and Schlieren photographs.

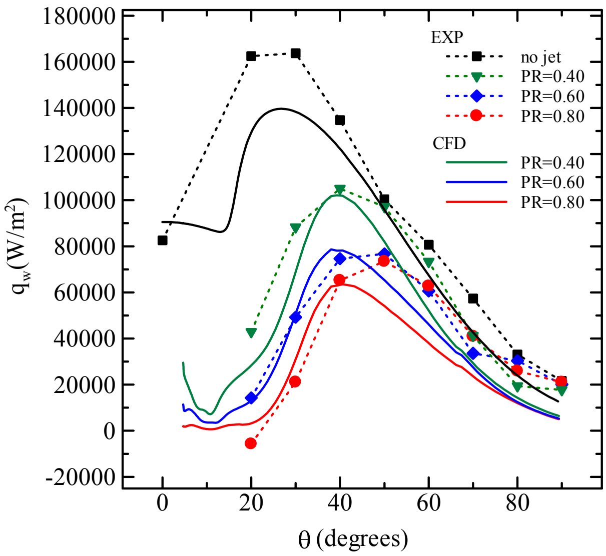


Fig. 4 Heat flux distributions ($M_j = 1.0, d_j = 4mm$).

3.2 Effects of total pressure ratio

Figure 4 shows heat flux distributions of each total pressure ratio. Mach number of jet is unity at nozzle exit and total temperature of jet is 300K. As the total pressure ratio, PR, increases, heat flux decreases.

Figures 5, 6 and 7 show Mach number contours, temperature contours and streamlines respectively. As the total pressure ratio increases, a mass flow ratio of jet increases and a shape of barrel shock wave changes. As a result, cool supersonic jet flows which pass through barrel shock wave increase. **Figure 7** shows that the main stream flows outside of cool supersonic jet flow, and then it passes through recompression shock wave. Observing temperature contours, temperature around a reattachment point increases. Temperature around a reattachment point decreases as the total pressure ratio increases. This is caused by cool jet flow near the wall. As the total pressure ratio increases, the mass flow ratio of the cool jet increases. Hence increase of jet temperature caused by hot free stream becomes small. This is the reason why the temperature of recirculation region decreases.

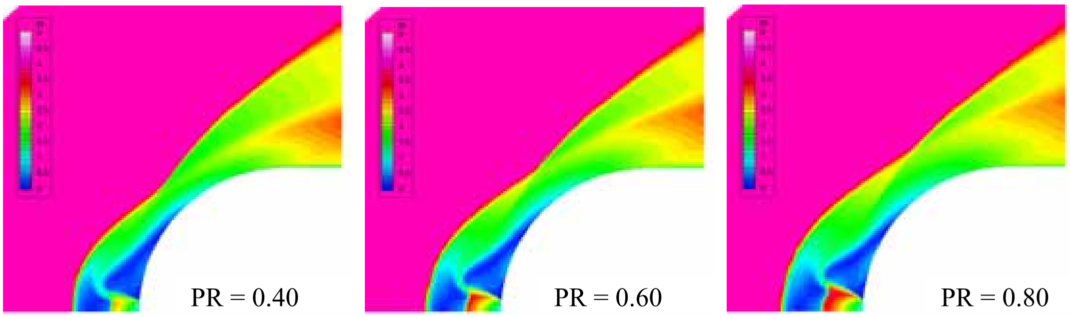


Fig.5 Mach number contour at various PR ($M_j = 1.0, dj = 4mm$).

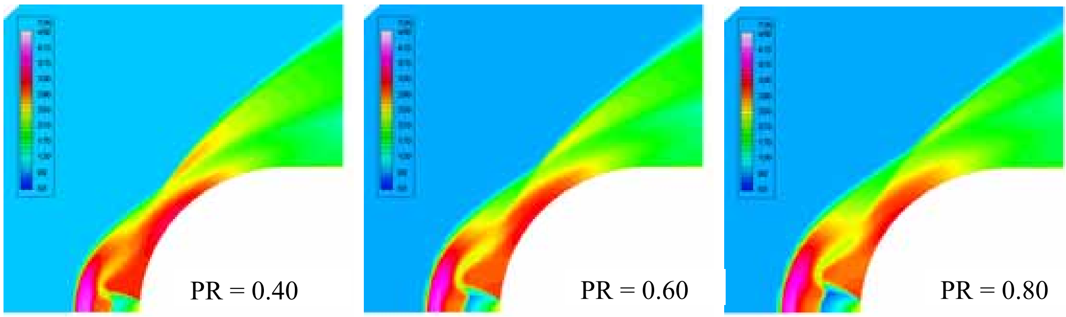


Fig.6 Temperature contour at various PR ($M_j = 1.0, d_j = 4mm$).

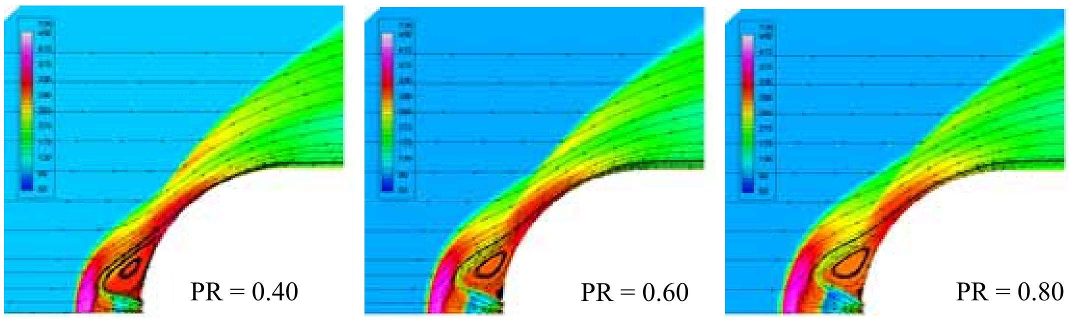


Fig.7 Streamline at various PR ($M_j = 1.0, d_j = 4mm$).

3.3 Effects of Mach number of jet

Exit Mach number of jet (Mj) are changed to study the effect of ext Mach number. Mass flow rates are same in all cases. **Figures 8, 9 and 10** show heat flux distributions of each exit Mach number. **Figures 11, 12, 13 and 14** show Mach number contours, temperature contours, density contours and streamline respectively.

In the case of d_j=2mm, heat flux decreases as the exit Mach number increases, where d_j is a diameter of nozzle exit. In the case of d_j = 4mm and 6mm, heat flux increases as the exit Mach number increases.

When exit Mach number increases, Mach disk and detached shock wave moves upstream. Three phenomena are observed when exit Mach number increases. The first is that a density of jet, which passes through Mach disk, decreases as exit Mach number increases. It causes the decrease of heat capacity of the jet near stagnation point. As a result, increase of temperature at stagnation region becomes large. Hot jet causes increase of temperature in the recirculation region. It leads to an increase of surface heat flux.

The second is that the density of the recirculation region decreases as the exit Mach number increases. It leads to a decrease of heat capacity of the recirculation region. As a result, the increase of temperature in the recirculation region becomes large and surface heat flux increases. The third is that temperature of supersonic jet layer decreases as the exit Mach number increases. It leads to the decrease of temperature of the recirculation region. As a result, heat flux decreases. It seems that surface heat flux depends on these three phenomena.

In the case of small exit diameter (d_j=2mm), main change is a decrease of temperature of cool jet layer which passes through barrel shock wave. On the contrary, the density of the jet which passes through Mach disk hardly changes. As a result, heat flux decreases when exit Mach number increases.

In the case of large exit diameter (d_j=4mm and 6mm), density of the jet which passes through Mach disk also changes when exit Mach number increases. Hence heat capacity at stagnation region becomes small. For this reason effect of small heat capacity is larger than that of cool supersonic jet layer. As a result, it seems that heat flux increases.

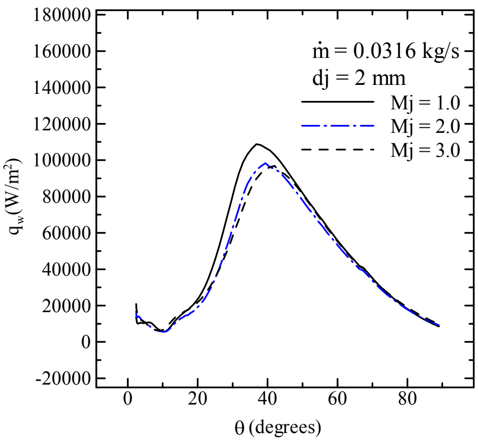


Fig.8 Heat flux distributions for various M_j

($\dot{m} = 0.0316 kg/s, d_j = 2mm$).

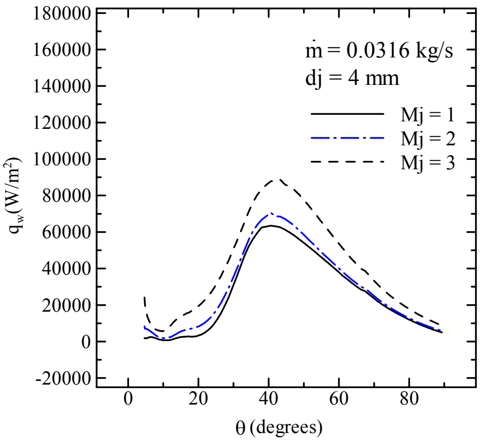


Fig.9 Heat flux distributions for various M_j

($\dot{m} = 0.0316 kg/s, d_j = 4mm$).

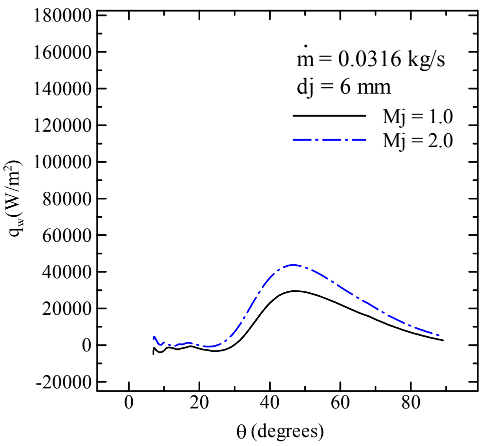


Fig.10 Heat flux distributions for various M_j

($\dot{m} = 0.0316 kg/s, d_j = 6mm$).

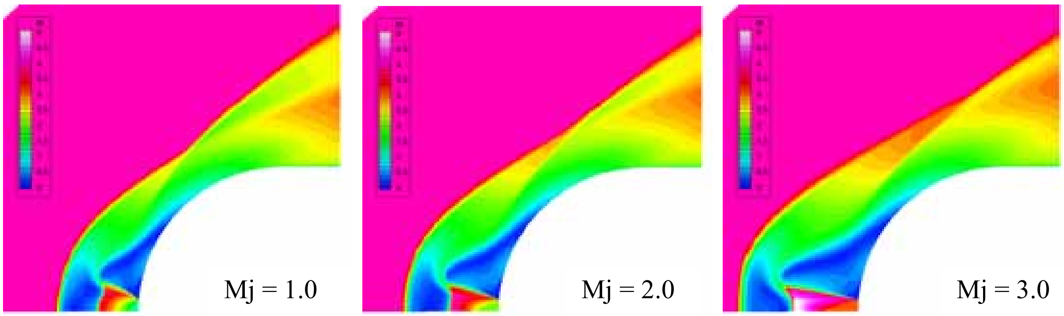


Fig.11 Mach number contour (Mj changes, $\dot{m} = 0.0316kg/s, d_j = 4mm$).

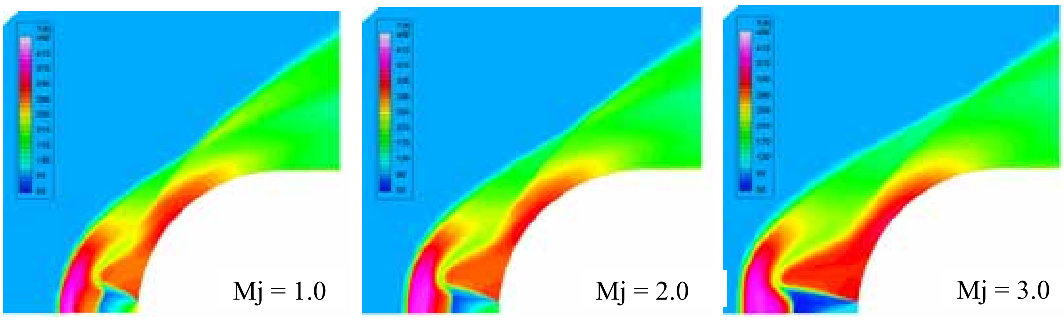


Fig.12 Temperature contour (Mj changes, $\dot{m} = 0.0316kg/s, d_j = 4mm$).

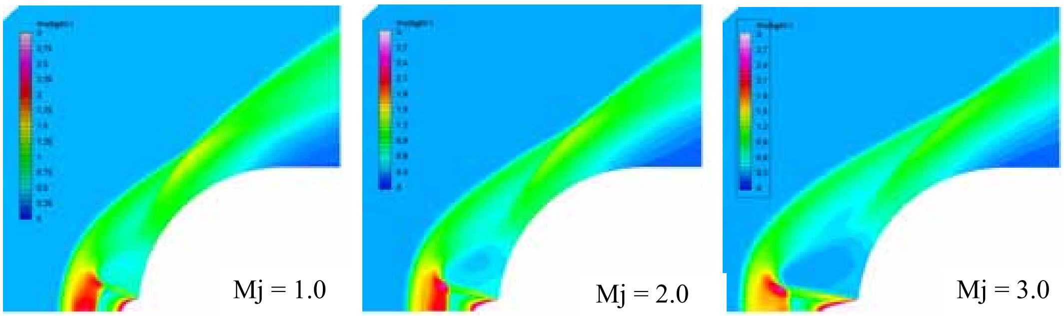


Fig.13 Density contour (Mj changes, $\dot{m} = 0.0316kg/s, d_j = 4mm$).

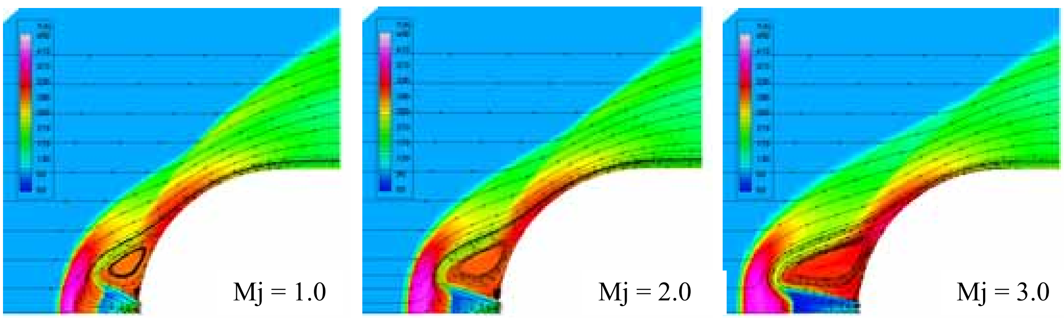


Fig.14 Streamline (Mj changes, $\dot{m} = 0.0316kg/s, d_j = 4mm$).

3.4 Effects of diameter of nozzle exit

In order to study the effect of d_j, the flow fields at various d_j are calculated. Mass flow rates are same in all cases. **Figures 15 and 16** shows heat flux distributions of each exit diameter. **Figures 17 and 18** show Mach number contours, temperature contours, density contours and streamline respectively.

As the exit diameter of jet becomes larger, heat flux decreases. Comparing to effect of exit Mach number, the effect of exit diameter is remarkable.

When exit diameter changes, a detached shock wave and Mach disk does not move in the case of M_j=1, however it moves in the case of M_j=2. When diameter becomes large, cool region becomes large at stagnation region. In addition, because a condition of under expansion jet changes, the shape of the barrel shock wave changes and cool supersonic jet layer which passes through oblique shock waves becomes large. For this reason, temperature of the recirculation region decreases, and then heat flux decreases.

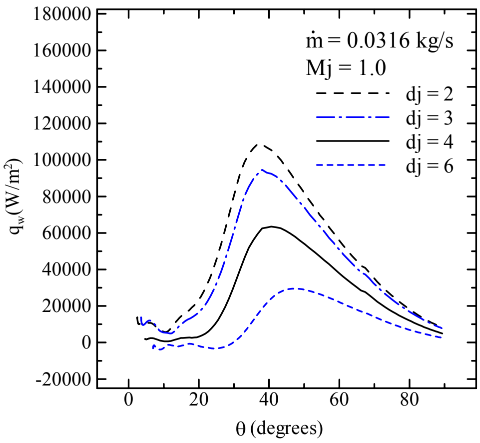


Fig.15 Heat flux distributions for various d_j ($\dot{m} = 0.0316 kg/s, M_j = 1.0$).

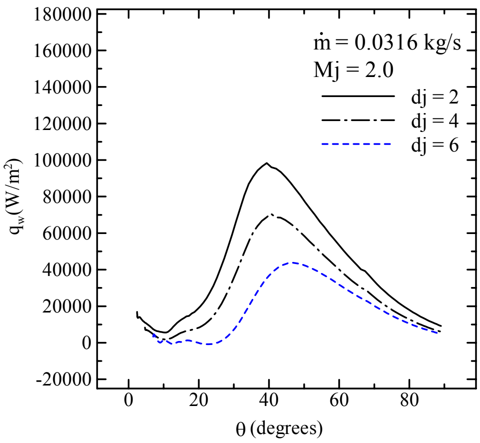


Fig.16 Heat flux distributions ($\dot{m} = 0.0316 kg/s, M_j = 2.0$).

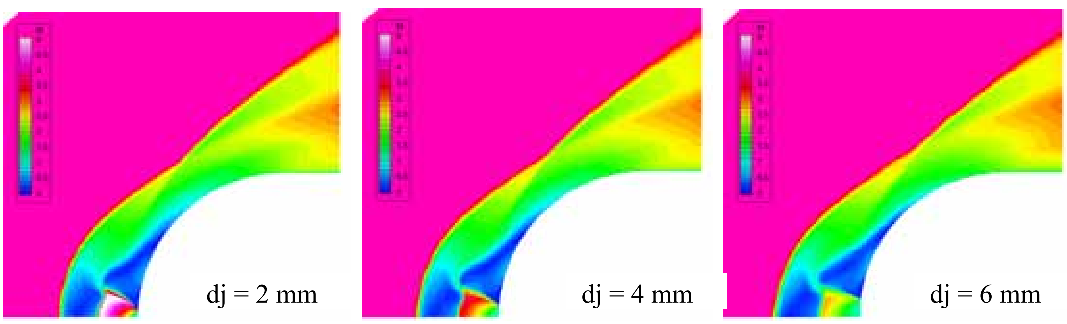


Fig.17 Mach number contour for various d_j ($\dot{m} = 0.0316 kg/s, M_j = 1.0$).

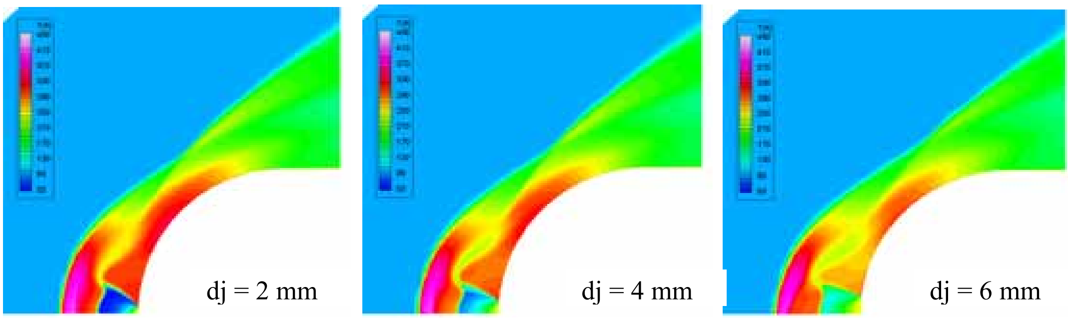


Fig.18 Temperature contour for various d_j ($\dot{m} = 0.0316kg/s, M_j = 1.0$)

3.5 Effects of total temperature of jet

The effects of total temperature of opposing jet are investigated. Total temperature of jet is changed with same total pressure ratio (PR=0.40, 0.80). Therefore mass flow ratio changes when total temperature changes. The diameter of nozzle exit is 4mm, exit Mach number is unity. In the case of PR = 0.40, mass flow ratio is between 0.0140 ~ 0.0198 kg/s, PR = 0.80, it is between 0.0274 ~ 0.0388 kg/s. **Figures 19 and 20** shows heat flux distributions of PR = 0.40 and 0.80. **Figures 21 and 22** show Mach number contours, and temperature contours of each PR.

Although mass flow ratio changes, the shapes of the detached shock wave and Mach disk does not change at each total pressure ratio. The flow field hardly changes except temperature field.

Temperature of flow field dramatically changes. In the case of Tj=200K, cool jet layer is clearly observed in **Figure 26**. Temperature of the jet layer affects temperature of the recirculation region and downstream region near the wall.

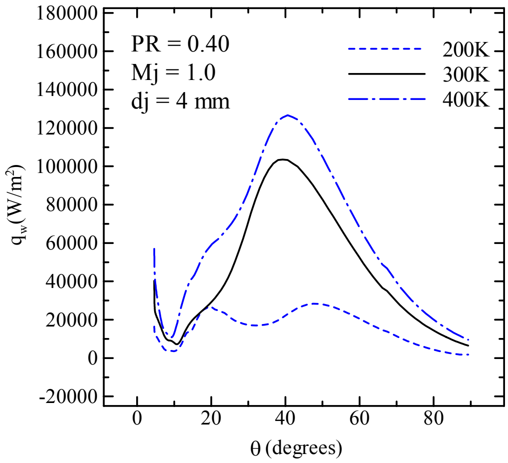


Fig.19 Heat flux distributions (*PR*=0.40)

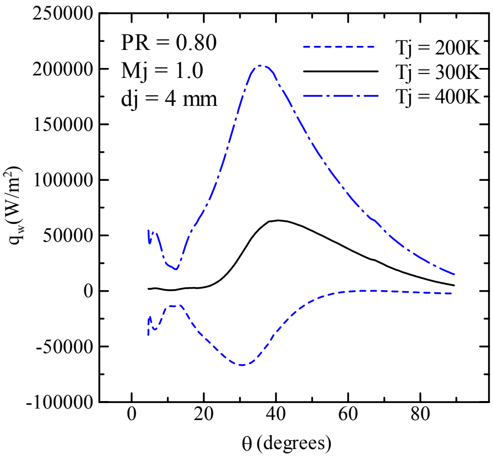


Fig.20 Heat flux distributions (*PR*=0.80)

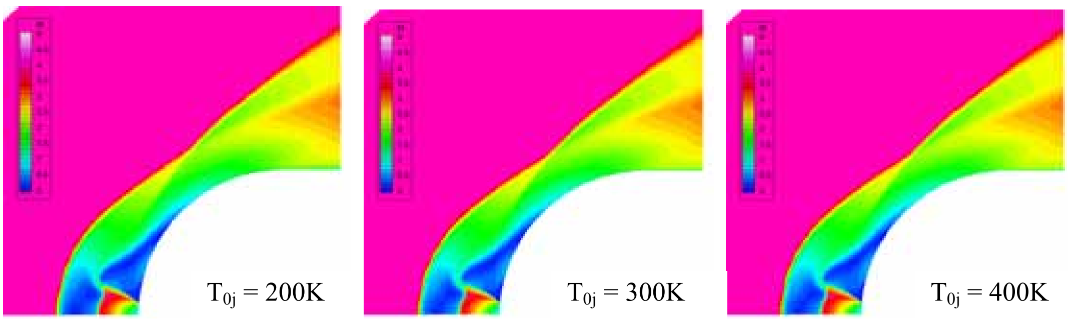


Fig.21 Mach number contour for various T_j ($PR = 0.80, M_j = 1.0, d_j = 4mm$)

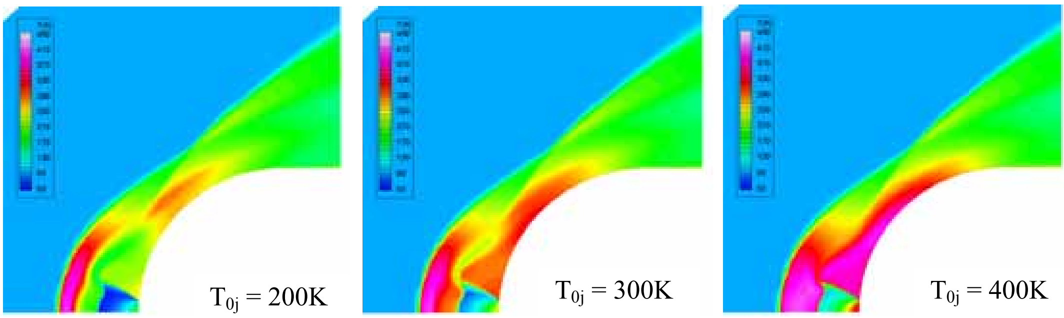


Fig.22 Temperature contour at various T_j ($PR = 0.80, M_j = 1.0, d = 4mm$)

3.6 Summary

According to the investigation of various conditions of opposing jet, we found following phenomena.

- The larger cool jet region becomes at stagnation point, the lower temperature of the recirculation region and surface heat flux become.
- Large cool supersonic jet layer leads to low temperature in the recirculation region. As a result, surface heat flux becomes small.
- Temperature of jet at stagnation region becomes large when exit Mach number is large. The temperature rise is caused by decrease of heat capacity of jet layer. In addition, as the diameter of nozzle exit becomes larger, density slightly increases, and then temperature rise becomes small.
- The cool supersonic jet layer becomes large when exit Mach number is large. This jet layer also becomes large when diameter of nozzle exit is large.
- Total temperature of jet greatly affects heat flux.

From these results, following jet conditions are effective for thermal protection by opposing jet.

- Using cool jet
- jet condition which achieves low temperature at the stagnation point
- jet condition which accomplishes a large amount of cool supersonic jet passing through the barrel shock wave

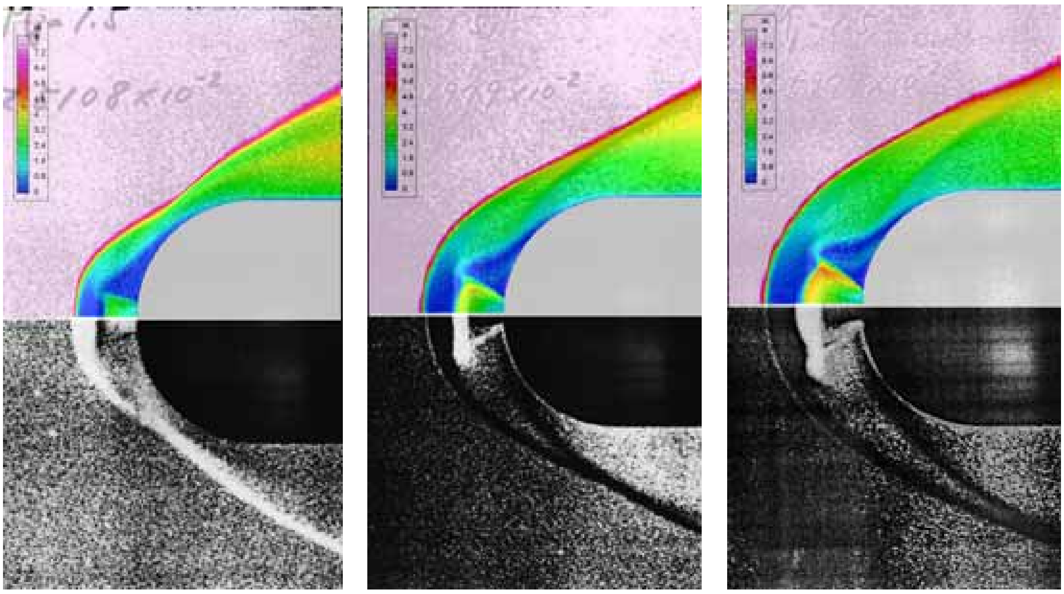
In order to achieve large low temperature region at the stagnation point, exit Mach number should be small. It leads to high density and high heat capacity of the jet passing through Mach disk. Large exit diameter also leads to high density of the jet.

In order to achieve a large amount of cool supersonic jet passing through the barrel shock wave, exit diameter should be large. As a result, jet becomes weak under expansion condition. However, when diameter is too large, flow field is unstable. In our previous study, heat flux became large at unstable flow field. Therefore, it is important to form stable flow field. Exit diameter should be set maximum diameter which can form stable flow field.

4. Aerodynamic Heating Reduction in Hypersonic Flow

4.1 Comparisons between experiments and numerical results

Figure 23 shows comparison between Mach number contours and Schlieren photographs. Numerical results show good agreement with experiments. **Figure 24** shows pressure distribution of each total pressure ratio. It also shows good agreement.



PR = 0.0251 PR = 0.0549 PR = 0.0859

Fig.23 Mach number contours and Schlieren photographs

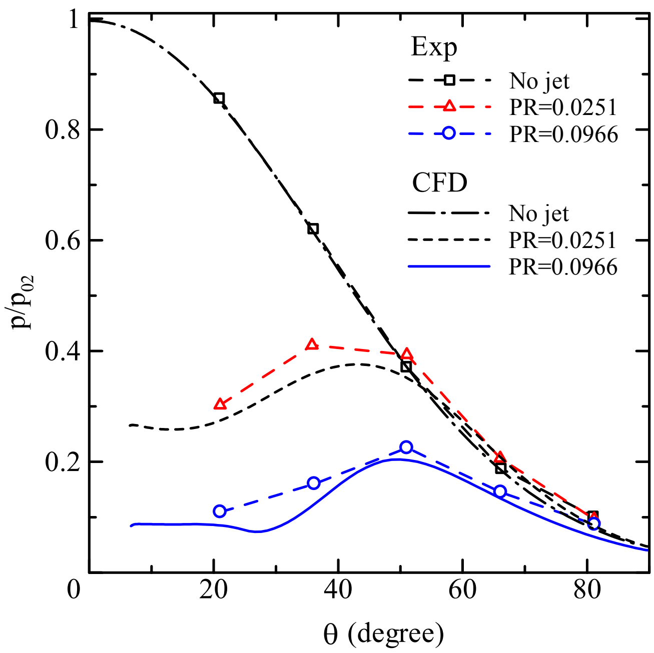


Fig.24 Pressure distributions

4.2 Results and discussions

Figure 25 shows heat flux distributions of numerical analysis and stagnation heat flux calculated by Fay & Riddell theory[14). In the case of blowing opposing jet, heat flux significantly decreases. As the total pressure ratio increases, heat flux significantly decreases.

Flow fields of each total pressure ratio are shown in **Figure 26 and 27**. Density of jet flow passing through the Mach disk is higher than that of free stream passing through the detached shock wave at stagnation point. A difference of density is clearly observed at contact surface. At high Mach number, total pressure loss is high behind the detached shock wave. Because the total pressure loss behind a normal shock wave at Mach 8 is 0.0085, pressure at contact surface is very low. Therefore pressure of jet flow is also low. According to this, the density of jet flow is higher than that of free stream at stagnation point. Pressure of jet flow is about three times higher than that of free stream at PR = 0.0859.

Discontinuity of temperature corresponds with the contact surface. This discontinuity of temperature is caused by high density of jet flow. Because of high density, heat capacity of jet flow is high. Therefore temperature rise of jet flow is small.

Compare to the case of Mach 4, the discontinuity of temperature is clear at contact surface in the case of Mach 8. As a result, temperature of the recirculation region which is formed by jet flow remains low. This is the reason why heat flux at body surface decreases.

The results are summarized as follows.

- It is clear that opposing jet is effective for a reduction of aerodynamic heating in hypersonic flow.
- If temperature does not increase at stagnation region, temperature of recirculation region remains low. As a result, heat flux at body surface decreases.
- As the total pressure ratio increases, heat flux significantly decreases.

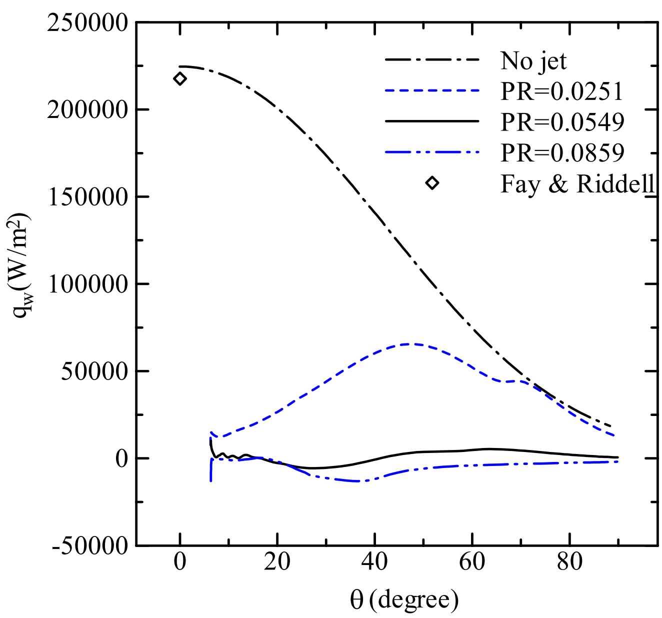
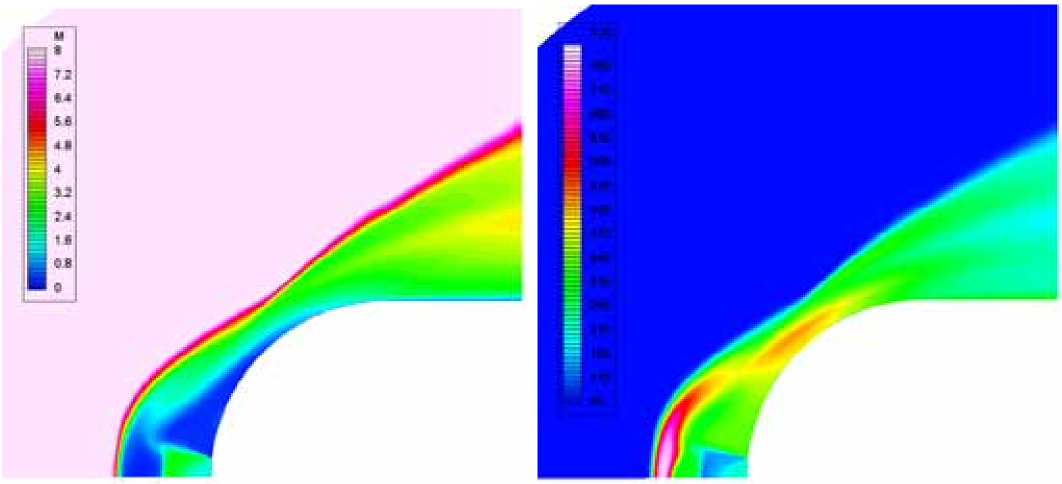
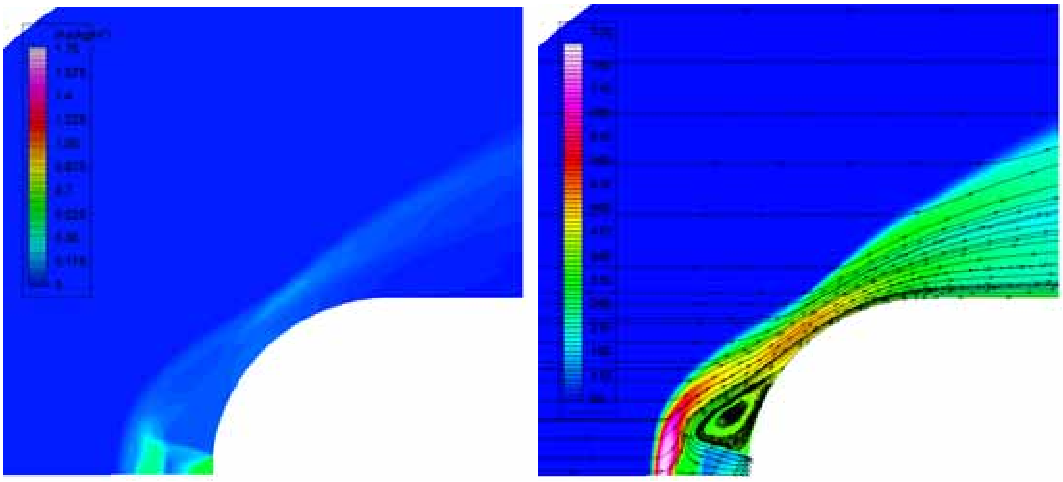


Fig.25 Heat flux distributions

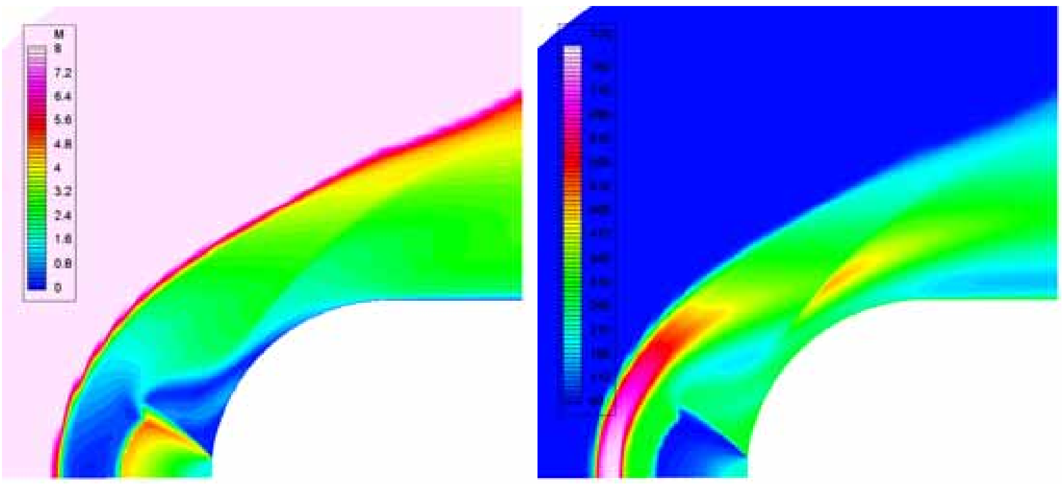


(a) Mach number (b) Temperature

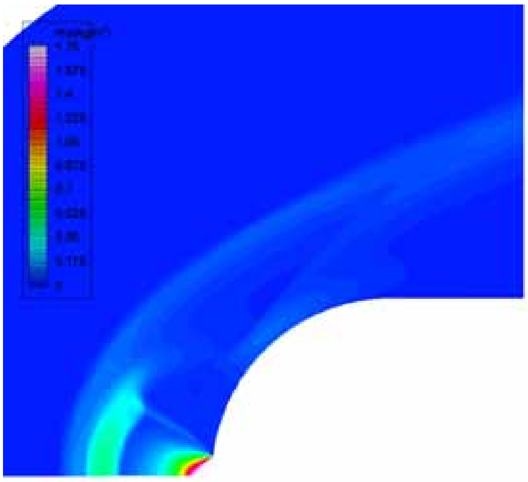


(c) Density (d) Streamline

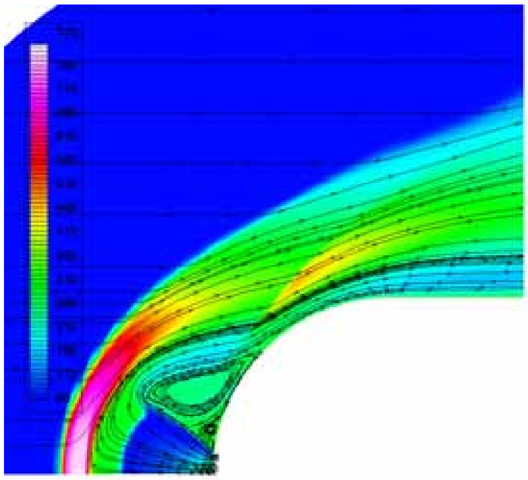
Fig.26 Flow field (PR = 0.025)



(a) Mach number (b) Temperature



(c) Density



(d) Streamline

Fig.27 Flow field (PR = 0.086)

5. Conclusions

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

1. Detailed flow fields of opposing jet in supersonic and hypersonic flow are calculated by solving axisymmetric Navier – Stokes equation with k - ω turbulence model. The numerical results show good agreement with experiments.
2. It is clear that opposing jet is quite effective for a reduction of aerodynamic heating in supersonic and hypersonic flow.
3. According to the investigation of various conditions of opposing jet in supersonic flow, some effective jet conditions are found. Exit Mach number should be small. Exit diameter should be set maximum diameter which can form stable flow field.
4. In hypersonic flow, density of jet flow plays an important role for reduction of aerodynamic heating. Since temperature does not increase at stagnation region caused by high heat capacity of jet flow, temperature of recirculation region remains low. As a result, heat flux at body surface decreases.

References

1) N. Motoyama, K. Mihara, R. Miyajima, T. Watanuki and H. Kubota; Thermal Protection and Drag Reduction with use of Spike in Hypersonic Flow, AIAA Paper 2001-1828 (2001).
2) M. Laptoff; 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 (1951).
3) E.S. Love; The Effects of a Small Jet of Air Exhausting from the Nose of a Body of Revolution in Supersonic Flow, NACA RM L52I19a (1952).
4) E.S. Love, C.E. Grigsby, L.P. Lee, and M.J. Woodling; Experimental and Theoretical Studies of Axisymmetric Free Jets, NACA TR No.6 (1959).
5) N. Charczenko and K.W. Hennessy; Investigation of a Retrorocket Exhausting from the

Nose of a Blunt Body into a Supersonic Free Stream, NASA TN D-751 (1961).

6) C.H.E Warren; An Experimental Investigation of the Effect of Ejecting a Coolant Gas at the Nose of a Bluff Body, *J. Fluid Mech.*, Vol.8, pp.400-417 (1960).
7) P.J. Finley; The Flow of a Jet from a Body Opposing a Supersonic Free Stream, *J. Fluid Mech.*,Vol.26, pp.337-368 (1966).
8) K. Hayashi and S. Aso; Effect of Pressure Ratio on Aerodynamic Heating Reduction due to Opposing Jet, *AIAA paper* 2003-4041 (2003)
9) K. Hayashi and S. Aso; A Study on Reduction of Aerodynamic Heating by Opposing jet in Supersonic Flow, Journal of the Japan Society for Aeronautical and Space Sciences, Vol.52, pp.38-44 (2004)
10) D.C. Wilcox, Reassessment of the Scale-Determining Equation for Advanced Turbulence Models, AIAA Journal, Vol.26, No.11, pp.1299-1310 (1988).
11) Y. Wada and M.S. Liou; A Flux Splitting Scheme with High-Resolution and Robustness for Discontinuities, *AIAA paper* 94-0083 (1994).
12) S. Obayashi, K. Matsushima, K. Fujii and K. Kuwahara; Improvements in Efficiency and Reliability for Navier-Stokes Computations Using the LU-ADI Factorization Algorithm, *AIAA paper* 86-0338 (1986)
13) T.J. Craft, B.E. Launder and K. Suga; Development and application of a cubic eddy-viscosity model of turbulence, International Journal of Heat and Fluid Flow, Vol.17, No.2, pp.108-115 (1996).
14) J.A. Fay and F.R. Riddell; Theory of Stagnation Point Heat Transfer in Dissociated Air, Journal of the Aeronautical Sciences, Vol.25, pp.73-85 (1958).