Effect of Counter Flowing Jet on Heat Transfer and Drag in Hypersonic Re-entry Vehicle

P Visakh¹, J. Akhil¹ and Nagaraja S. R.¹,²

¹ Department of Mechanical Engineering, Amrita School of Engineering, Bengaluru, Amrita Vishwa Vidyapeetham, India
² e-mail: sr_nagaraja@blr.amrita.edu

Abstract. All the re-entry vehicles usually travel at hypersonic velocities, there are some design constraints pertaining to them such as aerodynamic drag and heat, which are mutually conflicting. A sharp edged slender body offers longer range and lesser drag but it causes higher aerodynamic heat. Generally, blunt shaped nose is used for the re-entry vehicles as it can disperse the accumulated heat to the surroundings better than sharp edged bodies. The current work focuses on reduction of aerodynamic heating by the introduction of a single jet injection on the stagnation point. Different jet angle configurations are used for the analysis by maintaining the angle of attack of the body constant. It is found that for body at 6-degree angle of attack the optimum jet angle is 9 degrees and the corresponding reduction in peak heat flux is 22.7% compared to the heat flux when the jet is at zero degrees and 33% compared to the case without jet.

1. Introduction

The re-entry vehicles are mainly used as a part of spacecrafts to return through Earth after the completion of the mission. The major challenges observed during design of re-entry vehicles are the aerodynamic heat and drag. These factors are minimised to some extent by a blunt shaped nose design. The drag and heat are contradictory to each other. While designing if either of the constraints are modified beyond a certain extent it might lead to adverse effects on the other. When the re-entry vehicle enters the Earth’s atmosphere, it releases excess heat that can affect the material of the vehicle. This can be avoided by a thermal protection layer. A blunt shaped body is enough to reduce the heat, but at hypersonic velocities the wave drag is very large. As the Mach number increases wave drag also increases, which is the most dominating among all the other forms of drag. The maximum heat flux occurs at the stagnation point. There are several methods to mitigate both the aerodynamic drag and heat such as introduction of spikes, jets or the combination. Spikes are needle shaped bodies which replace the strong bow shock with oblique shock waves. The introduction of spikes in blunt bodies reduces the aerodynamic drag and heat as it replaces the strong bow shock but it leads to high heat flux at the stagnation point of the spike. The other method is introduction of jet at the stagnation point. Zhen-guo Wang et al. [1] have given survey of different experimental methods for reducing drag and heat flux.

This work focuses on the effects of heat transfer and drag when a jet is placed at the stagnation point. The jet is placed in such a way that it counteracts the free stream movement and pushes the bow shock.
away from the body. As a result, the counteracting jet can sustain the wave drag and heat at hypersonic velocities. This jet reduces the heat flux at the stagnation point but this causes heat flux at the reattachment point. As the momentum flux increases, the heat flux at the reattachment point also increases. In this work the analysis is based on a single orifice jet placed at the stagnation point.

2. Literature Survey

A blunt body with forward facing lateral jet injection and spike was experimented on a wind tunnel with steady and unsteady conditions by Jiang Zhanget al. [2]. When the critical pressure is lesser than the supersonic jet pressure, the flow is treated as steady otherwise unsteady. The model proposed was a blunt body with a spike including a jet at the stagnation point. The experiments were conducted for Mach numbers 2.5 and 4. From the experimental study it was observed that as the spike length increases, the coefficient of drag decreases and when the jet pressure ratio increases, the reattachment shock is weakened, thereby reducing the hotspots on the body. An equal polygonal jet approach was proposed by Shi-bin Li et.al.[3] instead of single orifice jet. In this work, the effect of equal polygon on heat transfer and drag has been investigated. The polygon used in the experimental model has 3 to 9 legs and the observations show that the polygon with 7 legs gives the best results. The drag reduction of 26.4% is achieved and peak heat flux is reduced by 60.6%. It has been identified that the optimum performance is obtained with odd number of legs than even number. Wei Huang [4] has done a survey on counter flowing jet in supersonic flow. This work compares three combinations of counter flowing jet i.e. jet alone, jet with forward facing cavity and aero spike with jet. With a higher pressure ratio, 45% drag reduction can be obtained with jet alone. With the combination of jet with cavity, the cavity acts as a nozzle and speeds up the incoming jet, thereby an oscillating effect is obtained which enhances the cooling process. An increased pressure ratio leads to the decrement in heat and drag effects. The cavity diameter plays a major role in heat reduction than cavity length. The combination of aero spike with jet pushes the bow shock away from the body thereby avoiding reattachment on the body. It also avoids the peak heat flux happening at the stagnation point of the aero spike.

R. Sriramet.al [5] proposed an experimental investigation on film cooling using micro jets in hypersonic flow. In this experimental work, a blunt model with single jet and micro jet opposite to the freestream has been used. In the case of micro jets, they take the same area of the single jet injection and replace it with array of micro jets, for the jet injection nitrogen and helium has been used. The experiments were conducted at a freestream Mach number of 5.9 and with jet pressure ratios of 1.2 and 1.45. The result obtained from the experiment suggests that micro jets than single jet of the same area offer better cooling effect and nitrogen gas gives better cooling performance than helium gas. Jagadeesh Gopalan et.al [6], have investigated drag reduction in a large angle blunt cone using different spikes. They found that a drag reduction of 40-55% with the hemispherical and flat aero discs. The flow characteristics around a blunt body with and without spike in hypersonic flow has been investigated using DSMC method by J. S. Jiss et.al [7]. The model with various L/D ratios were investigated with a Mach number of 6 and gives better results with the spike. When L/D is 2, the peak pressure and temperature are lower than when L/D is 1.5. The work done by M. Tahani et.al [8] provides the idea of maintaining pressure far field around the body for better convergence. Gopala Krishnan et.al [9] have studied the heat flux distribution over a blunt body with different spikes with and without jet. They have found that using twin conical spike reduces the heat flux by 42%. Srekanth et.al [10] have studied the effect of secondary spike on reduction of drag and heat flux.

3. Model and Boundary conditions
The 3D model simulated is a blunt cone with 58-degree apex angle and nose radius 35mm. An additional 53mm cylindrical structure is created to visualize the shockwave fully. A jet injection is provided at the stagnation point of the model with a diameter of 2mm, air is taken as injection gas. The entire simulation is done with a freestream Mach number of 5.9, the static temperature 230K and a static pressure of 1013 N/m². The jet pressure ratio used is 1.2, which is the ratio of total pressure of the jet to the freestream pitot pressure. These free stream conditions are chosen so that the present simulation results can be compared with experimental results of R. Sriram et al. [5]. A density based study was used with Spalart Allmaras one equation model for the simulation. The body wall temperature is kept 300K to calculate the heat transfer from the body. For the simulation, a rectangular enclosure is created around the body and pressure far field is fixed at the inlet, outlet and sides of the enclosure. An unstructured mesh with around 850000 grid points is used, an additional inflation near the boundaries of the body is created for better accuracy. Ideal gas is taken in the freestream, which follows Sutherland’s viscosity equation.

4. Results and Discussions

The work proposed by R. Sriram et al. [5] includes the graph of heat flux along the body of the model with and without jet injection. The model developed is validated by comparing the simulated results with the experimental results of R. Sriram et al. [5]. The simulations are done for the model with and without jet. The heat flux values are plotted along the surface of the model as shown in figure 2 and 3. In these figures the negative values in the x axis represents the lower part of the body and positive values represents the upper part of the body in the x-y plane and origin represents the stagnation point of the body. It can be observed that the present simulated results and experimental results closely agree with each other. Figure 4 and 5 show the pressure contours with and without the jet. In these simulations, the blunt body is at zero angle of attack and the jet is introduced at the stagnation point with a jet pressure ratio of 1.2. The jet was introduced at the stagnation point. It can be observed that the introduction of jet pushes the shock away from the body there by reducing the heat flux. The introduction of jet shows that there is a decrease in heat transfer and aerodynamic drag. The drag values obtained are Cd (without jet) = 0.815 Cd (with jet) =0.798.
Figure 2: Comparison of simulated values of heat flux with experimental [5] values without jet

Figure 3: Comparison of simulated values of heat flux with experimental [5] values with jet

Figure 4: Pressure contours of model without jet.

Figure 5: Pressure contours of model with jet
After validating the model, simulations are carried out with and without jet at zero angle of attack and 6-degree angle of attack of the blunt body. As the angle of attack of body changes the stagnation point shifts. Figure 6 shows the heat flux values on the surface of the body when it is at 6-degree angle of attack without the jet. It can be seen that the stagnation point has moved to the lower part of the body. Introduction of jet at the stagnation point corresponding to zero angle of attack i.e. parallel to the axis of the body reduces the maximum heat flux by 13% (figure 7), but it may not be the maximum reduction obtained as the jet is no longer at the stagnation point corresponding to the 6-degree angle of attack. It is not practically feasible to introduce the jet at different stagnation points corresponding to different angles of attack. Instead, it is practically feasible to change the angle of the jet as angle of attack of the body changes.

Figure 6: Heat flux - body at 6 degrees angle of attack without jet

Figure 7: Heat flux - body at 6-degree angle of attack with jet at zero degree

In order to find the optimum angle of jet at which the heat flux is minimum, the jet angle is varied from 0 to 12 degrees (in counter clockwise direction) in steps of 3 degrees, keeping its position at the original stagnation point i.e. stagnation point corresponding to zero angle of attack of the body. Figure 8 shows the heat flux values on the surface of the body when the jet angle is 9 degrees. It can be observed that
the maximum heat flux value has reduced by 22.7% compared to the maximum heat flux value when the jet is at zero angle. Figure 9 shows the peak heat flux values for varying jet angle. Figure 10 shows the percentage reduction in the peak heat transfer varying angles of jet. The percentage is calculated with maximum heat flux corresponding to body at 6-degree angle of attach and jet at zero angle. It can be observed from these figures that the optimum angle of jet at which the peak heat flux is minimum is 9 degrees. In all these simulations the free stream conditions and the angle of attack of the body is kept constant.

![Figure 8](image1)

**Figure 8:** Heat flux body at 6-degree angle of attack and jet at 9 degrees.

![Figure 9 and 10](image2)

**Figure 9:** Peak Heat flux variation for different jet angles  
**Figure 10:** Percentage Reduction in Peak heat flux

5. Conclusions

The introduction of a single jet at the stagnation point of the body reduces the heat flux when compared to the model without jet. There is 13.3% reduction in peak heat flux when the jet is introduced in the model at 6-degree angle of attack. In order to find the optimum angle of jet at which the heat flux is
minimum, the jet angle is varied from 0 to 12 degrees in steps of 3 degrees. The optimum jet angle was 9 degrees and corresponding reduction in the peak heat flux is 22.7% compared to the peak heat flux when the jet angle is zero degrees and a reduction of 33% compared to the case without jet. For all these simulations drag coefficient varied from 0.79 and 0.82, showing that there is no drastic change.

6. References

[1] Zhen-guo Wang, Xi-wan Sun, Wei Huang, Shi-bin Li and Li Yan 2016 Experimental investigation on drag and heat flux reduction in supersonic/hypersonic flows: A survey Acta Astronautica 129 95-110
[2] Jiang Zhang, Junfei Wu, Wenbin Ni, Handong Ma and Yongming Qin 2017 Experimental Investigation on Flow Characteristic of Combination of Forward-facing Jet and Spike 21st AIAA International Space Planes and Hypersonics Technologies Conference (AIAA 2017-2402)
[3] Shi-bin Li, Zhen-guo Wang, Wei Huang, and Jun Liu 2016 Drag and Heat Reduction Performance for an Equal Polygon Opposing J. Aeroesp. Eng 30 1-10
[4] Wei Huang 2015 A survey of drag and heat reduction in supersonic flows by a counterflowing jet and its combinations Huang J Zhejiang Univ-Sci A (ApplPhys&Eng)16(7) 551-561
[5] R. Sriram, G. Jagadeesh 2015 Film cooling at hypersonic Mach numbers using forward facing array of micro-jets International Journal of Heat and Mass Transfer 52 3654-3664
[6] JagadeeshGopalan, Viren Menezes, K.P. Jagannatha Reddy, Tokitada Hashimoto, Mingyu Sun, Tsutomu Saito and Kazuyoshi Takayama 2004 Flow Fields of a Large-angle, Spiked Blunt Cone at Hypersonic Mach Numbers Trans. Japan Soc. Aero. Space Sci. 48 110–116
[7] J. S. Jiss, E. J. Sandeep, R. V. Reji, A. Suryan, H. D. Kim 2016 Study of Hypersonic Flow Past Spiked Blunt Body using Direct Simulation Monte Carlo Method Topical Problems of Fluid Mechanics Prague February 10-12
[8] M. Tahani, M. S. Karimi, A. MahmoudiMotlagh, S. Mirmahdian 2013 Numerical investigation of drag and heat reduction in hypersonic spiked blunt bodies Heat Mass Transfer 49 1369–1384.
[9] Gopala Krishnan G, Akhil J and Nagaraja S R 2015 Heat Transfer Analysis on Hypersonic Re-entry vehicles with spikes Proceedings of the 23rd National Heat and Mass Transfer Conference and 1st International ISHMT-ASTFE Heat and Mass Transfer Conference.
[10] Sreekanth N, Akhil J and Nagaraja S R 2016 Design and Analysis of Secondary spike on blunt head Indian Journal of Science and Technology 9(43).