Study on Aerodynamic Drag Reduction of A Blunt Body Using Hot-Gas Injection in Hypersonic Flow

Srinidhi Sudhindra[1], R Aravind Ganesh[2], Shriharsha R[3], S. Saravanan[4], Prashant Manvi[5], Santosh Hosur[6]

Nitte Meenakshi Institute of Technology, Department of Aeronautical Engineering, Yelahanka, Bangalore 560064, India

Indian Institute of Science, Department of Aerospace Engineering, Laboratory for Hypersonic and Shock Wave Research (LSHR), Bangalore 560012, India

***Abstract:** A logical examination on the diminishment of streamlined wave drag by hot gas infusion in hypersonic stream has been investigated. In the present investigation, dimensional axisymmetric Euler conditions are utilized as overseeing condition. Investigation has been completed for an axisymmetric limit module with and without infusion cooling for Mach number of 5.7. The scientific outcomes has been effectively approved utilizing standard exploratory information for hot- gas infusion. The outcome appears around 15%– 30% diminishment in drag coefficient for various gas infusion temperature. It turns out to be evident that an execution of the decrease of streamlined wave drag by restricting plane is incredibly influenced by jet condition.*

Keywords: Hypersonic flow, Gas temperature, Aerodynamic wave drag, Boundary layer

1. Introduction

The hypersonic research activities have re-emerged around the globe in recent times in the backdrop of renewed focus on exciting concepts such as global range hypersonic re-entry vehicles, orbital transfer vehicles, reusable launch vehicles, hypersonic wave riders and space recovery experimental modules. In general, a hypersonic vehicle needs to have a blunt nose in order to withstand high convective surface heating loads that are expected to be very severe both during the ascent and re- entry phase of the flight path. But, this results in a significant performance penalty due to increased wave drag. Hence precise information on both aerodynamic forces and surface heat transfer rates are essential in deciding on both the requisite propulsion system and thermal protection system (TPS). So suitable measurement techniques must be incorporated in the ground-based testing facilities for such type of body configurations in order to address both convective heating rates and aerodynamic drag simultaneously. In addition, carrying out separate tests for force and heat transfer measurements is quite expensive. Also, in most of the cases, the information on the surface heat transfer rates and the basic aerodynamic drag coefficients obtained from careful experiments in the ground-based test facilities are complemented by CFD studies. So the cumulative effects of experimentally measured data essentially lead to erroneous conclusions when used for validating the CFD codes. Hence, a more reliable, innovative and carefully designed experimental techniques/methodologies are needed for the generation of data used validation of CFD code, especially at hypersonic Mach numbers. In this backdrop, an experimental program has been launched to develop a novel method of measuring the aerodynamic forces. The technique is used to measure both aerodynamic data for a blunt cone using FLUENT analysis software. The Large Wave drag on an blunt body as a result of the increase in entropy across the shock wave surrounding the body in high-speed flight is a serious consideration in its aerodynamic design. These large drag components affect on an aircraft by loss of performance, increased fuel consumption, reduction in speed range, and reduced payload capacity. The wave drag reduction in aerodynamic applications by either a structural spike or a jet spike on a blunt nosed body and is known hot gas injection. It is defined as the injection of a gas with higher temperature from the nose of a vehicle in the upstream direction. An analytical study on the drag reduction on a blunt body has been carried out by using the techniques of hot gas injection in hypersonic flow. Analysis has been carried out for an axisymmetric blunt module with and without hot gas injection. The study is carried out at a free stream of Mach no 5.7. The counter flow is injected at different Mach numbers and at different temperatures. The configuration of design giving best results has been selected for studying the drag reduction characteristics. This study is done in comparison with the base model in which these hot gas injections are not introduced. The results show a decrement of the drag by around 10% and the bow shock wave is pushed away by around 40% from the blunt body. The analytical results are verified using the reference paper with certain percentage of error. Further the areas of application of such blunt bodies are identifies as hyper sonic aircraft, UAV, re-entry vehicle etc.

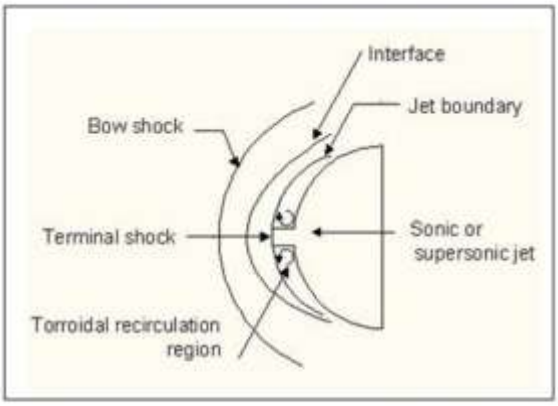

Figure 1: Blunt Body

2. Governing Equations

In the present study, the investigation of solution procedures for the inviscid flow region is studied. The governing equation is known as the Euler equation. In two-dimensional Cartesian coordinates, these can be written as

$$\frac{\partial \vec{U}_i}{\partial t} + \frac{\partial \vec{F}_i}{\partial x} + \frac{\partial \vec{G}_i}{\partial y} = 0$$

$$\vec{U}_i = \begin{bmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \\ \rho m_i \end{bmatrix} \quad \vec{F}_i = \begin{bmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uH \\ \rho u m_i \end{bmatrix} \quad \vec{G}_i = \begin{bmatrix} \rho u \\ \rho uv \\ \rho v^2 + p \\ \rho uH \\ \rho v m_i \end{bmatrix}$$

The Euler equations governing the 2D flow in the absence of body forces with species transport equation in the conservative and differential form are,

$$\begin{aligned} &\frac{\partial(\rho)}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \\ &\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2 + p)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \\ &\frac{\partial(\rho E)}{\partial t} + \frac{\partial(\rho uH)}{\partial x} + \frac{\partial(\rho vH)}{\partial y} = 0 \\ &\frac{\partial(\rho m_i)}{\partial t} + \frac{\partial(\rho u m_i)}{\partial x} + \frac{\partial(\rho v m_i)}{\partial y} = 0 \end{aligned} \qquad (3)$$

3. Numerical Methods

Boundary conditions

Boundary conditions are specifications of properties or conditions on the surfaces of fluid domains and sub-domains and are required to fully define the flow simulation. Boundary condition decides the solution of the governing Equation. For two-dimensional inviscid flow problem, the commonly encountered boundary conditions are, 2D solid boundary fluxes, Inviscid or slip wall boundary condition, Pressure extrapolation boundary condition, Mirror image boundary condition, Far field boundary condition.

Grid and flow conditions

A typical grid used for the computations of flow fields around blunt body model with 2 mm jet diameter in the nose region, which are shown in Figure 2. The flow is assumed to be axisymmetric for blunt cone, for 3D models. The number of grid points is 205 in the x- direction (along the body) and 150 in the y- direction (perpendicular to the body). 6 points in the y- direction are distributed to express the exit of the coolant gas at the nose of the body. Design parameters are shown in Table 1 and in Figure 2.

Table 1: Blunt Body Parameters

Parameters	Values
Radius	38.1mm
Diameter	76.2mm
Height	76.2mm
Angle of Attack	0 degrees
Model Area	0.0114 mm^2
Jet Opening nozzle	2mm

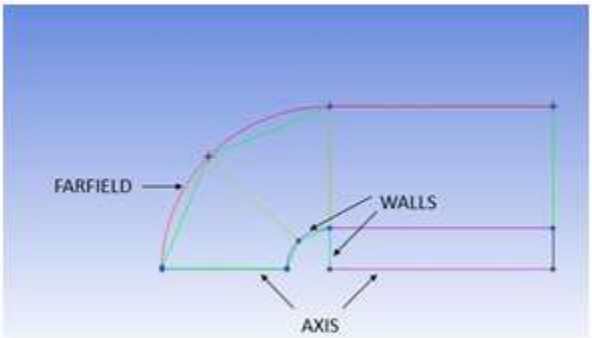

Figure 2: Blunt Body Parts

Table 2: Freestream flow conditions

Static Pressure	2.24kPa
Static Temperature	149.2K
Mach Number	≈ 5.7
Stagnation Enthalpy	≈ 5.0 MJ/Kg

Table 3: Hot-Gas Injection Conditions

Gas	Mach Number	Velocity (m/s)	Pressure (atm)	Temperature (K)
Air	4.4	1497.28	0.0918	6000
Air	2.565	871.14	0.0918	2000
Air	1.4	476.41	0.0918	600

4. Results and Discussions

The impact of the hot gas infusion from a surface of the limit body on the outer stream in the streamlined drag of the model was considered. The computational investigation demonstrates that the infusion of hot gas can be utilized to diminish the protection of a limit body into subsonic, transonic and hypersonic Mach numbers. The drag has been calculated for a blunt body without the influence of hot gas. The blunt body shows a higher drag co- efficient (Cd) of 0.6 without any influence of hot gas and in free-stream of 5.7 Mach and pressure at 2.24 kPa and free-stream temperature of 149.2 K. The bow shock is seen from figure 3 closed to the body when it is being exposed to the free-stream parameters.

As the hot gas is injected towards the free-stream flow the bow shock wave moves away from the model creating a greater pressure drop which can be seen in figure(s) 4,5,6 in different temperatures of hot gas. This pressure drop decreases the overall drag of the body. Different hot gas temperatures have been chosen like 6000K, 2000K and 600K. Drag for these hot gasses influenced body have been initialized and the drag have been calculated. This calculation shows that there is a drag reduction of 12-35% compared to the drag of the body without the influence of hot gas injection.

5. Formulation of Wave Drag Reduction

To determining the wave drag coefficient at the body it is clarify the reduction of drag force due to injection cooling. Drag, the fluid dynamics refers to forces which act on a solid object in the direction of the relative fluid flow velocity. The net drag is approximated by a non-dimensional parameter called wave drag coefficient which is defined by

$$C_d = \frac{2F_d}{\rho_\infty V_\infty^{\,2} A} = \frac{1}{\rho_\infty V_\infty^{\,2} A}\int_s (P - P_\infty).\vec{n}\, dA$$

Table 4: Drag Co-efficient of Different Hot- Gas Injection Conditions

Jet Stagnation Temperature (K	Mach N of Free-Stream	Mach No of Hot Gas Injecti	Co-Efficient of Drag	% of Drag Reduction
Without Ga Injection	5.7	-	0.58	-
600	5.7	1.4	0.43	15.4
2000	5.7	2.565	0.36	22.3
6000	5.7	4.4	0.28	30.2

5.1 Contour Representation

Figure 6 and Figure 7 show the contour of Mach number for blunt configuration in the absence of jet and in the presence of jet. From the Figure 6 clear that strong bow shock wave originates at the nose region that reduces the speed of the body. This causes a higher amount of drag at the nose region. Figure 7 displays the contours of Mach number to survey the actually existing amount of drag over the body in the presence of jet.

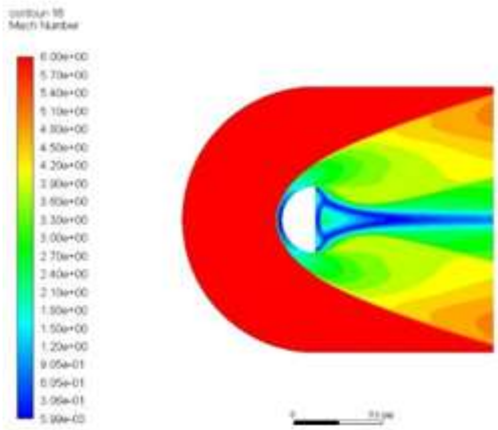

Figure 3: Mach Number of Blunt Body

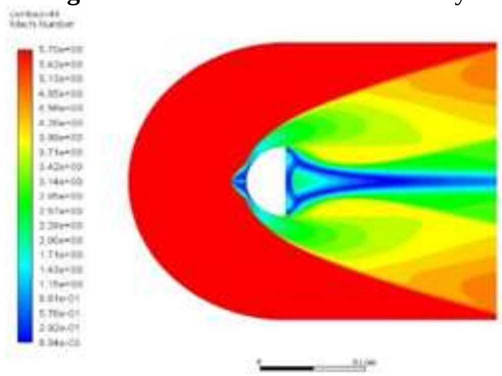

Figure 4: Mach number of 6000 K hot-gas injection

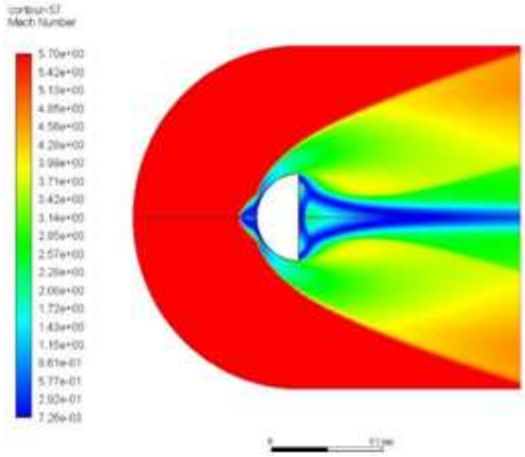

Figure 5: Mach number of 2000 K hot-gas injection

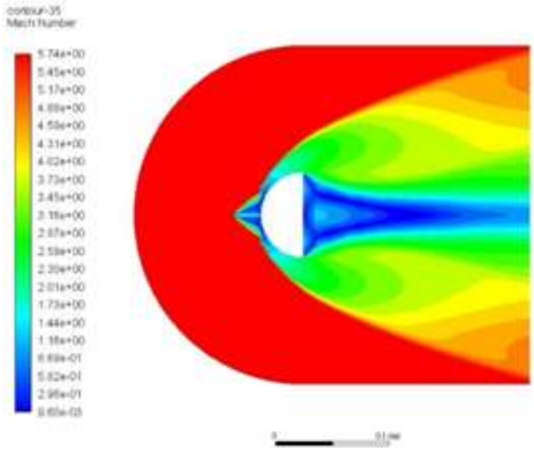

Figure 6: Mach number of 600 K hot-gas injection

5.2 Counters for Static Temperature

A larger recirculation region has formed at the blunt region. This larger recirculation region expels the bow shock far away from the nose region and enlarges the shock stand-off distance. The jet coming out from the blunt nose with the high velocity pushes the bow shock away from the blunt region of the body. This shows the way to reduction in drag.

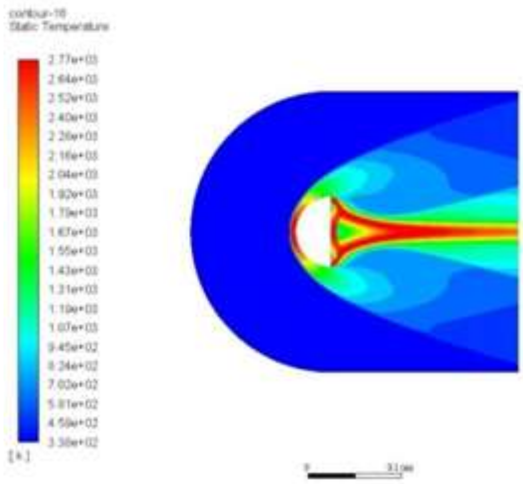

Figure 7: Static Temperature of Blunt Body

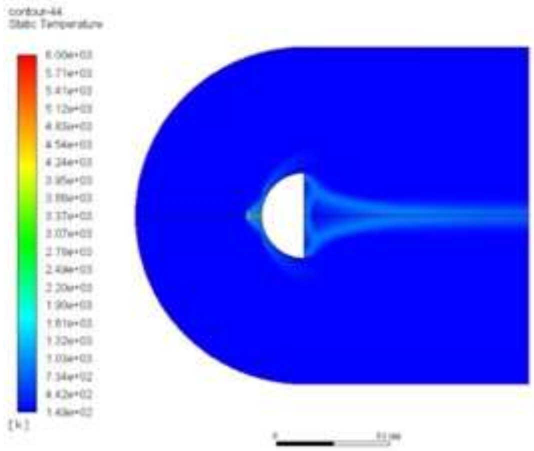

Figure 8: Static Temperature of 6000 K hot-gas injection

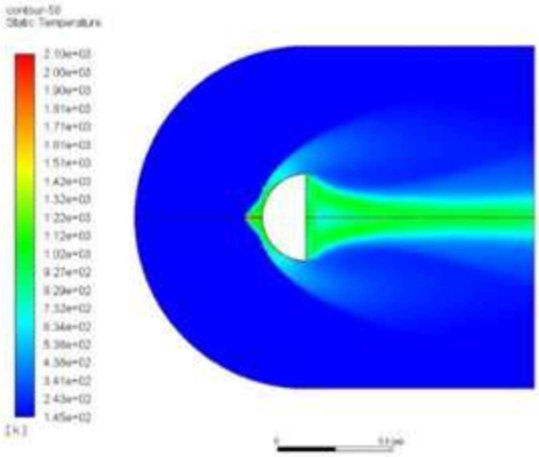

Figure 9: Static Temperature of 2000 K hot-gas injection

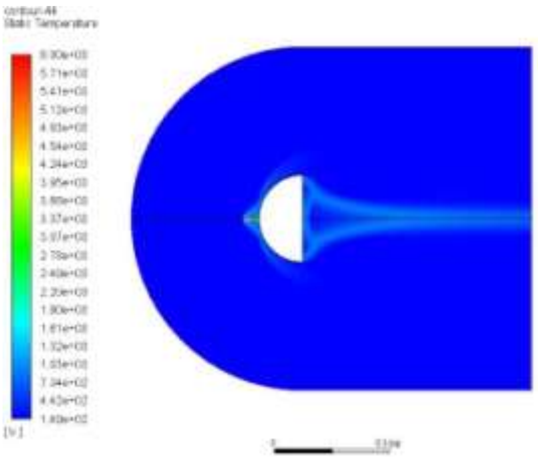

Figure 10: Static Temperature of 600 K hot-gas injection

The graphical representation of pressure drags and static temperature distribution which illustrate the comparative study between the case in the absence of jet and in the presence of jet for the blunt body are given here. The optimum condition for jet which is given in the Table 3 have been executed in the presence of the jet case, blunt body which is consider in the present work. The comparison of static temperature distribution in the absence of jet and in the presence of jet, with different jet conditions is presented through Figures. From the figures, very high amount of static temperature distribution is clearly seen at the nose region in without injection case. And in the injection case, evaluation of the static temperature distribution shows the nose static temperature distribution is minimum at the tip then slightly increases due to recompression shock wave.

6. Conclusion

The report has discussed aerodynamic drag reduction by using hot gas injection technique. The main objective of the project is to analyse the reduction percentage of aerodynamic drag using hot gas injection technique and to analyse the effect of gas injection in the opposite direction of the free stream and evaluate the amount of drag reduced using this technique. Two models are designed for drag analysis. Model one is analysed for closed section and the other model for the open section from which hot gas of 6000K, 2000K, 600K is injected from the nose of the model towards the freestream flow and drag is analysed.

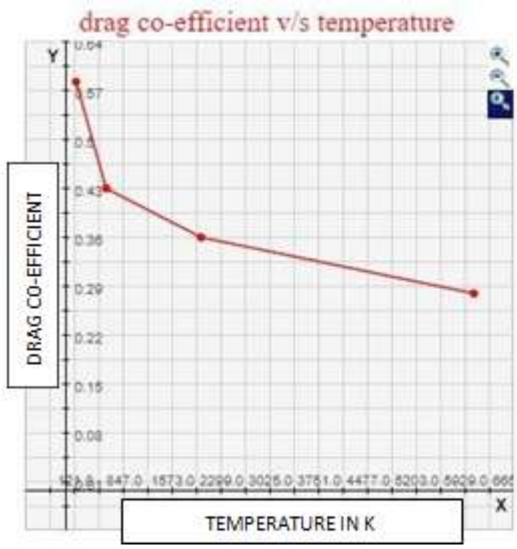

Figure 11: Drag co-efficient v/s Static Temperature

References

[1] Y.C.Ganiev, V. P. Gordeev, A.V.Krasilnikov, V.I.Lagutin, V.N.Otmennikov, and A.V.Panasenko Central Research Institute of Machine Building, Russian Space Agency, 141070,Korolev, Moscow Region, Russia. JOURNAL OF THERMOPHYSICS AND HEAT TRANSFER. Vol. 14, No. 1, January–March 2000.

[2] V.M. Fomin , P.K. Tretyakov, , J.-P. Taran, Institute of Theoretical and Applied Mechanics, Siberian Branch, Russian Academy of Sciences, Novosibirsk, Russia. ONERA, 29 Avenue de la Division Leclerc, 92320 Châtillon, France.

[3] R Sosa, J D'Adamo and G Artana Laboratorio de Fluidodinámica, Universidad de Buenos Aires, Av. Paseo Colon 850, C1063ACV, Buenos Aires, Argentina. CONICET.

[4] IOP Publishing. Jian Wang, Yinghong Li, Bangqin Cheng, Changbing Su, Huimin Son and Yun Wu Engineering College, Air Force Engineering University, Xi'an 710038, People's Republic of China.

[5] IOP Publishing. Plasma physics and Controlled Fusion.D Caruana. ONERA-Aerodynamics and Energetics Models Department (DMAE), Centre Midi-Pyrenees,2 av. Edouard Belin, 31055 Toulouse, France

[6] JOURNAL OF SPACECRAFT AND ROCKETS,Vol.

39, No. 3, May–June 2002 Hypersonic Flow over a Blunt Body with Plasma Injection by J. S. Shang and J. Hayes U.S. Air Force Research Laboratory,Wright–Patterson Air Force Base, Ohio and J. MenartWright State University, Dayton

[7] J. S. Shang U.S. Air Force Research Laboratory,Wright–Patterson Air Force Base. AIAA JOURNAL Vol. 40, No. 6, June 2002 Plasma Injection for Hypersonic Blunt-Body Drag Reduction

[8] Timothy N. Jukes and Kwing-So Choi Through Velocity and Temperature Measurements