International Journal of Applied Engineering Research ISSN 0973-4562 Volume 10, Number 18 (2015) pp 39488-39495

Review of Drag Reduction Techniques for Hypersonic Flows

Bibin John

Associate Professor, School of Mechanical and Building Sciences,
VIT University, Vellore, Tamil Nadu, India-632014 bibin.john@vit.ac.in

Vinayak Kulkarni,

Associate Professor, Department of Mechanical Engineering
Indian Institute of Technology, Guwahati, Assam, India-781039 vinayak@iit.ernet.in

Abstract
Generally hypersonic space vehicles are designed with blunt nose to reduce the surface heatflux. However this passive technique enhances the wave drag thus makes the space flight costlier. Hence the space vehicle design has to be carried out by considering the best tradeoff between surface heat load and wave drag. Therefore over the past sixty years development and optimization of hypersonic drag reduction technique is emerged as a major research area in the hypersonic field. Many passive and active techniques have been proposed by different researchers in this field. Efforts are made herewith to summarize the investigations of various researchers about such drag reduction studies. A brief review of the literature reported techniques based on aero-spike, counter-flow injection, energy deposition and surface coating is presented in this article.

Keywords: Hypersonics, Drag reduction, Aero-spike, Counter-flow injection, Energy addition.

Introduction
Man's desire to fly higher and faster has led to the development for hypersonic flights. This in turn accelerated the research in the field of hypersonic aerodynamics. The aerodynamic design of hypersonic vehicles has to be carried out by accounting the complexities involved in hypersonic flow. Generally hypersonic flows are characterized by thin shock layer and high temperature field near the body surface [1]. Such shock and viscous dissipation resulted elevated temperature near the space vehicle surface gives rise to excessive heating of space crafts and ballistic missiles flying at hypersonic speeds. This makes the design and implementation of thermal protection system of such vehicles more costly. Moreover higher thickness of heat shields reduces the payload capacity of the vehicle, thus makes the space flight uneconomical. The nose region of space vehicle being the most critical place of space vehicle that experience highest heatflux, blunt design of the same has been identified as an effective method to push the shock layer away from the body, so as to reduce surface heatflux. Since stagnation point aerodynamic heating varies inversely to the square root of the nose radius [1], keeping large blunt nose section is advantageous as far as surface heat reduction is concerned. However this design criterion enhances the drag penalty. For space mission increase in drag creates higher fuel consumption and lesser pay load capacity. Thus the cost of space mission increases drastically. Therefore, reduction of wave drag has remained as an important field of research in the area of hypersonics. Moreover, the dreamt hypersonic flight or mission needs reduced heat load for safety and reduced drag for economic advantage in order to reduce the fuel requirement or to increase the payload capacity of the space vehicle. Thus primarily investigated passive drag reduction techniques based on aero-spike and reactive surface coating are reviewed along with active drag reduction techniques like stagnation point injection and energy deposition in the subsequent sections.

Aero-Spike Based Drag Reduction Studies
The idea of structural aerodynamic spike is an effective means to reduce the pressure drag of blunt bodies in supersonic stream. Structural spike is a simple slender device that extent outward from the nose section of the space vehicle as shown in Fig. 1. Flowfield in the nose region of the blunt body get considerably altered with the introduction of aero-spike. The standing bow shock in front of the blunt body get recast by the spike into a conical shock thus forms a low pressure recirculation region over there. The recasted shock structure and recirculation region together forms an aerodynamic body profile similar to that of a slender body ahead of blunt body. Thus developed low pressure flow ahead of the body substantially reduces the dynamic pressure on the body, hence wave drag too. However the separated flow further reattaches on the surface of the blunt body somewhere downstream of nose section. Spike length, spike configuration and freestream Mach and Reynolds numbers play crucial role in changing the flow-reattachment location on the blunt body. At reattachment point flow turns across a reattachment shock to facilitate attached smooth flow over the downstream body surface.
For the last Six decades, aero-spike based drag reduction of blunt bodies has been a topic of research among hypersonic research community. Since 1950s a considerable amount of experimental and computational investigations have been taken place in this field. Many researchers have carried out parametric studies to understand the effects of spike length, spike configurations, freestream conditions and different blunt body configurations on wave drag acting on the body. Most of the important findings of past studies related to aero-spike based drag reduction are consolidated in this section.

International Journal of Applied Engineering Research ISSN 0973-4562 Volume 10, Number 18 (2015) pp 39488-39495
© Research India Publications. http://www.ripublication.com

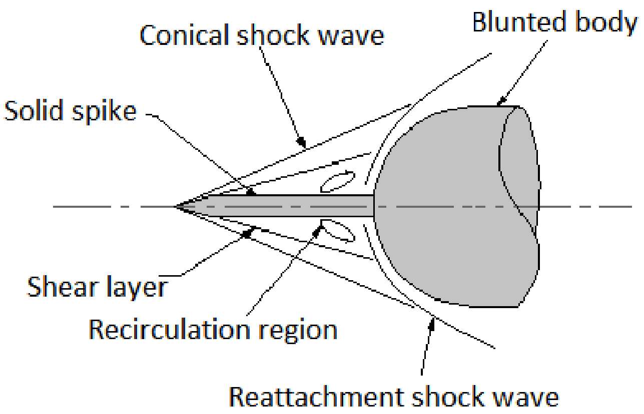

Fig. 1: Schematic diagram of hypersonic flow around a blunt body fitted with aero-spike.

Stadler and Nielsen[2] performed experimental studies on a hemisphere-cylinder configuration at freestream Mach numbers of 1.5, 2.67, and 5.0, and a Reynolds number in the range of 0:16 $\times 10^6$ - 0.85 $\times 10^6$ based on the diameter of the cylinder. They have observed a considerable amount of reduction in wave drag with the attachment of forward facing spike to the blunt body. Investigations of Bogdonoff and Vas [3] shown that the longer the spike is extended, the more the drag reduction can be achieved. Crawford [4] studied the Reynolds number effect on the drag coefficient of a spiked hemispherical configuration and observed the potential of reduction in the drag coefficient in presence of spike for all Reynolds numbers. Drag reduction has been found to be saturated if length of the spike is more than four times the body diameter. Maull [5] carried the experiments for drag reduction studies in a gun tunnel. In this study the author has performed analysis of the flow over axisymmetric spiked bodies at a Mach number of 6.8. Investigations have been carried out to explore the role of roundness of a spiked flat cylindrical model's shoulder on the flow oscillations. He performed experiments with models of various shoulder roundness, ie., shoulder roundness varied from zero (sharp shoulder) to 0.5 (hemisphere cylindrical model). It was observed that, reduction in drag is possible due to spiked configuration but in some cases this reduction could be unsteady, and hence selection of body geometry should be carefully designed to avoid the unsteadiness. From the study it was also noticed that, a hemisphere cylindrical model yields a stable flow regardless to the spike length. During the extension of these investigations for spiked cone-cylinder configuration, Wood [6] explored five distinct types of flows concerning spiked bodies. These studies revealed that flow separation in the presence of spike is independent of the shape and size of the body and depends on freestream Mach number, Reynolds number and spike geometry. Comment has also been made that maximum possible drag and heat transfer reduction is only possible if the region of separation covers the wetted surface of the body or in other words if re-attachment takes place very close to the body shoulder. However the same separated region was observed to be oscillatory for configurations with semi-cone angle greater than the conical shock detachment angle. Experimental study of Holden [7] was another important milestone in this field. He has performed Mach 10 and 15 flow experimental studies with main focus on the heat transfer to the spiked bodies. Hemisphere cylinder, flat cylinder, and cone cylinder were employed as models in his studies. From the investigation Holden concluded that, heat transfer at the reattachment point is directly proportional to reattachment angle

To increase the lifetime of network, harvesting the energy from wind, solar, vibration has a lot of attraction in the field of research. However energy harvesting technique has limitation because extraction of energy from the environment is subject to availability of the resources. The wireless power transfer technology has several applications medical sensors, image sensors and in implantable device in health care industry has already using the wireless power transfer technology to replenish battery energy in sensor node. Thurman [8] explored spike induced drag reduction for various spike deflection angles. In his study he employed a spike attached to the stagnation point of the blunt body which can deflect about the axis. By slightly deflecting the spike he could get some enhancement in lift too. The freestream conditions of the investigated flow field were Mach 11.76 and Reynolds number 14000. Zorea [9] used various spike lengths for experimental studies and concluded that, for maximum reduction in drag, there exists certain spike length termed as 'critical spike length'. Hutt and Howe [10] reported their experimental investigations for critical spike length mainly for different cross-sectional configurations during the explorations for reduction in the drag coefficient. Aerodynamic characteristics of hypersonic flows in the presence of aero-spikes attached with aero-disks have been studied by Motoyama et al.[11]. He performed multiple experiments with hemispherical and flat faced aero-disks attached to the aero-spike for a freestream of Mach number 7 and Reynolds number 4×10^5/m. Aero-spikes of L/D=0.5 and 1.0 were employed to investigate the angle of attack effect in drag reduction. Experiments were repeated for different angle of attack varying from 0 to 8 degree. From the experiments it has been found that, the aero-disk spike with L/D=1 and having 10mm diameter aero-disk has a superior drag reduction characteristic as compared to the other aero-spikes. Aero-spike is found to be useful for drag reduction at angle of attack flight too; however, care must be taken to adjust the increased pitching moment resulting from the spike. Viren et al. [12] carried out shock tunnel based experimental studies for various types of forward facing aero-spikes attached to a 1200 blunt cone configuration at stagnation point for Mach number 5.75 test conditions. Flat disc tipped aero-spike was noted as the best configuration for reducing wave drag at all the tested angles of incidences. Kalimuthu et al. [13], investigated experimentally the effect of spike length, spike nose configuration and angle of attack on the reduction of drag with hypersonic wind tunnel at Mach 6.They have separately analyzed the drag, lift and pitching moment coefficient's variation with angle of attack for conical aero-spike, hemisphere faced aero-disc, and flat faced aero-disc with various L/D ratios. After series of experiments, from the stand point of drag reduction they concluded that, the aero-disc with L/D=2 is most effective for flow situations with angle of attack.

Vinayak et al. [14] extended the studies of Viren et al. to access the performance of the disc tipped aero-spike at the critical length for different freestream stagnation enthalpies and observed that reduction in wave drag for this configuration is independent of the stagnation enthalpy. Recently d'Humieres, G and Stollery, J. L. [15] carried out the experimental study by considering laminar hypersonic flow over a spiked, conical body terminated by a spherical cap. In their experiments, for L/D values of 1/8 and 1/4, no drag reduction has been observed, even though a conical shock appears for L/D equal to 1/4. A further increase of L/D up to 1 reduces the drag coefficient linearly to 44% of the no spike C_D. For values of L/D above 1, the drag reduction was found to be more gradual but a drag reduction of 77% is obtained for the longest spike tested, with a L/D ratio of 2.125.

CFD based simulations for drag reduction studies have also been reported in the literature [16]. Numerical simulations and wind tunnel tests for pressure measurement and flow visualization are reported by Kubota et al [17] for a hemisphere cylinder with sharp/blunt nosed spikes of different lengths at Mach 7. Yamauchi et al. [18], have performed further investigation of Crawford's spiked hemisphere cylindrical model using computational fluid dynamics, for Re_D = 140,000 and Mach numbers of 2.01, 4.15 and 6.8. A three dimensional Navier-Stokes solver has been employed in this analysis. It was observed that, the calculated variation of the drag coefficient with L/D agrees well with the experimental trend from Crawford. Later Mehta [19, 20] conducted a series of numerical studies on the same model namely, a hemisphere cylindrical model equipped with a pointed spike. In those studies, he solved the unsteady Navier-Stokes equations in an axisymmetric computational domain and the flow was assumed laminar. Through his studies he confirmed the findings of Yamauchi et al. and made validations with their results. Recently Gerdroodbary and Hosseinalipour[21] numerically analyzed the effectiveness of effectiveness of aero-disk/aero-spike assemblies as drag-reduction devices for large-angle blunt cones flying with Mach 5.75 at various angles of attack. In this analysis 3D Navier-Stokes equations have been solved along with k-ω turbulence model. Effectiveness of various spike tips also analyzed in this study. They have reported different percentage of reduction in drag with different combination of spike length, angle of attack and spike tip shape.

Oscillation or pulsation in the flow has been observed by few researchers in the presence of spiked configurations. Among such investigations, study of Kenworthy [22] is more significant. Through his studies he could explain the cycle, mechanism, and cause of oscillation mode. He pointed out that the cause of flow oscillation mode is a continuous pressure imbalance on both sides on the shear layer at the reattachment point. Another important reported research in this field is experimental studies of Panaras [23] for pulsating flows with axisymmetric spiked bodies. During these studies at Mach 6 in a blow down wind tunnel, it was observed that there are two types of instabilities called as 'pulsation' and 'oscillation' types. Formal reason for pulsation in the flowfield was proposed due to the effect of the annular supersonic jet appearing at the shock intersection of the foreshock and aftershock on the separation bubble. The oscillation type instability was claimed mainly due to the stability issues of the shear layer that surrounds the conical separation bubble. The volume of the axisymmetric cavity and the geometry of the shoulder at the point of reattachment of the shear layer were noted to be the critical parameters for the occurrence this instability. Supersonic buzz phenomenon was investigated by Shang and Hankey [24] through the CFD based simulations at a Mach number of 3 for spike-tipped configurations. Temporal variation of pressure has been studied by Mehta [25] during numerical simulations at Mach 6.8 conditions for various spike lengths corresponding to hemispherical diameter ratios of 0.5, 1.0 and 2.0. Increase in length of the spike was found to be effective in reducing surface pressure which was observed to be highly time dependent. Later Feszty [26, 27] further investigated Kenworthy's experimental findings through numerical studies. His results were also agreeing with Kenworthy's explorations. Laminar supersonic axisymmetric flow (M=2.21 and Re_D =120000) over forward facing cylinder attached with a spike of L/D=1.0 was considered in his analysis. He has proposed a new driving mechanism with main feature of a vortical region in the vicinity of the foreshock-aftershock intersection.

Counter-flow Drag Reduction Studies

Use of Jet spike is an active technique proposed for the reduction of wave drag along with surface heatflux. In this technique, a counter-flow jet injected from the nose region of blunt body considerably alters the external oncoming flowfield and weakens the strong shock structure in front of the blunt body. Schematic representation of flow structure ahead of a blunt body with counter-flow injection is depicted in Fig. 2.

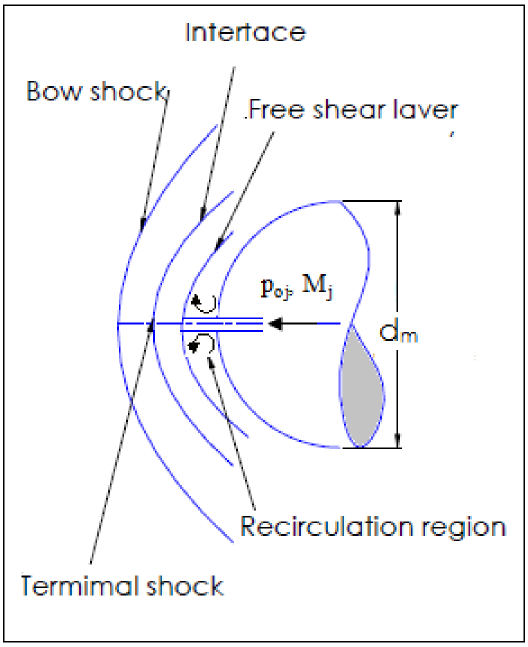

Fig. 2: Schematic diagram of hypersonic flow around a blunt body with stagnation point injection.

Here a gas of total pressure p_{0j} is injected from the stagnation region of the blunt body through a small orifice. Depending on the pressure ratio ($p_{0j}/p_{0\infty}$) the jet velocity at orifice exit may attain sonic or even supersonic speed. Here $p_{0\infty}$ represents freestream total pressure. If the injection pressure 'p_{0j}' is maintained as sufficiently higher than the freestream stagnation pressure '$p_{0\infty}$', then the orifice exit may get chocked and the jet comes out in the opposite direction to the freestream flow. This high velocity jet pushes the shock layer further ahead of the body. This interaction of counter-flow jet with freestream supersonic or hypersonic flow creates a recirculation zone in front of the blunt body that contains large scale vortices. The counter-flowing jet terminates with a Mach disk (also known as terminating shock) and reverses its direction of flow to form a nearly conical free shear layer. An interface gets created in front of the Mach disk with a second stagnation point (free stagnation point) where the jet meets the freestream which has already decelerated across the bow shock. The position of Mach disk and steady nature of flow structure largely depend on the injection pressure ratio '$p_{0j}/p_{0\infty}$'. Mass, momentum and energy transfers are expected to take place across the interface. The deflected jet from the free stagnation point may reattach the blunt body surface by passing across a reattachment shock (not shown in the schematic). The recirculation region which is bounded by the deflected jet and blunt body surface has a great impact on wave drag reduction as well as surface heatflux decline.

The earliest study of counter flow injection can be seen in the experimental work of Warren [28]. In this study, the author presented experimental investigations for straight and swirl injection for a blunt body. Heat transfer and pressure measurements for these studies were conducted in a hypersonic wind tunnel at freestream Mach number of 5.8. Changes in the flow structure had not been observed with increase in jet total pressure until it reached a certain injection total pressure at which the bow shock ahead of the body starts bulging. The different gases used for injection were found to have same momentum coefficient corresponding to the injection pressure at which change in shock shape was observed. When heat transfer rate was calculated along the model surface, it was found that, heat transfer rate decreases in the stagnation region after which severe heating was observed. It was also concluded that the heat capacity of the gas is an important parameter to define the gas as an effective coolant. Libby and Cresci [29] carried out an experimental investigation of downstream influence of stagnation point mass transfer for a hemisphere-cylinder combination in the wind tunnel at freestream Mach number of 6.0. Different injecting gases such as helium, nitrogen, argon and krypton were used during the experiments. Approximately 15% increase in shock stand-off distance was noticed for the higher mass flow rates. Romeo and Sterrett [30] did exploratory investigation to understand the forward facing jet for two different cases in a Mach 6 wind tunnel. In one case, it was found that the bow shock moves away from the body without altering its shape while in other, the jet displaces the bow shock and changes its shape. For the shock displacement cases, larger shock displacements were recorded at higher jet Mach number or for the higher ratio of model diameter to jet diameter. It was also observed that the shock displacement reduces with increase in angle of incidence. Later Barber [31] performed an experimental investigation of stagnation point injection for heat transfer and shock stand-off distance measurement in a plasma tunnel. He observed three regimes of the flow depending on the relative mass flow rate coefficient. These three regimes were termed as 'boundary layer', 'viscous' and 'shock regimes'. During the transition from first to second regime, increase in shock stand-off distance was observed; while from second to third regime transition decrease in shock stand-off distance was found. These experiments were conducted with hydrogen and helium as injection gases. Since hydrogen is a combustible gas, the average heat transfer rate was found to be increased while for helium injection a decrease in average heat transfer rate was reported due to noncombustible nature of the gas. Finley [32] conducted experiments to study the effect of injection on pressure measurement on a blunt body in an intermittent wind tunnel of Mach 2.5. Injection of a jet from the stagnation point showed three different flow regimes with increasing injection pressure ratio ($p_{0j}/p_{0\infty}$) during schlieren flow visualization. The first regime with multiple jet cells shows steady reduction in force. Flow structure shifts to the second regime with increase in injection pressure ratio. This regime showed continuous reduction in the force, and also has the characteristic of multiple jet cells. However, the flow field is unsteady in nature. Drag reduction reaches to its maximum value during the transition from second to third flow regime. Steady decrease in drag force is the characteristic of the third regime with further increase in pressure ratio. The possibility of aerodynamic drag reduction was studied experimentally in the subsonic, transonic and supersonic freestream using plasma and hot gas injection in transonic and supersonic wind tunnel by Ganiev et al. [34]. During these experimentation strain gauges were used for force measurement. Modification in the flow field in the presence of a forward facing jet was computationally studied by Benjamin et al. [33]. Maximum drag reduction of 55 % was reported for a specific jet exit condition. Numerical analysis has revealed the reduction in heat transfer and skin friction along the body. Shang et al. [24] conducted wind tunnel testing for plasma injection experiments at freestream Mach number of 6.0. The two regimes of the flow structure were found to be separated by jet bifurcation, before which self-sustained oscillatory motion of the jet was observed and after bifurcation, the jet was found to be steady. The ratio of the tunnel stagnation pressure to jet stagnation pressure was suggested as the governing parameter for the jet bifurcation at which a maximum reduction of 40 % in the drag force was observed using the cold jet injection. Plasma and cold jet injection studies were also conducted by Shang et al. [36] in blow down wind tunnel. It was observed that injection of plasma gives lower drag reduction at a given stagnation pressure and in the absence of magnetic field, when compared with room temperature air injection. However, under the same mass flow rate consideration, plasma injection showed an enhancement in reduction of drag by 6.1-13.4 % and this drag reduction is due to the thermal effect of the plasma.

Considering the interest in the area of jet injection, drag and heat transfer measurement experiments in the presence of forward facing supersonic jet were conducted in the High

Enthalpy Aerodynamics Lab of IISc. Gas injection studies were carried out earlier in HST2 shock tunnel by Sahoo et al. [37] and they reported an increase in wave drag for lower injection pressure ratios at freestream Mach number of 5.75. Balla Venukumar et al. [38] also carried out jet injection experiments in the same experimental facility at freestream Mach number of 8.0 and stagnation enthalpy of 2 MJ/kg. They reported the possibility of reduction in wave drag at higher injection pressure ratios. During the extended investigation by Vinayak and Reddy [39] for the same sphere-cone configuration in the free piston driven shock tunnel, it was noticed that percentage reduction in wave drag increases for the same injection pressure ratio with increase in freestream stagnation enthalpy. Energy interaction between high enthalpy dissociated fluid behind the shock and injected cold jet was claimed as one of the possible reasons for this enhanced reduction in drag. Vinayak and Reddy [40] also claimed the possibility of reduction in heat transfer in the presence of stagnation point injection of cold sonic jet.

Freestream Energy Deposition based Drag Reduction Studies

Complexities involved in the implementation of aero-spike and counter-flow jet based drag reduction techniques motivated the researchers in this field to explore a flexible technique for drag reduction. Investigations on that theme resulted in the proposal of a new drag reduction technique, called localized energy deposition based drag reduction technique. In this technique a suitable magnitude of concentrated energy source is added upstream of the configuration of interest as shown in Fig. 3. Such an energy deposition at the suitable location, upstream of the desired blunt body, creates alterations in the upstream flowfield and provides low pressure region ahead of the vehicle. Thus resulted low surface pressure distribution lowers the wave drag. Substantial amount of contributions have been reported in the open literature about drag reduction using concentrated energy deposition.

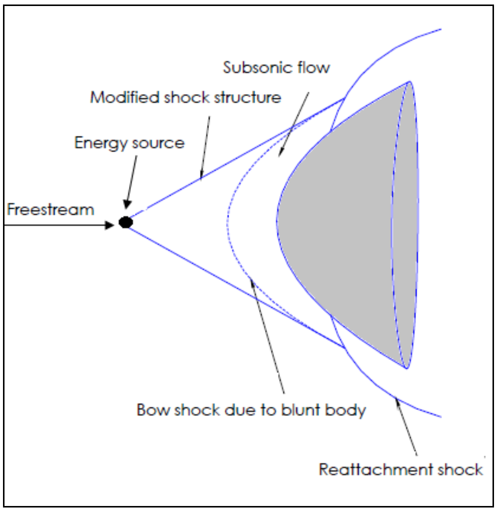

Fig. 3: Schematic of flowfield with upstream energy addition

The studies of Miller and Carlson [41] can be considered as the earliest reporting about the use of energy addition to alter the supersonic flow field. In this work the potential use of heat source for sonic boom mitigation is presented. For these investigations, 1-D inviscid flow was modeled for the feasibility of using suitably placed heat source to change the effective shape of the aircraft by deflecting the oncoming streamlines. From these studies, it had been observed that, very large amount of energy (almost twice the amount required to keep the aircraft in steady flight) is required to achieve noticeable effects. Myrabo and Raizer [42] performed a theoretical study to explore the possibility of laser induced upstream flow alterations to reduce the strength of the bow shock ahead of the blunt body. Blast wave analysis was used to predict the shape of the shock wave generated by the point energy source in the flow field and subsequently to estimate the location of the focused energy spot and the power requirement for flow control on a given blunt body configuration. It was identified from the study that the power requirement reduces as ambient pressure reduces with altitude. However it was also observed that, at higher altitude, the power required for the initial breakdown of the flow is much higher than that required to sustain plasma and this energy required will increase with altitude. Numerical methods have also been used to analyze the effects energy deposition to compressible flow field. Explorations of Levin et al. [43] are of such kind. In this work, they carried out a parametric study of a supersonic flow around a spherically blunted body in the presence of an energy-supply source. Calculations have been carried out on the basis of the Navier-Stokes equations for the model of thermally equilibrium air for a wide range of freestream parameters and the intensity and size of the heat source. It has been noticed that, heat added to the incoming supersonic freestream has the effect of reducing wave drag. They have reported that the limiting values of drag reduction decreases from 0.68 for M_∞ =1.5 to 0.32 for M_∞=6 conditions. However the power of the heat source necessary to reach these limiting values exceeds considerably in comparison with the power of an engine required to overcome the aerodynamic drag when moving in air without heat supply. It has also been reported that, the efficiency of heat supplied for drag reduction increases as the relative size of the heat source decreases and the Mach and Reynolds numbers increase. Riggins et al. [44] carried out a 2-D numerical parametric study using different energy sources located at different positions in front of the model. It was observed that maximum drag reduction of 50% is possible with a power input as low as 3% of the total propulsive power. Adelgren et al. [45] conducted experiments in a Mach 3.45 flow, using a pulsed laser beam to deposit energy in the freestream, and observed around 40% decrease in surface pressure on the sphere exposed to the flow, due to the interaction of the energy spot. The numerical and experimental studies by Kolesnichenko and Brovkin [46] address the effect and efficiency of energy deposition on drag reduction in a Mach 2 flow. In these studies, two types of heating were considered, viz a quasi-static isobaric heating and a fast isochoric explosive heating. Based on the results obtained, it was suggested that, in the first case, a region of low density develops, which interacts with the shock wave and this interaction is independent of the location of the energy spot. In the second case a shock wave is formed from the energy deposition region resulting in the flow field modification. In this case nature of interaction is dependent on the location of the energy spot. Hartley et al. [47] carried out experiments at low enthalpy conditions in a shock tunnel using an electric arc struck between two electrodes placed in front of the model as energy source. The effect of the arc power on the aerodynamic drag has been measured. Around 70% of drag reduction has observed for an arc power of 40 KW after which increase in power did not produce any appreciable change in drag. Doyle et al. [48] has developed a gas dynamic model for microwave energy deposition in air. The model included detailed kinetics and thermo-chemistry using 23 species and 238 reactions. Satheesh and Jagadeesh [49] provided an experimental proof for drag reduction by energy deposition using electrodes during shock tunnel experiments. These studies were conducted on a 120^0 apex angle blunt cone configuration for argon and air hypersonic flows. Maximum drag reduction of 50% was noticed for argon freestream due higher capacity of energy absorption of argon gas.

Numerical studies of energy addition based drag reduction have also been carried out by the present authors [50]. In this study we considered hypersonic flow over sphere of 30 mm radius. Computational studies were carried out by employing an in-house developed finite volume compressible flow solver. Effects of energy source size, strength and location of energy addition on amount of drag reduction were analyzed through this study. This investigation revealed that at low magnitude of energy addition this active drag reduction technique is more efficient. The drag magnitude is seen to be decreasing with increase in energy source strength. However the power effectiveness, which is an efficiency representative of energy addition based drag reduction technique, reduces with increase in energy source strength. The optimum location of energy source that gives maximum drag reduction is noticed to be a point in the stagnation line which is at a base diameter distance in front of the stagnation point. This study also showed that the smaller the energy bubble size larger the drag reduction. However the size of the energy bubble must be decided with the practical feasibility of implementation of this technique.

Reactive Coating Based Drag Reduction Studies

In the area of passive drag reduction technique, a new method has been recently proposed by Vinayak et al. [51]. This method although exhibits similarity with drag reduction using localized energy deposition, the basic difference in the two techniques lies in two ways viz. method of energy deposition and location of energy deposition. Chromium coated 30^0 sphere-cone configuration was tested in shock tunnel during these investigations. It was observed that deposition of energy is indeed possible through exothermic reactions in the shock layer at high stagnation enthalpy conditions between the dissociated shocked gas and the coating material. Upstream displacement of the bow shock was noticed during flow visualization while reduction in drag was measured using accelerometer force balance. Forty seven percent reduction in wave drag was reported for stagnation enthalpy of 3.4 MJ/kg. This technique being simple and passive can be easily integrated with the other drag reduction techniques. In line with this view, drag reduction for spiked configurations has been observed by active deposition of energy in the shock layer (Reding and Jecmen [52] and Srinivasan Chamberlain [53]). Hence further scope of research exists for this technique to get integrated with the contemporary techniques. However this technique although reduces drag, but at the cost of marginal increase in surface heat flux (Vinayak et al. [51]). Hence this technique, unlike other techniques when solely implemented for drag reduction, care should be taken to incorporate for efficient cooling mechanism. A few techniques which employ combination of different drag reduction techniques were also reported in literature. The investigation of Jiang et al. [54] is one of such kind. In this study he proposed a new technique of combining the aero-spike and lateral jet to protect the spike tip from overheating and to push the conical shock away from the blunt body when a pitching angle exists during flight. The idea was to enlarge the spike recasted conical shock angle in such a way that the shock/shock interaction point occur in space not at the blunt body. He pointed out that in the absence of lateral jet the spike become ineffective at non-zero angle of attack. With his newly proposed idea he could achieve about 77% reduction in drag for a hemispherical cylinder at 4^0 angle of attack.

Conclusion

Drag reduction studies for passive and active techniques have been summarized in this article. Simplicity of the passive techniques, mainly spike and surface coating based drag reduction, make them easier to implement in the real flights. Most of the researchers have noticed that maximum reduction in drag is possible with the flat disc aero-spike of critical length. Drag reduction technique based on aero-spike was also seen to have higher potential of heat flux reduction in many investigations; however the heat flux reduction with aero-spike is not always guaranteed. Moreover to obtain the maximum possible advantage of aero-spike technique, the laminar nature of incoming flow field must be ensured. Counter flow drag reduction technique was seen to be an important technique which can offer drag reduction along with heat flux reduction. However the requirement of maintaining a continuous supply of a jet with a total pressure higher than the stagnation pressure makes counter flow drag reduction technique more complicated in practical implementation. In addition the aero-spike and counter flow techniques are limited to zero or small degrees of angle of attack. Combination of spike and lateral injection was found to be advisable as drag reduction technique in angle of attack flight conditions. Drag reduction based on localized energy deposition has been observed to be equally promising as the counter flow injection in the area of active techniques. Surface coating based drag reduction has also been the promising due to its uniqueness. However implementation of this technique should be done in the presence of efficient heat alleviation method due to marginal enhancement in heat flux offered by this technique.

References

[1] J.D. Anderson Jr., Hypersonic and High Temperature Gas Dynamics. McGraw-Hill series in Aeronautical and Aerospace Engineering, New York: The McGraw-Hill Company, 1989.

[2] J.R. Stalder and H.V. Nielsen, Heat transfer from a hemisphere-cylinder equipped with flow-separation spikes. National Advisory Committee for Aeronautics, 1954.

[3] S.M. Bogdonoff, I.E. Vas, Preliminary investigations of spiked bodies at hypersonic speed, Journal of the Aerospace Sciences 26 (2) 65-74, 1959.

[4] D.H. Crawford, Investigation of the Flow over a Spiked-Nose Hemisphere-Cylinder, NASA TN-D-118, 1959.

[5] D.J. Maull, Hypersonic Flow over Axially Symmetric Spiked Bodies J. Fluid Mech., Vol. 8(4), pp. 584-592, 1960.

[6] C.J. Wood, Hypersonic Flow over Spiked Cones. J. Fluid Mech., Vol. 12(4), pp. 614-624, 1961.

[7] M.S. Holden, Experimental studies of separated flows at hypersonic speeds. I-Separated flows over axisymmetric spiked bodies. AIAA Journal, 4(4), 591-599, 1966.

[8] W.E. Thurman, A Flow Separation Spike for Hypersonic Control of a Hemisphere- Cylinder. AIAA J., Vol. 2(1), pp. 159-161, 1964.

[9] C. Zorea and J. Rom, Effect of a Spike on the Drag and on the Aerodynamic Stability of Blunt Bodies in Supersonic Flow. J. Spacecraft, Vol. 7, pp. 1017-1019, 1970.

[10] G.R. Hutt and A.J. Howe, Forward Facing Spike Effects on Bodies of Different Cross Section in Supersonic Flow. The Aeronautical J., Vol. 93(921-930), pp. 229-234, 1989.

[11] N. Motoyama, K. Mihara, R. Miyajima, W. Watanuki and H. Kubota, Thermal Protection and Drag reduction with Use of Spike in Hypersonic Flow, AIAA Paper 1828, 2001.

[12] Viren Menezes, S. Saravanan, G. Jagadeesh and K.P.J. Reddy, Aerodynamic drag reduction using aerospikes for large angle blunt cone flying at hypersonic Mach number. AIAA Paper 2002-2709, 2002.

[13] R. Kalimuthu, R.C. Mehta and E. Rathakrishnan, Drag reduction for spike attached to blunt-nosed body at Mach 6. Journal of Spacecraft and Rockets, 47(1), 219-222, 2010.

[14] Vinayak Kulkarni, Viren Menezes and K.P.J. Reddy, Effectiveness of forward facing spike for drag reduction on a large angle blunt cone in hypersonic flow. Journal of Spacecraft and Rockets, Vol.43, No. 3, pp 542-544, 2010.

[15] G. d'Humieres and J.L. Stollery, Drag reduction on a spiked body at hypersonic speed, Aeronaut J, 114, (1152), pp 113-119, 2010.

[16] M. Fujita and H. Kubota, Numerical Simulation of Flow Field over a Spiked Blunt-Nose. Computational Fluid Dynamics J., Vol.1 (2), pp. 187-195, 1992.

[17] H. Kubota, T. Watanuki, S. Matsumoto, T. Fukui and H. Suenaga, Experimental Study on Flow over Spiked Hemisphere at Komaba Hypersonic Wind Tunnel. J. Faculty of Engineering, Tokyo University (B), Vol. XLII (1), pp. 65-71, 1993.

[18] M. Yamauchi, K. Fujii and F. Higashino, Numerical investigation of supersonic flows around a spiked blunt body. Journal of Spacecraft and Rockets, 32(1), 32-42, 1995.

[19] R.C. Mehta, Numerical heat transfer study over spiked blunt bodies at Mach 6.8. Journal of Spacecraft and Rockets; 37(5):700-703, 2000.

[20] R.C. Mehta, Peak heating for reattachment of separated flow on a spiked blunt body. Heat and Mass Transfer;36 pp:277-283, 2000.

[21] M.B. Gerdroodbary and S.M. Hosseinalipour, Numerical simulation of hypersonic flow over highly blunted cones with spike. Acta Astronautica, 67(1), 180-193, 2010.

[22] M.A. Kenworthy, Study of unsteady axisymmetric separation in high speed flows. PhD dissertation, Dept. of Aerospace and Ocean Engineering, Virginia Polytechnic Inst. And State Univ., Blacksburg, VA., 1978.

[23] A.G. Panaras, Pulsating Flows about Axisymmetric Concave Bodies. AIAA J., Vol.19(6), pp. 804-806, 1981.

[24] J.S. Shang and W.L. Hankey, Flow Oscillations of Spike-Tipped Bodies. AIAA J., Vol. 20(1), pp. 25-26, 1982.

[25] R.C. Mehta, R. C., Numerical Analysis of Pressure Oscillations over Axisymmetric Spiked Blunt Bodies at Mach 6.80. Shock Waves, Vol.11, pp. 431-440, 2002.

[26] D. Feszty, K. Badcock, and B. Richards, Driving mechanisms of high-speed unsteady spiked body flows, part 1: pulsation mode. AIAA Journal, 42(1), pp:95-106, 2004.

[27] D. Feszty, K. Badcock, and B. Richards, Driving mechanisms of high-speed unsteady spiked body flows, part 2: oscillation mode. AIAA Journal 42(1), pp:107-13, 2004.

[28] C.H.E. Warren, An experimental investigation of the effect of ejecting a coolant gas at the nose of a bluff body. Journal of Fluid Mechanics 8 (Part 3). pp 400-417, 1959.

[29] P.A. Libby and R.J. Cresci, Experimental investigation of downstream influence of stagnation point mass transfer, J. Aerospace Sci, Vol. 26 No. 6. pp 51-63, 1961.

[30] D.J. Romeo and J.R. Sterrett, Exploratory investigation of the effect of forward facing jet on the bow shock of a blunt body in a Mach number 6 free stream. NASA TN 1605, 1962.

[31] E.A. Barber, An experimental investigation of stagnation point injection. Journal of Spacecraft and Rockets, Vol. 2. No. 5. pp 770-774, 1965.

[32] P.J. Finley, The flow of a jet from a body opposing a supersonic freestream. Journal of Fluid Mechanics 26 (Part 2) pp 337-368, 1966.

[33] Benjamin Meyer, H.F. Nelson and W.D. Riggins, Hypersonic drag and heat transfer reduction using a forward facing jet. Journal of Aircraft, Vol. 38, No 4, pp 680-686, 2001.

[34] Y.C. Ganiev, V.P. Gordeev, A.V. Krasilnikov, V.I Lagutin, V.N. Otmennikov and A.V. Panasenkp, Aerodynamic drag reduction by plasma and hot gas injection. Journal of Thermophysics and heat transfer, Vol. 14, No 1, pp. 10-17, 2000.

[35] J.S. Shang, J. Hayes, K. Wurtzler and W. Strang, Jet spike bifurcation in high speed flows. AIAA Journal, Vol. 39, No. 6, 2001, pp1159-1165, 2001.

[36] J.S. Shang, J. Hayes and J. Menart, Hypersonic flow over a blunt body with plasma injection. Journal of space craft and rockets, Vol. 39. No. 3, pp 367-374, 2002.

[37] Sahoo Niranjan, Vinayak Kulkarni, S. Saravanan, G. Jagadeesh, and K.P.J. Reddy, Film cooling effectiveness on a large angle blunt cone flying at hypersonic Mach number. Phys. Fluids 17. Art. 036102, 2005.

[38] Balla Venukumar, G. Jagadeesh and K.P.J. Reddy, Counterflow drag reduction by a supersonic jet for a blunt body in hypersonic flow. Physics of fluids, 18, 118104., 2006.

[39] Vinayak Kulkarni and K.P.J. Reddy, Enhancement in counterflow drag reduction by supersonic jet in high enthalpy flows. Physics of Fluids 20, 016103, 2008.

[40] Vinayak Kulkarni and K.P.J. Reddy, Effect of counterflow supersonic jet on blunt body heat transfer rates for oncoming high enthalpy flow. Journal of engineering physics and thermophysics Vol. 8, No. 1, pp 3-7, 2009.

[41] Miller.David.S and Carlson.Harry.W., A study of application of heat or force fields to the sonic boom minimization problem NASA-TND 5582, 1969.

[42] L.N. Myrabo and Y.P. Raizer, Laser-induced air spike for advanced transatmospheric vehicles. 25th AIAA plasma dynamics and lasers conference. AIAA Paper-2451, 1994.

[43] V.A. Levin, V.G. Gromov and N.E. Afonina, Numerical analysis of the effect of local energy supply on the aerodynamic drag and heat transfer of a spherically blunted body in a supersonic air flow. Journal of Applied Mechanics and Technical Physics, Vol 41, No. 5, 2000.

[44] D.W. Riggins, H.F. Nelson and E. Johnson, Blunt body large wave drag reduction using focused energy deposition. AIAA Journal 37, pp.460-504, 1998.

[45] R.G. Adelgren, G.S. Eliot, D.D. Knight, A.A. Zheltovodov and T.J. Beutner, Energy deposition in supersonic flows. 39th AIAA aerospace science meeting and exhibit. AIAA- 2001-0885, 2001.

[46] Y.F. Kolesnichenko and V.G. Brovkin, Microwave energy release regimes for drag reduction in supersonic flows. 40th AIAA Aerospace science meeting and exhibit. AIAA- 2002-0353, 2002.

[47] C.S. Hartley, T.W. Portwood, M.V. Elipelli, L.N. Myrabo and H.T. Nagamatsu, Experimental/computational investigation of drag reduction by electric arc air spikes at Mach 10. 42nd Aerospace science meeting and exhibit. AIAA-2004-35, 2004.

[48] Doyle, D. Knight, Y.F. Kolesnichenko, Vadim Brovkin and Dmitri Khmara, High Speed Flow Control Using Microwave Energy Deposition. 16th Australasian Fluid Mechanics Conference Crown Plaza, Gold Coast, Australia, 2007.

[49] K. Satheesh and G. Jagadeesh, Effect of energy deposition on aerodynamic drag of a blunt body in hypersonic flow. Phys. Fluids 19. Art. 031701, 2007.

[50] J. Bibin, and K. Vinayak, Investigation of Energy Deposition Technique for Drag Reduction at Hypersonic Speeds. In Applied Mechanics and Materials, Vol. 367, pp. 222-227, 2013.

[51] Vinayak Kulkarni, G.M. Hegde, G. Jagadeesh, E. Arunan and K.P.J. Reddy, Aerodynamic drag reduction by heat addition into the shock layer for a large angle blunt cone in hypersonic flow. Physics of Fluids 20, 081703, 2008.

[52] J.P. Reding and D.M. Jecmen, Effects of external burning on spike-induced separated flows, J. Spacecraft and Rockets, 20, 452, 1983.

[53] G.R. Srinivasan and R.R. Chamberlain, Drag reduction of spiked missile by heat addition. AIAA, Atmospheric flight mechanics conference, 4714, 1, 2004.

[54] Jiang Zonglin, Yunfeng Liu, Guilai Han,Wei Zhao, Experimental demonstration of a new concept of drag reduction and thermal protection for hypersonic vehicles. Acta Mech Sin. 25:417-419 Springer-Verlag GmbH, 2009.