CHAPTER 1

INTRODUCTION

Flying like a bird has been a long cherished dream of humans which is slowly but progressively being fulfilled as can be observed in the history of evolution. The quest for flying higher and faster is the new unending challenge. A rapid progress in this regard was seen during Second World War onwards when V–2 became the first manmade object to fly at hypersonic speed in 1949. V-2 was a two stage sound rocket by the Nazi Germany. The second stage of V-2, separated after thirty seconds, could attain a speed of 16099.344 m/s. After the removal of second stage, the slender upper stage of V-2 accelerated to a speed of 2303.25 m/s up to an altitude of 392.67 km. This was a land mark success in the field, the first man made mission which flew five time faster than the speed of sound and could attain a reasonable height. The first U. S rocket launched from White Sand Proving Ground in New Mexico could reach a height of 99.78 km, above Karman line that divides the earth atmosphere and space. April 12, 1961 was another historical day in the aerospace field. The Russian Cosmonaut Yuri Gagarin made his successful 108-minute orbital flight by using the Vostok 1 spacecraft. The landing of the re-entry capsule was success but the outer surface become black in colour due to partial burning of the small ports covered with thermal insulation. Vostok 1 flew with a weight of 4725 kg, which excluded the weight of the final stage of carrier rocket and attained a 302 km distance at apogee and 175 Km distance at perigee with the inclination of the orbit plane to equator is $65^0\ 4^{'}$. Less than a month later astronaut Alan Shepard became the first U. S pilot in space with Mercury attaining at an altitude of 122.97 km with re-entry speed of more than five times the speed of sound. In the subsequent year 23rd June 1961, Major Robert White of US Airforce flew the first X-15 Flight which exceeded Mach number 5, recording a maximum velocity of 1610.68 m/s. This was further extended by Major White himself in the same year by flying X-15 at Mach number 6 on 9th November 1961.

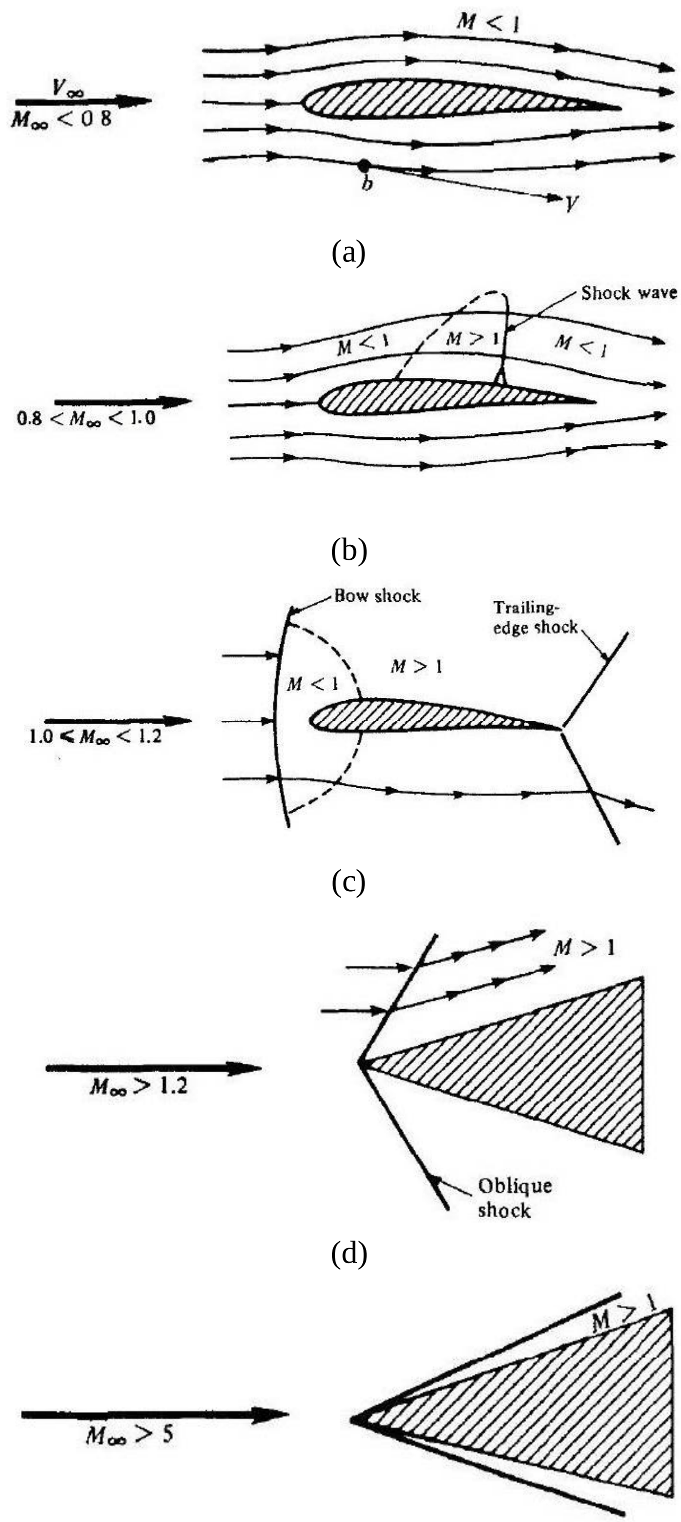
Although the hypersonic flight seems to be an established practice from these historical achievements, the challenges associated with it are paramount and still an area of major concern. The inability of conventional supersonic aerodynamic theories to explain the complex flow fields associated with hypersonic flows had been the major hindrances in the development of hypersonic vehicles for decades. With the advent of high speed digital computers and hypersonic ground test facilities, the hypersonic research is at the centre stage once again. Thus the new age of hypersonic flight regime has begun, and the challenges are abundant to be overcome step by step towards a faster and higher flight of the human race.

1.1. Aerodynamic flow Regimes

Aerodynamics plays a major role in the design of the aerospace vehicle. The design of any flight vehicle is governed by the Mach number regime of flow field in which the object is flying. When speed of sound is less than the speed of flying object i.e. when freestream Mach number M_∞< 1, the flow in the regime is termed as subsonic flow, when the object is flying at a range 0.8≤ M_∞ ≤1.2, which is close to the speed of sound, the flow in the regime is termed as transonic flow, when the object flying at a range 1.2≤ M_∞ ≤5, the flow in the regime is termed as supersonic flow, and when M_∞ > 5 the flow is referred as hypersonic flow. When the freestream Mach number is slightly less than unity, i.e. M_∞ ≤0.8, the density changes are significant but the shock wave does not appear and the flowfield everywhere remains subsonic and the flow in this regime is referred as subsonic flow as shown in Fig. 1.1a [1]. When the freestream Mach number increases, but remains close to unity i.e. in subsonic regime, the flow expands over the airfoil and a region of local supersonic flow first experiences on the surface of the airfoil. The local Mach number is either less than or more than unity i.e. a mixed regime of flow is considered called a sonic pocket as shown in

Fig. 1.1b. If the freestream Mach number is increased to slightly above unity, the shock will move towards the trailing edge of the airfoil and a second shock wave appear in the leading edge of the airfoil, which is known as bow shock as can be seen in Fig. 1.1c. Ahead of the bow shock, the streamlines are straight and parallel with a uniform supersonic freestream Mach number, just behind the shock the flow over the airfoil becomes subsonic and it further expands to supersonic value and finally terminates with trailing edge shock in the downstream. These types of regime where shock wave appears and divides the regime into subsonic and supersonic flow are referred as transonic flow. Thus, transonic flow is characterized by mixed regions of subsonic and supersonic flow such that $0.8 \leq M_\infty \leq 1.2$ as shown in Fig. 1.1b and c.

When the speed everywhere in the flowfield is above the speed of sound, then it defined as supersonic flow. The supersonic flowfield is characterized by a straight oblique shock which is attached to the sharp nose of the wedge. The streamlines are straight, horizontal and parallel. Behind the oblique shock, the streamlines remains straight and parallel but takes the direction of wedge surface. The flow is supersonic both upstream and downstream of the oblique shock as shown in Fig. 1.1d. When the freestream Mach number increases to a higher supersonic speed exceeding almost five times the speed of the sound, the pressure, density and temperature increases explosively across the shock. The oblique shock moves very closer to the body surface as shown in Fig. 1.1e. Because of the formation of shock wave very close to the body for $M_\infty > 5$, the flowfield between the body and shock becomes hot enough to ionize the gas. These effects of thin shock layer aerodynamic heating add more complexity to the analysis of the flowfield. Such complex region of flow for $M_\infty > 5$ is specially characterized as hypersonic flow. The the aerodynamic performance of a flying body is governed the flowfield and therefore the forces and aerodynamic heating experienced by the body is different in the different regions of flow.



(e)

Fig. 1.1: Different regimes of flow [1]

1.2. The Hypersonic Flowfield

In the hypersonic flight regime, the phenomena of shock layer, viscous interactions, entropy layer, high temperature gradients, low density flow effect becomes significant. The formation of shock waves, in such flows, results in sharp increment in drag due to the discontinuity jump in pressure across it. Hence, sufficient power and specific design is required to overcome the pressure barrier. With increasing speeds, the effect of the huge pressure gradient across the shock becomes stronger and stronger, resulting in an increase in temperature, entropy and density. The aerodynamic phenomenon related with very high speed flow is physically different from supersonic flow. High speed flows often contribute to the thickening of boundary layer. The boundary layer thickness as well as the shock layer thickness are of same magnitude, since the shock layer is fully viscous, the shock wave and the surface pressure is fully affected by such viscous effects. The other prominent aspect of hypersonic flow is the high temperature in shock layer along with large aerodynamic heating of the vehicle. Thus the thin shock layer viscous interaction and high temperature flows distinguishes hypersonic aerodynamics from the more moderate supersonic aerodynamics.

The details of the physical phenomena associated with hypersonic flows are discussed below.

1.2.1. Thin Shock Layer

The shock layer is the region of flowfield between the flight vehicle body and shock wave, which is quite thin for hypersonic flows. It is quite clear from $\theta - \beta - M$ relation that with an increase in Mach number the shock angle decreases, i.e. the thickness of the shock angle decreases with increase in Mach number for the same wedge angle or flow deflection angle. Thus the hypersonic flow can be distinguished from supersonic flow by the thinness of the shock layer. Higher density exists across the shock wave which further becomes more progressive with the increase of Mach number, and a constant mass flow rate can be attain only by squeezing the mass flow behind the shock through a smaller area. This brings the oblique shock closer to the body making the shock layer quite thin. This is one of the prominent characteristic of the hypersonic flow that the distance between the shock and the body is highly reduced and it leads to a thin shock layer as shown in Fig. 1.2. Thus hypersonic flow becomes more complex as the shock wave merges with a thick viscous boundary layer [2]

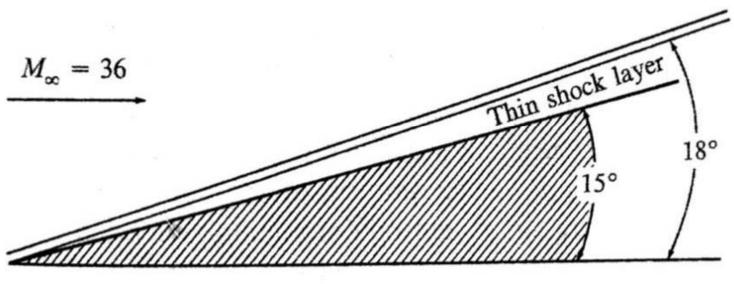


Fig. 1.2: Thin shock layer [2]

1.2.2. Entropy Layer

A very thin shock layer is formed at hypersonic speeds and the shock detachment distance is also very small for blunt bodies. The shock formed ahead of the blunt nose is highly curved wherein each streamline passing through the shock experiences a different entropy rise. Streamlines experience large entropy difference as it passes through the normal portion than the weaker portion of the oblique shock wave. Hence near a nose section a strong entropy gradient exist, which trails downstream and wets the body for larger distances from the nose as shown in Fig. 1.3 [2]. Therefore a layer of entropy variation forms downstream of the shock and is termed as entropy layer, where the boundary layer growth happens inside of it due to the wetting generated by the same. Entropy layer is a region strong vorticity as per Crocco's theorem as well as a region of high viscous interactions. Since for blunt bodies the shock detachment distance decrease with increase in Mach number, hence a high entropy gradient exists in the entropy layer which gives rise to higher vorticity at higher Mach number. Therefore it is an important aspect of hypersonic flow to predict the boundary layer properties due to interaction of the entropy layer vorticity and boundary layer vorticity. This Interaction is also termed as vorticity interaction.

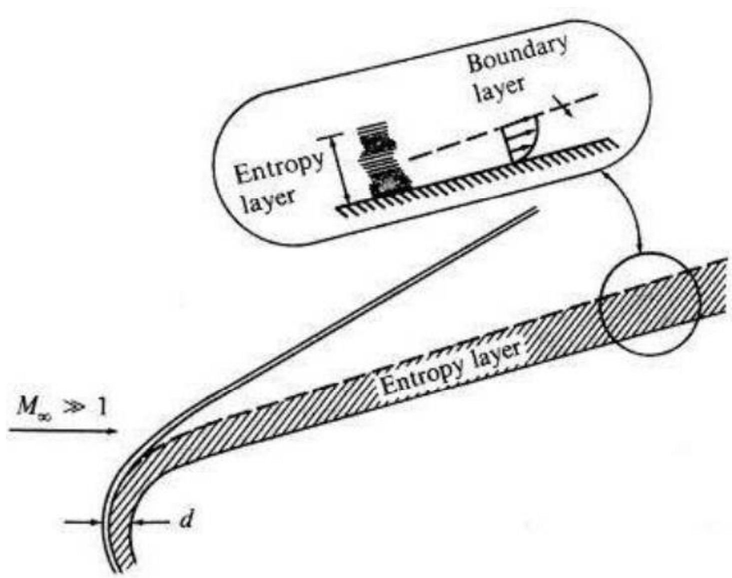


Fig. 1.3: Entropy layer [2]

1.2.3. Viscous Interaction

The development of the boundary layer takes place near the wall due to no slip condition of the viscous fluid flow. The hypersonic flow contain large amount of kinetic energy and when this comes in contact with surface, because of the viscous effect the flow, velocity gradually decreases which results in the enormous loss in kinetic energy. This loss is transmitted into internal energy which gives rise to increase in temperature of the flow near the surface. Fig. 1.4 shows a typical temperature profile within a boundary layer and the phenomenon is known as viscous dissipation. This phenomena give rise to increase in the boundary layer thickness as the coefficient of viscosity increases with increase in temperature which itself make the boundary layer thicker. Also as per boundary layer theory the pressure almost remains constant in the normal direction of the boundary layer and the corresponding increase in temperature results in decrease of density. For a mass flow to pass through boundary layer at reduced density, thickness of boundary layer must be large. This thickened boundary layer displaces the effect in inviscid flow outside boundary layer and hence the hypersonic flow encounters an object with much thicker than it really is. Because of extreme thickness of the boundary layer, the outer inviscid flow is greatly changed, therefore the growth in the boundary layer is affected by the changes in the inviscid flow. The interaction between the boundary layer and the outer inviscid flow is termed as viscous-inviscid interaction. This becomes important as the shock layer fully tends to becomes viscous in nature due to this interaction [2].

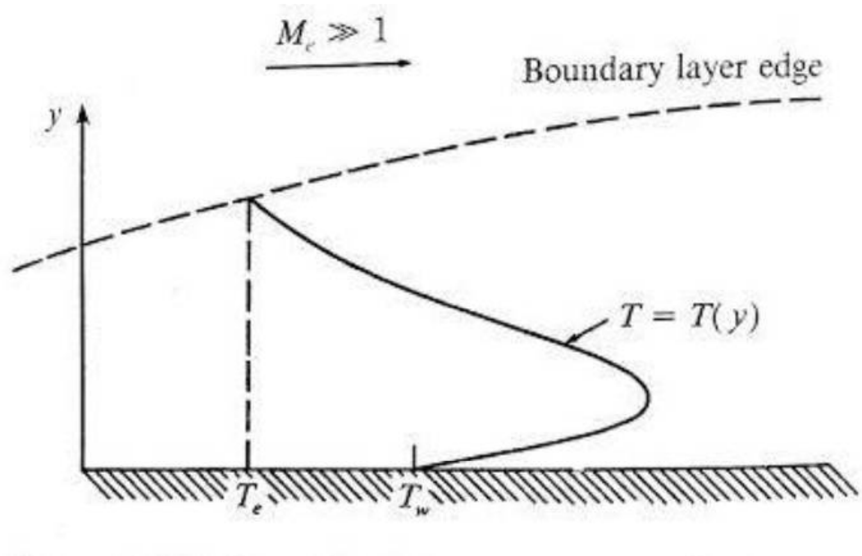


Fig. 1.4: Temperature profile [2]

1.2.4. High Temperature Flow

In hypersonic flows, the viscous dissipation can leads to higher temperatures and boundary layer thickness of the fluid. Thus any hypersonic flight experiences the high temperature flied in the surrounding area of the flight vehicle. Apart from this, blunt nose configurations encounter very high temperatures due to normal shock present at the stagnation point which can cause large aerodynamic heating at the leading edge. For re-entry vehicles the temperature due to viscous effects and shock layer interactions are very high that make the value of γ no longer to be constant. At these elevated temperatures the ionization of gases takes place and it leads to plasma state in front of the nose cone. For re-entry bodies such as ICBMs at high velocities convection is the only possible meaningful mode of heat transfer, at the same time at high temperature radiation heat transfer is dominant because it transfers heat proportional to fourth power of temperature. Fig. 1.5 shows the stagnation region at the nose of the blunt body wherein it can be observed that the bow shock is almost normal at the nose region , and a very high temperature exist behind the shock wave at hypersonic speed.

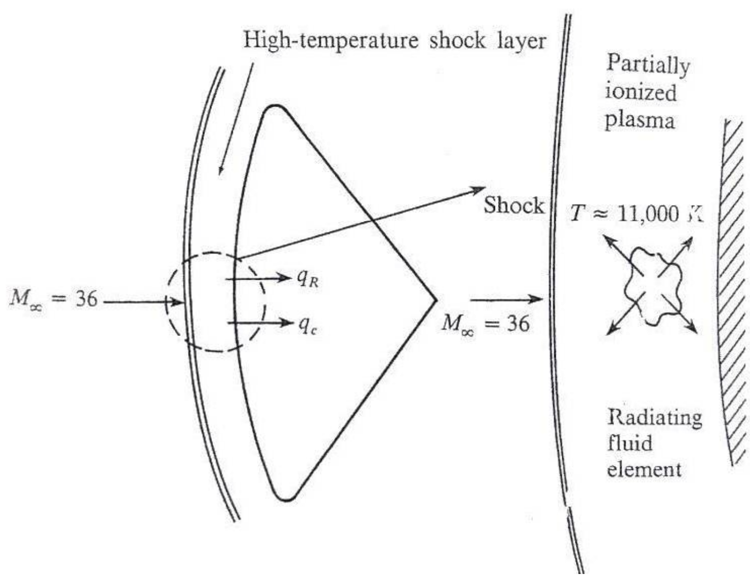


Fig. 1.5: High temperature shock layer on a blunt body [2]

1.2.5. Low Density Flows

Hypersonic flights generally occur at high at higher altitudes where very low density flows are encountered. The similarity parameter that governs these different regimes is the Knudsen number, which is defined as the ratio of the mean free path to the characteristic length of the object ($K_n = \lambda/_L$), where λ is the mean free path and L is the characteristic dimension of the body. The mean free path is termed as the mean distance traveled by the fluid molecule between two successive collisions with other molecules. Since density of the air is very high on the earth's surface, therefore Knudsen number is close to zero. Hypersonic flight taking off from the earth surface encounters change in the density with increase in altitude. Therefore up to 90 km from the earth surface the use of continuum flow and the use of usual governing equations remain applicable where the Knudsen number is below 0.3. From 90 km to 150 km the density becomes lower as an effect of which the fluid velocity and temperature at the surface doesn't remain in equilibrium with the surface. Therefore flow for Knudsen number in range 0.3 to 1 is treated in the transitional regime where slip wall boundary conditions should be used along with the usual governing equations based on continuum assumption. However above 150 K from earth's surface density of air becomes very low therefore this regime is called as free molecular flow where Knudsen number becomes more than or equal to unity. This flow regime is governed by the kinetic theory of gases where the continuum hypothesis breaks. [2]

Therefore the hypersonic flow is best defined as that regime where all or some of the preceding physical phenomena become important as the Mach number is increased to high values. Fig.1.6 elaborates the important characteristics associated with hypersonic flights.

1.3. Forces and Heat Transfer at Hypersonic Speeds

Sir Isaac Newton postulated a fluid dynamics theory in 1687, wherein fluid flow can be modelled as a rectilinear motion as shown in Fig. 1.7. Newton derived a "law" that that force on any object offered by the fluid flowing is proportional to the square of the sine of the deflection angle, given by equation 1.1.

$$F = \rho_\infty V_\infty^2 A \mathrm{Sin}^2(\theta) \quad 1.1$$

Newton proposed the model for low speed flows but it was found to hold good for hypersonic flows. The famous sine-squared law for pressure coefficient as postulated by Sir Isaac Newton is given by equation (1.2),

$$C_p = 2\mathrm{Sin}^2\theta \quad 1.2$$

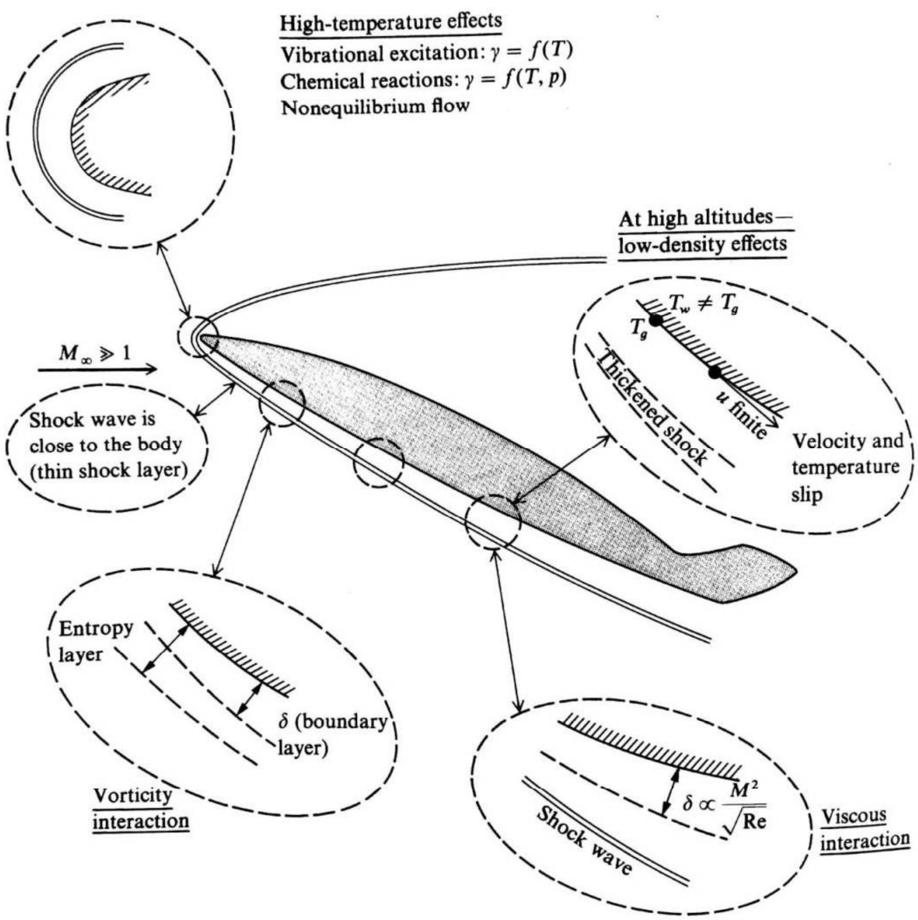


Fig. 1.6: Characteristics of hypersonic flow [2]

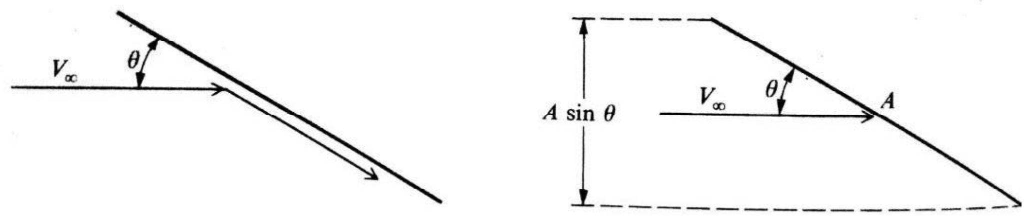


Fig. 1.7: Schematic of the hypersonic flow over flat plate [2]

where θ is the local deflection angle and C_p is the local pressure coefficient. This relation holds good for the coefficient of pressure at the stagnation point and for a large Mach numbers. Later Lester Lees [4] proposed a correction to the C_p values given by the Newtonian theory, as given by equation 1.3.

$$C_p = C_{p\,max}\ \mathrm{Sin}^2\theta \qquad 1.3$$

where $C_{p\,max}$ is the maximum coefficient of pressure at the stagnation point located on the back of normal shock wave which is given by equation 1.4

$$C_{p\,max} = \frac{2}{\gamma M_\infty^2}\left\{\left[\frac{(\gamma+1)^2 M_\infty^2}{4\gamma M_\infty^2 - 2(\gamma-1)}\right]^{\gamma/\gamma-1}\left[\frac{1-\gamma+2\gamma M_\infty^2}{\gamma+1}\right]-1\right\} \qquad 1.4$$

Lester Lees in 1956 [5] studied the effect of laminar heat transfer rate over blunt-nosed cone bodies at high stagnation temperature or at hypersonic flight speeds. Lees developed a relation between the stagnation point heat transfer rates and wall heat transfer of the body. The generalized Lee-Dorodnitsyn transformation for hypersonic laminar boundary layer as suggested by Lees can be given by Equations 1.5 and 1.6.

$$\xi = \int_0^x \rho_e\, u_e \mu_e dx \qquad 1.5$$

$$\eta = \frac{u_e}{\sqrt{2\xi}} \int_0^y \rho dy \qquad 1.6$$

where $\rho_{e\,,}$ u_e and μ_e are the coefficients of density, velocity and viscosity respectively at the edge of the boundary layer and x, y and ξ, η are the coordinates in physical and transformed planes respectively.

For an axisymmetric body, the above Dorodnitsyn transformation is slightly modified as per Equations 1.7 and 1.8,

$$\xi = \int_0^x \rho_e\, u_e \mu_e r^2 dx \qquad 1.7$$

$$\eta = \frac{u_e r}{\sqrt{2\xi}} \int_0^y \rho dy \qquad 1.8$$

with r as the vertical coordinate measured from the centreline. This leads to an axisymmetric compressible stagnation point boundary layer as given by Equations 1.9 and 1.10.

$$(Cf'')' + ff'' = \frac{1}{2}[(f')^2 - g] \qquad 1.9$$

$$\left(\frac{C}{Pr} g'\right)' + fg' = 0 \qquad 1.10$$

The expression for the heat transfer rate for axisymmetric bodies in hypersonic flow follows from the solution of above equations which for a spherical body is given by Equation 1.1.

$$q_w = 0.763\, Pr^{-0.6} (\rho_e \mu_e)^{0.5} \sqrt{\frac{du_e}{dx}} (h_{aw} - h_w) \qquad 1.11$$

where the velocity gradient,

$$\frac{du_e}{dx} = \frac{1}{R}\sqrt{\frac{2(P_e - P_\infty}{\rho_e}} \qquad 1.12$$

This states that heat transfer at stagnation point is inversely proportional to square root of the nose radius. Hence to reduce heating, the nose radius should be increased.

$$q_w \propto \frac{1}{\sqrt{R}} \qquad 1.13$$

1.4. Need for Blunt Body

While designing a hypersonic vehicle flow, the reduction in aerodynamic drag heating poses a great challenge. The increase in the drag indicate substantial effect in the performance of flying vehicle, due to formation of shock wave which results in high pressure region in front of the fore body. Aerodynamic heating becomes crucial for hypersonic vehicles as it can lead to structural disintegration or deformation which can lead to aerodynamic and stability problem and prove to be complete disaster. The large aerodynamic heating at hypersonic speeds is primarily due to thinness of the shock layer as can be seen in Fig. 1.8. Initial hypersonic researches lead to the development of sharp pointed slender body, which unfortunately those went in vain since the sharp pointed slender body produced excess heating aerodynamically and eventually burnt down in the Earth's atmosphere itself. Fig. 1.8 displays how the frictional boundary layer near the surface, transfers heat to the body and also demonstrates the attachment of oblique shock wave for a sharp point slender body.

When a vehicle is re-entering to Earth’s atmosphere the large amount of potential gets converted into kinetic energy which is further dissipated into air as thermal energy due frictional traction at the surface and due to stagnation at the nose. The part of the heat dissipated heats up the vehicle and the airflow around it. In a slender body with pointed nose, the shock wave generated is very thin and very close to the body, the temperature rise to the extreme value near the stagnation point, which may result in malfunctioning of important electronics payload and may even cause permanent structure failure if the material cannot withstand the amount of heat. On the other hand for a blunt body at hypersonic speeds, a strong bow shock wave is formed ahead of body by which is at a relatively large distance from the body, and reduces the aerodynamic heating over body as most of the heat is lost to airflow around it. Fig. 1.9 shows the clear advantage of using a blunt body over a slender body at hypersonic speeds.

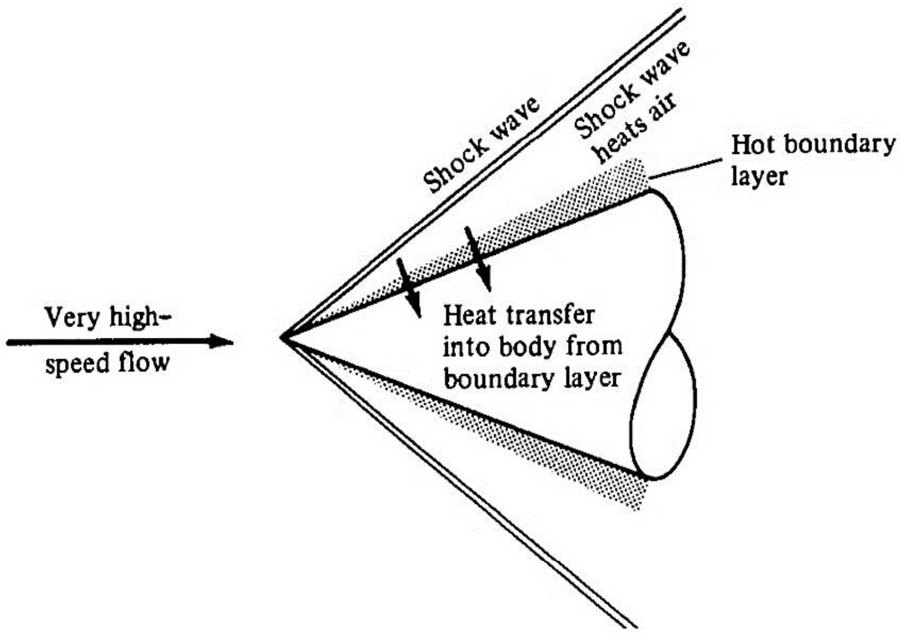


Fig. 1.8: Aerodynamic heating of body and air around it [1]

The fact that a blunt nose reduces aerodynamic heating is also established by Equations 1.11, 1.12 and 1.13. A blunt body provides a larger heat transfer area to airflow which reduces heating to the body as compared to sharp point slender body but results in an increase in drag due to large pressure in the stagnation region. A study of various nose cone designs with different bluntness ratios, as shown in Fig. 1.0, has been investigated for a freestream Mach number of 2.72 at the Langley Gas Dynamic laboratory in 1952. The bluntness ratio is defined as the ratio of the hemispherical nose radius (r_n) to the base radius of the model (r_b) i.e. $^{r_n}/_{r_b}$.

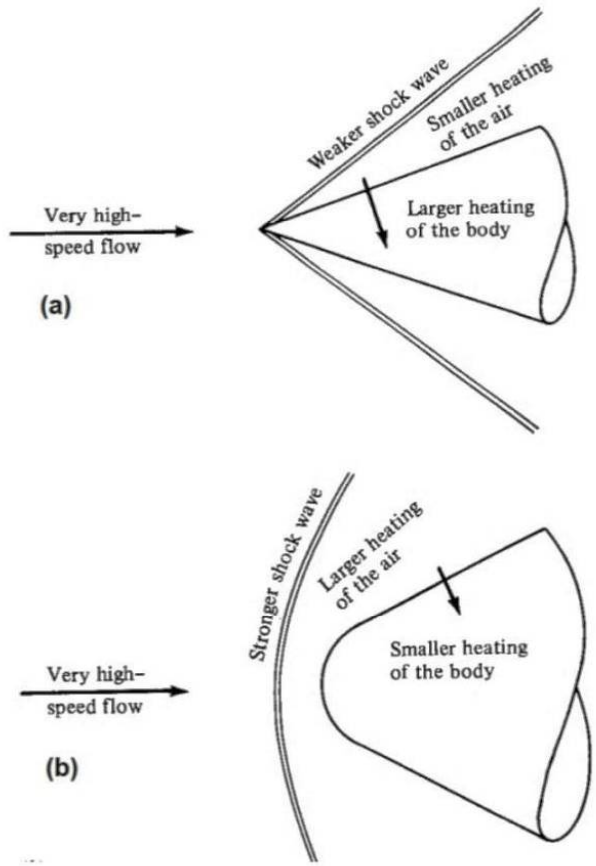


Fig. 1.9: Comparison of blunt and slender body [1]

The variation drag coefficient with the bluntness ratio, as reported in reference 7, is depicted in Fig. 1.11. As can be seen in Fig. 1.11, the maximum drag is observed for the most blunted nose and as the nose radius is reduced a corresponding reduction in drag can be observed. It can also be observed that if the nose radius is less than one quarter of the maximum radius of the base body, the increase in the drag is not severe. The reason for the increase in drag as the nose radius increases is that the compression provided by the surface becomes large as bluntness is increased. For re-entry vehicles a blunt nose is essential so as to reduce the aerodynamic heating which results in large pressure drag due to detached shock wave.

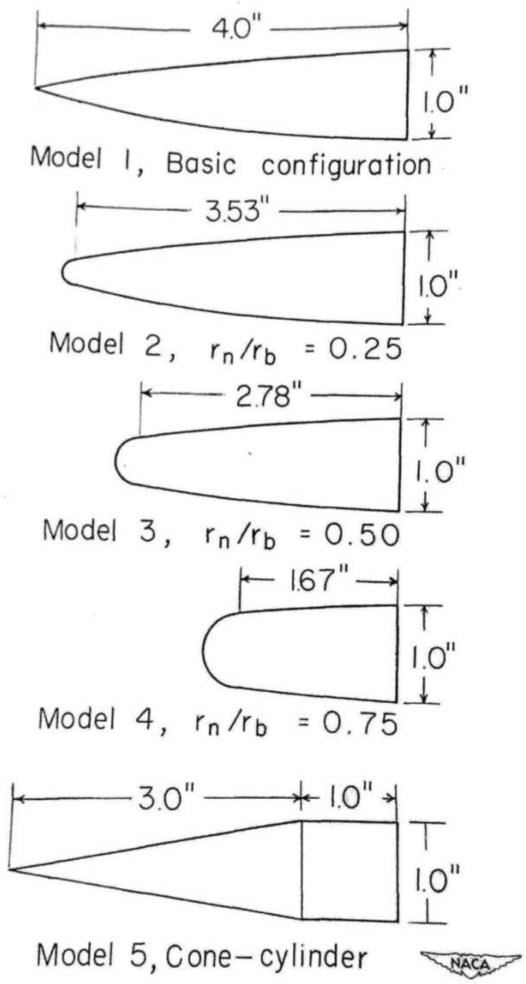


Fig. 1.10: Sketches of models with different bluntness ratio [7]

Modifying the high pressure region ahead of the blunt body can favourably reduce the high pressure drag as well as aerodynamic heating for blunt vehicles. Many studies on the reduction of wave drag and aerodynamic heating for blunt bodies have been reported in open literature. These methods are broadly classified as active and passive methods.

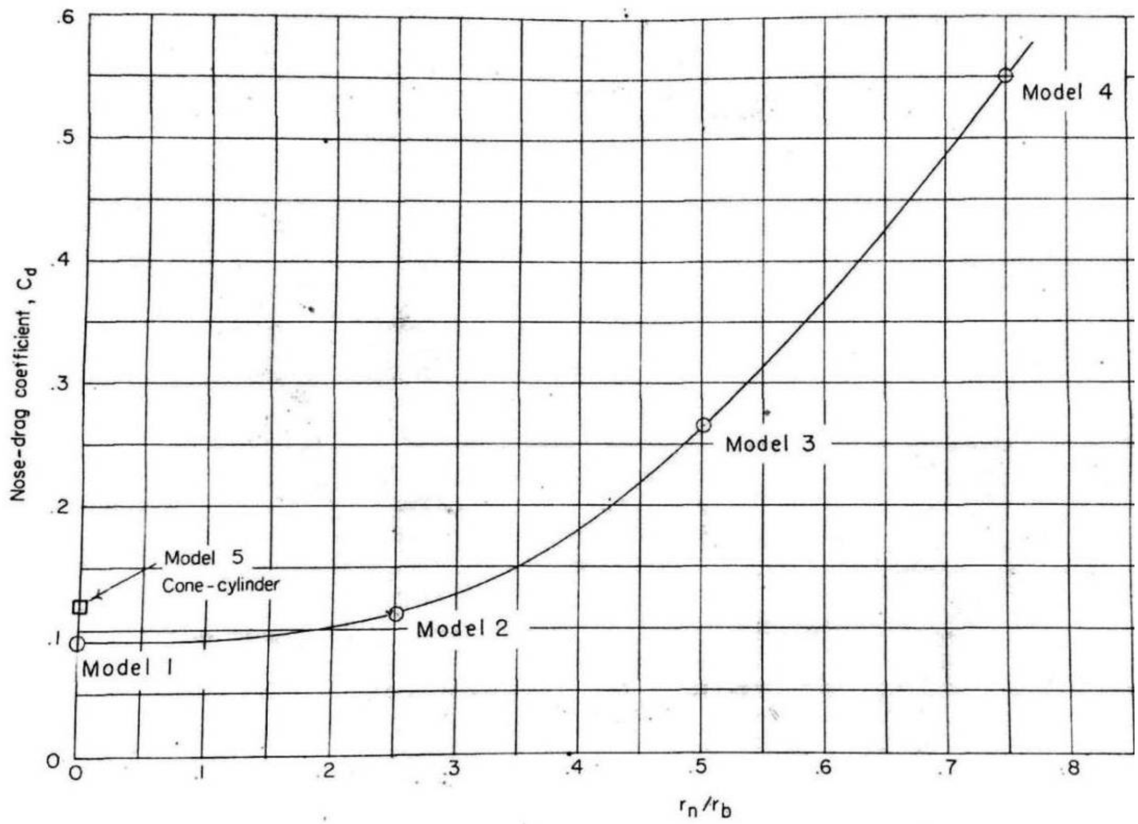


Fig. 1.11: Effect of nose radius on drag coefficient [7]

Active methods rely on the expenditure of extra energy to modify the flowfield whereas passive methods use geometric modifications. One such method is the use of aerospike – a passive method control which is the subject of the current research. Aerospike is a thin cylindrical rod protruding from the stagnation point of a blunt nosed body. Sometimes, a hemispherical or a flat circular disk is attached at the tip of the aerospikes so as to enhance its efficiency. It is then referred to as aerodisks. Besides aerospikes, there are many other methods that have been widely investigated for reduction of aerodynamic heating and wave drag. Few of these mechanisms are explained below.

1.5. Mechanisms of Heat and Drag Reduction

The various mechanisms for reduction of aerodynamic heating and drag can be broadly classified into two groups, viz. active and passive mechanisms. Both these mechanism alter the flowfield ahead of the nose of a blunt body in a favourable way so as to reduce one of the two menaces of hypersonic flows. The studies are mainly focused to reduce these effects so as to make hypersonic vehicles reusable and cost effective.

1.5.1. Active Flow Control Methods

Active methods are those mechanisms that work from the expenditure of energy from the external source, which is an auxiliary power unit that can be used to control the drag on the body. One of the popular active mechanisms is a forward facing counter jet wherein a pressurized mass of air, gas or some coolant is ejected out from the stagnation point of the nose. The ejected mass takes a large chunk of dissipated heat and keeps the blunt body cooler. Ejection mass also creates a recirculating flow ahead of the blunt nose which reduces the drag of the blunt body. Recently, many investigations combining various mechanisms to reduce aerodynamic heating and drag have been reported.

1.5.2. Combination of Counter Flowing Jet And Forward-Facing Cavity

In this method a counter flowing jet is introduced through a forward facing cavity. Forward facing cavity is a passive mechanism which acts by creating a recirculating zone in the stagnation region. As the bow shock wave is distant from the stagnation region a cool ring is formed with reduced stagnation point heat transfer rates. The forward facing cavity however does not affect the drag of the blunt body and is also associated with large local heat fluxes at the lips. The combination of forward facing cavity and counter-flowing jet adds the benefits of both mechanisms as shown in Fig. 1.12.

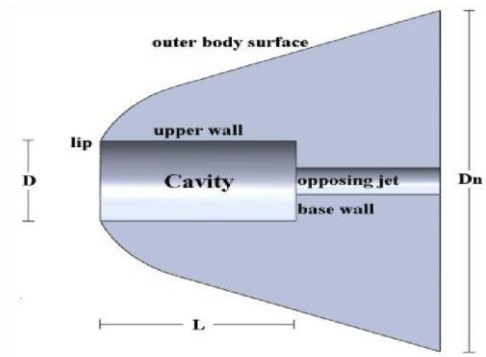


Fig. 1.12: Combined mechanism of counter flowing jet and forward-facing cavity

1.5.3. Combination of Counter Flowing Jet and Energy Deposition

In this mechanism, energy is deposited upstream in the front of blunt body along with a counter flowing jet. The energy deposition is done wither by heating the flow upstream through a microwave or by an electric discharge. This combination helps to settle down the instability occurring during the introduction of counter flowing jet alone and also enhances the stability of penetration of counter flowing jet is shown in Fig. 1.13.

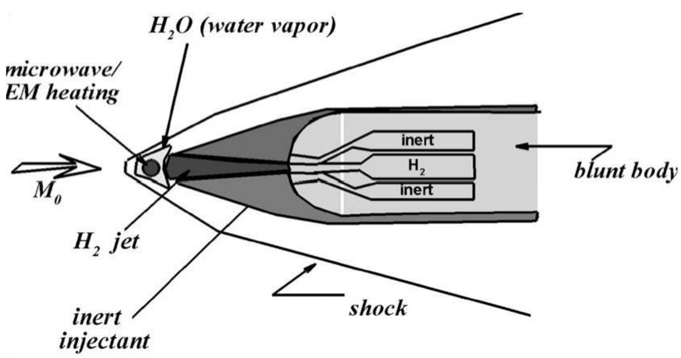


Fig. 1.13: Schematic representation of the combination of counter flowing jet and energy deposition [8]

1.5.4. Combination of Counter Flowing Jet and Aerospike

The aerospikes or aerodisks efficiently reduces the aerodynamic drag but fails to address the aerodynamic heating problem. This method combines the effectiveness of forward facing jets to avoid few drawbacks of aerospike such as high local temperature experienced at the aerospike nose due to small radius of curvature, surface heat flux at blunt body shoulder, generation of high heat flux area at angle of attack on body as can be seen in Fig. 1.14.

1.5.5. Passive Flow Control Methods

In passive methods, the flowfield ahead of the blunt body is modified favourably without spending any additional energy. This is achieved by modifying the geometry of the nose cone slightly with the help of some protuberances or some concavities. Two of the most popular passive flow control devices used for hypersonic flight, are the forward facing cavity and aerospikes. Forward facing cavity acts by creating a recirculating zone in the stagnation region.

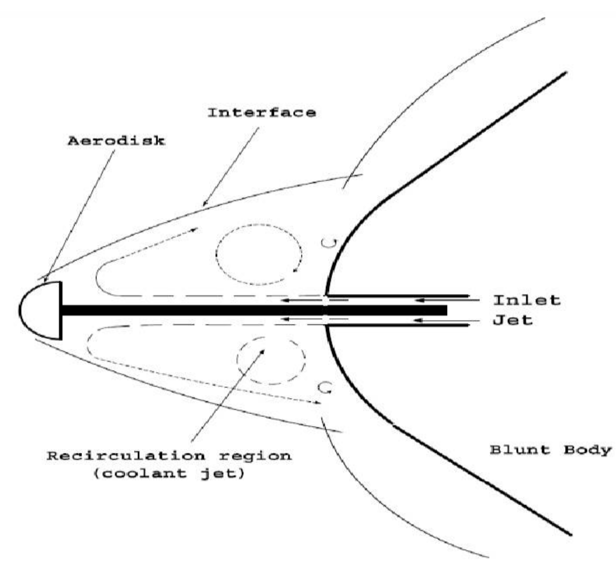


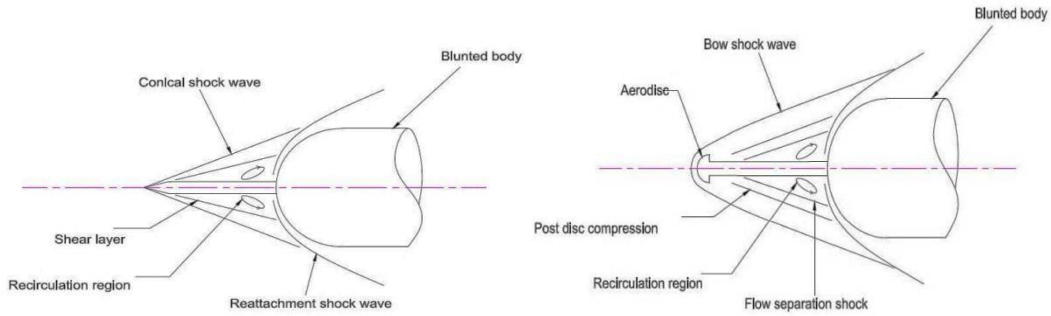
Fig. 1.14: Schematic representation of the flowfield properties around the aero-disked blunt cone with counter flowing jets [8].

As the bow shock wave is distant from the stagnation region a cool ring is formed with reduced stagnation point heat transfer rates. Aerospikes on the other hand is primarily meant for reducing the pressure drag by creating a region of low pressure ahead of the blunt nose.

1.6. Need for a Multidisk Aerospike

Hypersonic vehicle with a blunt nose experiences a detached bow shock and the total drag associated with a blunt body is mainly because of high pressure zone ahead of the blunt nose. Therefore, reducing the high pressure ahead of the body by modifying the flowfield can substantially reduce the drag. This can be achieved if the detached shock can be shifted away from the nose by placing a thin cylindrical rod called Aerospike. An aerospike replaces a strong bow shock into a weaker shock creating a large recirculation region along its length. This reduces the dynamic pressure in the recirculating region which contributes to the reduction in both aerodynamic drag and heating. As can be seen in Fig. 1.15, both the aerospike and aerodisk creates a region of recirculating flow in front of the main body. The pressure and temperature in the recirculating zone is greatly reduced, which leads to decreased heat loads and aerodynamic drag.

An aerodisk is an aerospike with a blunt nose mostly hemispherical or circular in shape as shown in Fig. 1.15b. For the conical spikes shown in Fig. 1.15a, the shock wave of conical shape originates at the spike tip and creates an adverse pressure gradient along the spike length. The flow along the spike stem detaches with a detachment shock wave. Both the primary shock wave and the detachment shock wave merge before striking the shoulder of the main body. The flow between the detachment shock wave and reattachment point on the main body is a recirculating fluid zone with reduced pressure and temperature as can be seen in Fig. 1.15 a. With an aerodisk of same length the primary shock wave is not attached to the aerodisk and the oblique portion of the detached shock wave is inclined at a lower wave angles to the flow, resulting in a weaker shock wave for major portion.



(a) Conical Aerospike (b) Hemispherical Aerodisk

Fig. 1.15: Flowfield around Aerospike and Aerodisk [9]

This results in much lower temperatures and pressure in the recirculating zone. The also means an improved ability of the aerodisk to reduce aerodynamic drag and heat transfer rated. The extent, shape and size of the recirculating regions depend upon the geometrical parameter of the aerospike and a generally larger recirculating zone is associated with aerodisks as compared to aerospikes.

Many researches in past have addressed the hypersonic drag reduction with the use of aerospikes or aerodisks. It has been established that both aerospikes and aerodisks reduce the wave drag substantially, but there is a lot of confusion regarding the heat reduction capabilities of aerospikes or aerodisk. Many researchers have established that for turbulent floes the aerodynamic heat transfer rates to the main body is increased detrimentally by the use of aerospikes. It is this factor that the practical usage of aerospikes is limited. As such, the current research aims at providing an alternate modification to the existing aerodisk mechanism so as to address both the aerodynamic heating and wave drag problems optimally. In this regard, multidisk aerospikes have been proposed and thoroughly studied numerically. This research also take opportunity to address other less reported problems of the spiked blunt bodies such as the effect of aerodisk size on the aerodynamic drag and heat transfer rates. It is obvious that size of aerodisk significantly modifies the flowfield ahead of the blunt body and hence affects the aerodynamic parameters. For a given *l/D* ratio and size of the front aerodisks, the intermediate location of the rearward aerodisk can have a significant effect on the flowfield ahead of the blunt body. So the effects of most of the geometrical parameters of multidisk aerospikes have been studied in this research.

1.7. Organization of the Thesis

The current chapter has highlighted the various aspects of hypersonic flowfield and introduced the existing concepts for heat and drag reduction at hypersonic speeds. The classical analytical and empirical formulations for hypersonic flow computations have also been presented so as to familiarize the reader with the subject.

In Chapter 2, a literature review of past work is done so as to explore the state of the art on the aerodynamics of spiked blunt bodies at hypersonic speeds. This includes a large spectrum of journal papers, research memoranda and technical notes related to experimental, analytical and numerical analysis of spiked blunt bodies at supersonic and hypersonic speeds. Chapter 3 provides the motivation behind choosing the current subject for research and also highlights the key objectives of the research based on the literature review in Chapter 2.

In Chapter 4 the numerical tools that forms the CFD methodology used in this investigation is explained in detail. This includes the physical, mathematical and numerical model of the fluid dynamic problem under investigation along with the pertinent numerical procedure.

Chapter 5 presents detailed numerical results for the work carried out in the research. This includes discussion on the flowfield structure around multidisk aerospikes and its effect on aerodynamic heating and drag.

Finally, in Chapter 6 the main outcomes of the current research are outlined along with recommendations for future work directions.