Parametric Study of Hypersonic Flow over Different Bi-Cone Configurations

Vishal S¹, Varun V Nair¹, Peri Sai Teja¹, Sangeeth Rajan¹, J. Akhil¹, Nagaraja S R¹,²
¹Department of Mechanical Engineering, Amrita University, Bengaluru, India
²e-mail: sr_nagaraja@blr.amrita.edu

Abstract. A series of numerical simulations are conducted on bi-cone shaped axisymmetric geometries using ANSYS FLUENT commercial code. Eight bi-cone geometries were simulated with first cone half-angle fixed at 25° and varying the second cone half-angle varying from 10° to 45° with an increment of 5°. The simulations were conducted in hypersonic flow regime at three different Mach numbers of 6, 8, 10 with geometries approximated as 2-D axisymmetric. Shock interactions observed essentially fall in the TYPE VI and TYPE V category. The simulations accurately predicted these shock interactions, the size of flow separation regions, the surface pressure, and density variations across the geometries considered. It is observed that when the second cone angle is greater than the first cone angle larger values of pressure and heat flux were encountered at flare angle whereas when the second angle is less than the first cone angle, smaller values of pressure and heat flux were encountered at the second cone.

1. Introduction

Hypersonic flow finds applications in diverse fields from ballistic missiles to atmospheric re-entry vehicles. The higher temperatures and adverse pressure gradients encountered in these flows renders these flows difficult to predict. In hypersonic flow, a blunt nose is preferred over a pointed tip, as the latter produces an attached shock, heating the tip close to the stagnation temperature of the flow. But a blunt nose produces a separated shock. This increases the drag and heat flux over the body. However, the heat flux is spread over a larger area. The objective of the design is to create a hypersonic geometry with low aerodynamic heating and minimum drag. A detached shock wave in front of the blunt nose at hypersonic speeds converts all the kinetic energy into enthalpy or thermal energy. Heat transfer rate near the stagnation point depends on the magnitude of velocity gradient along the body surface as studied by Lees [1]. Santos [2] found significant differences between edges on the flow field structure and on the aerodynamic surface quantities. The work also found that the upstream effects have different influence on velocity, density, pressure and temperature along the stagnation streamline ahead of the leading edges. Changes on the leading-edge shape disturbed the flow field upstream and its domain of influence decreased as the leading edge became sharp. The domain of influence for temperature was found to be larger than that observed for pressure and density. A modified geometry of a simple cone is a double cone as used by Wright et al [3] in their study of shock interactions. They have done both numerical and experimental studies to observe shock interactions. The experiments were conducted with free stream Mach number of 8. The numerical simulations were done using k–ε turbulence model. They have

² To whom any correspondence should be addressed sr_nagaraja@blr.amrita.edu

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd
reported that the experimental results agree well with the numerical results. They have reported six types of shock interactions. Computational results were in good agreement with experimental data for type VI shock interaction. However, for type V shock interaction laminar computations over predicted the size of the separation region. Pasha et al [4] have conducted experiments to study shock/boundary layer interactions in hypersonic flows. They have studied a range of interaction from the weak case in which there is no flow separation and strong interaction in which there is large separation bubble. Visualization of high enthalpy separated flow field over double cone is done by Jagadeesh et al [5]. The experiments are conducted for stagnation enthalpies of 4.2 MJ/kg and 1.6 MJ/kg. A triple shock structure in front of the second has been reported. Further surface convective heat transfer measurements were made at Mach 5.75. The surface heat transfer on the second cone in the vicinity of transmitted shock fluctuates between 100 W/cm² - 400 W/cm² (± 10 %) for nearly identical (± 8 %) free stream conditions, indicating the severe unsteadiness in the flow field. Similar unsteady fluctuations in the heat transfer and oscillatory shock structure in the flow field around the double cone are also observed in the numerical simulations carried out by solving the axisymmetric Navier-Stokes equations. Cheng et al [6] studied the effects of displacement of boundary layer in high temperature hypersonic flows. It was found that the displacement thickness of the boundary layer increase with decreased Reynolds number. Olejniczak [7] conducted high enthalpy double wedge experiments. They have reported that large changes in the size of the separation zone occur even for small changes in the shock angle and impingement point. Saravanan et al [8] have studied convective heat transfer distribution over missile shaped body. They have compared experimental heat transfer data with values obtained using Fay-Riddell equation and simulations. The tests are conducted for free stream Mach numbers of 5.75 and 8 with the missile shaped body at zero angle of attack. It is concluded that the heat flux over the body is dependent on the free stream Mach number. In their experimental studies on variation of drag coefficient of a sphere with respect to Mach number and Reynolds number Bailey and Hiatt [9,10] found that the coefficient of drag, for a constant Reynolds number, reduces with an increasing velocity up to Mach 10 and it increases with further increase in Mach number.

2. Numerical Modelling and Boundary Conditions

The governing equations solved are the continuity, momentum and the energy equations. In the differential for these equations are

\[
\frac{\partial p}{\partial t} + \nabla \cdot (\rho u) = 0
\]

\[
\rho \frac{\partial u}{\partial t} + \rho(u \cdot \nabla)u = \nabla \cdot [-pI + \tau] + F
\]

\[
\rho C_p \left( \frac{\partial y}{\partial x} + (u \cdot \nabla)T \right) = -(\nabla \cdot q) + \tau : S - \frac{T}{\rho} \frac{\partial y}{\partial x} \frac{\partial y}{\partial x} + (u \cdot \nabla)p \right) + \phi
\]

In the present study, the two-equation Standard k-\(\epsilon\) turbulence model is used. It is a two-equation model used to model turbulence in supersonic and hypersonic flows. The first variable k represents the turbulent kinetic energy and the second variable represents the turbulent dissipation. This two-equation model incorporates the effects of convection and diffusion of turbulent kinetic energy.

The transport equation for this model is

\[
\rho \frac{Dk}{Dt} = \frac{\partial}{\partial x_i} \left[ \alpha_k \frac{\partial k}{\partial x_i} \right] + G_k - \rho \varepsilon
\]

\[
\rho \frac{De}{Dt} = \frac{\partial}{\partial x_i} \left[ \alpha_\varepsilon \frac{\partial \varepsilon}{\partial x_i} \right] + C_1 \frac{\varepsilon}{k} G_k - \rho C_2 \frac{\varepsilon^2}{k} + S_\varepsilon
\]
Nine 2D axisymmetric models with different configurations were considered for the simulation. All the models have a constant first cone half-angle of 25 degrees and a face length of 0.026 m. The second cone half-angles vary from 10 degrees to 45 degrees with an increment of 5 degrees to cover a broad spectrum of flow variations. The second cone has a constant face length of 0.026 m. An after body parallel to the axis with a constant length of 0.038 m was added after the second cone. The models are represented as θ₁, θ₂ where θ₁ represents the half cone angle of the first cone and θ₂ half cone angle of the second cone. Example 25_35: the first cone half angle is 25 degrees and second cone half angle is 35 degrees. Figure 1 shows a general shape of a bi-cone and figure 2 depicts the general geometric configuration used for modelling. The model developed is validated with the results of Wright et al [3]. For the validation the 25_35 model is used with boundary conditions as in the reference work [3]. Figure 3 shows the variation of non-dimensional pressure $P/P_\text{fs}$ with $S/L$ ratio, where $P$ is the pressure at the given location, $P_\text{fs}$, free stream pressure, $S$ is the distance along the cone surface and $L = 0.026$ m is the reference length (length of the first cone surface).

In this section, the boundary conditions used are explained. For the inlet and outlet conditions a pressure far field condition of Mach 6, Mach 8 and Mach 10 are given. The model used is a 2D axisymmetric model with a 4-sided wall, 2 cones, after body and base. At the walls, adiabatic no-slip boundary condition is applied. The free stream temperature of 57 K and free stream pressure of 358 Pa are taken for all the simulations. Specific heat of 1006.43 (J/kg-K) and specific heat ratio of 1.4 were used for all the simulations. The numerical scheme was assumed converged when the residuals of continuity equation, velocities and energy equation are $\leq 10^{-3}$. 

![Figure 1: Typical shape of a bi-cone](image1)

![Figure 2: General model dimensions](image2)

![Figure 3: $P/P_\text{fs}$ vs $S/L$ validation between results by Wright [3] and results present work](image3)
3. Results and Analysis

In the simulations conducted, different types of shocks and their interactions with each other were observed. The fluid flow interactions can be broadly distinguished based on different geometrical configurations of the bi-cone, the first configuration being ‘Second cone half-angle greater than first cone half-angle’ and the second configuration being ‘Second cone half-angle lesser than first cone half-angle’.

![Diagram of shock interactions](image)

Figure 4. General shock interactions when second cone half angle is greater than first cone half angle

![Diagram of shock interactions](image)

Figure 5. General shock interactions when second cone half angle is less than first cone half angle

Figures 4 and 5 respectively show interaction of different shock waves generated when half cone angle of the second cone is greater than and less than that of the first cone. In the figure 4 zone I and zone II correspond to the oblique shock generated by first and second respectively. The adverse pressure gradient causes separated flow region at the corners of the cones. The two shocks coalesce to a third shock with downstream as in zone 3. The conditions upstream of zone 3 are the free stream conditions. The static pressure in zone 2 is not equal to the static pressure in zone 3 due to different upstream conditions and shock strengths. Due to this mismatch, an expansion fan is generated with downstream conditions as zone 4. The zones 3 and 4 have different total pressure and Mach numbers. This due to different shock strengths and downstream conditions causing a slip surface to be generated between zone 3 and zone 4. The conditions in figure 5 are similar to figure 4. The upstream conditions of the shock are conditions in zone 0. There is a formation of an expansion fan at the flare angle for which the downstream conditions are zone 2 and upstream conditions are zone 1.

Figures 6 and 7 show density contours for cases where second cone angle is greater than and less than the first cone. With decrease in the second cone half-angle, the difference in density between different zones becomes more prominent. The change in density is higher with decrease in flare angle due to stronger expansion fans that are formed at the flare angles. The density reduces as the flow passes through an expansion fan.
Figure 6: Density and temperature contours for second cone angle greater than first cone angle

Figure 7: Density and temperature contours for second cone angle lesser than first cone angle

Figure 8: Density vs x/L different models

Figure 9: Pressure ratio vs x/L for different models
Figure 8 shows variation of density with x/L, where x is the axial distance and L = 0.026m is the face length of the first cone. All the curves have similar values till they hit the flare-angle. They have the same increase in density in the beginning due to the shock formed by the first cone half-angle, which is constant for all the bi-cone configurations. The configurations with second cone half-angle more than 25deg show an increase in density after hitting the flare-angle because of the presence of another shock while transition from zone1 to zone 2, then there is a slight dip in density due to the presence of an expansion wave while transition from zone 2 to zone 4. Another dip in density is due to the presence of an expansion wave when the flow reaches the after body. The configurations with second cone half-angle less than 25deg show a decrease in density after hitting the flare-angle because of the presence of an expansion wave while transition from zone1 to zone 2. Another dip in density is due to the presence of an expansion wave when the flow reaches the after-body. The configurations with second cone half-angle equal to 25deg show a decrease in density only once due to presence of an expansion wave when the flow reaches the after body. For the same bi-cone configuration with increase in Mach number, the density change across shock is higher.

Figure 9 shows variation of pressure ratio P/P\textsubscript{fs} with x/L. The pressure ratio across zone 1 remains the same with change in second cone half-angle, this is because zone 1 occurs on the first cone and any change in the second cone will not affect this zone. The pressure ratio for zone 2 has a gradual increase with increase in second cone half-angle due to small increase in shock strength. Zone 3 is the result of downstream conditions of a shock formed due to the coalesce of two shocks and it changes drastically with increase in second cone half-angle, making it a very strong shock with a very high-pressure ratio. The numerical value of pressure ratio for zone 4 is lesser than 1 due to the presence of an expansion fan. As the difference in pressure between zone 3 and zone 2 changes, the strength of the expansion wave also changes accordingly.
Figure 10 shows pressure contours when angle of second cone is larger than first cone. It can be observed that the highest pressure occurs at the flare angle. This is because the flow at zone 2 has passed through two shocks, the static pressure between zone 3 and zone 4 are equal due to the expansion during the transition between zone2 and zone3. Though the pressure ratio to zone 3 is the higher than that to zone 2 it has a lower static pressure because the upstream conditions for zone3 are the freestream conditions whereas the upstream conditions for zone2 are the conditions for zone1. In figure 1, the highest pressure is observed in zone1 due the presence of a shock in transition to zone1. An expansion occurs at the flare-angle due to the presence of an expansion wave. As the second cone, half-cone half-angle decreases the pressure drop across the expansion wave increases.

The drag coefficient can be calculated using the equation
\[
F_d = \frac{1}{2} \rho_\infty v_\infty^2 C_d A,
\]
where \(\rho_\infty\) is the free stream density, \(v_\infty^2\) is the free stream velocity, \(A\) is the projected area and \(F_d\) is the drag force which includes both pressure and viscous drag. Figure 12 shows the variation of the drag coefficient with Mach number of different models. For a particular Mach number the Coefficient of drag reduces as the second cone half-angle reduces. For Mach number 6 there is 57.7% reduction in the drag for model with second cone half-angle 35deg compared to the model with15deg. Similarly, for Mach number 10 the reduction is 60.6%. Also for a particular bi-cone configuration, the Cd reduces by 3% to 10% as Mach number increases. The total heat flux is measured over the surface of the bi-cone. Figure 13 shows the variation of heat flux with x/L. It is observed that the heat flux is constant in the beginning because the geometry of the first cone half angle is constant for all the models. As the second cone half angle is varying the heat flux varies and is has the maximum value for the highest second cone angle.

4. Conclusions

A series of simulations have been conducted to study variations in density, pressure, drag coefficient, temperature and heat flux of hypersonic flow over a bi-cone. In cases where the second cone half-angle is more than first cone half-angle, the density and pressure increases with increase in second cone half-angle and Mach number. The pressure was found to be highest at the flare angle (zone2). The static pressure between zone3 and zone4 was equal. The total pressure between zone 3 and zone 4 is found to
be different due to difference in Mach numbers. The temperature increases with increase in second cone half-angle and Mach number. The highest temperature is observed at zone3. In cases where the second cone half-angle is less than first-cone half-angle, the density and pressure at flare angle (zone2) decreases with decrease in second cone half-angle. It is found that heat flux distribution along the model surface is directly proportional to the free stream Mach number and second cone half-angle. The drag coefficient decreases for a decreasing second cone half angle and for an increasing Mach number. The reduction in drag coefficient due to the reduction in second cone half angle is because the projected frontal area of the bi-cone reduces. Its reduction with an increasing Mach number is as predicted by the drag coefficient formula where the drag coefficient is directly proportional to pressure force acting on the body and inversely proportional to the square of the velocity. Even though the pressure increases with Mach number, the ratio of $P/v^2$ decreases thereby reducing the drag coefficient.

5. References

[1] Lester L. Hypersonic Flow 2003 *Journal of Spacecraft and Rockets* 40 5 700-735.

[2] Santos, Wilson FN. Simulation of blunt leading edge aerothermodynamics in rarefied hypersonic flow 2007 *Journal of the Brazilian Society of Mechanical Sciences and Engineering* 29 2 123-135.

[3] Wright, M. J., Sinha K., Olejniczak, J, Candler G. 2000 Numerical and experimental investigation of double-cone shock interactions *AIAA journal* 38 12 2268-2276.

[4] Pasha, Amjad A., and Krishnendu Sinha. 2012 Simulation of hypersonic shock/turbulent boundary-layer interactions using shock-unsteadiness model *Journal of Propulsion and Power* 28 1 46-60.

[5] Jagadeesh, G, K.P.J Reddy, T. Hashimoto, K. Naitou, M. Sun and K. Takayama 2002 Study of the separated high enthalpy flow around a double cone 2002 40th *AIAA Aerospace Sciences Meeting & Exhibit*.

[6] Cheng, H. K. Boundary-layer displacement and leading-edge bluntness effects in high-temperature hypersonic flow 2012 *Journal of the Aerospace Sciences*

[7] Olejniczak, J, Candler G., Wright M., Hornung H., and Leyva I 1996 High enthalpy double-wedge experiments *Advanced Measurement and Ground Testing Conference*. 1996.

[8] Saravanan, S., G. Jagadeesh, and K. P. J. Reddy Convective heat-transfer rate distributions over a missile shaped body flying at hypersonic speeds 2009 *Experimental Thermal and Fluid Science* 33 4 782-790.

[9] Bailey, A. B., and J. 1972 Hiatt Sphere drag coefficients for a broad range of Mach and Reynolds numbers *AIAA Journal* 10 11 1436-1440.

[10] Bailey, A. B., and J. Hiatt 1971 Free-flight measurements of sphere drag at subsonic, transonic, supersonic, and hypersonic speeds for continuum, transition, and near-free-molecular flow conditions *No. AEDC-TR-70-291* Arnold Engineering Development Center Arnold AFS TN