Numerical simulation of flow field over slender bodies at transonic Mach number and low angle of attacks

Jai Makhija¹, D Siva Krishna Reddy² and Rishabh³
Department of Mechanical Engineering, SRM Institute of Science and Technology, Kattankulathur, Pin code: 603203, TamilNadu, India
E-mail: sivakriv@srmist.edu.in

Abstract. The transonic flow field over a launch vehicle is complicated due to presence of a normal shockwave on the heat shield. To predict the aerodynamic loads, the shockwave position needs to be predicted accurately. The present work aims at predicting the flow field over a typical launch vehicle at an angle of attack (AOA), and analyzing the effect of AOA on flow field, shockwave position and aerodynamic and moment coefficients. The launch vehicle considered here is a long slender body with spherical nose followed by the conical, cylindrical and boat-tail portions. Simulations are performed at a Mach number of 0.95 with AOA of 0°, 2°, and 4°. Commercial Computational Fluid Dynamics package ANSYS Fluent is used for simulations. Density-based algorithm is used to obtain the steady state solutions with explicit time stepping. κ-ω SST turbulence model is used to close the turbulent stresses terms. The surface pressure distribution in axial direction is analyzed to understand the effect of AOA on the aerodynamic and moment coefficients. It is observed that as AOA increases, the normal shockwave moves towards nose. The drag coefficient and pitching moment coefficients are found to increase with increase of AOA.

1. Introduction
A satellite launch vehicle experiences aerodynamic loading, heating and buffeting load [1] during its ascent phase. Hence, the heat shield is provided to protect the payload from the above forces. Flow field over the heat shield at transonic Mach numbers is characterized by the presence of a normal shockwave. Usually the shockwave will interact with the turbulent boundary layer and can result in flow instabilities [2]. The boundary layer interaction and associated effects depends on many factors such as freestream Mach number, angle of attack, shape of the nose and heat shield, and boat-tail angle. Hence it is required to understand the flow pattern over a typical launch vehicle at transonic speeds.

Purohit, S.C. [3] computed the flowfield over a typical heat shield by solving compressible Navier-Stokes equation with McCormack’s explicit finite difference method. The location of the shockwave and separation point, size of separation bubble is predicted. Surface pressure fluctuations arising from the shock-turbulent boundary layer are analyzed. R.C.Mehta [4] computed the flow field over a payload shroud at transonic Mach numbers. The simulations are performed by solving axisymmetric, compressible and Reynolds-averaged Navier-Stokes (RANS) equations in strong conservation form. The discretization was performed using finite volume method. Baldwin-Lomax turbulence model was used for the closure of turbulent stresses. A good comparison is obtained between the predicted location and strength of normal shockwave compared to that of experiments. In addition, the
predictions are also compared to that of flight measurement of surface pressure on heat shield and comparison found to be good.

Gireesh et al. [5] studied effect of nose shape on the slender bodies. The first variant has hemispherical nose connected to a 20° cone. In the second variant, the hemispherical nose is replaced by an Ogive nose. All other geometric parameters are kept constant. Axisymmetric simulations are performed over the two configurations for Mach numbers of 0.8, 0.9, 0.95 and 1.1. Commercial CFD package ANSYS was used for meshing and subsequent simulations. The predicted coefficient of pressure over the surface is compared with those obtained from experiments. The comparison was found to be good between experiments and predictions. Based on the numerical studies, it was concluded that flow over the ogive shape nose is smoother compared to that of hemispherical nose cone and suggested to have ogive shape nose for transonic flow applications.

Murugan, K. N., et al. [6] conducted experiments in wind tunnel to investigate the shockwave pattern over typical launch vehicle at Mach number of 0.95. The launch vehicle consists of conical-cylindrical core body and strap-on boosters. Experiments were conducted for AOA spanning from 0° to 4° with a step of 1. It was observed that a standing normal shockwave is observed on the cylindrical portion for AOA 0°. Downstream of the shockwave, flow separation was observed. As AOA increases, the shockwave moves upstream, and a separation shockwave also observed. At AOA 4°, the shockwave was found to oscillate showing the unsteady pressure loads on the launch vehicle. In the present work, simulations are performed over a blunt nosed long slender body. This represents the core body of a launch vehicle. The objective of the work is to investigate the effect of AOA on the flowfield, shockwave location and associated aerodynamic coefficients. Figure 1 shows geometry of the launch vehicle. It has spherical and conical fore body, followed by a cylindrical heat shield portion. The boat tail region is present towards the end of cylindrical portion. The boat tail region is followed by another cylindrical portion, whose diameter is lower than that of first one. The freestream conditions are: Pressure is 96900 Pa, Temperature is 254.1 K. These conditions are identical to those presented in Ref [6].

2. Simulation Methodology

ANSYS Fluent is used for simulating the flowfield over the core body. Simulations are performed on structured grids, which are generated using ICEM CFD meshing tool. The compressible RANS equations are solved with density-based algorithm. The inviscid fluxes are discretized using AUSM method. κ-ω SST model is used to close the turbulence terms in RANS equations. At the inlet, free stream conditions are imposed. Extrapolation boundary conditions were applied at outlet boundary. No-slip boundary conditions were applied on walls of the corebody. A typical mesh consists of 225, 120, 120 points in the X, Y and Z directions respectively. The total number of cells in the grid are 29,65,638. Figure 2 shows a plane view of the grid in XY and YZ planes respectively. Fine mesh is used at the near wall region of nose stagnation and boat-tail. The grids are progressively stretched towards outlet boundary so that a coarse grid can be employed. The first grid point in the wall-normal direction is placed at a distance of 0.001 mm from wall. The y+ value in majority portion of the wall is below 2.

![Figure 1. Geometry of the slender body considered in the present study.](image)
Figure 2. Schematic of the structured grid used in the simulations: a) YZ plane and b) XY plane.

Table 1. Drag coefficient obtained on three successively refined grids.

| Grid            | Number of cells | Cd     |
|-----------------|-----------------|--------|
| Coarse mesh     | 22,03,734       | 0.567541 |
| Medium Mesh     | 29,65,638       | 0.526626 |
| Fine Mesh       | 36,91,896       | 0.525586 |

Figure 3. The predicted drag coefficient as a function of iterations for a typical simulation on a log scale.

Table 1. Drag coefficient obtained on three successively refined grids.

Figure 3. shows the reduction in the predicted drag coefficient (Cd) as a function of the iterations for a typical simulation on logarithmic scale. There is a rapid drop in Cd from 0 to 1000 iterations. In the subsequent iterations, the reduction in the Cd is slower. The change in Cd from 29000 to 30000 iterations is less than 0.07%. As the Cd variation is negligible beyond 30000 iterations, solution is treated as converged. The residual history for AOA 2° and 4° is also similar. It takes nearly 30000
iterations for iterative convergence, and on 8 core 2.4 GHz processor the simulation takes 18 hours for convergence. This accounts 144 CPU hours.

3. Grid Convergence
To assess the grid point density to resolve flow gradients, a grid convergence study is performed. Simulations are performed by reducing and increasing the number of points by 25%. The near wall spacing is kept constant in the all the three grids. Table 1 shows $C_d$ obtained from the three grids. The change in $C_d$ when grid point density decreased by 25% is 7.1%. On the other hand, when grid point density increased by 25%, the change in $C_d$ is below 0.2%. It indicates that refining the mesh, beyond the medium grid, has not yielded significant changes. Hence the medium grid is used for further simulations, as it takes a smaller number of CPU hours compared to that of fine mesh.

4. Results
4.1 Mach number Contours for AOA 0°
Figure 4 shows Mach number contours over core body at AOA 0° for the mid-plane, $Z=0$. The free stream Mach number is 0.95. In the nose stagnation point region, flow decelerates and attains zero velocity at stagnation point. As the flow moves downstream of the stagnation point, expansion occurs and results in the increase of Mach number to 0.78 at end of spherical portion. At the conical and cylindrical portion, flow further expands and leads to formation of Prandtl-Mayer expansion fan centered on the junction. Towards end of expansion, the Mach number is 1.61. As the flow approaches to boat-tail region, it cannot follow the profile of geometry and as a result a Normal shockwave. Downstream of the shockwave, the Mach number decreases to 0.9. Flow separation occurs at about beginning of boat-tail region and results in the formation of separation bubble. Streamlines shows the extent of separation bubble, and it extends up to 1.1 times the body diameter. A shear layer can be clearly seen, that originates at the start of the boat-tail region. The shear layer re-attaches on the conical body. Wake continues to develop in the downstream of the cylindrical portion, and the Mach number of the flow in the wake is about 0.96. Flow again separates at the end of core body, and subsequent separation bubble and wake can be clearly seen. Mach number of the flow in the separation bubble spans from 0.10 to 0.15.

Figure 4. Normalized surface pressure along the body. The critical parts of the body i.e., spherical, conical, cylindrical and boat-tail are identified.
Figure 5. Numerical visualization of flow field in terms of Mach number contours for AOA = 0° at Z = 0 plane. Streamlines are drawn to demarcate separation bubbles.

Figure 5 shows Mach number contours over core body at AOA 0° for the mid-plane, Z=0. The free stream Mach number is 0.95. In the nose stagnation point region, flow decelerates and attains zero velocity at stagnation point. As the flow moves downstream of the stagnation point, expansion occurs and results in the increase of Mach number to 0.78 at end of spherical portion. At the conical and cylindrical portion, flow further expands and leads to formation of Prandtl-Mayer expansion fan centered on the junction. Towards end of expansion, the Mach number is 1.61. As the flow approaches to boat-tail region, it cannot follow the profile of geometry and as a result a Normal shockwave. Downstream of the shockwave, the Mach number decreases to 0.9. Flow separation occurs at about beginning of boat-tail region and results in the formation of separation bubble. Streamlines shows the extent of separation bubble, and it extends up to 1.1 times the body diameter. A shear layer can be clearly seen, that originates at the start of the boat-tail region. The shear layer re-attaches on the conical body. Wake continues to develop in the downstream of the cylindrical portion, and the Mach number of the flow in the wake is about 0.96. Flow again separates at the end of core body, and subsequent separation bubble and wake can be clearly seen. Mach number of the flow in the separation bubble spans from 0.10 to 0.15.
4.2 Surface Pressure data
Surface pressure along the wall of the body in the mid-plane, Z=0 is shown in Figure 6. On the X-Axis, the arc-length of the point from nose stagnation point is normalized by the body diameter. On the Y-Axis, surface pressure is normalized by the freestream static pressure. The critical geometric points are identified. At the stagnation point, $P_w/P_\infty$ is 1.788. The isentropic relations for $M = 0.95$ are 1.789. Hence the error in prediction of stagnation pressure is less than 1%. Due to the flow expansion around the spherical portion, the pressure decreases to 1.017 times $P_\infty$. Further expansion of the flow on the conical portion, decreases the pressure to 0.376 times $P_\infty$. Pressure progressively increases along the cylindrical portion. Due to the normal shockwave at $s/D = 2.56$, the pressure increases sharply. In the downstream of the shockwave, pressure continuously increases till $s/D = 4$. This portion corresponds to boat-tail region. After boat-tail, pressure decreases for the entire length of the body.

Figure 6. Normalized surface pressure along the body. The critical parts of the body i.e., spherical, conical, cylindrical and boat-tail are identified.

4.3 Validation
Figure 6 shows comparison of flow pattern obtained from experiments and predictions. The experimental image [6] is obtained by shadowgraph method, hence the lines represent the density gradients. The numerical visualization is presented in the form of density contours where the data corresponds to AOA 0°. In both cases, the free stream conditions are identical. The simulations capture the critical flow features such as expansion, boundary layer and normal shockwave. Similar flow features can be identified in the experimental visualization. The predicted flow pattern around the cylindrical portion matches well with the experiments. The shockwave patterns observed for AOA 2° and 4° are similar to those observed in the Ref [6]. Comparison is not presented for the sake of brevity.

4.4 Effect of AOA
Simulations are performed at AOA 2° and 4° with free stream conditions same as that of AOA 0°. Changes in the computed flowfield are analyzed to understand the effect of AOA on the flowfield and surface properties.
Figure 7. Comparison of flow visualization obtained from (a) Experiments, and (b) CFD for AOA 0°.

Figure 8. Computed Mach number contours at mid-plane for (a) AOA 0°, (b) AOA 2°, (c) AOA 4°.

Figure 7 shows Mach number contours at Z = 0 plane for AOA 0°, 2° and 4°. Stream lines are shown to identify the extent of separation bubble. The size of separation bubble for AOA 2° and 4° are 1.15 and 1.17 times the body diameter respectively. As the AOA is increased from 0° to 4°, on the windward side, the size of separation bubble decreases. Separation shock is not observed for AOA 0° and 2°. The intensity of the flow expansion at the beginning of the boat-tail decreases and, that on end of boat-tail increases. On the leeward side, the size of the separation bubble increases. Separation shock wave can be clearly noticed for AOA 4° condition. Further, the shockwave moves upstream and normal shockwave transforms into oblique shockwave. The extent of flow expansion at the beginning of the boat-tail progressively increases as AOA increase. Mach number contours for AOA 4° indicated the presence of asymmetry in the flow field due to angle of attack.
Figure 9. Simulated surface streamlines on the payload fairing of the core body for (a) AOA 0° and (b) AOA 4°.

Table 2. Shock location in terms of non-dimensional distance.

| AoA | Shock Location (m) | Shock Position in X/L | Range for X/L   |
|-----|--------------------|-----------------------|-----------------|
| 0   | 0.09047698         | 0.947139605           | 0.84 < x/L < 0.92 |
| 2   | 0.0862494          |                       |                 |
| 4   | 0.060193648        | 0.341472969           | 0.2 < x/L < 0.62 |

Figure 9 shows surface streamlines on the surface of the body obtained from the simulations. Enlarged view near the boat tail region is shown. The separation and re-attachment points are marked by the ‘S’ and ‘R’ respectively. At separation point, the stream lines diverge, whereas at re attachment point, stream lines converge. The distance between ‘S’ and 'R' represents extent of separated zone. The size of the separation bubble for AOA 4° is higher than that of the AOA 0°. A similar observation was noticed in Ref [6].

Table 2 shows the comparison of predicted shock location with that measured in Ref [6]. The shock position is detected based on surface pressure variation. The location at which sharp pressure rise occur can be taken as shockwave location. The beginning of cylindrical portion is taken as origin. The shock position for AOA 0° is over the predicted. For other two angle of attacks, the predicted shock position is within the range that observed in the experiments. As the Mach number increases, shockwave moves upstream. The upstream movement is much higher for AOA 4°. Note that in the experiments [6] the model consists of boosters also. The presence of boosters can alter the shock location. Further, in experiments it is observed that, shock oscillation for AOA 4° are dominant, which is a transient phenomenon. Hence for accurate prediction of shock position, unsteady simulations need to be performed.
Figure 10 shows comparison of surface pressure obtained from the three simulations. The data presented corresponds to upper surface (leeward) of the body at Z=0 plane. The expansion ratio across the spherical part is highest for AOA 4° case as indicated by the lowest pressure. At the beginning of the conical part, pressure rise is observed for 3 cases. In the immediate downstream, pressure decreases again for the three cases. The rapid expansion around the cylindrical portion can be noticed. At the beginning of the cylindrical portion, pressure is approximately same for the three cases. In the downstream pressure increases and a deviation can be noticed among the three solutions. Upstream of boattail region, a change in the rate of pressure rise can be noticed. For AOA 0°, the pressure rise is relatively steep, whereas for AOA 2°, it is gradual. The pressure rise at this location is due to the presence of normal shockwave. The solution obtained for AOA 4° shows a different trend on cylindrical portion. The sharp pressure rise at s/D = 1.37 indicates is due to the flow separation and associated flow phenomena. The flow separation leads to formation of separation shock and re-attachment shock. In the downstream side of the boattail region, pressure obtained from the solution of AOA 0° and 2° is approximately same. At the base region (9.72 < s/D < 10.33) again a deviation among the three cases is observed (not shown here due to space constraints).

Development of wake for the three cases is shown in Figure 11. The contours correspond to Turbulent Kinetic Energy (TKE). Figure 10 shows TKE at Z = 0 plane for AOA = 4°. In the downstream of the boat tail, TKE has significant values, and further in the downstream, it progressively decreases. Maximum TKE is observed in the near-wake region. Development of TKE along the different cross-sectional planes is also presented in the figure. The chosen sections are X = 0.063 m, 0.103 m and 0.403 m represent cross sectional planes near to beginning of boat-tail, end of boat-tail and end of core body respectively. Streamlines are shown for plane X = 0.403 m. As the AOA increased, symmetry in the TKE contours is disturbed. This can be noticed at sections X = 0.103 m and 0.403. For AOA 0°, a sink flow is observed which is nearly symmetric. For AOA 4°, the size of the windward vortex reduces and that of leeward increases. Further, magnitude of TKE is higher on windward side compared to that of leeward side. This is mainly due to presence higher cross flow velocity in the windward side.
Figure 11. Numerical visualization of the wake development in terms of turbulent kinetic energy contours.

Table 3. Numerical drag and lift coefficient and center of pressure for AOA 0°, 2° and 4°.

| AoA | Cd    | Cl   | COPx | COPy  | COPz  |
|-----|-------|------|------|-------|-------|
| 0   | 0.526626 | 0.001219 | 0 | -0.000844 | -0.005608 |
| 2   | 0.531369 | 0.072327 | 0 | -0.008035 | -0.001997 |
| 4   | 0.586411 | 0.155667 | 0 | -0.039368 | -0.014871 |

Table 4. Comparison of predicted moment coefficients for the three cases of AOA 0°, 2° and 4°.

| AoA | Cm roll  | Cm yaw  | Cm pitch |
|-----|----------|---------|----------|
| 0   | -4.90E-05 | -0.07676 | 0.009975 |
| 2   | 0.000217  | -0.032729 | 0.102994 |
| 4   | 0.000485  | 0.188261  | 0.554452 |

4.5 Aerodynamic Coefficients

Table 3 shows the drag and lift coefficients and coordinates of center of pressure for AOA 0°, 2° and 4°. At AOA 0°, the drag coefficient is about 0.526626. As the AOA is increased, flow separation on leeward side enhances the drag force. Hence drag coefficient also increases. The drag coefficient at AOA 4° is 11.35% higher than that of AOA 0°. The lift coefficient is nearly zero for AOA 0° and it increases as the AOA increases. This is mainly due to the asymmetric flow field. The coordinates of the center of pressure for AOA 0° are (0, -0.000844, -0.005608). The reference point to define the center of pressure is nose stagnation point. As AOA is increases, the center of pressure shifts towards nose stagnation point.

Table 4 shows moment coefficients obtained from the present simulations. The slope of the pitching moment coefficient with respect to AOA is high compared to other moment coefficients. The pitching moment coefficient for AOA 4° is about 0.55 and is more than 50% higher than that of AOA.
0°. The roll moment generated is comparatively lower. The predicted yaw moment is significant and maximum moment is observed for AOA 4°.

5. Conclusions

CFD simulations over a blunt nosed long slender body at Mach number of 0.95 and at low Angle of Attack were performed. The comparison between the predicted pressure at nose stagnation point and that of isentropic flow relations is good and deviation is below 1%. The position of the normal shockwave on the cylindrical portion also compared well with those of experimental values. As AOA is increased, the normal shockwave moves upstream and separation bubble size on windward size is reduced. The AOA introduces asymmetry in the pressure distribution over the entire length of body. The drag coefficient at AOA 0°, 2° and 4° are 0.526626, 0.531369, and 0.586411 respectively. A rapid increase in the drag coefficient and pitching moment coefficient with respect to AOA is observed. The center of pressure found to shift towards nose region as AOA increases. The changes in the roll moment coefficients is negligible. However, yaw moment coefficient found to change, as AOA changes.

Acknowledgments

The authors acknowledge CFD Laboratory, Department of Mechanical Engineering, SRMIST, Kattankulathur for providing computational resources for performing the numerical simulations.

Nomenclature

| Abbreviation | Description |
|--------------|-------------|
| CFD          | Computational Fluid Dynamics |
| AOA          | Angle of Attack |
| k            | Turbulent Kinetic Energy, J/kg |
| ω            | Specific turbulent dissipation rate, 1/s |
| SST          | Shear Stress Transport |
| M            | Mach number |
| C\text{d}    | Drag coefficient |
| C\text{l}    | Lift Coefficient |
| COP          | Centre of Pressure |
| C\text{m}    | Moment coefficient |

6. References

[1] Rainey A G 1965 Progress on the launch vehicle buffeting problem *J. Spacecraft and Rockets* 2 289–99
[2] Bogdonoff S 1955 June Some experimental studies of the separation of supersonic turbulent boundary layers *In Proceedings of the Heat Transfer and Fluid Mechanics Institute* University of California, Los Angeles 1–23
[3] Purohit C S 1986 Navier-Stokes solution for a Conical Payload Shroud *J. Spacecraft and Rockets* 23 590-96
[4] Mehta R C 2009 Numerical study of flow field visualizations over a payload shroud at transonic speeds Part G *J. Aerospace Engineering* 223 179-185
[5] Yanamashetti G, Suryanarayana G K and Mukherjee R 2016 Development of Flow over Blunt - Nosed Slender Bodies at Transonic Mach Numbers *Journal of Physics: Conference Series* 822
[6] K N Murugan, Arunkumar T, Prasath M and Ganesan V R 2018 Shockwave oscillations over the conical heat shield region of a typical launch vehicle at Mach 0.9 *Symposium on Applied Aerodynamics and Design of Aerospace Vehicles (SARODA)*