Computation of three-dimensional transonic flow fields over a three-body launch vehicle configuration for low angle of attack conditions

D Siva Krishna Reddy1,*, Pradip Shah1

Department of Mechanical Engineering, SRM Institute of Science and Technology, Kattankulathur 603203, Tamil Nadu, India

*E-mail: sivakriv@srmist.edu.in

Abstract. The present work aims at the prediction of shock wave position on the heat shield region of a three-body launch vehicle in transonic region. The launch vehicle consists of a cylindrical core body and two boosters. The core body is made of a spherical nose, followed by a cone and heat shield region which is a flat surface. The heat shield is followed by a boat tail. Simulations are performed with commercial CFD package, ANSYS FLUENT. Reynolds stresses are computed using the k-ω SST turbulence model. Explicit time integration is used to obtain steady-state solutions. A structured grid over the vehicle is generated using ICEMCFD software. In the near-wall region of the launch vehicle fine mesh is employed to capture the boundary layer. Simulations are performed for angle of attacks (AOA) 0° and 4° for Mach number 0.95. The computed nose stagnation pressure compared with the isentropic relations and deviation found to be .067%. A standing normal shock wave is observed for AOA 0 whereas an oblique shock wave is observed for angle of attack 4°. The predicted flow structure in terms of density distribution is compared with experimental schlieren images. The predictions captured essential flow features such as expansion around the conical part, boundary layer over the heat shield, shock wave on the heat shield, and flow separation in the boat tail region. The predicted flow structure and shock wave position for AOA O matched well with the experiments. However, for AOA 4°, the deviation between flow structure and shock position is noticed. Possible reasons for the deviation are explained.

1. Introduction
In spaceflight, a launch vehicle is a rocket-powered vehicle that transports a spacecraft beyond Earth's atmosphere for example to orbit to other location in space. Since the 1950s, launch vehicles have been used to launch crewed spacecraft, and satellites into orbit. For launch vehicles, the flow field in the transonic regime is important to study as the body experiences various aerodynamic problems. The majority of launch vehicles fly at very low angles of attack and during the flight, the loads such as, drag, buffeting that these vehicles experience are significant, and should be studied and precisely calculated. It is necessary to estimate the aerodynamics of these vehicles at an angle of attack throughout the design phase because this gives the loads required for the vehicle's structural design as well as the flight dynamics [1]. Boosters are a frequent way of application in spaceflight, however, they cause problems with the vehicle's aerodynamics. Increased drag, a complicated flow field, and interference effects among bodies are the most significant aerodynamic issues [2]. These concerns highlight the need of doing aerodynamic studies throughout the development of a launch vehicle using boosters. The three-body launch vehicle complicates the grid generation and satisfaction of boundary conditions. Furthermore,
the interference between the core and boosters, as well as the interaction of distinct shocks with each other and the boundary layer causes flow discontinuities throughout the flow field [3]. Transonic flow is typically difficult to understand since it includes subsonic supersonic flows and shock/turbulent boundary layer interactions. Usually, shock movements are frequently paired with a large dynamic pressure, the flow field is particularly significant for ascent flight within the transonic speed range [4]. The estimation of flow field behaviour in such a system is both practical and important for research.

The payload is usually placed downstream of the nose, and a heat shield will be provided to protect the payload against aerodynamic loading and heating during ascent flight. During the transonic flight, a portion of the heat shield's external surface is subjected to steady and transient pressure loads due to the presence of normal/oblique shock waves.

A certain configuration of the heat shield is linked to the severity of the terminal shock and the mechanism of their interaction. The payload is placed at the nose, i.e. around the region of the heat shield. During the flow, the heat shield's external structure is subjected to both steady and erratic pressure loads [5].

Murugan K N., et al. [6] conducted experiments to evaluate the shock wave pattern over a typical launch vehicle at a Mach number of 0.95 for AOA ranging from 0° to 4° at 1° interval. In the case of AOA 0°, a standing normal shock wave was detected on the cylindrical part. The shock wave advances upstream as AOA rises, and a separation shock wave is also visible. The shock wave was found to oscillate at AOA 4°, causing uneven pressure loads on the launch vehicle.

For aerodynamic analysis of the launch vehicle, analytical approaches, CFD, and wind tunnel testing is often utilized, depending on practical issues and the level of precision required. With significant advancements in CFD, it is useful to take advantage of the capabilities of CFD tools to obtain a greater understanding of complex transonic flows of launch vehicles following experimental validation. In the present work, CFD is used to understand the shock wave pattern and its location at transonic Mach number and low angle of attack. The launch vehicle consists of a cylindrical core body and two strap-on boosters. Schematic diagram of the vehicle is shown in Fig.1

2. Simulation methodology

ANSYS-Fluent is used for the computation of three-dimension flow fields over the launch body. Simulations are performed on structured grids. ICEM CFD meshing tool is used to create grid for flow computational domain. The RANS equation is solved with a density-based algorithm under steady-state conditions. K-ω SST model is used to close the turbulence terms in the RANS equation. The free-stream Mach number is 0.95, and the flow would be compressible hence pressure far field is chosen for the inlet. At the outlet, 0 gauge pressure is applied. For the launch vehicle and boosters, no slip wall boundary condition is applied. The turbulent intensity and turbulent viscosity ratio are 5% and 10 respectively. Simulations were computed for angle of attacks for 0 and 4. The three body launch vehicle is created in Solid works whose dimensions are taken from [6].

![Figure 1: Geometry of three body launch vehicle [6].](image-url)
2.1 Grid
The grid was created in ICEM CFD software. The total number of cells equals 40,823,21. In the near wall of the launch vehicle, fine mesh is employed to capture the boundary layer effect properly. Coarse grids are used towards the outlet computational domain. The width of first grid normal to the wall is 0.0009 mm. The Y+ value majorly on the core body wall is maintained below 2. The mesh sizing lies between 1e-7 m and 1e-3 m. Fig 2(a) shows an isometric view of the computational domain. Figure 2(b) shows plane view of the grid in XY plane.

![Figure 2: Schematic of the structured grid: (a) computational fluid domain (b) XY plane](image)

2.2 Iterative convergence
The simulations are computed for angle of attacks of 0° and 4°.

![Figure 3: $C_d$ versus Iterations for AOA 0° and 4° on logarithmic scale](image)
Figure 3 shows the predicted drag coefficient ($C_d$) versus iterations for both AOA 0° and 4°. There is significant drop in $C_d$ for both cases for first 1000 iterations. $C_d$ then slowly drops in the successive iterations. For last 10,000 iterations percentage change in the value of $C_d$ is less than 0.0311% for AOA 0° and 1.236% for AOA 4°. Variation is insignificant for AOA 0°, hence the solution can be considered convergent. However, with AOA 4°, convergence is difficult to achieve.

3. Results and discussions

3.1 Mach number contours

![Mach number contours](image)

Figure 4: Computed contours of Mach number along launch vehicle at $Z = 0$ plane for
(a) AOA 0° (b) AOA 4°.

The free stream Mach number is 0.95. The stagnation point is seen at the nose of the launch vehicle where the flow is retarded completely to zero velocity. As the flow moves downstream from the nose, it expands along the spherical, and Mach number increases at the end. Further, the Mach number goes on increasing as the flow expands in the conical and cylindrical regions. Finally at the end of expansion higher Mach number is attained.

For AOA 0° near the boat tail region, the expanding flow cannot attain smooth flow due to the geometry profile. As a result, the normal shock wave is formed as shown in fig 4(a). Downstream to this region the Mach number drops. The flow separation is observed from the beginning of the boat tail region. For AOA 4°, the flow separation is observed when the flow moves from the start of the cylindrical region. The Mach number is seen to increase to value greater than 1 indicating the presence of shock wave. Then the flow downstream changes to subsonic. It is observed that for AOA 4° the normal shock wave is changed to oblique. Moreover, separation points are seen implying the presence of a separation shockwave. The separation shock wave merged with oblique shock wave forming lambda shape pattern shown in fig (b). It's also seen that when the AOA rises, the shock position shifts closer to the nose stagnation point. Also, the shear layer due to the velocity gradient present in the region of the boat tail is vividly seen hitting the booster’s nose.

Development of the flow along the vehicle is analysed at axial locations of $X = 0.103m$, $0.153m$, $0.403m$. These locations correspond to boat tail region, the centre between the boat tail and the nose of the booster, and the end of the launch vehicle. The circumferential contours show the boundary layer effect on the downstream flow. It is observed that near the wall in boat tail region, the flow decelerates and are attached to the wall uniformly in case of AOA 0° as shown in fig (a). The downstream of flow past boat tail region gradually breaks in small bubbles for AOA 0° fig (b). In case of AOA 4° asymmetry in flow is observed in windward side. The flow near the wall is greatly affected by higher angle of attack causing uneven velocity distribution over the wall as shown in fig (d) and (e). This emphasizes that presence of boosters retards the velocity in the windward side increasing boundary layer thickness as the angle of attack increases from 0° to 4°. The velocity variation at locations $X=0.153m$ for both AOA 0° and 4° shows the effect of boosters on boundary layer thickness. The end of the launch vehicle is
not affected as the flow is moving over the wall smoothly and uniformly shown in fig (c) and (f) for both angle of attacks 0° and 4°.

Figure 5 (a–c): Numerical visualization of Mach number contours at location of X for AOA 0°, (d–f) Numerical visualization of Mach number contours at location of X for AOA4°

3.2 Static Pressure contour

Figure 6: Contours of Static Pressure taken from Z=0 plane: (a) AOA 0° (b) AOA 4°.
Variation of static pressure over the Launch Vehicle is shown in the figure 6. The incoming flow is decelerated at nose where stagnation point is observed. The static Pressure is highest in region of nose where Mach number is zero. As the flow expands along the conical region, the drop in the static pressure is observed. Lowest static pressure band is observed over the heat shield region indicating acceleration of flow. The low band of static pressure is seen till the boat tail region. The static pressure then increase and at the nose of booster the sharp increase is observed in both windward and leeward side for AOA 0° as shown in fig 6(a). In case of AOA 4°, the lowest band of static pressure is observed near the start of heat shield region where shock wave is forming. The static pressure distribution is asymmetrical for AOA 4° as shown in fig 6(b).
3.3 Stagnation pressure comparison

The stagnation pressure from isentropic relation

\[
\frac{P_o}{p} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\gamma/\gamma-1}
\]
\[
= \left(1 + \frac{1.4-1}{2} 0.952\right)^{1.4/1.4-1}
\]
\[
= (1.1805)^{3.5}
\]
\[
= 1.7874
\]

From surface pressure data for AOA 0 from fig 4.4, the stagnation pressure at nose is 1.7862.

\[
error = \frac{1.7874 - 1.7862}{1.7874} \times 100\% 
\]
\[
= 0.067\%
\]

3.4 Density contours

![Figure 7: Comparison of density contours for AOA 0° and AOA 4°, (a,b) Experimental and (c,d) CFD](Image)
Figure 7 shows a comparison of flow patterns produced by tests and simulations. The density gradients are represented by the shadowgraph approach in the experimental image (a) and (b). The data corresponds to AOA 0° and 4°, and the numerical representation is plotted in the form of density contours. The free stream conditions are identical in both circumstances. All the necessary flow properties for example expansion, boundary layer, and normal shockwave are captured in the simulations. Similar flow characteristics can be seen in the experimental visualisation. The flow pattern around the cylindrical part matched the experiments quite well for AOA 0° fig (c) and (d). However, in case of AOA 4°, the flow pattern differs as the oblique shockwave is located as beginning of heat shield which can be observed from Figs. 7 (b) and (d).

From the experimental shadowgraph image, the shock wave is found to be moving over the heat shield region for angle of attack 4°. Due to this oscillating nature of shock wave it can be concluded the phenomena is transient. Since in this present work, steady state simulations have been used to predict the shock location, the shock is observed to be settling near the start of heat shield region. To precisely predict the location of shock wave transient simulations need to be performed. Furthermore, present work is based on RANS model with K-ω SST turbulence which may not accurately predict precise location of shock wave. So, some other refined simulations models may be used to predict the shock location on the launch vehicle. It is also found that shock wave- turbulent boundary layer interaction is sensitive to wall temperature. In the present work, effect of wall temperature on shock location is not studied. However in the future works, these factors can be considered and studied to precisely predict the shock waves over the launch vehicle.

3.5 Surface pressure Data

![Normalized surface pressure along the body in the mid plane Z = 0. The critical parts of the body, spherical, conical, cylindrical and boat-tail are identified.](image-url)
Figure 8 shows surface pressure on the core body along the wall on z=0 mid plane. The arc length of
the point from the nose stagnation point normalised by the core body diameter is shown along the X-
axis. The coordinates along the Y-axis show the non-dimensional pressure, which is the ratio of surface
pressure to freestream pressure. For Mach number 0.95 stagnation pressure is 1.789 from the isentropic
relations. At nose, where stagnation pressure is seen, the value is 1.7862 for AOA 0° computed from
simulation. The error in the prediction of stagnation pressure at nose is 0.067%. The plot shows that as
the flow expand around spherical region away from nose, the pressure drops. Further expansion of flow
in conical region, the pressure continue to decrease. However from the start of cylindrical region the
pressure progressively increase and sudden jump in pressure is seen at s/D=2.5. This sharp jump is due
to formation of shock wave in cylindrical region for AOA 0°. For AOA 4°, the sudden jump is seen at
of s/D= 1.3 over heat shield region. This is the region where shock is formed. Shock oscillations are
strong for AOA 4° due to which, the location of shock wave is hard to capture in steady state solutions.
It implies that unsteady simulations are required for precise prediction of shock position. The
downstream flow behind the shockwave, yields increase in pressure in boat tail region. After that
pressure decreases.

3.6 Turbulent kinetic energy

Figure 9: Contours Turbulent Kinetic energy taken from Z=0 plane for (a) AOA 0° (b) AOA 4°.

Fig 9 (a), (b) shows TKE contours for z=0 plane for AOA 0° and AOA 4° respectively. For AOA 0°
vortices formation are not prominent hence the energy associated due to turbulence is low. As the angle
of attack increases, the flow around mixes between the heat shield region and nose of booster leading
to velocity fluctuations causing formation of vortices. The vortices are dominant in the flow over launch vehicle as AOA increases and these accounts for high turbulence energy in the flow shown in fig (b). The intensity of turbulence is more prominent in windward side.

Figure 10: Contours of turbulent kinetic energy at three locations of X for AOA 0° (a), (b), (c) and AOA 4° (d), (e), (f).

Three sections X = 0.103m, 0.153m, 0.403m are chosen to turbulent kinetic energy. The boat tail region, the centre between the boat tail and the nose of the rocket, and the end of the launch vehicle configuration are represented in these sections. The turbulence in the first part is fairly minimal, thus the effects of asymmetry cannot be seen clearly. The impact of asymmetry in the flow can be seen in the X = 0.153 m section by the shift in the position of vortices to the right side. Four vortices can be seen when AOA = 0° and they are symmetrically placed shown in fig (a), (b). With AOA = 4° is used, the vortices travel much closer to each other, and the size of the vortices increases as well as shown in fig(d), (e). The influence of pressure variations caused by the presence of boosters is seen in the X = 0.403 m section shown in fig (c), (f). Locally high-pressure air characterises the vortex zone, exerting greater strains on the launch vehicle. The symmetry of the flow is disrupted as the AOA increases, resulting in a shift in the position of the vortices seen in fig (e). The vortices are also more apparent on the windward side and less noticeable on the leeward side.

4. Conclusion

Transonic flow field over a launch vehicle which consists of a core body and two strap-on boosters computed using commercial CFD package, ANSYS – FLUENT. The freestream conditions are Mach number = 0.95, Temperature =254.1 K, pressure =96900 Pa. Simulations are performed for Angle of Attack (AOA) 0 and 4. Three-dimensional simulations are performed to account for AOA. Compressible RANS equations are solved with a density-based algorithm. K-ω SST turbulence model is used for turbulence computations. Structured grids are employed for discretization. Fine mesh is used...
in the nose stagnation region, heat shield, and boundary layer so that flow gradients are captured properly.

The resulting grid is having 265,158,105 cells in the X, Y, and Z directions respectively. The first cell is placed at a distance of 0.0009mm from the surface and the resulting y+ value is below 2 over the majority portion of the vehicle. The flow field is visualized in terms of Mach number, static pressure, and turbulent kinetic energy (TKE) contours. From Mach number and static pressure contours, critical flow features such as expansion, shock wave, boundary layer, and separation region are identified. For AOA 0°, a normal shock wave is located at a distance of 0.864m from the origin of the heat shield part. The downstream of the shock wave, the flow separates and resulting in a complicated flow pattern. The separation is identified with the aid of velocity vectors. An abrupt increase in the surface pressure is noticed at the shock wave location. The further flow field is analysed at a different axial location with the help of Mach number and TKE contours. For AOA 4°, an oblique shock wave is formed and located at the beginning of the heat shield region. Flow separation is noticed downstream of the shock wave. The separation region extends up to boosters. The resulting shear layer hits the nose of the boosters. Comparisons of Mach number and TKE contours at different axial locations showed strong asymmetry in the flow field. The flow structure in terms of density distribution compared with experimental flow visualizations for identical conditions for AOA 0° and 4°. The flow structure compared very well for AOA 0°, however, for AOA 4°, a deviation is noticed in terms of the shock wave position and associated flow features. In the experiments, it was observed that for AOA 4°, shock waves continuously fluctuating on the heat shield region. This suggests that transient simulations need to be performed to predict the shock wave position accurately.

Nomenclature:

| Abbreviation | Description |
|--------------|-------------|
| CFD          | Computational Fluid Dynamics |
| RANS         | Reynolds Average Navier Stokes |
| AOA          | Angle of attack |
| κ            | Turbulent Kinetic Energy, J/kg |
| ω            | Specific turbulent dissipation rate, 1/s |
| SST          | Shear Stress Transport |
| Cd           | Coefficient of drag |
| TKE          | Turbulence Kinetic Energy |

Acknowledgments

The authors acknowledge High Performance Computing centre and CAD lab of SRM Institute of Science and Technology, Kattankulathur for providing computational resources for performing the numerical simulations.

References

[1] Bigarella, E. D. V and Azevedo, J. L. F (2005), “Numerical Study of Turbulent Flows over Launch Vehicle Configurations”. *Journal of Spacecraft and Rockets*, **42**(2), 266–276

[2] Mani, M., Naghib-Lahouti, A., & Nazarinia, M. (2004), “Experimental and numerical aerodynamic analysis of a satellite launch vehicle with strap-on boosters”. *The Aeronautical Journal*, **108**(1085), 379–387
[3] K. P. Singh, T. S. Prahlad, “Numerical Simulation of Inviscid Supersonic Flow over a Launch Vehicle with Strap-On Boosters”, 25th AIAA Aerospace Sciences Meeting AIAA-87-0213

[4] R C Mehta (1997), “Flow field study over a bulbous payload shroud in transonic and low supersonic mach number”, 15th Applied Aerodynamics Conference AIAA paper 2256

[5] Yanamashetti, G., Suryanarayana, G. K., & Mukherjee, R (2017), “Development of Flow over Blunt-Nosed Slender Bodies at Transonic Mach Numbers”. Journal of Physics: Conference Series, 822, 012071

[6] Murugan K.N., Arunkumar, T., Prasath, M. and Ganesan, V.R., (2018), “Shockwave oscillations over the conical heat shield region of a typical launch vehicle at Mach 0.95”, Symposium on Applied Aerodynamics and Design of Aerospace Vehicles (SAROD).