NUMERICAL INVESTIGATION OVER A TYPICAL LAUNCH VEHICLE WITH PROTRUSION

M. Sasi Kumar[1], R. Santhanakrishnan[1], M. Makesh Kumar[2] and S. Karthick Ram[1]
[1]Department of Aeronautical Engineering, SNS College of Technology, Coimbatore, Tamil Nadu, India
[2]Department of Mechanical Engineering, Sri Guru Institute of Technology, Coimbatore, Tamil Nadu, India

ABSTRACT

The effect of shape of protrusion on aerodynamic pressure distribution at subsonic to supersonic. The grid generation is carried out using multi block structured grid near the wall to capture the viscous boundary layer. Computational Fluid Dynamic Simulations of flow over a typical launch vehicle protrusion has been carried out using CFD++ RANS solver. K-epsilon turbulence model has been used for turbulence closure. The flow features like distribution of pressure, Mach number, shock –boundary layer interaction, expansion waves has been captured. Simulation have been also been performed to understand the flow features and to access the force over typical launch vehicle protrusions at selected Mach number.

Keywords: protrusions, separation, reattachment, SWBLI, RANS.

1. INTRODUCTION

Satellite launch vehicles external surfaces are embedded with wire tunnels, destruct channel, feed lines, antennae, Stringers, and pressurizations ducts etc. The protrusions are likely to disturb the local aerodynamic flow field distribution and the overall aerodynamic characteristics of the vehicle. The locations of protrusions become important depending on size and shape. The size of the protrusion is small compared to the size of the vehicle. In that sense strap –on boosters are not considered as a protrusion.

In general coefficient of drag increases due to protrusion .but the normal force coefficient can increase or decrease depending on the size, shape and location of the protrusion. If the protrusion is in the fore body and size is large, then they would affect the contribution of the Strapons stages, fins and other protrusions if they are in the flow field created by the fore body protrusion.

The effect of protrusions can be studied in wind tunnel, but there scale restriction exists. Hence, as a preliminary effort, a literature survey was made and it was found that most of the protrusions were studied by mounting them on flat plates. Most of the studies are carried out in supersonic, transonic, subsonic Mach number condition.

Therefore in this the protrusionsof both transonic and supersonic Mach numbers are carried out to understand the local flow field nature at the critical Mach number regimes faced by a launch vehicle.

2. CONFIGURATION DETAILS

The Protrusions configuration comprises of a blunt nose, sharp edge, semi cone, compression corner, Expansion corner, compression corner followed by expansion corner, etc. Typical Protrusion on a launch vehicle is shown in Figure-1, There are some many protrusion on launch vehicle used for various purpose like clamping, communication, fuel filling, ground servicing and fuel draining/venting, etc.

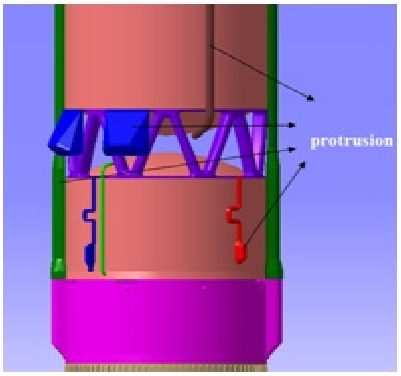

Figure-1. Images of protrusions on launch vehicle.

a) Configuration modeling

The Protrusions configuration was modeled using CATIA V5, multi-platform CAD commercial software. It is a feature-based, parametric solid modeling design tool used to create fully associative 3D solid models, with or without constraints, while using automatic or user-defined relations to capture the design intent.

The geometrical details of protrusions are given in Figure-2. The protrusions are mounted on a Flat surface are modeled. To study the wall proximity and interference effect's.

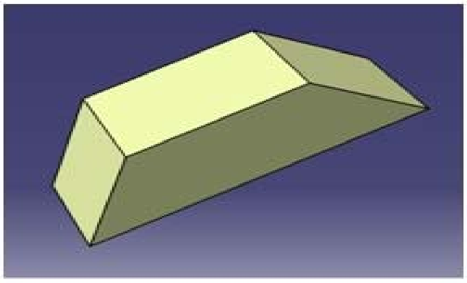

Figure-2. Geometrical of Protrusion.

3. GRID GENERATION

The multi block grid was generated using Pointwise grid generation software.

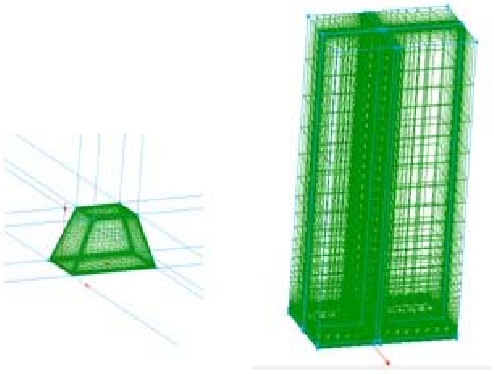

Figure-3. Structured gridsover protrusion configuration.

4. SIMULATION AND BOUNDARY CONDITIONS

Turbulent flow is simulated with a free stream Mach number of 0.8 AND 2. Realizable k-ε turbulence model is used to close the RANS equation. The boundary condition are set in CFD++ solver preprocessor for all the domain (inflow, outflow, far field, wall, symmetry) of the volume grid. The pictorial representations of applied boundary condition are shown in Figure-4. The physical boundary conditions prescribed on the domain are given in Table-1.

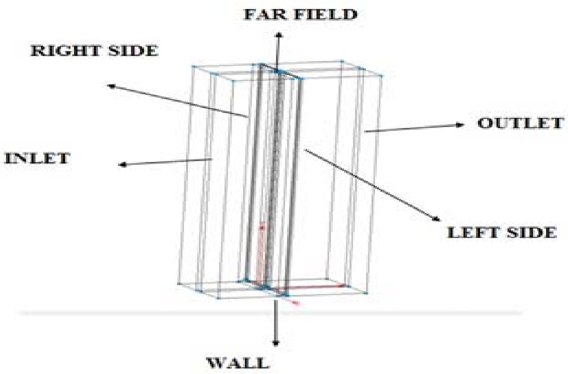

Figure-4. Boundary condition.

Table-1. Boundary condition used for CFD++ simulation.

Boundary	Description	Condition
Inflow	Subsonic/Supersonic Inflow	Characteristic based Inflow/ Outflow
Outflow	Subsonic/Supersonic Outflow flow	
Top	Far field	
Side	Far field/Symmetry	
Plate and Model	Wall	Adiabatic Viscous wall

5. RESULTS AND DISCUSSION

CFD simulations have been carried out for the various protrusion configurations using *CFD++ RANS* solver with realizable *k-ε* turbulence model at transonic and supersonic Mach numbers. The results have been analyzed after the convergence of the residual.

a) Volume lines and streamlines over protrusion at M=0.8 and 2.0

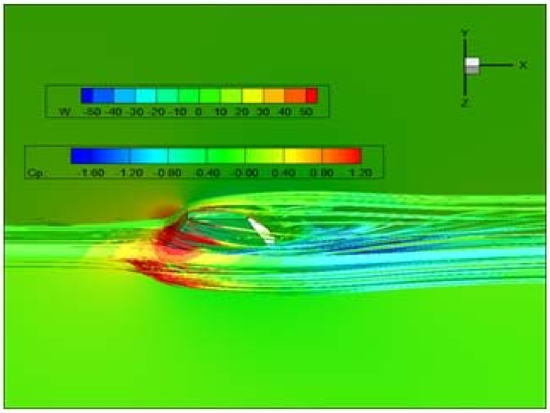

Figure-5. Volume lines and C_p distribution over Protrusion at M=0.8.

Figure-5 shows volume lines and c_p over the symmetry plane. The volume lines show flow separation ahead of the protrusion, on the top face, sides and in the wake region. Symmetry plane Cp palette indicates rise in pressure over the ramp and the subsequent fall over front face as well as the top face. Low pressure prevails in the base region.

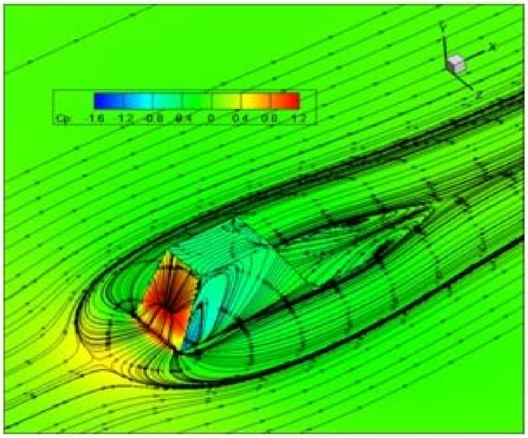

Figure-6. Streamlines over Protrusion at M=0.8.

Figure-6 shows the surface streamlines over Protrusion and over the flat plate. The flow is separated ahead of protrusion and reattaches over the forward facing ramp surface. The separation distance in this 3D flow is slightly shorter (0.872D) as compared to that in 2D (0.97 D). There is a complex updraft of flow on the side plate/

surface. A small corner vortex is seen on the sides of the protrusion and is stretched along the flow direction. The top portion of Protrusion / the expansion corner following the forward face is submerged in separated flow. Separated flow region exists in the lateral sides also. The base region shows long and narrow separated flow region.

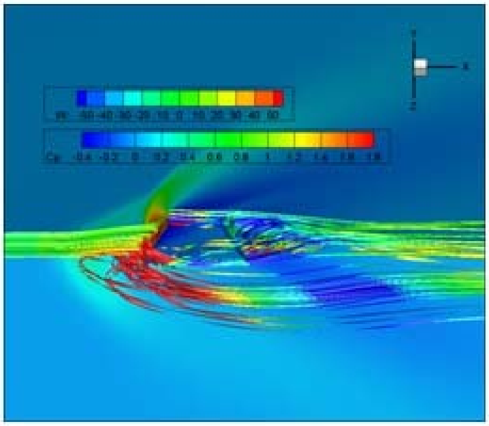

Figure-7. Volume lines and C_p distribution over Protrusion at M 2.0.

The volume lines in Figure-7 indicate separated flow region ahead of the protrusion, bifurcation of flow over the forward face, the flow movement around the protrusion and in the lateral sides and the complex flow pattern in the base. In the symmetry plane, the oblique shock induced due to separation region ahead of the protrusion, the high Cp in the reattachment region, drop in Cp over top face and base region are clearly seen.

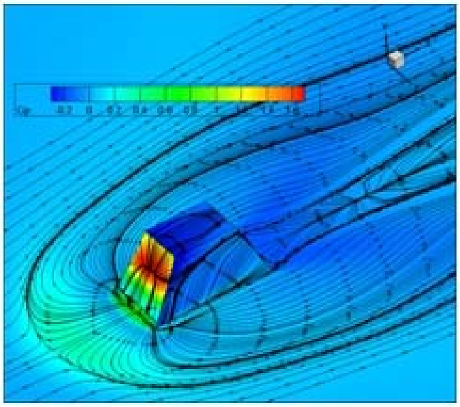

Figure-8. Streamlines over Protrusion at M= 2.0.

Figure-8 shows streamlines over plate and Protrusion at M=2. At M=2, the flow is separated well ahead of the ramp and reattaches on the ramp face. Separation region at M=2 is higher as compared to M=0.8 case. The separation distance in this 3D flow is very short (1.434 D) as compared to that in 2D (3.237 D). There is a small counter rotating flow ahead of the ramp. The flow is swept in upward direction in the lateral sides. A counter-rotating vortex stretching along the sides is also seen. The flow is attached over the top face of the protrusion for this Mach number. The base recirculation region is very narrow, and lengthy.

b) Force coefficient

The integrated force coefficient is given for protrusions in Tables-2 for the given protrusion. The entire force coefficients are normalized to reference area of 1 sq.m. Normal force coefficient of protrusion is positive at M=0.8. The value decreases continuously with increase in Mach number and becomes negative.

Table-2. Force coefficient for Protrusion.

M	C_A	C_N
0.8	0.0698	0.0265
1.05	0.0835	0.0243
1.2	0.0895	0.0185
2.0	0.0795	-0.0107

6. CONCLUSIONS

Flow over a 3D protrusion exhibits rich flow features topologies such as separation and reattachment lines and junction vortices, wake vortices, and horseshoe vortices. With this study as comprehensive data base related to compression and expansion corner at subsonic to supersonic Mach number has been generated provably for the first time.

7. SCOPE OF THE FUTURE WORK

3D simulation was carried out to get an understanding of the flow field around a typical launch vehicle protrusion at transonic and supersonic Mach number. Some of the suggestion work is outlined below:

- Effect of various turbulence models can be investigated.
- Experimental work can also be carried out.
- Configuration of other protrusion simulation work can be carried out.

REFERENCES

[1] Asmelash Haftu Amaha and Amarjit Singh. 2011. “Numerical Analysis of Shock Wave Turbulent Boundary Layer Interaction over a 2-D Compression Ramp”.

[2] Oliver A. B. Lillard, R. P. Blaisdell, G. A. and Lyrintzis A. S. 2008. “Effect of Three – Dimensionality in Turbulent Compression Ramp Shock –Boundary Layer Interaction”, AIAA Paper No.2008-720, January.

[3] Oliver A. B. Lillard, R. P. Schwing, A. M. Blaisdell, G. A. and Lyrintzis A. S. 2007. “Assessment of Turbulent Shock Boundary Layer Interaction Computational Using The Over Flow Code”, AIAA Paper No.2007-0104, January.

[4] Kung-Ming Chung. 2002. “Investigation on Transonic Convex-Corner Flows”. November-December.

[5] Menter F. 1997. Grotjans H. And Unger F. 1997. “Numerical Aspect Of Turbulence Modelling For The Reynolds Averaged Navier-Stokes Equation”, VKI Lecture Series, March.

[6] Panov Yu. A. and Shvets A. L. 1966. “Separation of A Turbulent Boundary Layer in A Supersonic Flow”, Prikladnaya Mekhanika, Vol. 2, No, 1, pp. 99-105.

[7] Usebelkack H. T. ”Turbulent Flow Separation Ahead of Forward Facing Steps In Supersonic Two Dimensional And Axisymmetric Flows”, Von Karman Institute For Fluid Dynamics, Technical Note 64.

[8]Zukoshi E. E. 1964. “Turbulent Boundary Layer Separation In Front of a Forward Facing Step”, AIAA Journal, Vol.5, No.10. pp. 1746-1753.

[9] Chapman D. R., Kuhen D. M. and Larson H. K. 1957. “Investigation of Separated Flows in Supersonic and Subsonic Streams with Emphasis on the Effect of Transition”, NACA Technical Notes No. 3869.