

# Numerical Studies On Effect Of Changing Mach Number On Aero Disk Model On Flow Contours And Drag Force

**K V N B Sai Krishna Mohan, Kushaal Bandaru and Prabhudev B M <sup>1</sup>**

Department of Mechanical Engineering, Amrita University, Bengaluru, India.

E-mail: bm\_prabhudev@blr.amrita.edu.

**Abstract.** A supersonic aircraft needs to travel in the atmosphere at different Mach numbers. In this paper the effect of Mach number on a body in hypersonic flow is studied. Also the effect of scaling on the flow behaviour is studied. As the original geometry taking more computational time, scaled model is used for further research. The geometry is modelled, meshed and solved in Ansys tool. Results are compared with available literature and further parametric study is done. There was no appreciable change found in the results obtained by scaled model compared with actual model. Hence Actual body<sup>[1]</sup> was scaled to 1/10<sup>th</sup> of its original scale body and simulated at different Mach numbers. The spike of the cone is fitted with an aero disk. Results shows that the angle of the bow shock inclined to body is found to be decreasing as the Mach number is increasing. Two separate shockwaves from disc and cone were found merged at higher Mach numbers. At starting Mach numbers the separation region was unaltered but after Mach number 3, Mach number contours show that flow deflected from aero disk has greater role in deciding the separation region ahead of cone. Further surface pressure graphs are plotted and effect on drag force were discussed.

## 1. Introduction

So much work has been carried out from past to till date on studying different designs of aerodynamics of the spiked hypersonic vehicles. As we know that the front part i.e. nose of the hypersonic vehicles is larger, so the drag force produced by this is very high. This also increases the heat flux on the surface and result in reducing the overall performance of the vehicle. This drag can be reduced by attaching a spike on the nose of the body. The aerodynamics of spike on high speed vehicles is in such a way that it is able to withstand severe heating conditions. Many experimental investigations were done with different aerodynamics for the drag reduction. The investigations include that the drag on the spiked body is influenced by the Reynolds number. A blunt body creates a bow shock when travelling at higher Mach numbers, which creates a very high pressure and results in the high drag during the high projectile's flight through the atmosphere. It is good to have low drag which in turn minimizes the thrust required from the propulsion system during the supersonic and hypersonic regime. Presently, much experimentation is being made on the spike with aero disc in the hypersonic blunt body, it is held at stagnation point and projected upstream side. By means of aero

<sup>1</sup> bm\_prabhudev@blr.amrita.edu.



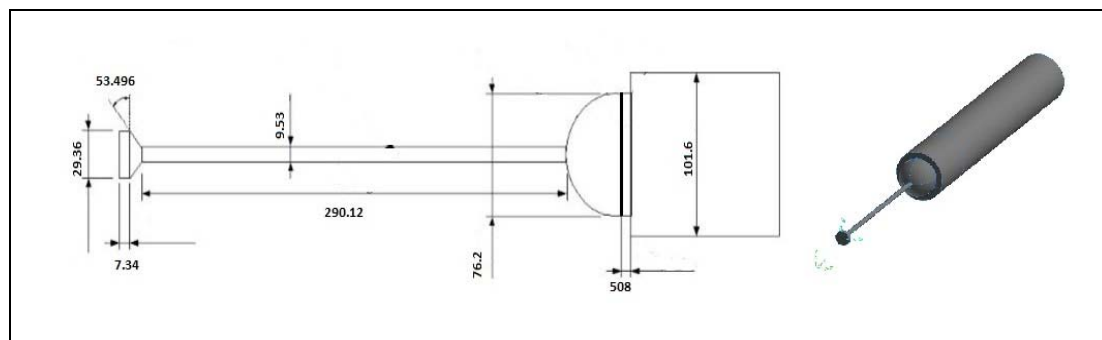
disc with spike, the strong bow shock has changed into weak. Hence this aero disc is helpful in producing oblique shock which in turn reduces the drag force. The high temperature regions in the flow field around a blunt-nosed body and the massive amount of flow kinetic energy in a hypersonic free stream are converted into internal energy of the gas across the strong shock wave, hence reducing very high temperatures in the shock layer nearer the nose. The dynamic pressure on the surface of the blunt body can be reduced by creating a low pressure region in front of the blunt body by placing the spike.

## 2. Literature Review

A flow field around two aero spike hemisphere-cylinder configurations has been successfully simulated by Roberto Roveda[4] for a high-supersonic free stream Mach range and some angles of attack. The first model was simulated with the conditions specified by Huebner[9] and the second model was simulated with the conditions specified by Gnemmi[10]. His observations explain the variation in results of Numerical tools CFD++ (k- $\epsilon$ -Rt) and Cobalt (k- $\omega$ ) under normalized static pressure in xy plane and observed the variation of shear zone with different angle of attacks. The effects of secondary spike on drag and heat flux reduction is studied in the paper Reference[8]. The results also shows that the secondary spike can be employed for the drag and heat flux reduction. In Reference[7] detailed work on the heat flux distribution over blunt body section of a re-entry model for Mach number 10.1 is shown. For the L/D ratio 1.5, it is observed that the local heating at the leading edge of the aero spike was higher than the heating which would have occurred at the nose without spike. The reduction of heat flux was found to be 27% in case of hemispherical tip and 31% for the conical tip. These values changed to 33% and 29% respectively when a forward jet was injected from the nose. The flow over spike will experience buzz phenomena which was simulated and Shown in Reference [9]

## 3. Geometry and Simulation

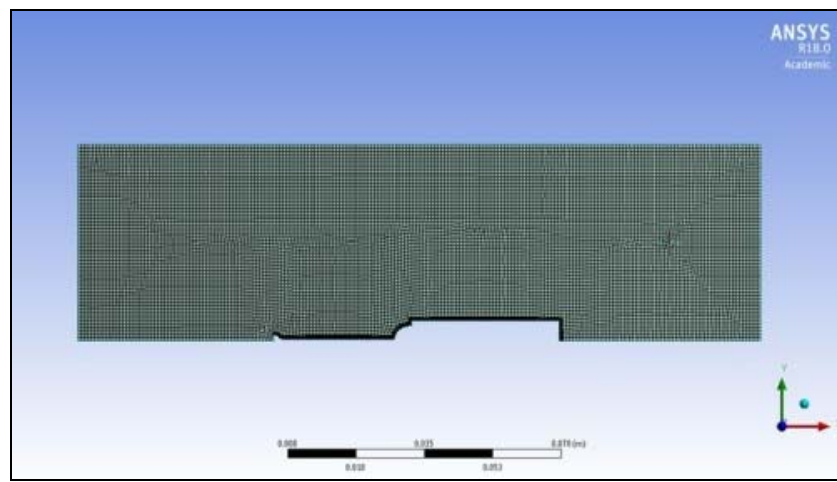
In our study we have used the same geometry which was used by Huebner<sup>[9]</sup> in his experimentation same physics has been simulated by Roberto Roveda<sup>[1]</sup>. Fig.1 shows the two dimensional view of the geometry that we have used to simulate in our case.



**Figure 1.** Geometry of Spiked body from Roberto Roveda<sup>[1]</sup>

Then the geometry is meshed using Ansys meshing. The original geometry size (~1m) was scaled to Ten times smaller geometry (~0.1m) to see effects of scaling on flow simulations and to get the benefit of obtaining results in lesser computational time. Both geometries were meshed for same mesh parameters and boundary conditions. The mesh has all its elements as quadrilateral element but very few triangular elements in between as seen in Fig.2 due to the variant shape of the profile. Mesh

density is increased near the areas of interest to obtain the exact results. In order to check the reliability on our mesh three different meshes for different element sizes were simulated, among which the mesh which gave results as shown by Reference[1] and [2] is finalized to carry out our research. Further the computational flow studies were being made using Ansys Fluent by solving RANS equations with inlet and side wall as pressure far field. Axis symmetric boundary condition is used as the body is circular. Turbulent SST turbulence model is used because of its greater capacities to simulate flows near boundaries and in main stream. Convergence criteria given  $1e-4$  and took half an hour in Intel i7 processor with 7 core parallel processing.

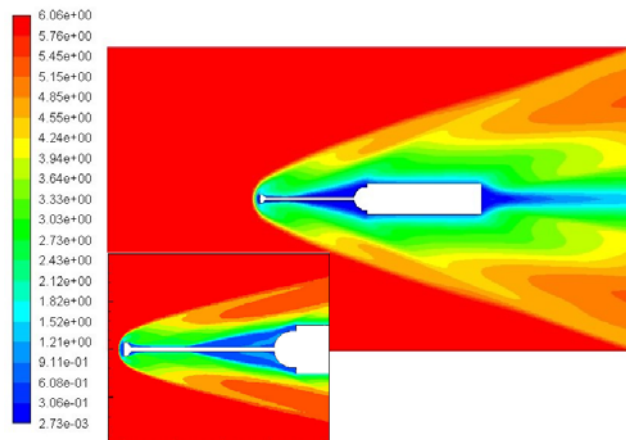


**Figure 2.** Meshed Flow Field and Refined Mesh at surface can be seen

#### 4. Results and Discussion

The author Roberto Roveda<sup>1</sup> has worked on simulating two models and effect of different codes on results comparing with the two different geometries, but it is for a fixed Mach number i.e. Mach Number 6.06. In our study we have varied Mach number 2,4,6.06,8. These values are chosen to see the higher and lower side of Mach numbers  $M=6.06$  that has been used in our reference work.

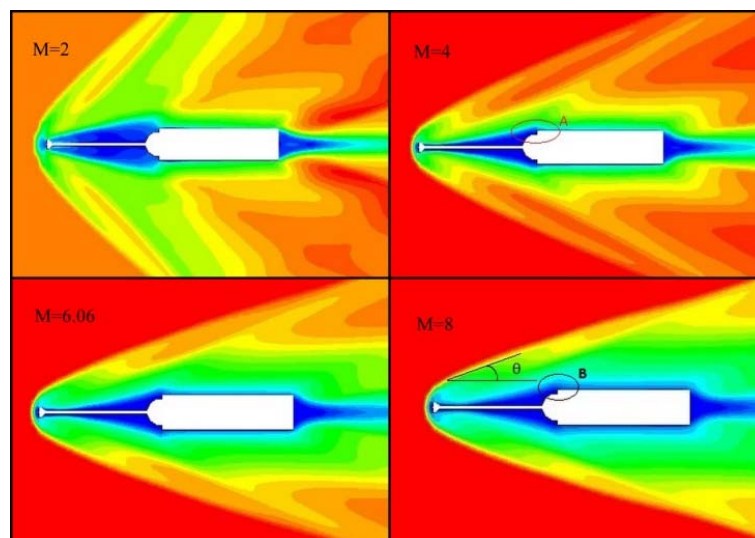
The results were validated with the temperature contours that is given in the Roberto Roveda<sup>1</sup> and it is seen that the flow simulations that are carried out are matching with the results given by Roberto Roveda<sup>1</sup> and is shown in Figure3



**Figure 3.** Validation of Mach Number contour (a) with the contour from Roberto Roveda<sup>[1]</sup> PIP (b).

So in further studies we have found that with the increasing Mach number the inclination of shockwave with body decreases it is evident in the Fig 4 this effect can also be seen in Results of Reference [8], this means the shock wave is coming close to the body which increase thermal loads on the body.

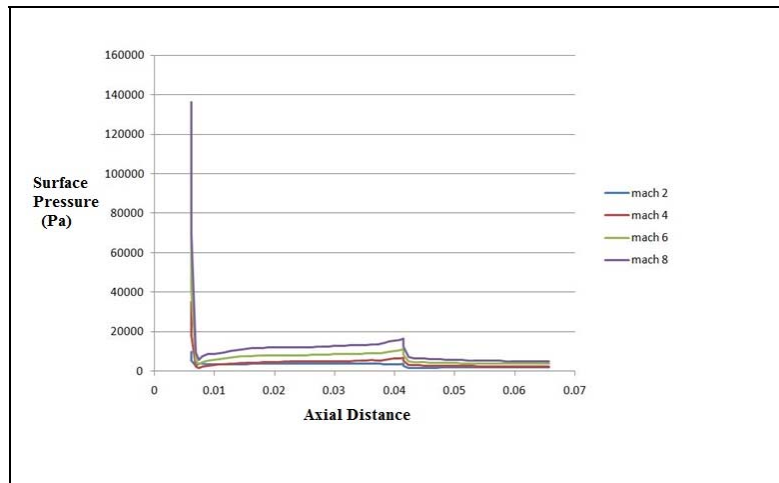
It is also observed that the region of shear separation near base of spike is influenced by the deflected flows behind the aero disk. At higher Mach numbers more flow is circulating along the spike and body because of deflected flows and smaller separation region. Flow has detached from location of step in the main body as seen in Mach number 4 contour as shown in Figure 4(Mach contour locations A and B in the Figure).



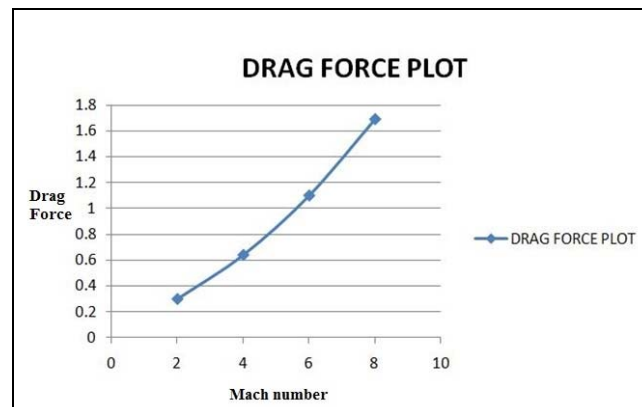
**Figure 4.** Contours of Mach number on the stepped spiked body

From surface pressure graph Fig.5 it is evident that at the base of hemisphere is showing high pressure loads as usual the tip is in its peak pressure loads. Graph also shows that pressure loads increases with

Mach number. From Fig.6 we see drag force is more at high Mach number and decreases with decreasing Mach number



**Figure 5.** Variation of Surface Pressures along the Specimen



**Figure 6.** Variation of Drag force with respect to Mach number

## 5. Conclusion

The overall purpose of the present study was to demonstrate the effect of Mach number variation on stepped hypersonic vehicle with aero disk

1. As the Mach number increases the inclination of shock wave with body decreases
2. It is observed that the region of shear separation near base of spike is influenced by the deflected flows behind the aero disk.
3. At higher Mach numbers more flow is circulating along the spike and around the body because of deflected flows and smaller separation region is seen.
4. Increase in Mach number results in increase in the thermal load on the body of the hypersonic vehicles.

## Acknowledgments

Authors wishes to express their Gratitude to Department of Mechanical Engineering, Amrita School of Engineering Bengaluru, Amrita Vishwa Vidyapeetham for providing facilities and support to carry out this work

## 6. References

- [1] Roberto Roveda, “*Benchmark CFD Study of Spiked Blunt Body Configurations*,” American Institute of Aeronautics and Astronautics 5 - 8 January 2009.
- [2] Bogdonoff, S. M., and Vas, E. I., “*Preliminary Investigations of Spiked Bodies at Hypersonic Speeds*,” Journal of the Aeronautical Sciences, Vol. **26**, No. 2, 1959, pp. 65-74
- [3] Crawford, D. H., “*Investigation of the Flow Over a Spiked-Nose Hemisphere-Cylinder at a Mach Number of 6.8*,” NASA TN-D118, Dec. 1959.
- [4] Mikhail, A.G. “*Spike-Nosed Projectiles: computations and Dual Flow Modes in Supersonic Flight*,” Journal of Spacecraft and Rockets, Vol. **28**, No. 4, July-August1991, pp. 418-424.
- [5] Mikhail, A.G. “*Spike-Nosed Projectiles with Vortex Rings: Steady and Nonsteady Flow Simulations*,” Journal of Spacecraft and Rockets, Vol. **33**, No. 1, January-February1996, pp. 8-15.
- [6] Reding J. P., Guenther, R. A., and Richter, B. J. “*Unsteady Aerodynamic Considerations in the Design of a Drag-Reduction Spike*,” Journal of Spacecraft and Rockets, Vol. **14**, No. 1, January 1977, pp. 54-60.
- [7] G.Gopala Krishnan, Akhil J, Nagaraja. S. R “*Heat transfer analysis on hypersonic re-entry vehicles with spike*,” Proceedings of the 23rd National Heat and Mass Transfer Conference and 1st International ISHMT-ASTFE Heat and Mass Transfer Conference 17-20 December, 2015.
- [8] N. Sreekanth, J. Akhil, S.R.Nagaraja. “*Design and Analysis of Secondary Spike on Blunt Head*” Indian Journal of Science and Technology,” Vol. **9(45)**, December 2016.
- [9] R. Ramachandran, R. Sirangala Ganesh. “*Effect of dimensions of sharp spiked cylinder on the buzz phenomenon subjected to hypersonic flows*,” International Journal of Fluid Mechanics Research, **44(5)**: 1–17 (2017)
- [10] Huebner, D. L., Mitchell, A. M., and Bodreaux, E. J., “*Experimental Results on the Feasibility of an Aerospike for Hypersonic Missiles*,” AIAA Paper **95-0737**.
- [11] Gnemmi, P., Srulijes, J. and Roussel, K. “*Flowfield around Spike-Tipped Bodies for High Attack Angles at Mach 4.5*,” Journal of Spacecraft and Rockets, Vol. **40**, No. 5, September-October 2003, pp. 622-631.