International Journal of Research
Available at https://edupediapublications.org/journals
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 05 Issue 07
March 2018

Report on CFD Analysis of a Spiked Blunt Body

Saad Ahmed, Mohammed.Samiuddin Khan, Khaja Hamiuddin Mujahid
Mohd Ayazuddin, Mr. Sunil Manchala

ABSTRACT: *Supersonic vehicles operate at very high speeds and are exposed to problem of drag and heating. Employment of aerospike in the current body can reduce problem of drag and heating of body. Imposed bluntness at the nose of the supersonic vehicle is necessary to alleviate the oncoming heat load.In current analysis, flow around a blunt body which is fitted with an aerospike is analyzed using software ANSYS Fluent, at a high Mach number (M=4.5) at an angle of attack 0°and 15°. Aerospikeis placed in front of the body, this tackles strong shock waves. The region of recirculation develops which is depicted in contour plots below.This acts as as streamline contour thus reducing drag and wall heat flux.*

1. INTRODUCTION:

To reduce drag force and high heating, vehicles like space planes,missiles and interplanetary probes which fly at super and hypersonic speeds usually employ blunt nosed bodies. The greatest challenge faced in designing such vehicles is reduction of both drag and aerodynamic heating. For high speed supersonic vehicles, blunt geometries are given superiority over slender due to increased volumetric efficiency, effective accommodation of crew or on-board equipment

This blunt body creates a bow type shock wave ahead of it at high Mach numbers. Now an aerospike is fitted infront of this blunt body which reduces drag acting on body. This flow surpassingaerospike creates a conical shock wave, replacing the bow shock, which will be far away from the body. Flow behind the shock wave separates on the aerospike and a recirculation region is created near the body surface.

In the present study, high speed flow at a Mach number, M = 4.5, past a blunt body fitted with aerospike at 0 and 15 degree angle of attack are studied numerically using CFD analysis.

2. LITERATURE SURVEY:

Langley pilotless aircraft division is said to be the first to suggest aerospike for drag reduction on blunt bodies at supersonic speeds [1]. Study performed by Mair is said to be the first landmark study in this field [2]

He studied experimentally the flow around spiked flat cylindrical and hemisphere cylindrical models at a Mach number of 1.96 and Reynolds number of

1.65x105. He recorded flow instability around the spiked bodies and proposed an explanation for this flow oscillation based on the pressure difference between the flow downstream of the reattachment shock and the flow inside the recirculation zone. The label 'spike' was used first by Piland and Putland [3]. They concluded that no drag reduction was achieved for transonic range at Reynolds numbers from 1.44x105 to 2.64x105 flight conditions regardless of the aerospike lengths. Jones [4] studied that the drag reduction depends on both rod length and aero disk geometry. For longer rods, the aero disk was found to lose its benefits in reducing the drag. It is seen that, majority of the investigations regarding the use of aerospike or aero disks are limited to the assessment of drag reduction abilities. Stalder and Nielsen [5] carried out the first study on the aero-thermodynamic effects of the spikes and they measured the heat transfer to a hemisphere cylindrical model for Mach numbers ranging from 0.12 to 5.04.

They concluded that regardless of the length of aerospike and tip geometry, heat transfer to the spiked model was as twice as that of the unspiked model. They found the reason for the heat transfer rise to be the high turbulence level in the dead air zone and impingement of turbulent shear layer on the model surface. Crawford performed an experimental study of drag and aerodynamic heating on a spiked hemisphere cylindrical model in Mach 6.8 flow [6].

The model's surface pressure and heat flux attained peak values at the reattachment point. Yamauchi et al [7]. studied a numerical simulation of the flow field around an aerospike fitted blunt body at free stream Mach numbers of2.01, 4.14 and 6.80 for different values of L/D. Holden studied that the peak local heat transfer rate at the reattachment point is in a direct proportionality with the reattachment angle [8]. Khlebnikov inferred that the value of peak heat flux varies inversely with its distance from the root of aerospike [9]. The studies done by Gerdroodbary et al. [10] shows a reduction in the total heat transfer rate by employing hemispherical aero disks in turbulent flow conditions. The shortest aerospike should be avoided as this length may lead to the same impingement location of bow shock and the reattachment shock.Paskonov et al. [11] solved the complete unsteady Navier-Stokes equations in a 2-D axisymmetric flowfield around cone cylindrical and flat cylindrical models equipped with pointed aerospikes of length to diameter ratios varying up to 1.Subsequently, a large number of investigations were carried out to understand the effect of high speed flows past a blunt body with a projecting aerospike at the tip.

Majority of the investigations conducted for spiked blunt bodies' have concluded that the use of aerospikes can drastically reduce the aerodynamic drag at very high speeds viz., supersonic and hypersonic speeds, for certain ratios of length of aerospike to the diameter of the base body.

3. NUMERICAL MODEL:

3.1. Governing Equations

Numerical simulations were obtained for analyzing a supersonic flow past blunt body fitted with aerospike. The present analysis is steady and hence, the governing equations are solved in its steady form. Governing equations used by ANSYS Fluent [12] for the numerical analysis are given in general form as follows:

Continuity equation:

$$\partial\rho/\partial t + \nabla.(\rho v) = 0$$

Momentum equation:

$$\partial(\rho v)/\partial t + \nabla.(\rho v v) = -\nabla p + \nabla.\mu[(\nabla v) - 2/3\, \nabla.vI]$$

Energy equation:

$$\partial(\rho E)/\partial t + \nabla.(v(\rho E + p)) = \nabla.(k_{eff}\, \nabla T + (\mu_{eff}\, [(\nabla v) - 2/3\, \nabla.vI].v)$$

Where, p is the static pressure, ρ is the density, μ is the molecular viscosity, I is the unit tensor, keff is the effective conductivity, ρE is the total energy per unit volume, T is the temperature.

$E = h - (p/\rho) + (1/2)\, v2$

keff = kg + kt, the effective thermal conductivity, where kt = (Cpgμt)/Prt, Prt is turbulent Prandtl number. Last term in the energy equation represents viscous heating and μeff = μ + μt.

3.2MODELLING AND METHODOLOGY:

A. GEOMETRY: The geometry of spike is symmetric and hence taken as a 2-D model in order to save time and computational costs. Shape of control volume is taken i.e. (shown in results) so as to utilize it in pressure far-field problem

B. MESHING: Automatic method and refinement are used to mesh with relevance = 50 to get a skewness of 0.7 and orthogonality equal to 0.4 which is satisfactory to get solution of spike. Nodes obtained are 47,711 and elements are 46,992.

C. SETUP: Density based solver is used and velocity formulation is set to absolute, "2-D space planar" is selected with "steady" analysis and then mesh quality is checked.

For modeling, energy equation is turned on and viscous model SST K omega is used. Viscosity is then set to Sutherland.

In Cell Zone operations conditions pressure is set to zero and in Boundary Conditions pressure far-field values are assigned as

1. Gauge pressure = 350 Pa
2. Mach number = 4.5
3. X component of flow direction =1 &
   Y component of flow direction =0 (For angle of attack = 0 degrees cos(0) =1 and sin(0)=0)

4. X component of flow direction= 0.96 &

Y component of flow direction = 0.25

(For angle of attack 15 degrees cos(15)=0.96 and sin(15) =0.25)

5. Temperature is set to 47.52 Kelvin. In solution methods formulation is set to explicit and in spatial discretization gradient is chosen as green gauss node method, flow and turbulent is set to second order. Then in monitors residuals is selected Cd and Cl(Drag and Lift coefficients whose value depends on angle of attack) are given.

NOTE: The values of pressure and temperature were extrapolated from the formulae involving ambient pressure, ambient temperature, Mach number and the isentropic coefficient as these values were provided in the base paper.

3.3 BODY GEOMETRY:

The geometrical dimensions of the spiked blunt body used in this study are shown in Fig.(Gnemmi [13]) . The model is two dimensional, with a cylindrical fore body having a hemispherical nose and an aerospike attached to the stagnation point of the nose cone. The diameter of the cylindrical fore body, D, is taken as 70 mm. The end of the aerospike is taken as hemispherical with a diameter 17.5 mm as shown in the Fig. The length to diameter ratio is taken as 3.5 to 4 depending on the case selection.

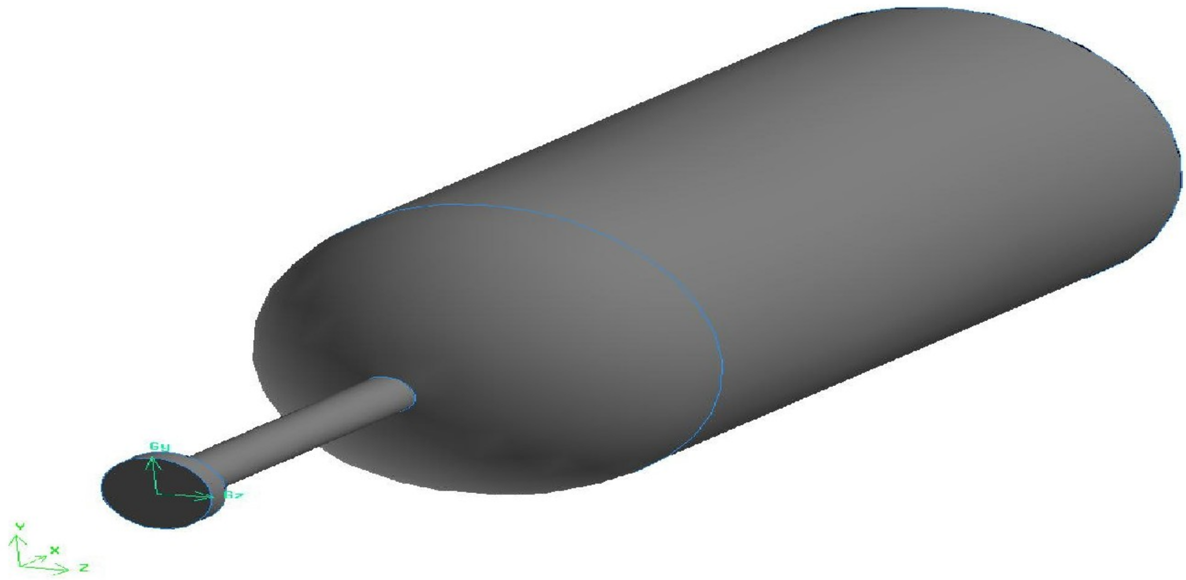

Figure 3: 3D Gambit rendition of Model

MODEL				
	Notes	Inches	mm	Deg
D Aerodisk		.6909	17.55	
L Aerodisk	Assumed	.1063	2.70	
D_Sting	Assumed	.2362	6	
L_Sting	Assumed	2.6496	67.30	
D_Seeker		2.7559	70	
L_Cylinder		0	0	
D_Missile		2.7559	70	

e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 05 Issue 07
March 2018

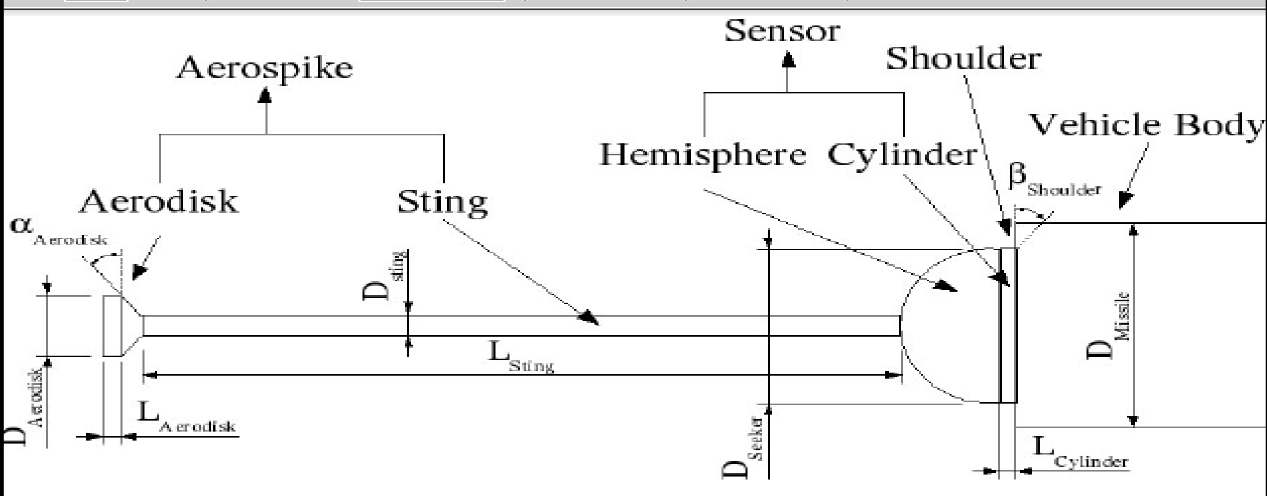

Figure 1: Sketch of generic aerospike configuration with relevant dimensions.

4. RESULTS AND DISCUSSION :-

This model uses free stream conditions identical to the wind tunnel tests detailed in (Gnemmi [13])a static pressure of 40000 Pa and static temperature of 240 K yield a free stream Mach number of 4.5.

From CFD numerical simulations carried out shows that a strong bow shock is generated at a small distance from the spiked blunt body. Flow behind the shock wave separates on the spike and a recirculation region is created near the body surface. As a result of this recirculation, the pressure and wall heat flux decrease in the region ahead of the blunt body. However, the reattachment of the shear layer on the shoulder of the blunt body increases the values of local heat flux and pressure. The pressure on the surface of the body can be significantly reduced when the aerospike replaces the bow shock with a system of weaker oblique shock waves at Mach number = 4.5 variation of flow over a blunt body without aerospike using ANSYS Fluent is shown in below two figures. A bow shock is formed in front of the body.

The model is tested at different angles at 0 and 15 degrees to get different aerodynamic coefficients i.e. value of lift and drag at the given temperature, pressure and Mach Number.

International Journal of Research
Available at https://edupediapublications.org/journals
e-ISSN: 2348-6848
p-ISSN: 2348-795X
Volume 05 Issue 07
March 2018

SHOULDERLESS AEROSPIKE (ZERO ANGLE)

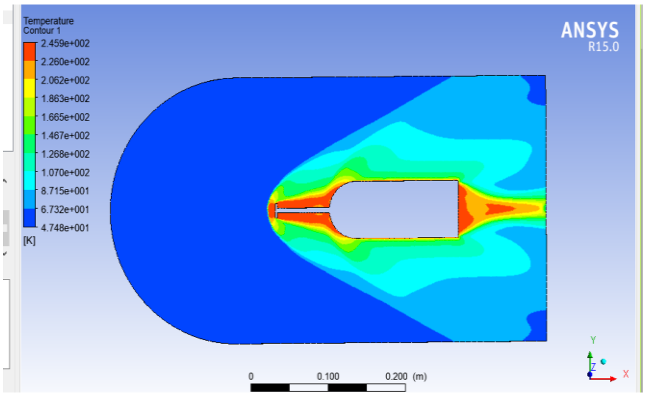

Figure 1.1 Temperature

Temperature is highest at sting and raises again at end of geometry

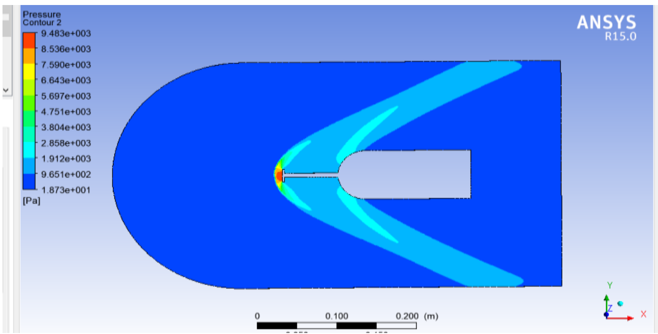

Figure 1.2 Pressure

Pressure increases symmetrically on upper and lower side and it is highest at the tip

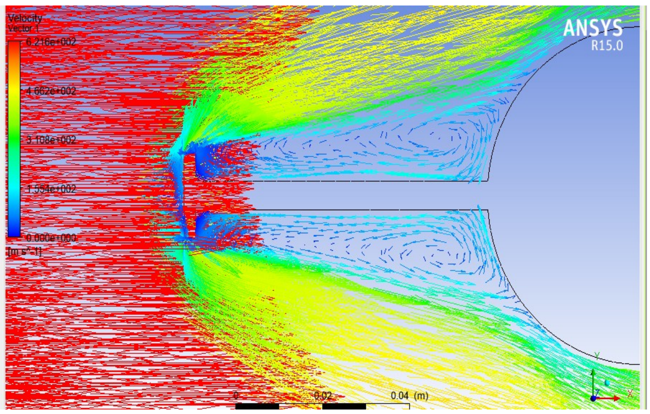

Figure 1.3 Velocity

Behind aerodisk , wake region experiences recirculation

SHOULDERLESS AEROSPIKE(AT 15 DEG)

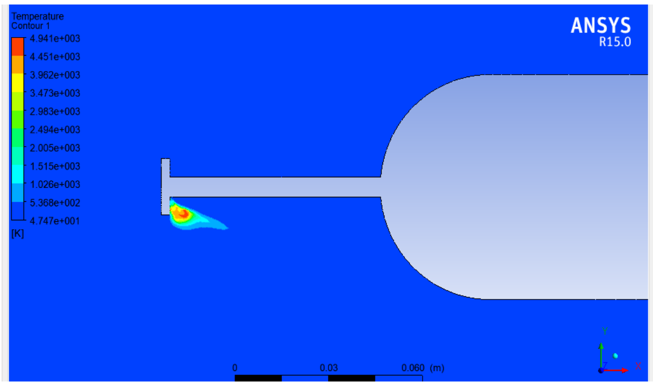

Figure 2.1 Temperature

Temperature is highest at sting part of the geometry only

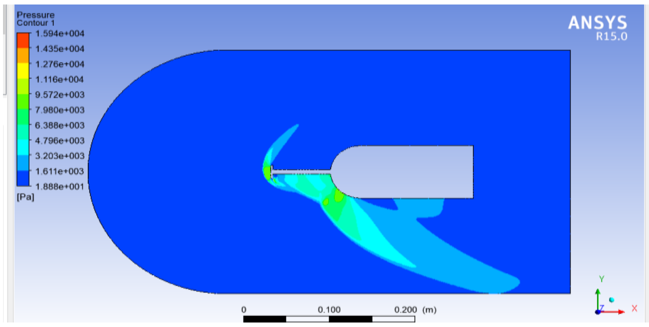

Figure 2.2 Pressure

Pressure increases along the lower side of sting and no change in pressure is on upper side

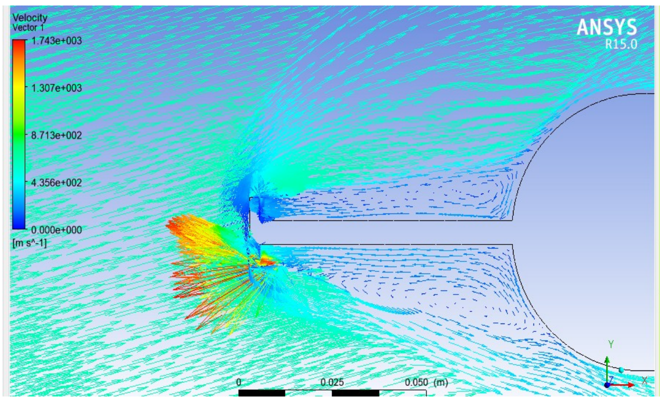

Figure 2.3 Velocity

Recirculation occurs with different values in velocity

AERODYNAMIC COEFFICIENTS(AT ZERO DEGREES)

S.NO.	Cd(Coeff. of drag)	Cl(Coeff. of lift)
1	4.41 X 10^2	-2.355 X 10^0

AERODYNAMIC COEFFICIENTS(AT 15 DEGREES)

S.NO.	Cd(Coeff. of drag)	Cl(Coeff. of lift)
1	4.73 X 10^2	8.04 X 10^2

4 .CONCLUSION :-

From results we conclude that best angle of attack for shoulder less aerodynamic spike is 0 degrees as the model is symmetrical from which we get lowest values of coefficients of drag and lift as given below. Aerospike is employed in supersonic flows where variation in drag is paramount.

1. **Cd = 4.41 * 10^2**
2. **Cl = -2.355**

5. REFERENCES :-

[1] Alexander, S. R., Results of tests to determine the effect of a conical wind shield on the drag of a bluff body at supersonic speeds, NACA RM L6K08a, 1947.

[2] Mair, W. A., Experiments on separation of boundary layers on probes in front of blunt-nosed bodies in a supersonic air stream, Philosophy Magazine, Vol. 43, No. 243, 1952, p. 695–716.

[3] Piland, R. O., Putland, L. W., Zero-lift drag of several conical and blunt nose shapes obtained in free flight at Mach number of 0.7 to 1.3, NACA RM L54A27, 1954.

[4] Jones, J. J., Experimental drag coefficients of round noses with conical wind- shields at Mach Number 2.72, NACA RM L55E10, 1955.

[5] Stalder, J. R., Nielson, H. V., Heat transfer from a hemisphere cylinder equipped with flow separation spikes, NACA TN 3287, 1954.

[6] Crawford, D. H., Investigations of the flow over a spiked nose hemisphere cylinder, NASA TN-D-118, 1959.

[7] Yamauchi, M., Fujii, K., Higashino, F., Numerical investigation of supersonic flows around a spiked blunt body, Journal

of Spacecraft and Rockets, Vol. 32, No. 1, 1995, p. 32-42.

[8] Holden, M., Experimental studies of separated flows at hypersonic speeds. Part I-separated flows over axisymmetric spiked bodies, AIAA Journal, Vol. 4, No. 4, 1966, p. 591-599.

[9] Khlebnikov, V. S., Distribution of heat flux on the surface of a sphere with a forward separation zone, Fluid Dynamics, Vol. 21, No. 4, 1986, p. 654-658.

[10] Gerdroodbary, M. B., Hosseinalipour, S. M., Numerical simulation of hypersonic flow over highly blunted cones with spike, ActaAstronautica, Vol. 67, 2010, p. 180-193.

[11] Paskonov, V. M., Cheranova, N. A., Numerical investigation of laminar separation in the case of supersonic flow of viscous gas past spiked bodies, Fluid Dynamics, Vol. 19, No.2, 1984, p. 281–285.

[12] ANSYS Inc., Fluent user's guide, [USA].

[13] Gnemmi, P., Srulijes, J. and Roussel, K. "FlowfieldAround 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.