International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 8, August - 2014

Recent Advancements in Aerothermodynamic Design Optimization of Spiked hypersonic Winged Re-Entry Vehicle using CFD

P. Harish
PG Student
Dept. of Mechanical Engineering
K.S.R.M College of Engineering
Kadapa,Andhra Pradesh-516003.

Naveen. H N
Research Assistant
High Speed Wind Tunnel complex
Dept.of Aerospace Engineering
Indian Institute of Science ,Bangalore-12

Dr. K Rajagopal
Professor. & HOD
Dept. of Mechanical Engineering
K. S. R. M College of Engineering
Kadapa, Andhra Pradesh-516003

***Abstract*—During the conceptual design of a Aerospikes in a re-entry vehicle, the spikes shape and geometry is varied and its impact on performance is evaluated, In this study, the shape optimization of two classes of spikes has been studied: A spike with tip as slender & blunt. On one hand, sharp slender spike design reduces the drag and ensures long ranges and more economic flights. However they are more vulnerable to aerodynamic heating. On the other hand, blunt spike produce more drag. However, they are preferred as aeroheating is concerned. In addition, in the content of hypersonic winged vehicles, blunt spike geometries are preferred over slenderones for practical implications such as higher volumetric efficiency, better accommodation of crew or on board equipment. Also, it is found that imposing pitch-stability for the winged vehicle at all angle of attacks results in spike shapes which require upward control surface deflections during the majority of the entry. These spikes are placed infront of the nose region to avoid stronger shocks for reducing the temperature at the nose. Cad Model was created in solid works, meshing in Gambit 2.4.6& post processing using FLUENT 6.3.**

Keywords: Aerothermodynamics, spike, heat flux; heat transfer co efficient.

1. INTRODUCTION

Hypersonic vehicles experience high levels of drag and aerodynamicheating during flight. Many efforts are devoted to minimizingthese effects for longer ranges, lower fuel consumption, andsafer flights. One approach is to usespikes. A spikeis a needle-like body mounted at the nose tip of the main bluntbody. The basic mechanism of drag and heating reduction is twofold: first, replacingthe strong detached bow shock ahead of the blunt body by amuch weaker oblique foreshock; second, encouraging separation ofthe flow downstream of the foreshock and creating a recirculationzone that screens a large portion of the main body nose surface.Fig. 1illustrates this mechanism. The shear layer that envelopesthe recirculation zone reattaches on the main body surface and ashock wave, called the reattachment shock, is created to turn theouter flow at the body shoulder.

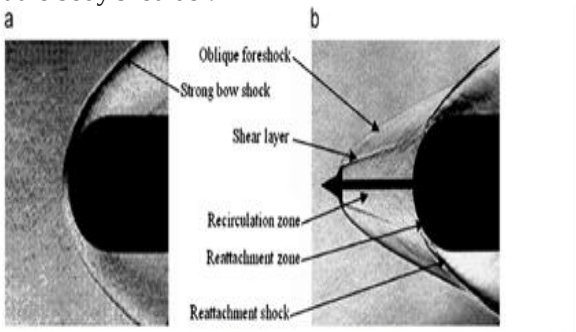

Fig. 1. Macroscopic features of the flowfield ahead of a blunt forebody in hypersonic speeds. (a) without [23] and (b) with the spike/aerodisk [4].

Thus, it was believed that the heating loads can be reduced by altering the flow field pattern ahead of the blunt body so as to replace it with a weaker system of shock waves. A variety of designs are considered. These designs include spikes and aero disks,and even supersonic projectiles fired

ahead of the blunt forebody. Out of these varieties of techniques, the use of spikes proved to be the simplest and the most effective technique in reducing aerodynamic heating.

2. GEOMETRY AND GRID

The 2D cad geometry was developed using solid works cad software as shown in fig 2.1

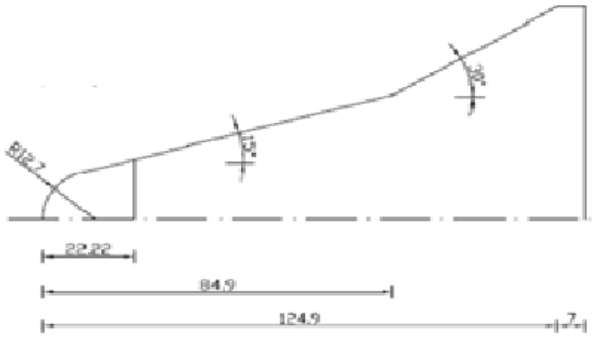

Fig2.1 shows the 2D geometry of the hypersonic winged re-entry vehicle(without Spike).

A spike of two designs was introduced at the nose region of hypersonic winged re-entry vehicle. Blunt spike with entire length 40mm and having radius of 3mm as shown in fig 2.2. Slender spike of length 40 mm and having slender angle of 17.06^0as shown in fig 2.3.

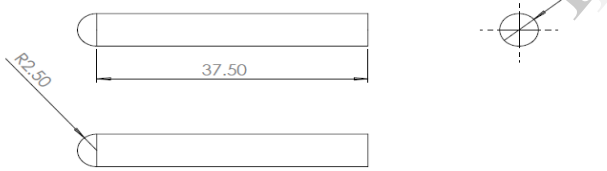

Fig 2.2 shows the 2D geometry of Blunt spike

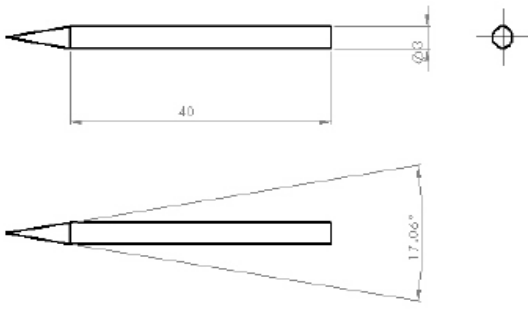

Fig 2.3 shows the 2D geometry of slender spike

Both the cad geometries of without spike and with spike (Blunt & Slender spike) was imported to the gambit software in the igs format .and generated the mesh as shown in the fig 2.4, 2.5.1 &2.5.2

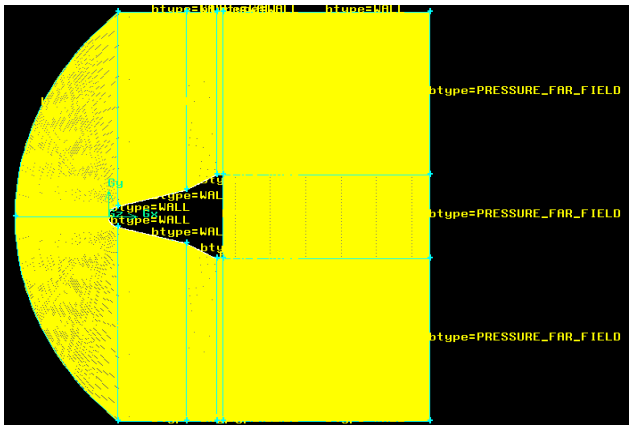

Fig 2.4: without spike meshed and boundary conditions

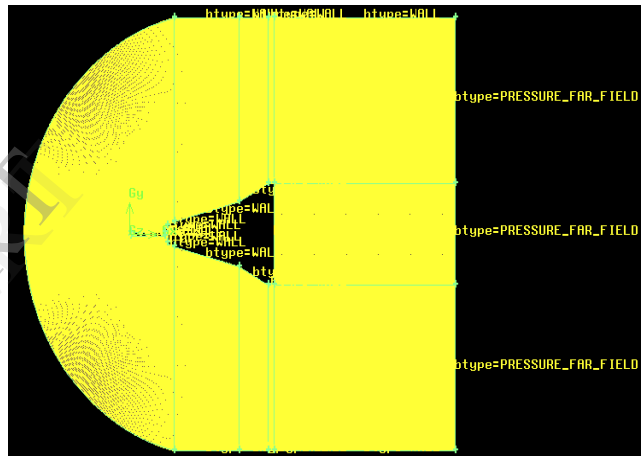

Fig 2.5.1: withBlunt spike meshed and boundary conditions

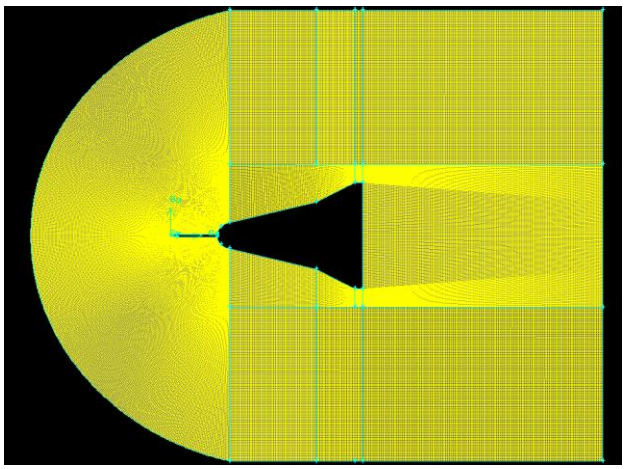

Fig 2.5.2: with slender spike meshed and boundary conditions

The grids were made very fine at the surfaces and coarse as it goes away from the body. Domain extending from 1.5*l* major length of the model at the face and 2.5*l* times behind the geometry and 2*l* of the top and bottom of the geometry modelPressure inlet condition was set to the inlet of the meshed model where inlet pressure are specified and pressure far field was set to the pressure outlet and top and bottom of model and body was set to the wall boundary condition.

2.1 BOUNDARY CONDITIONS

The flow conditions that was used in this analysis is shown in the below table.

Inlet	50 bar
Outlet	0.0.0022025 bar
Body	Wall
Temperature	773k
Mach No	9.1

The CFD analysis was conducted for density based implicit formulation 2D Flow, the compressible flow is coupled to the energy equation forthe flow simulationusing Spalart–Allmaras Model.

3. RESULTS & DISCUSSIONS

3.1 Experimental Results:

The experiment was done at Delft University of Technology, Netherlands .The tunnel was operated at a free stream of Mach number9.1. The settling chamber of the wind tunnel ispressurized at a total pressure P_0 ranging between 25 and 75 bars. The fluid (dry air) is heated to astagnation temperature of 773 K with a temperature controlled electrical heating system.

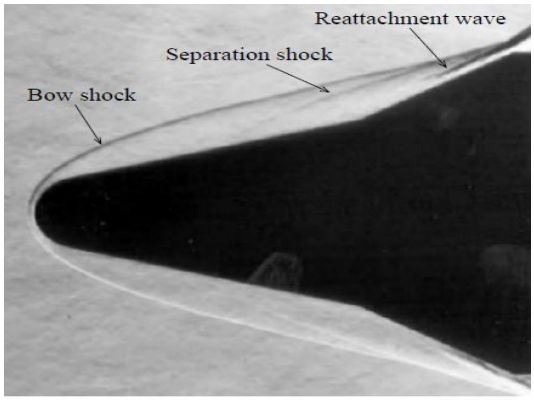

Fig 3.1.1 shows Schlieren image from the high speed Camera for a nose experiment at free M_∞ = 9.1.

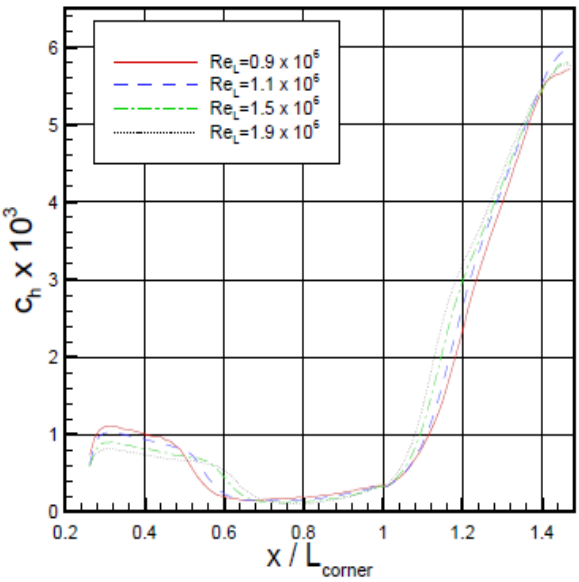

Fig.3.1.2 shows the heat transfer for the experimental model at different Reynolds number with free stream M∞ =9.1

3.2. CFD Results:

3.2.a Temperature contour plots for without spike configuration

From the temperature contour we observe that temperature is maximum at nose of the blunt body for without spike model, the bow shock temperature near the nose is observed to be 900K.This shock wave scatter the heat into the flow field so temperature decreases along the body as seen in the fig 3.2.1

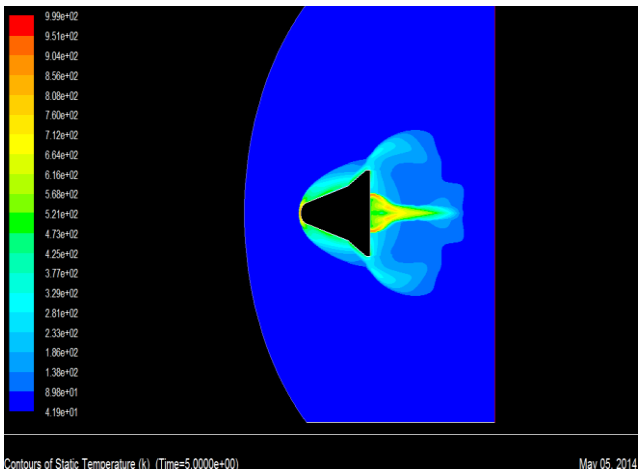

Fig 3.2.1: Temperature profile of without spike at 50 bar

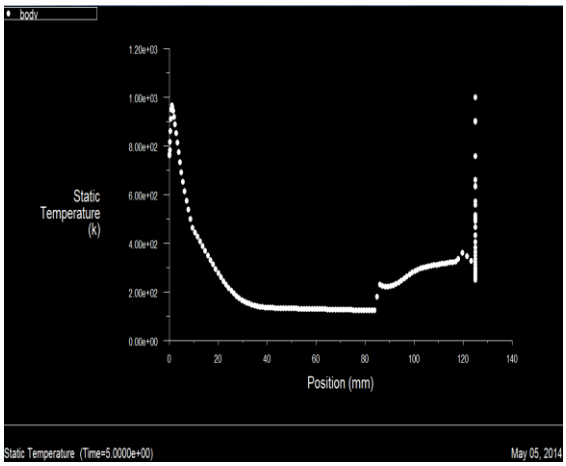

Fig 3.2.2 shows variation of temperature vs length for without spike model.

3.2.b Temperature contour plots of with spike(Blunt) configuration.

As the introduction of the Blunt spike at the nose of the blunt body we observed that the temperature at the nose was 625 k. The shock wave scattered along the length and decreases the temperature along physical domain.

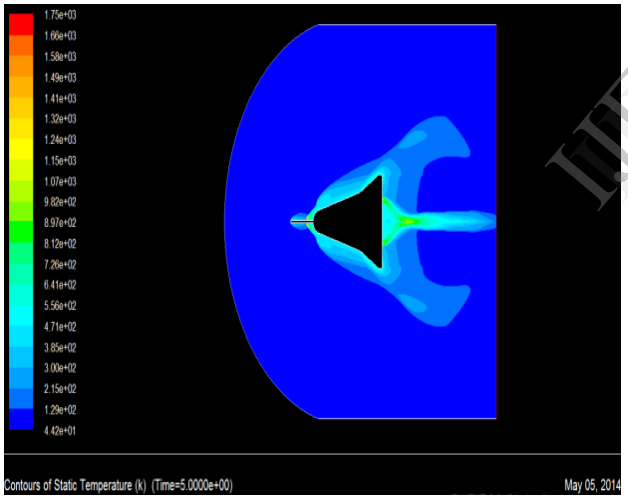

Fig .3.3.1: Temperature Profile of with spike (Blunt) configuration at 50 bar

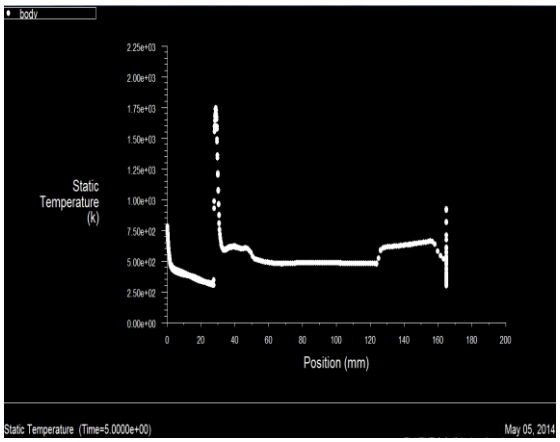

Fig 3.3.2: shows variation of temperature vs length for the Blunt spike model.

3.2.c Temperature contour plots of with spike (Slender) configuration.

As the introduction of the slender spike at the nose of the blunt body we observed that the temperature at the nose was around 800 k. The shock wave scattered along the length and decreases the temperature along physical domain

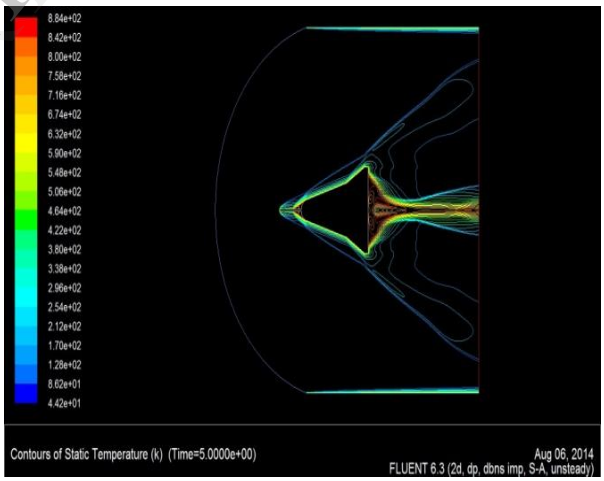

Fig .3.4.1: Temperature Profile of with spike (Blunt) configuration at 50 bar

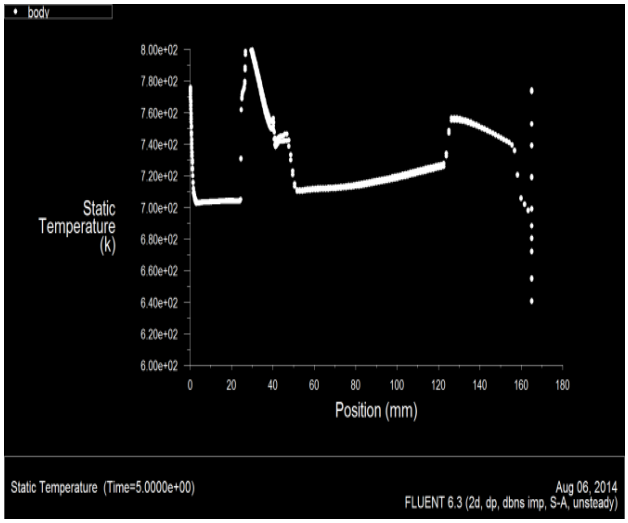

Fig 3.4.2: shows variation of temperature vs length for the Slender spike model.

3.2.d Calculation of Heat Transfer Co-efficient(C_H):

The Heat Transfer Rate is given by the relation

$$q = \rho c l (dT/dt) \quad \text{.....1}$$

Where the values of ρ (density), c(specific heat) and l(thickness) of the skin are

'ρ' density of Stainless Steel = 8000 Kg/m3

'c' specific heat of Stainless Steel = 502 J/Kg K

'l', thickness of the material = 1.0 mm

and the co-efficient of heat transfer is calculated by using the equation(2)

$$c_H = \frac{q}{\rho_\infty c_{p_\infty} U_\infty (T_o - T_w)} \quad \text{.....2}$$

Where, ρ∞, U∞, and Cp∞ are free-stream density, velocity and specific heat at constant pressure. To is stagnation temperature of the given run and Tw is wall temperature at a given instant time.

3.3 Comparison of Experimental Result with CFD results :

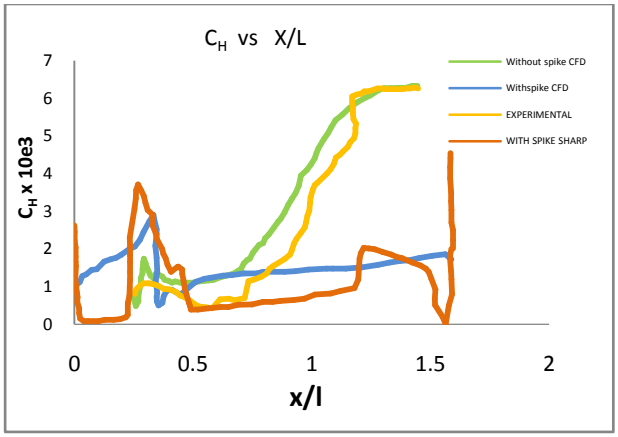

Fig 3.5: Shows the Variation of C_H with length at free stream Mach Number M_∞=**9**.1.

The above graph shows the variation of C_H with respect to length for both experimental and CFD result. As we notice that by introducing the Blunt spike to the model there is 30.5% reduction of heat at the nose of the model, and for slender spike it is around 11.5%.

From the above graph we can notice that slender spike heat reduction at nose is comparatively less, but at the body is high. When the slender spiked vehicles are good is compared with without spiked vehicles. But the Blunt spiked vehicles are comparatively good over slender ones.

4. CONCLUSION

In spite of the decades of research into hypersonic flow; there are still many challenges to analyzing and designing high-speed vehicles. Recent events such as the Columbia accident are clear evidence that there are still "unknown unknowns" in the field of hypersonic flight that even our best experimental or numerical analysis cannot adequately predict.

While analyzing both with spiked (Blunt & Slender) hypersonic vehicle the temperature of the Blunt spiked vehicle reduces at the nose compared to Slender, due to reduction in the bow shock wave and also pressure of the nose vehicle so that there is less possibility or more time to take disaster of the hypersonic re-entry vehicles.

5. SCOPE OF FUTURE WORK

\

Introduction of spike in hypersonic vehicles are there. Since work have been going in the hypersonic vehicle. very few people are doing work on introducing spike on the hypersonic re-entry vehicle . Since No static temperature data exists for Mach number in the range 8-14. Also data on large nose radius do not exist.

Keeping in view of future requirements in the country as well as to generate data base towards extending some of existing static temperature ,correlations , a detailed CFD study of hypersonic laminar surface flows on several spiked models this will have relevance to reentry missions

6. REFERENCES

[1]. Ferry F.J. Schrijer and Fulvio Scarano and Bas W. van Oudheusdenwas conducted on experiment on Hypersonic Boundary Layer Separation and Reattachment on a Blunted Cone-Flare using Quantitative Infrared Thermography 2003.

[2]. Antonio Viviani and Giuseppe Pezzella was studied Heat Transfer Analysis for a Winged Reentry Flight Test Bed 2005.

[3]. M.Y.M. Ahmeda N. Qininvestigated Recent advances in the aerothermodynamics of spiked hypersonic vehicles 2011

[4]. Zonglin Jiang and Yunfeng Liu and Guilai Hanwas conducted on In order to achieve efficient wave drag reduction under non-zero attack angles and avoid the severe aerodynamic heating, a new concept of the Non-ablative Thermal Protection System (NATPS) for hypersonic vehicles. 2011.

[5]. N.Sreenivasa Babu. Dr. K. Jayathirtha Raowas studied Aerodynamic drag and heating are the crucial in the thermal stability of hypersonic vehicles at various speeds.

[6]. Agosh M C had conducted on experiment Aerodynamic and Heat Transfer Analysis over Spherical Blunt Cone.

[7]. D. Dirkx, E. MooijDelft University of Technology, Faculty of Aerospace Engineering, Section of Aerodynamics and Space Missions, Kluyverweg

[8].D. Kinney, Aerodynamic Shape Optimization of Hypersonic Vehicles, 44th AIAA Aerospace Sciences Meeting and Exhibit, No. AIAA 2006- 239, 2006.