Aerothermodynamic Design Optimization of A 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.

Dr. K Rajagopal

Professor. & HOD

Dept. of Mechanical Engineering

K.S.R.M College of Engineering

Kadapa, Andhra Pradesh-516003

Abstract - **Hypersonic flows are characterized by high Mach number and high total enthalpy. An elevated temperature often results in thermo-chemical reactions in the gas, which play a major role in aerothermodynamic characterization of high-speed aerospace vehicle. Among variety of design requirements reducing the aero heating problem is most crucial one. The hypersonic winged re-entry vehicle flying at hypersonic speed generates strong bow shock ahead of nose which is responsible for aero heating problem at the nose region, sotwo designs are consider without spike and with spike for reducing aero heating problem, studied with different pressures. Spike is placed in front 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.

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.
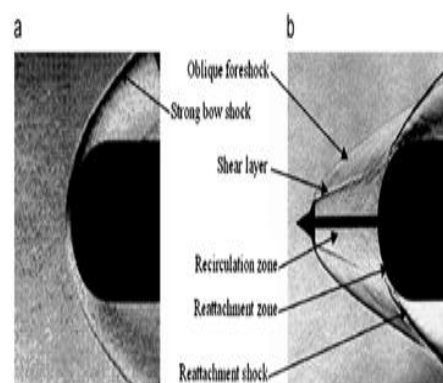

Fig. 1. Macroscopic features of the flowfield ahead of a blunt forebody in hypersonic speeds. (a) without [23] and (b) with the spike(aerodisk [4]

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 fore body. 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

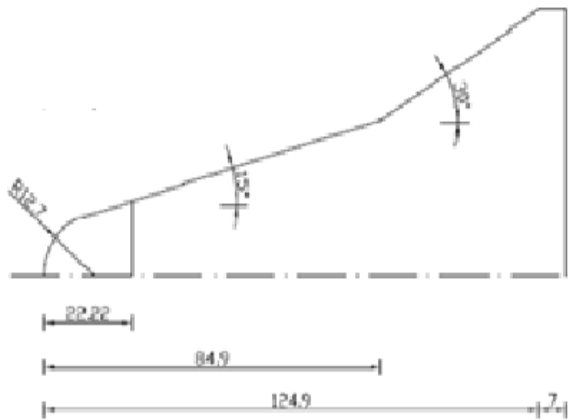The 2D cad geometry was developed using solid works cad software as shown in fig 2.1

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

A spike of length 40mm and having radius of 3mm was introduced at the nose region of hypersonic winged re-entry vehicle. Both the cad geometries (with and without spike) was imported to the gambit software in the igs format and generated the mesh as shown in the fig 2.2& 2.3
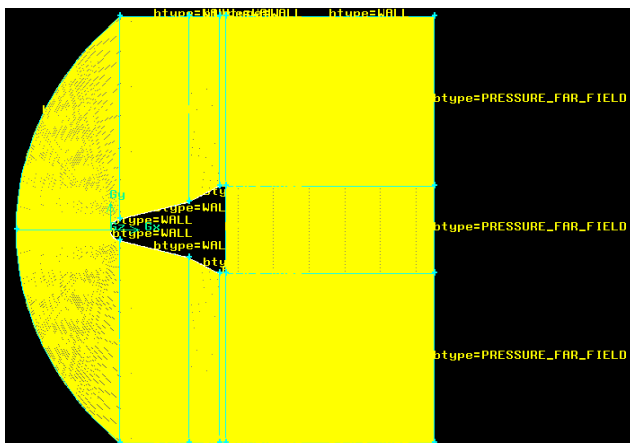

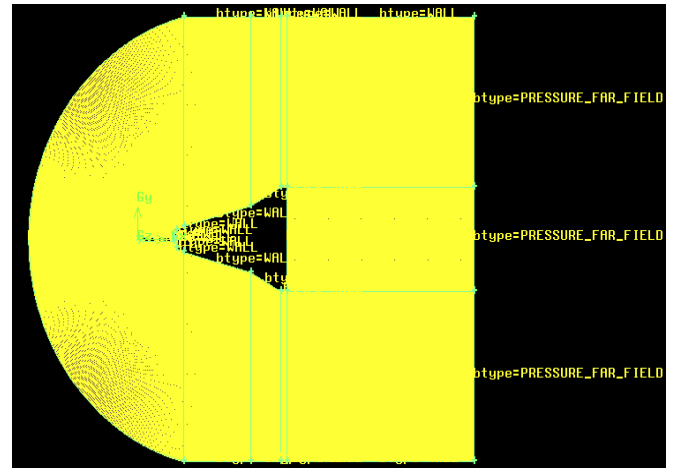*Fig 2.2: without spike meshed and boundary conditions*

Fig 2.3: with 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 $1.5l$ major length of the model at the face and $2.5l$ times behind the geometry and $2l$ of the top and bottom of the geometry model Pressure 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 simulation of the modelis shown in the below table.

Pressure Inlet	50 bar
Pressure Outlet	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 for the flow simulation using 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.
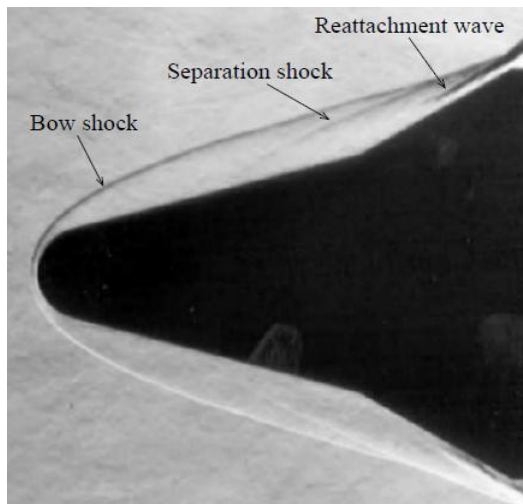
Fig 3.1.1 shows Schlieren image from the high speed
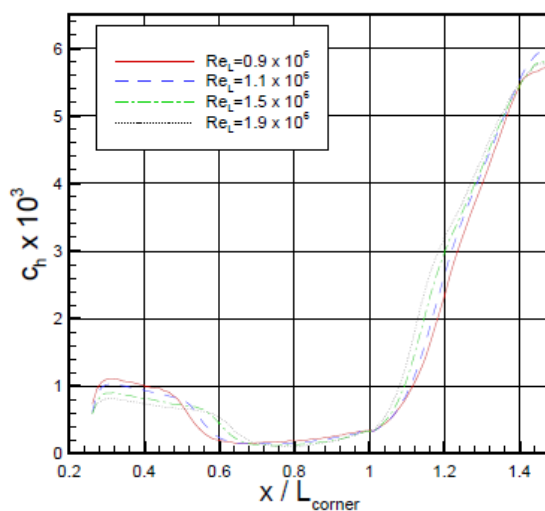Camera for a nose experiment at free $M_\infty = 9.1$.

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

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
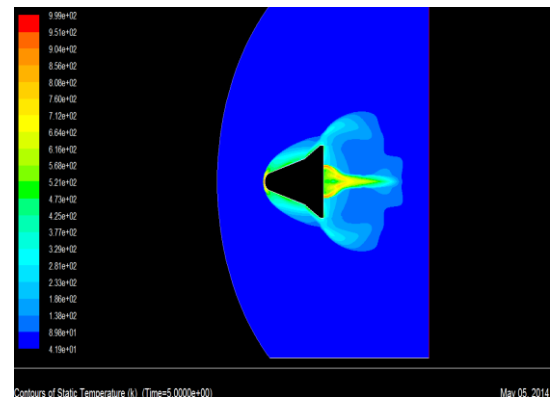the flow field so temperature decreases along the body as
seen in the fig 3.2.1

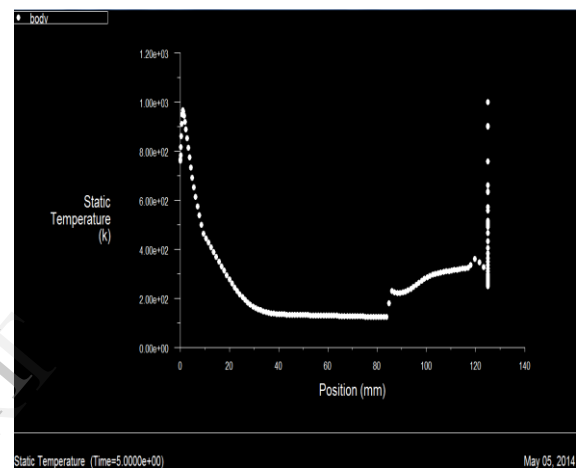Fig 3.2.1: Temperature profile of without spike at 50 bar

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

3.2.b Temperature contour plots of with spike configuration.

As the introduction of the 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
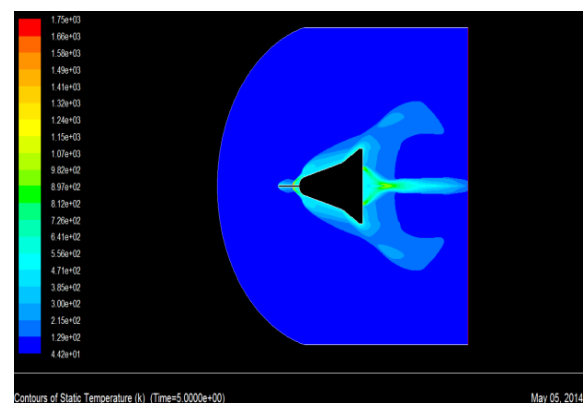the temperature along physical domain.

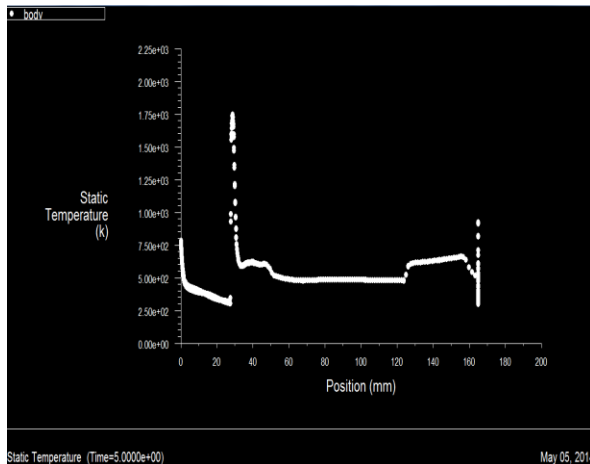Fig .3.2: Temperature Profile of with spike configuration at 50 bar

Fig 3.6 shows variation of temperature vs length for the spike model.

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

The Heat Transfer Rate is given by the relation

$$q = \rho cl(dT/dt) \ ....1$$

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)

$$c_H = \frac{q}{\rho_\infty\, Cp_\infty\, U_\infty\, (T_0 - T_w)} \ ....2$$

Where, $\rho\infty$, $U\infty$, and $Cp\infty$ 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.
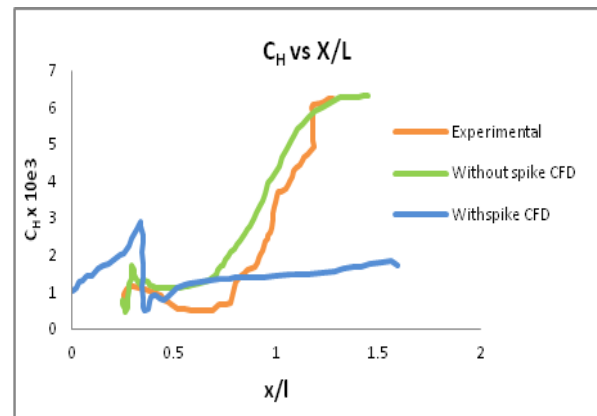
3.3 Comparison of Experimental Result with CFD result :

Fig 3.3.1: Shows the Variation of C_H with length at free stream Mach Number $M_\infty=9.1$.

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 spike to the model there is 30.5% reduction of heat at the nose of the model.

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 without and with spiked hypersonic vehicle the temperature of the spiked vehicle reduces at the nose 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 re-entry missions

6. REFERENCES

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

[2]. **Menezes V, Saravanan S, Reddy KPJ**. Shocktunnel study of spiked aerodynamic bodies flying at hypersonic Mach numbers. Shock Waves 2002;12(1): 197–204.

[3]. **Antonio Viviani and Giuseppe Pezzella** Heat Transfer Analysis for a Winged Reentry Flight Test Bed 2005.

[4]. **M.Y.M. Ahmeda N. Qin** Recent advances in the aerothermodynamics of spiked hypersonic vehicles 2011

[5].**Zonglin Jiang and Yunfeng Liu and Guilai Han**, A new concept of the Non-ablative Thermal Protection System (NATPS) for hypersonic vehicles. 2011.

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

[7]. **Agosh M C** had conducted on experiment Aerodynamic and Heat Transfer Analysis Over Spherical Blunt Cone.