Aerodynamic Analysis of Flow Field Around Typical Aerospike Missile at Supersonic Speed

Mr. T. Anbarasan.M.Tech.,
Head of the department,
Department of Aeronautical Engineering,
Parisutham Institute of Technology and Science,
Thanjavur, Tamil Nadu, India.

Mr. N. Vairamuthu.M.Tech.,
Assistant professor,
Department of Aeronautical Engineering,
Parisutham Institute of Technology and Science,
Thanjavur, Tamil Nadu, India.

Ms. V. Gayathri kanimozhi,
Department of Aeronautical Engineering,
Parisutham Institute of Technology and Science,
Thanjavur, Tamil Nadu, India.

Mr. R. Gopal,
Department of Aeronautical Engineering,
Parisutham Institute of Technology and Science,
Thanjavur, Tamil Nadu, India.

Abstract*–Pressure drag and aero heating stirred by the shock wave is the main challenge of hypersonic flight. We know that the sharp slender forebodies design reduces the drag and ensures longer ranges and more economic flights. At the same time, they are more vulnerable to aerodynamic heating. And also, blunt bodies produce more drag; however, they are preferred as far as aero heating is concerned. Therefore, aerospikes can be efficiently utilized as a mean for drag reduction. It is the simplest and the most reliable technique. A spike is a slender rod attached to the stagnation point of the vehicle's nose. It replaces the strong bow shock with a system of weaker shocks along with creating a zone of recirculating flow ahead of the forebody thus reducing both drag and aeroheating.In this project, three cases of aerospike missiles have been taken for an *Aerodynamic analysis of flow field around typical aerospike missile* at supersonic speed. One is aerospike with conical shape at 10 deg and the next is aerospike with conical shape at 5 deg and the other is aerospike with hemisphere cylindrical shape. Flow field around these three cases of aerospike have been investigated by adopting three dimensional computational fluid dynamic analysis by using CFD software tool packages (ICEM CFD, CFX- Pre-processor, solver, Post processor) and adopting k- Epsilon turbulence model to study the effects of flow separation through the bow shock wave over the aerospike missile in supersonic boundary layer condition. The Comparative studies of mach number contours, pressure contours, velocity contours and temperature contours of all the three cases of aerospike missiles have been studied. It is observed that the aerospike with hemispherical cylindrical shape has given the better aerodynamic drag reduction.

Keywords – Aerospike missile; drag reduction; Shock wave; CFD.

I.INTRODUCTION

The aerospike also known as spike is originally used as “flow separation spike”, which is caused by the adverse pressure gradient in the boundary layer region near the aerospike. However, the effect of drag and heat transfer reduction depend on the flow conditions, blunt body shape, and spike geometry.

Hence,The flow field around a spiked blunt body appears to be very complicated and complex and contains number of interesting flow phenomena and characteristic, which has yet to be investigated. The recirculating region is formed around the root of the spike up to the reattachment point of the flow at the shoulder of the hemispherical body. Due to the recirculating region, the pressure at the stagnation region of the blunt body will decrease. The flow past the spike creates conical shock wave and remains away from the body. Flow behind the conical shock wave separates on the spike and create a conical shape recirculation zone appears in the vicinity of the stagnation region. Due to formation of the recirculation of the flow, pressure and wall heat flux reduces in the forward facing region of the blunt body. However, the reattachment of the shear layer on the shoulder of the hemispherical body increases the local heat flux and pressure. The reattachment shock is moved downstream, which is function of the geometrical parameter of the spike. The flow field features captured by the mach and pressure, velocity and temperature contours are used to know the mechanism of the drag reduction. The influence of the spike shock wave generated from the spike interacts with the reattachment shock were also studied to understand the cause of drag reduction.

II.ICEM CFD GEOMETRY AND MESH REPORT

The solid modeling of the aerospike is carried by ICEM CFD modeling tools. The dimensions are taken for the aerospike as from the base paper. The solid model was drawn in ICEM CFD by the help of design parameters of the aerospike will be shown in following tables. In this analysis the flow over the aerospike requires the flow domain for the flow analysis. Therefore for flow analysis, a flow domain is created as for the dimensions required. Before starting the mesh need to create the boundary layer around aerospike shape. And then mesh the faces of the body by using unstructured mesh. To create 3D mesh of the domain the trihedral pave elements are used. Check the mesh of the domain for convergence. In this the flow domain selected as AIR for Outer region and SOLID for aerospike region. And the flow boundary is selected as INLET, OUTLET, and OUTER WALL for the outer region.For our aerodynamic analysis, we are taking three cases of aerospike shapes and corresponding design parameters are taken from base paper. They are,

1. Aerospike with conical body at 10 deg
2. Aerospike with conical body at 5 deg
3. Aerospike with hemisphere cylindrical body

Table 1.Model configuration

MODEL CONFIGURATION	DIMENSION
Length of the missile	7.32 m
Wing surface area	$0.21m^2$
Reference area	$0.13m^2$
Aspect ratio	3.59
Mean aerodynamic chord	0.53m
Mach number m	3
Diameter,d	0.42m
C.G from nose	4.5m
Nose fineness ratio	2.8
Body fineness ratio	17.42
Span	0.88m

Table 2. Mesh Information

Domain	Nodes	Element
Air	72561	365142
Solid	15624	142546
All domain	88185	507688

III.IMPLEMENTATION

For each case apply boundary conditions which is accumulate from base paper will be apply for all cases and it will be tabulated as follows. The flow characteristics value over the aerospike has been shown.After the mesh of the aerospike in ICEM CFD then it is imported to CFXPOST software for the flow analysis with following mentioned boundary conditions. After importing of the mesh file into the CFX-POST pre.we are checking the mesh for the accurate solution and applying accurate values for domains and boundaries.Then the CFX-POST file is imported to CFX-POST-solver, which it solving the corresponding iterations by using finite element analysis. And we can see the all types of flow characteristics and corresponding results has been categorized in CFX-POST after imported file from the solver.

Table 3.Boundary conditions

Domain	Boundaries							
AIR	Boundary – in		Boundary – out		Boundary – outerwall		Boundary - air Default	
	Type	INLET	Type	OUTLET	Type	OUTLET	Type	WALL
	Location	INLET	Location	OUTLET	Location	OUTER_WALL	Location	
	Settings		Settings		Settings		Settings	
	Flow Regime	Supersonic	Flow Regime	Supersonic	Flow Regime	Supersonic	Heat Transfer	Adiabatic
	Heat Transfer	Static Temperature					Mass And Momentum	No Slip Wall
	Static Temperature	330 K					Wall Roughness	Smooth Wall
	Mass And Momentum	Normal Speed and Pressure						
	Normal Speed	480m/s						
	Relative Static Pressure	$4.85*10^3$ Pa						

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 07, July-2018

	Turbulence	Medium Intensity and k-Ɛpsilon						
Solid	Boundary – wall							
	Type	WALL						
	Location	Aerospike missile						
	Settings							
	Heat Transfer	Adiabatic,Isentropic						

After get the successfully solved file from the CFX, Clearly have seen the consequent flow characteristics of three cases with respect to contours and streamline path of given boundary flow and have to obtain the exacting data's from the analysis,

MACH NUMBER CONTOURS:

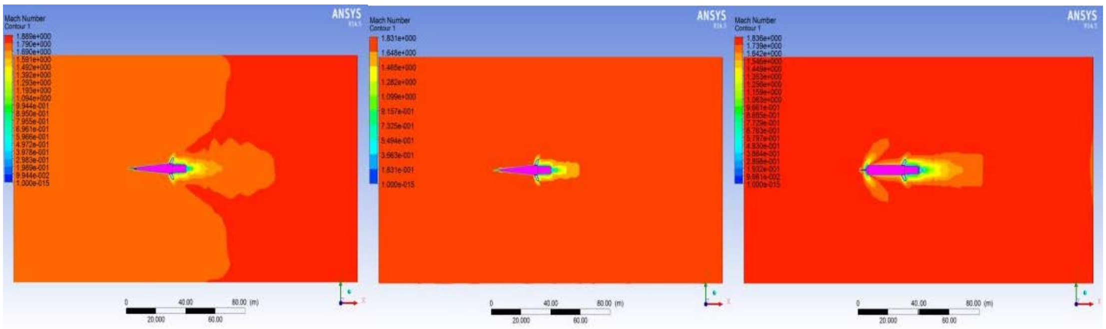


Fig.1.Mach number contours for case1, case2 and case3 respectively.

PRESSURE CONTOURS:

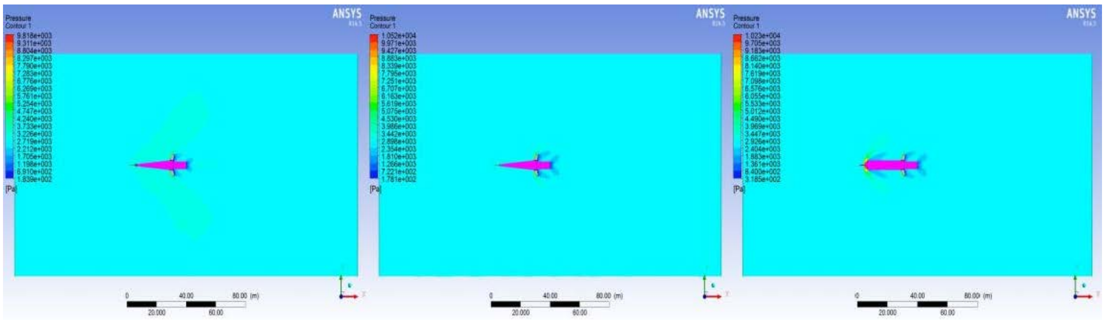


Fig.2.Pressure contours for case1, case2 and case3 respectively.

VELOCITY CONTOURS:

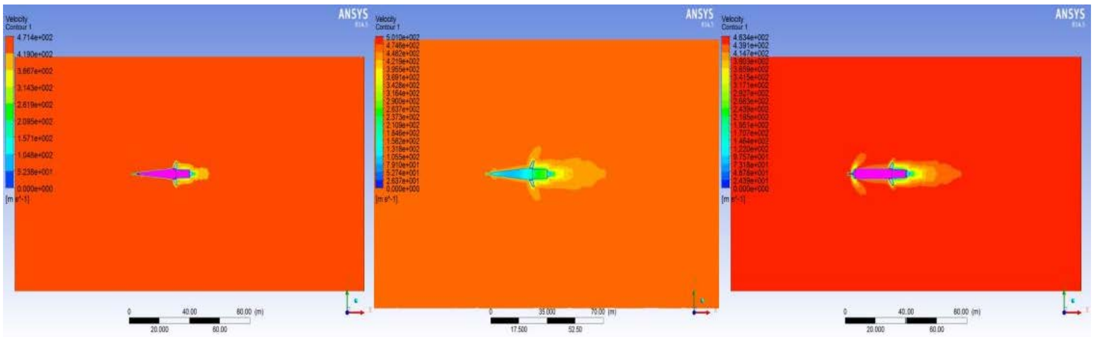


Fig.3.Velocity contours for case1, case2 and case3 respectively.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 07, July-2018

TEMPERATURE CONTOURS:

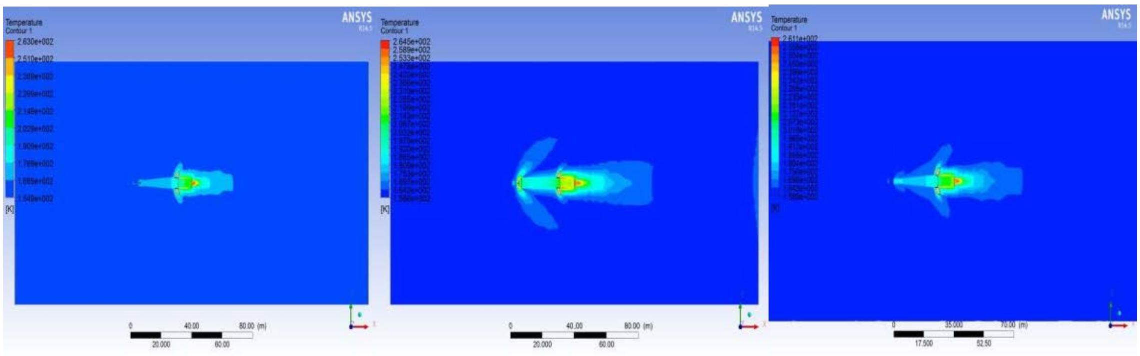


Fig.4.Temperature contours for case1, case2 and case3 respectively.

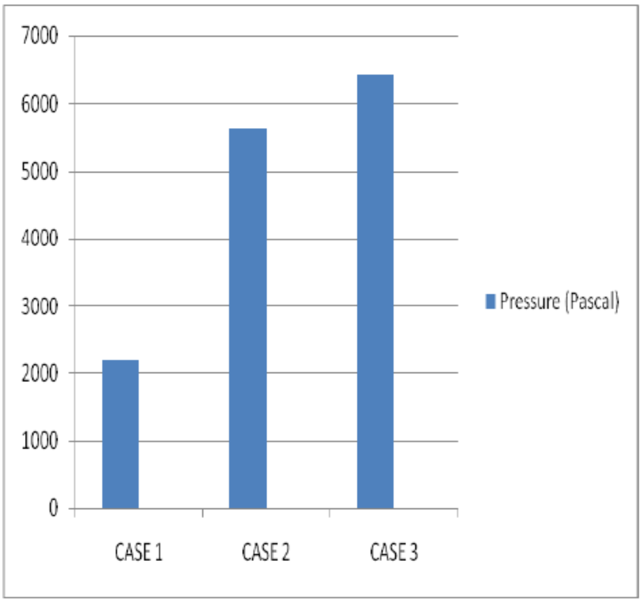
IV.COMPARITIVE RESULT

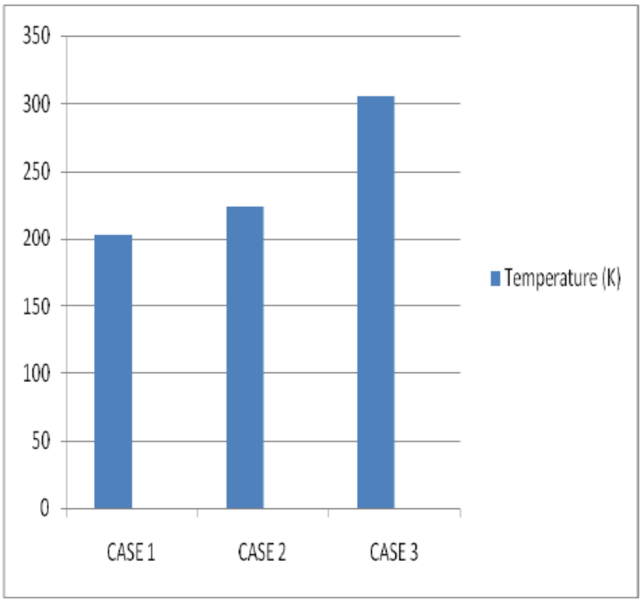
Table 4.The ideal values are tabulated from the flow characteristic values, as below

Flow characteristics	Case 1	Case 2	Case 3
Pressure (Pascal)	$2.21*10^3$	$5.619*10^3$	$6.42*10^3$
Temperature (K)	$2.029*10^2$	$2.235*10^2$	$3.052*10^2$
Velocity (m/s)	$4.19*10^2$	$3.164*10^2$	$2.542*10^2$
Mach number	1.82	1.64	1.24

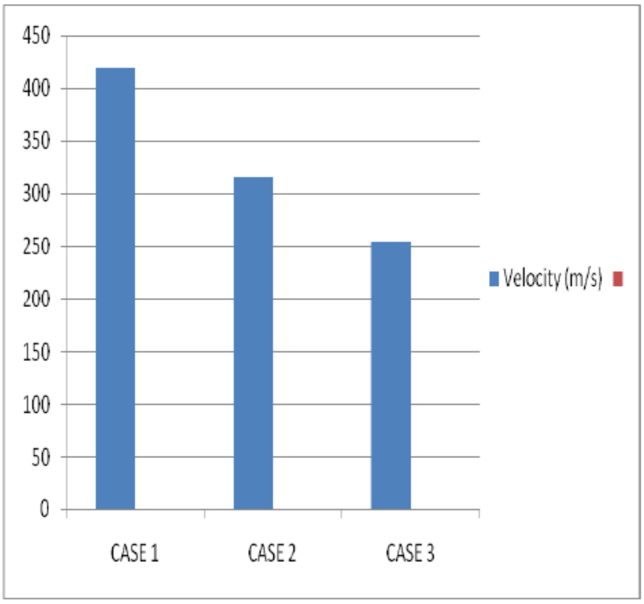
Hence, the flow characteristics of three cases are compared .From the validated results obtained by the flow field analysis on the three different aerospike designs it shows that 10 deg conical shape aerospike has given the higher drag reduction. The comparison of flow characteristics are as follows,

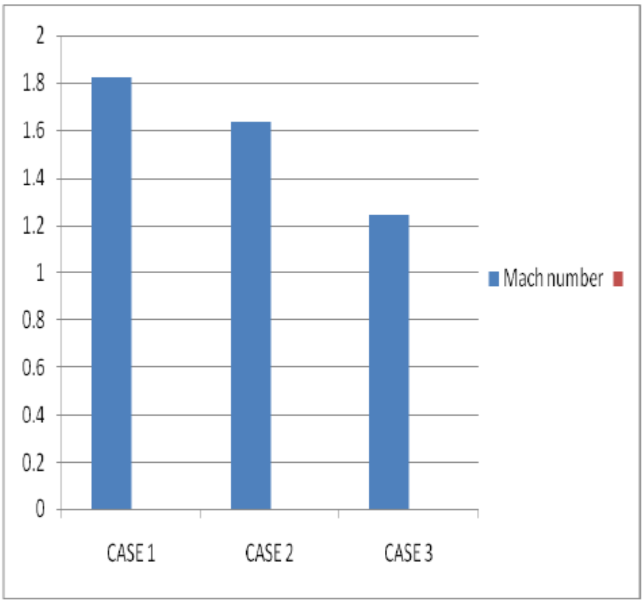
COMPARISON OF FLOW CHARACTERISTICS VALUES





International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 7 Issue 07, July-2018





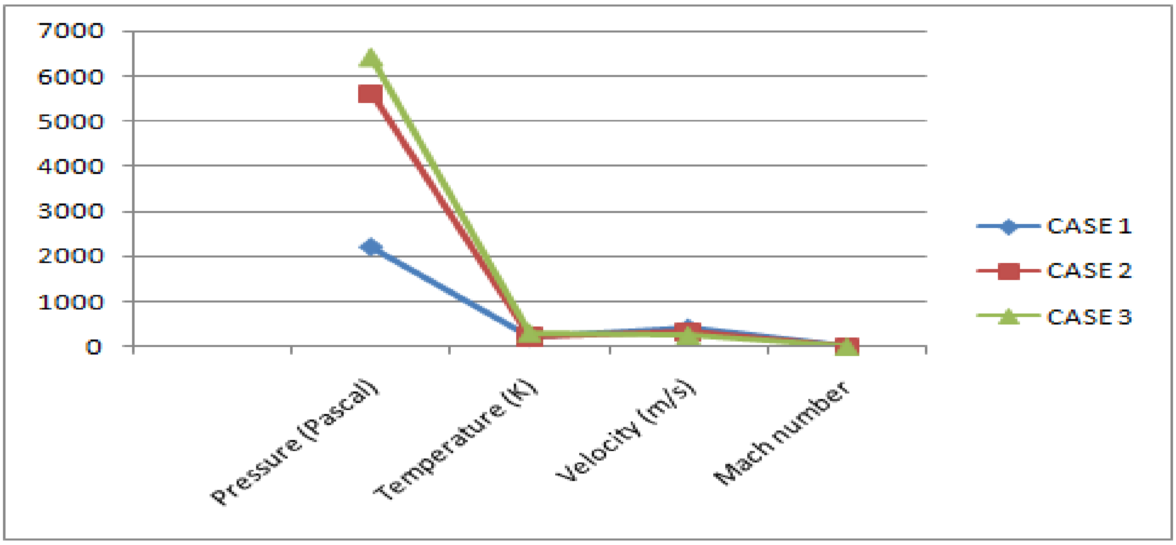


Fig.5.comparitive chart of flow characteristics

V.CONCLUSION

In this thesis work, aerodynamic analysis over aerospike missile at supersonic speed has been studied using CFD.Three cases has taken for analysis,the first is aerospike with conical shape at 10 deg and the next is aerospike with conical shape at 5 deg and the other is aerospike with hemisphere cylindrical shape. Comparative studies have done for these cases, among that the aerospike with 10 deg conical shape has better aerodynamic drag reduction.

VI.REFERENCES

[1] Anderson JD (2000) Hypersonic and high temperature gas dynamics.AIAA press,USA.
[2] Kalimuthu R,Mehta RC,Rathakrishnan E(2008) Experimental investigation on spiked body in hypersonic flow. Aeronautical J 112:593-598.
[3] Mehta R (2010) flow field computations over conical, disc and flat spiked body at mach 6.AIAA paper,USA.
[4] Roberto R (2009) Benchmark CFD study of spiked blunt body configurations .AIAA, press, USA.