Novelty Journals

ISSN 2394-9678

International Journal of Novel Research in Electrical and Mechanical Engineering
Vol. 2, Issue 3, pp: (139-145), Month: September-December 2015, Available at: www.noveltyjournals.com

CFD Analysis of Different Bluff Bodies

[1]Sibha Veerendra Singh, [2]Mr. M. Nirmith Kumar

[1,2] Department of Aeronautical Engineering, MLR Institute of Technology and Management, Hyderabad, India

Abstract: **The main objective of this thesis is to get a better understanding of flow behavior of a nose cone bluff Bodies of missiles. To do so the influence of pressure is investigated. Results are gained from calculations performed with CFD analysis program. Also there's a possibility of pressure decrement. This thesis focuses on the simulation of flow around bluff bodies in which unsteady nature of flow are commonly found using computational fluid dynamics (CFD). Various turbulence models have been tested to develop understanding and proper modeling techniques for the flow around bluff bodies.**

Keywords: **Different bluff bodies, simulation, Satosphere condition, CFD Analysis.**

I. INTRODUCTION

The topic of this thesis is the simulation of flow around bluff bodies and bluff bodies sections using computational fluid dynamics (CFD). CFD calculates numerical solutions to the equations governing fluid flow. Bluff bodies are structures with shapes that significantly disturb the flow around them, as opposed to flow around a streamlined body. Examples of bluff bodies include circular cylinders, square cylinders and rectangular cylinders. The study of the flow around bluff bodies and bluff bodies sections is important in terms of the effects of wind on such bodies.

II. LITERATURE SURVEY

To achieve the above-mentioned objectives, the work is split into various stages. Initially, modelling of the flow around a circular cylinder within range of Reynolds numbers (Re250- Re10,000) is done by using basic steady state simulation methods. This is done as a pilot study for the further application of CFD on the computation of a more complex flow using advanced CFD techniques at a later stage of the work.

The loads acting on a body immersed in a flowstream are produced by the normal and tangential stresses over its surface. When integrated, these stresses give rise to the resultant load components, which are usually expressed in non dimensional form by means of force and moment coefficients, defined as follows:

$$C_{F_i} = \frac{F_i}{1/2\rho U^2 S}\ ; \qquad C_{M_i} = \frac{M_i}{1/2\rho U^2 S l}$$

Where *Fi* and *Mi* are the components in the *xi* direction of the resultant force and moment acting on the body, *U* is the undisturbed upstream flow velocity, *S* is a reference surface and *l* a reference length. In the case of two-dimensional bodies (which approximately represent sufficiently long structural elements), the load coefficients are defined by using the load per unit distance along the span of the body in the numerator, and a reference length in the denominator (which is normally either the cross-flow or the along-flow dimension of the body cross-shape).

It is often useful to refer also to the pressure *p* acting at a certain point of the body surface, by using a pressure coefficient defined as

International Journal of Novel Research in Electrical and Mechanical Engineering
Vol. 2, Issue 3, pp: (139-145), Month: September-December 2015, Available at: www.noveltyjournals.com

$$C_p = \frac{p - p_\infty}{1 / \rho U^2}$$

where *p* is the pressure in the undisturbed upstream flow.

The above mentioned loads have, in general, mean and time-varying components (which may be characterized, e.g., by their r.m.s. values and by their frequency spectra). The fluctuating loads may be significant not only when the upstream flow is time-dependent (for instance due to the presence of turbulence), but also when the wake produced by the body itself has more or less regular fluctuations. In general one may say that, for steady upstream flow, aerodynamic bodies are characterized by steady wakes and loads, whereas for bluff bodies the opposite is true.

In terms of pressure coefficient, by using Bernoulli's equation one has

$$C_{ps} = 1 - V_s^2 / U^2$$

III. DESIGN AND ANALYSES

CATIA is a multi-platform CAD/CAM/CAE commercial software suite developed by the French company Dassault systems. CATIA offers a solution to shape design, styling, surfacing workflow and visualization to create, modify, and validate complex innovative shapes from industrial design to Class-A surfacing with the ICEM surfacing technologies. CATIA is a powerful tool to learn in depth designing of aerospace interior and exterior components and analyze aero structural parts. We can learn to implement engineering technical and validating our design.

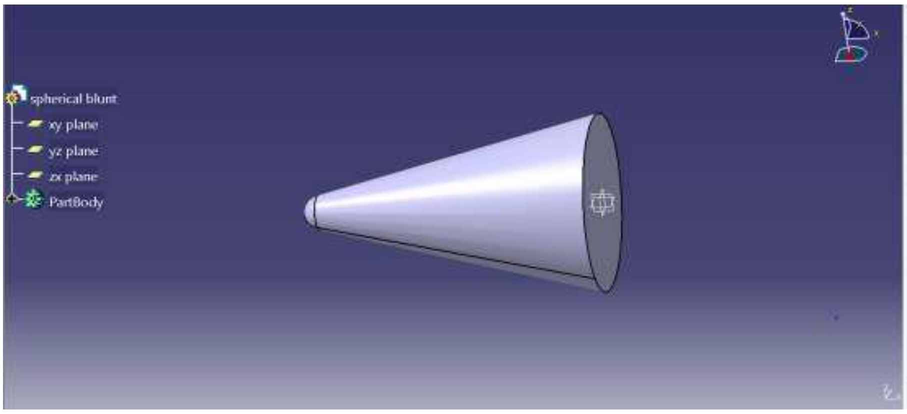

Fig:1 3D model of spherical blunt body

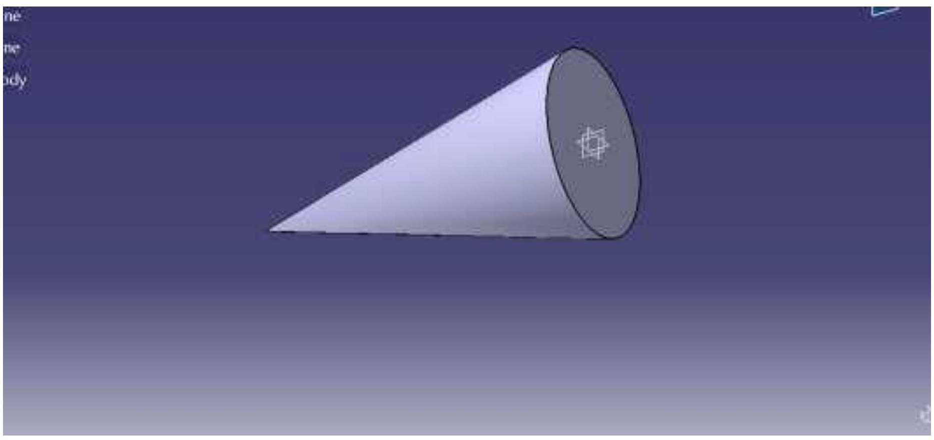

Fig:2 3D model of wedge blunt body

ISSN 2394-9678

International Journal of Novel Research in Electrical and Mechanical Engineering

Vol. 2, Issue 3, pp: (139-145), Month: September-December 2015, Available at: www.noveltyjournals.com

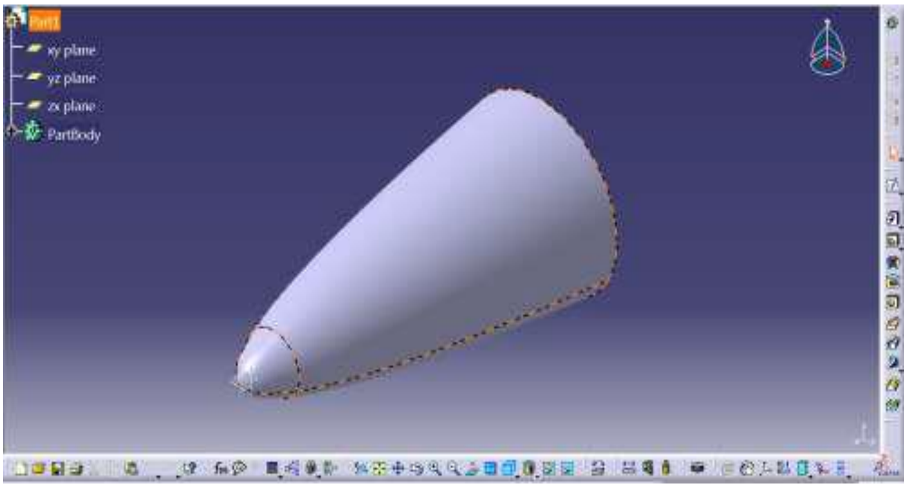

Fig:3 3D model of parabolic blunt body

ANALYSIS: Selection of CFD→ Importing of CAD model in geometry module and creating of fluid domain →Update model in mesh module for meshing and defining boundary limits → Open the solver and import the mesh file in fluent→ Define the input and flow requirements for analysis→ Initialize the flow in solver and run iterations until the flow gets converged → View the results in command and graphical window.

Pre-processing consists of the input to a flow problem by means of an operator friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The user activities at the pre-processing stage involve:

- Definition of the geometry for the region of interest generally known as the computation domain
- Grid generation the sub division of the domain in to a number of smaller, overlapping, subdomains, a grid of cells.
- Selection of the physical and chemical phenomena that needs to be modelled
- Definition of fluid properties
- Specification of appropriate boundary conditions at cells, which coincide with or touch the domain boundary.

Finally a postprocessor is used for the analysis and visualization of the resulting solution.

For the CFD analysis, the inlet is assumed to have no losses. Admission of air occurs with no entropy generation. In fact, for the analysis, it is assumed that the detonation chamber is closed at the front end for analysis. However, in order to perform the CFD analysis a description of the geometry at the downstream end is required.

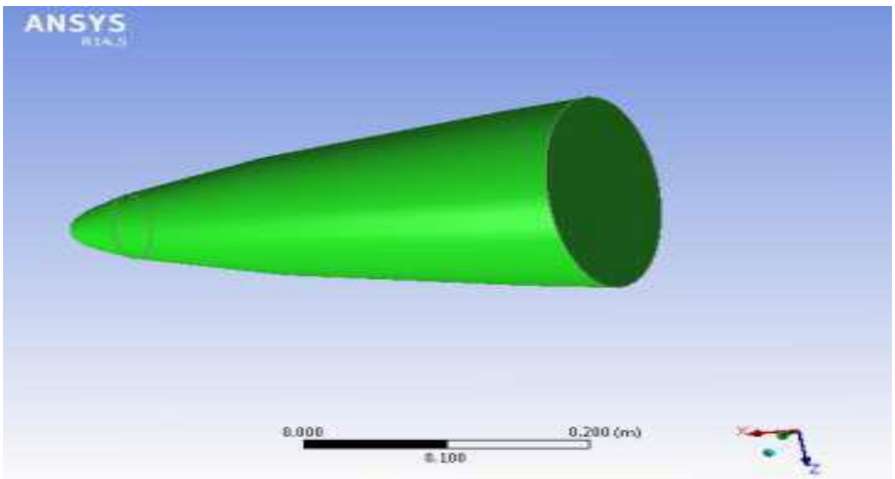

Fig 4. Importing CAD files in ANSYS Workbench

International Journal of Novel Research in Electrical and Mechanical Engineering
Vol. 2, Issue 3, pp: (139-145), Month: September-December 2015, Available at: www.noveltyjournals.com

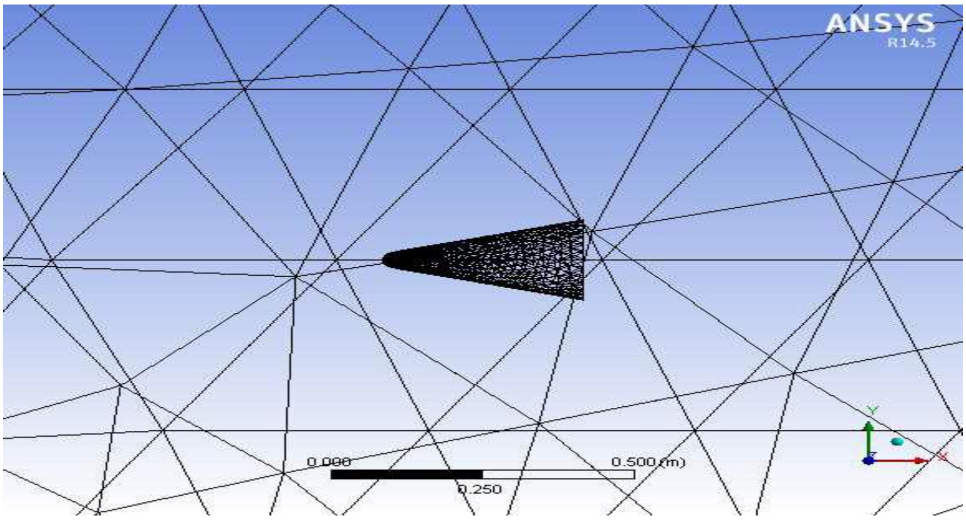

Fig 3.3(f) Meshing of wedge body in the fluid domain

The type of flow modelled in FLUENT to analyse the flow inside.

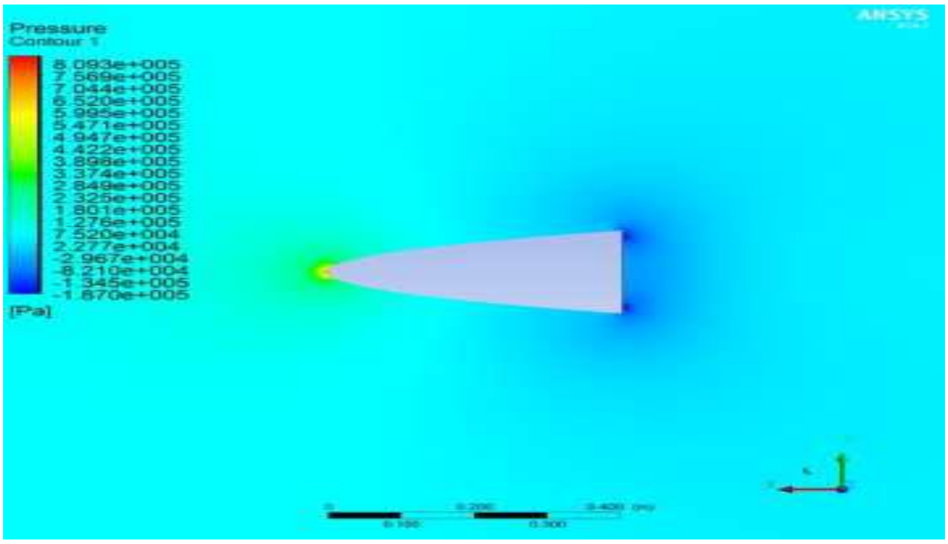

Fig:4.2(g) Pressure over parabolic cross-section

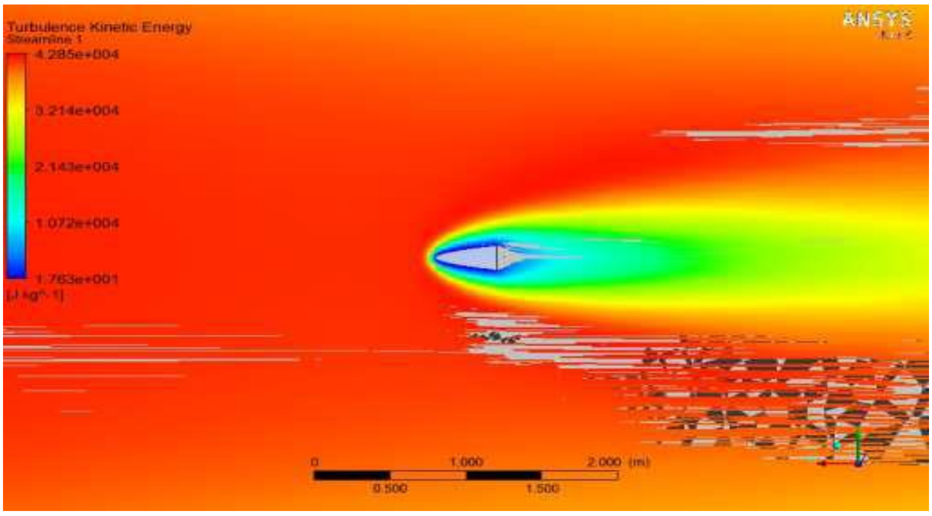

Fig:4.2 (h) Turbulent kinetic energy over parabolic body

International Journal of Novel Research in Electrical and Mechanical Engineering
Vol. 2, Issue 3, pp: (139-145), Month: September-December 2015, Available at: www.noveltyjournals.com

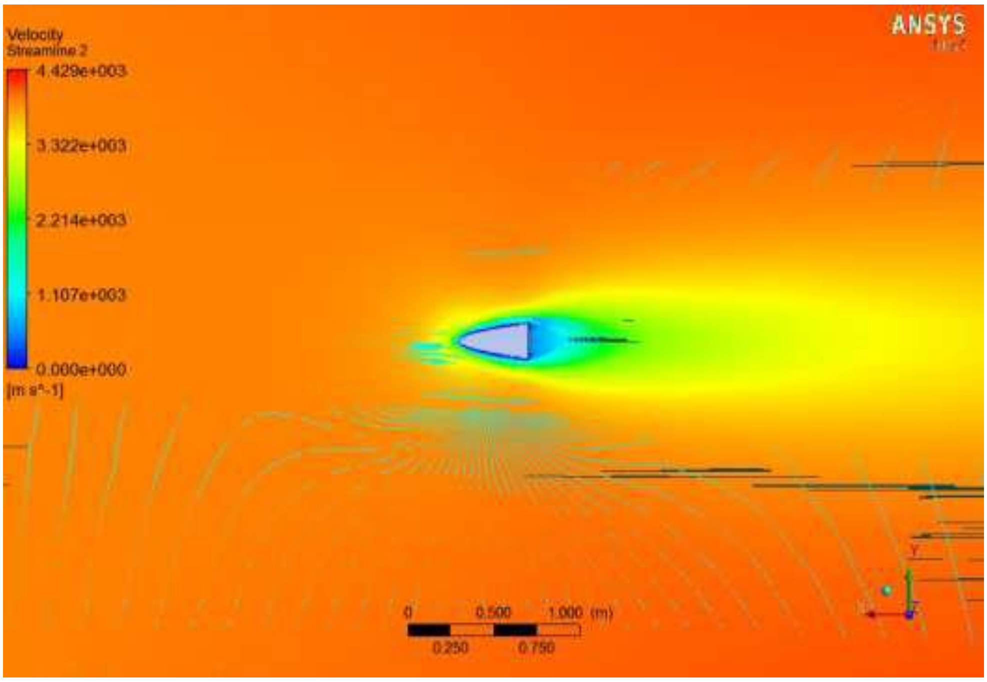

Fig:4.2(i) Velocity pattern over parabolic body

IV. RESULTS

In the flow field around the body, gas molecules which impact on the body experience a change in momentum, and by the random molecular collision of the molecule this change transmitted to the neighbouring molecules. In this fashion, information about the presence of bluff body transmitted to the surrounding flow via molecular collisions.

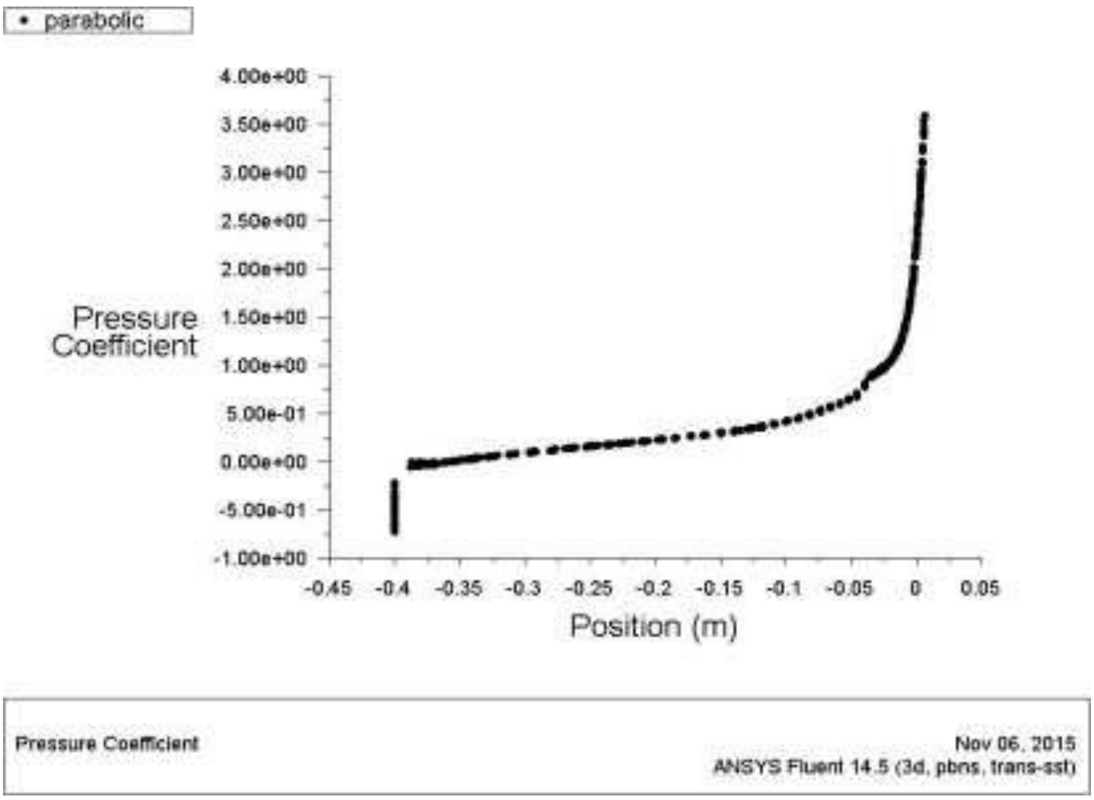

Fig 4 pressure coefficient of parabolic blunt body

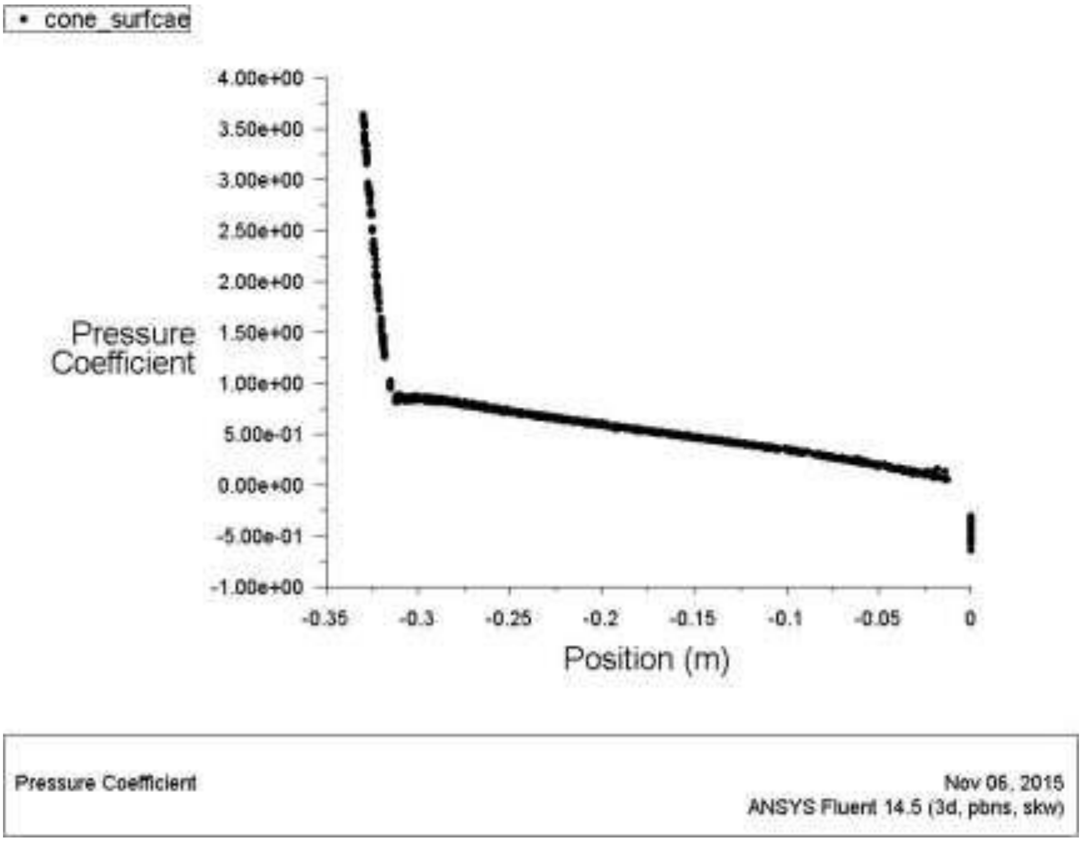

Fig 4.2(k) pressure coefficient of Spherical blunt (cone surface) blunt body

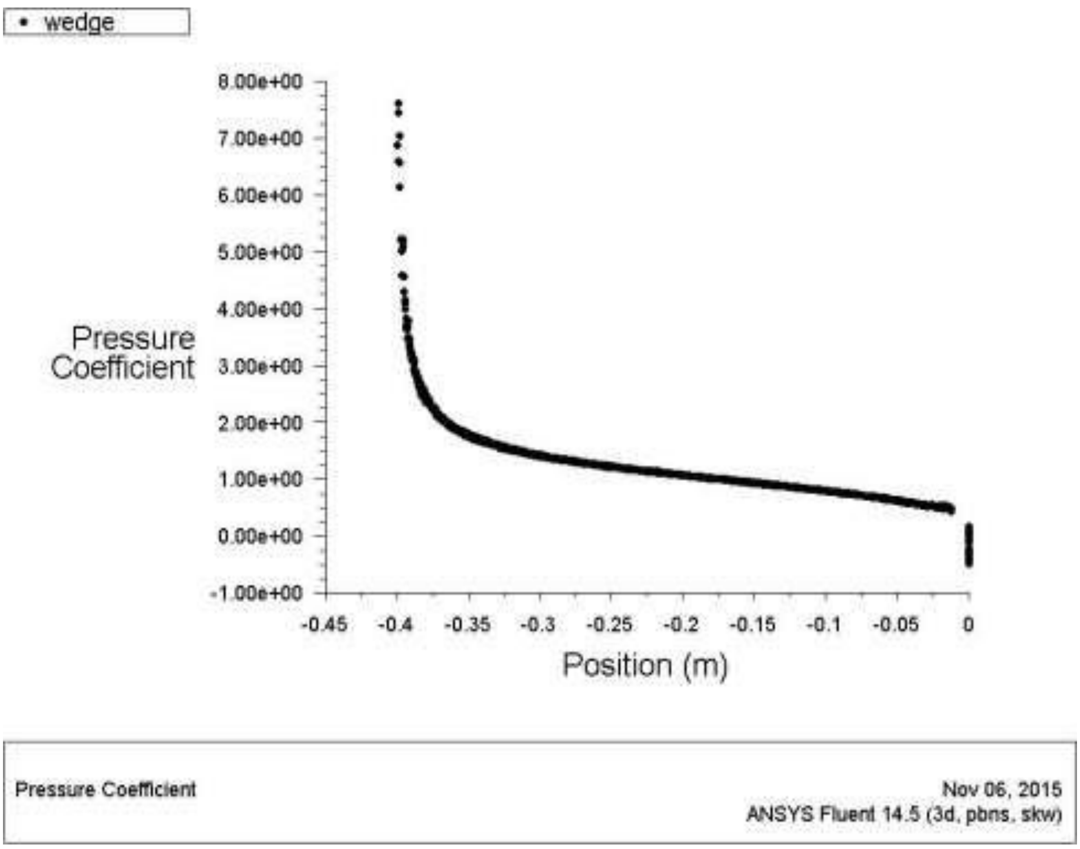

Fig 4.2(L) pressure coefficient of wedge blunt body

International Journal of Novel Research in Electrical and Mechanical Engineering
Vol. 2, Issue 3, pp: (139-145), Month: September-December 2015, Available at: www.noveltyjournals.com

V. CONCLUSION

From the fluent simulation it is concluded that at high Mach number a detached bow shock at the front of the body generates, which highly influence the flow properties around the body. Pressure suddenly increased drastically behind the wave and flow compressed to a high level at the stagnation point. Temperature rise at stagnation point is very high; At the apex of the body a sonic region is generates where the flow is subsonic, and the flow properties changes drastically at this region due to a strong bow shock. Calculation of flow variables at a point just behind the bow shock wave confirm that at the apex the bow shock wave can be treated as a normal shock.

From the theoretical formulation we conclude that aerodynamic drag body initially depends on its kinetic energy and bluntness of the nose cone decrease the aerodynamic drag over the body by generating the strong bow shock. Computational investigation of minimum-drag bodies at moderate hyper sonic speeds (Mach 10) confirms that, the bodies with the lowest wave drag formed on cone body and the drag has increased to 0.42% on spherical blunt body and 0.5% drag increment on blunt body.

ACKNOWLEDGEMENT

I owe a debt of gratitude to Nirmith Kumar Mishra at MLR Inst of Tech & Management for his valuable suggestions, vision and foresight which inspired me to conceive this project.

I express my sincere gratitude to Dr. K V Reddy, Principal and Dr. D. Muppala, Head of the department of Aeronautical department at MLR Institute of Technology and Management, for their encouragement in pursuing the project. Also I am very much thankful to Mr.Gupta, and Dr. Ratnakar, Associate Professors for Aeronautical department for their support through and through.

REFERENCES

[1] Moretti Gino and AbbtitMichael : “A Time- Dependent Computational Method for Blunt Body Flows”AIAA Journal ,Vol 4 No 12 ,1970.

[2] Anderson,john,D, Albacete M.Lorenzo and Winkelmann E Allen: “On Hypersonic Blunt body fields Obtained with A time dependent Technique:Naval Ordnance Lab Itr ,1971

[3] *D.A. Jones and D.B. Clarke (2008),* Simulation of Flow Past a Sphere usingthe Fluent Code.

[4] Fluent 14.5. Available: www.sharcnet.in. Last accessed 20th July 2013.

[5] BengtFornberg (2014), Some Observations Regarding Steady Lamina Flows Past Bluff_ Bodies.

[6] John C K Cheung, William H Melbourne, Effects of surface roughness on a circular cylinder in supercritical turbulent flow, Department of Mechanical Engineering, Monash University.

[7] Pijush.K.Kundu, Ira M.Cohen. Fluid Mechanics. New York: Academic Press. Pg.256-300.