IJISET - International Journal of Innovative Science, Engineering & Technology, Vol. 2 Issue 4, April 2015.
www.ijiset.com
ISSN 2348 – 7968

Flow Analysis on Various Bluff Bodies

Hemanth Raghav.P[1], Dinesh Kumar.G[2]
[1,] M.Tech scholars, School of Aeronautical Sciences, Hindustan University, Chennai, India.
[2,] Assistant Professor, Department of Aeronautical Sciences, Hindustan University, Chennai, India.

Abstract— **A bluff body of certain length and various sizes are taken perpendicular to the flow direction. It has been flow analyzed for aerodynamics forces which leads to pressure distribution, a wake and also vortex shedding. This results in flow separation and velocity variation. . Also a brief comparison is discussed from the available experimental data.**

Keywords*— *Bluff body, Aerodynamic forces, Pressure distribution, Wake, Vortex shedding, Flow separation, Velocity variation.

I. INTRODUCTION

Computational flow involves the study of an object under moving condition. In this paper computational flow of para-rec bluff body was analyzed at different Reynolds's number to determine the pressure drag and also the drag coefficients. This study would help scientists and researchers to know about the importance of the para-rec shape objects and its usefulness in designing a new type of re-entry vehicles.

II. DESIGNED MODEL

In an external flow such as flow over a blunt body, we have to define a far field boundary and mesh the region between the blunt body geometry and the far field boundary. It is good idea to place the far field boundary well away from the geometry since we will use the ambient conditions to define the boundary conditions at the far field.

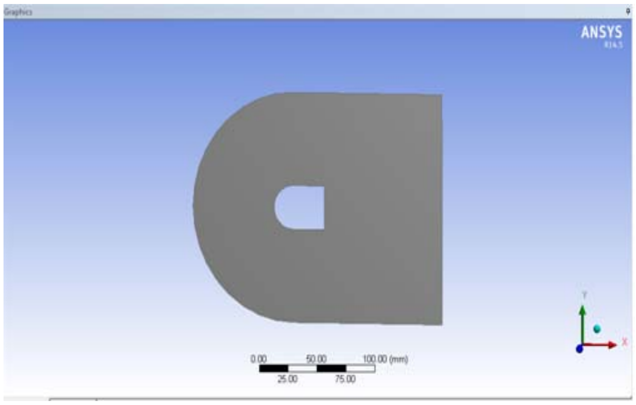

Figure 1 Geometry body in ANSYS WORK BENCH

A. MESHING

Step1: As we know that the flow property changes drastically in front of the body nose and on the sharp edges. There are basically two regions, first is flow field near the blunt body nose and other is behind the So we have to mesh the edges coarse, where the flow change are dominant and to do this we split the upper edge and the lower edge of the body in two sub sections and mesh with different interval count.

B. DEFINE BOUNDARY TYPES IN GAMBIT

Pressure far-field conditions are used in FLUENT to model a free-stream condition at infinity, with free-stream Reynolds and static conditions are specified. The pressure far-field boundary condition is often called a characteristic boundary condition, since it uses characteristic information to determine the flow variables at the boundaries Wall boundary conditions are used to bound fluid and solid regions. In viscous flows, where there is a no-slip boundary condition is enforced at walls by default, also you can specify a tangential velocity component in terms of the translational or rotational motion of the wall boundary, or model a "slip" wall by specifying shear.

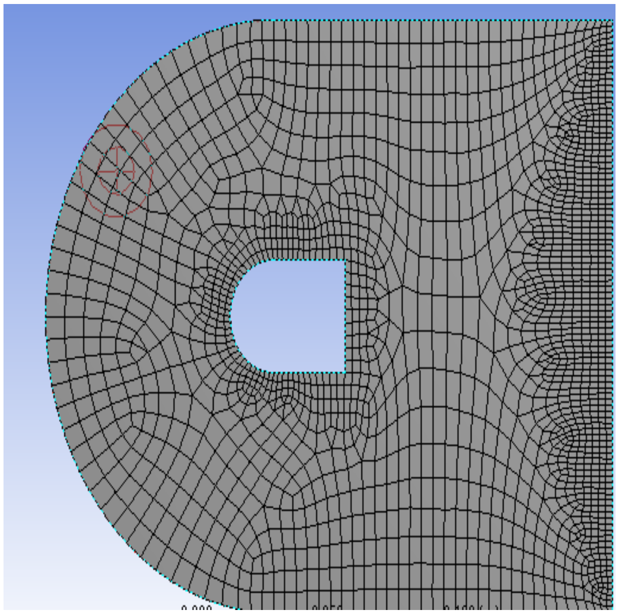

Figure 2 Boundary types in GAMBIT

III. EQUATIONS USED

This model was solved with three major equations.

Continuity equation, Momentum equation, Energy equation.

1. Continuity equation

$$\frac{d\rho}{dt} + \nabla.(\rho V) = 0 \quad \text{———} \quad 3.1$$

2. Momentum equation

X axis:

$$\partial\frac{\rho U}{\partial t} + \nabla.(\rho u V) = -\frac{\partial p}{\partial x} + \frac{\partial \tau xx}{\partial x} + \frac{\partial \tau xy}{\partial y} + \frac{\partial \tau xz}{\partial z} \text{—} \quad 3.2.1$$

Y axis:

$$\partial\frac{\rho V}{\partial t} + \nabla.(\rho v V) = -\frac{\partial p}{\partial y} + \frac{\partial \tau xy}{\partial x} + \frac{\partial \tau yy}{\partial y} + \frac{\partial \tau yz}{\partial z} \text{—} \quad 3.2.2$$

Z axis:

$$\partial\frac{\rho W}{\partial t} + \nabla.(\rho w V) = -\frac{\partial p}{\partial z} + \frac{\partial \tau xz}{\partial x} + \frac{\partial \tau zy}{\partial y} + \frac{\partial \tau zz}{\partial z} \text{—} \quad 3.2.3$$

A. REYNOLDS NUMBER

Reynolds number Re is a dimensionless number that gives a measure of the ratio of inertial forces to viscous forces and formulated as

$$\mathrm{Re} = \frac{\rho v D}{\mu} = \frac{v D}{\vartheta}$$

IV. RESULTS AND DISCUSSION

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 neighboring molecules. In this fashion, information about the presence of blunt body transmitted to the surrounding flow via molecular collisions. If the upstream flow is supersonic the disturbance wave pile up and coalesce, form a standing wave in front of the body. That’s why ahead of the blunt body a bow shock is generated because deflection angle is very large at the body nose. This shock wave is clearly seen in the figure 5.1.And ahead of this flow properties changes drastically.

Most frequently used properties in aerodynamic analysis of any object are given as:

1. Pressure
2. Density
3. Temperature

A. PRESSURE

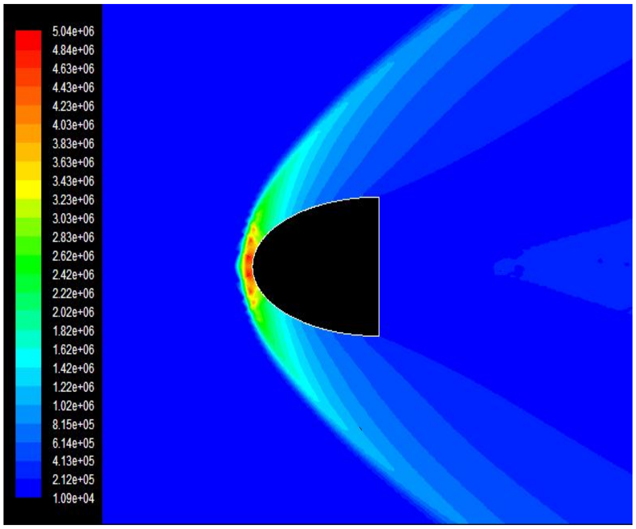

Figure 3 Contour of static pressure (Pascal)

In the flow field around the blunt body, due to hypersonic flow a shock wave generate ahead of the body. This shock wave is called bow shock .This shock is detached from the body due to the high deflection angle .Pressure changes drastically across the shock wave, at the stagnation point pressure is at peak value because at the stagnation pressure shock wave is normal to the body. Area of the contour shows the sonic region here velocity of the flow is subsonic.

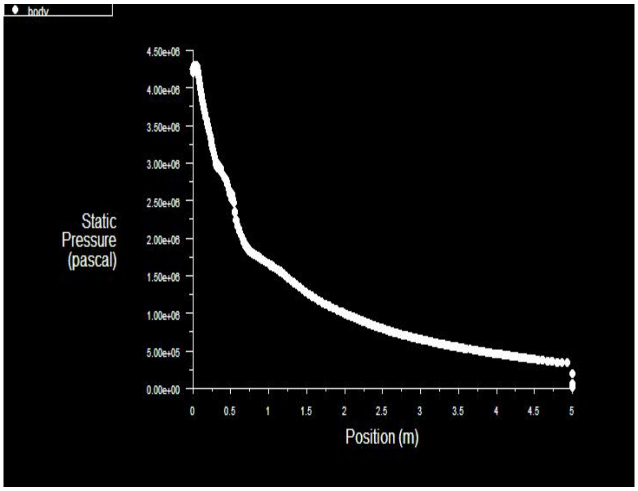

Figure 4 static pressure variations with position for blunt body

From the above graph it is concluded that along the blunt body pressure is maximum at the stagnation point, and decreasing along the body. Pressure rise at stagnation point due to the bow shock.

Between the bow shock and the body the pressure contour is reddish .it shows that at that region pressure is extremely high, flow compressed due to the normal shock. Because at the apex of the body shock is strongest and normal to the flow. Now from the normal shock relation we can easily find the value of pressure at stagnation point.

B. DENSITY

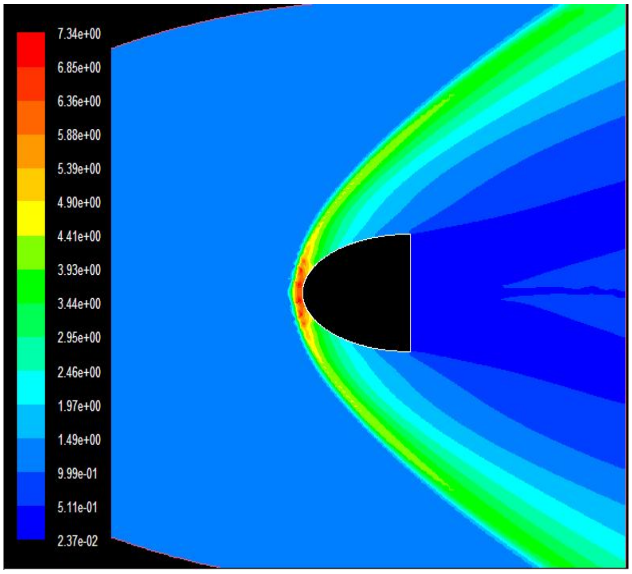

Figure 5 Contour of density (*Kg*/m^3)

The disturbance created by the body on the flow changes density drastically around the body. Reddish area just ahead of the body shows high density region because at the stagnation point shock wave is strongest and normal to the body. And along the density decrease to .5 Kg/ m^3.

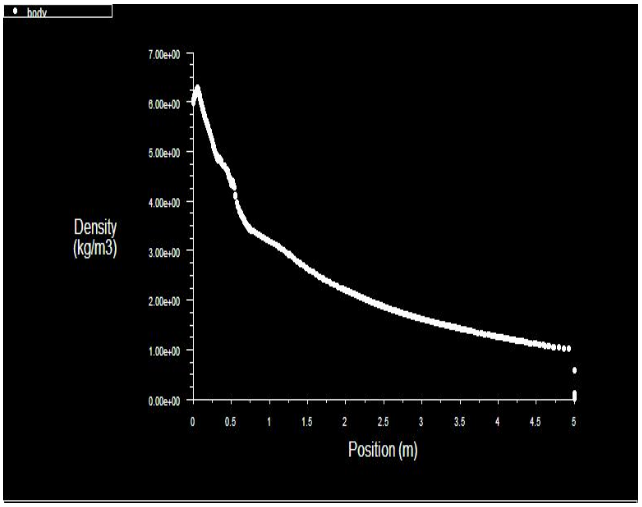

Figure 6 Density variations with position for blunt body

Above graph shows the variation of density along the blunt body, at the stagnation point density is approximately 6.4 *Kg*/ m^3 and at the rear point of the body density drop down to 1.0 *Kg*/m^3. From the normal shock equation we find that at the stagnation point the density is 6.479 *Kg*/m^3.

C. TEMPERATURE

Temperature takes an important role in high speed aerodynamics because at high Mach number the kinetic energy of flow reappear in the form of internal energy of the fluid, this phenomenon is called viscous dissipation. And when the fluid temperature increase, a high temperature gradient set up between body and fluid and this cause high heat transfer rate. Temperature contour over the blunt body is as follows.

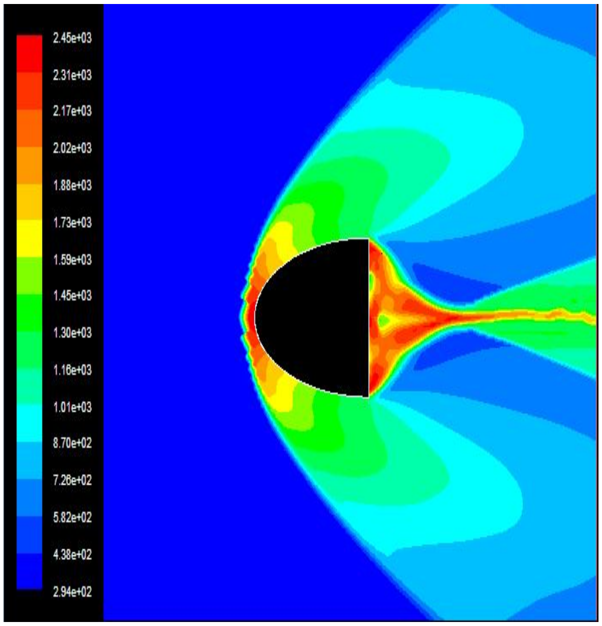

Figure 7 Contour of Temperature (k)

From the temperature contour we observe that temperature is maximum at just ahead of the blunt body, and at the starting point of the bow shock temperature is about the 1590k.This shock wave scatter the heat into the flow field so temperature decreases along the body.

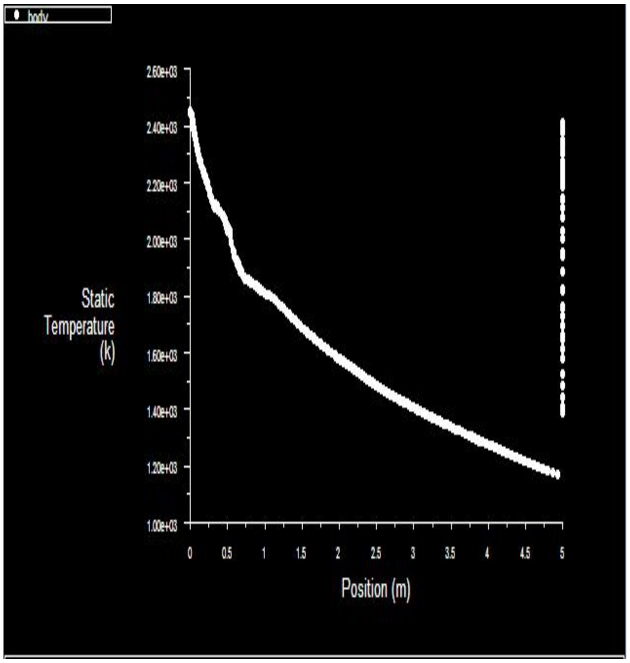

Figure 8 Temperature variations along blunt body

Above graph shows the variation of static temperature along the body, at the stagnation point temperature is about 2450k.and decreases to 1200 k at the aft.

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. Mach number suddenly decrease drastically behind the wave and flow compressed to a high level at the stagnation point. Temperature rise at stagnation point is very high; due to this high heat transfer rate is set up between the flow and body. 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 heating of the body initially depends on its kinetic energy and bluntness of the nose cone decrease the aerodynamic heating over the body by generating the strong bow shock. And aerodynamic heating varies inversely proportional to the radius of the nose cone. Another conclusion from the literature study is that the flow behind the shock wave is rotational, because each streamline pass through a different strength shock wave and entropy change behind the shock wave is different for every streamline. The computational investigation of minimum-drag bodies at supersonic and moderate hyper sonic speeds (Mach 3-12) confirms that, the bodies with the lowest wave drag have to be geometrically blunt. Aerodynamic heat in body depends over the ratio of coefficient of friction and coefficient of drag. or temperature rise at the stagnation point over the bodies Compare to all bodies first body is efficient at high mach numbers. From the all considerations of pressure velocity temperature over bodies.

VI. REFERENCES

[1] 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

[2] Anderson john,D.,jr: Computational Fluid Dynamics :The basic with applications, McGraw Hill ,New York ,1995

[3] Anderson J.D, Jr, 2005, McGraw Hill, Fundamental of aerodynamics, New York

[4] Anderson J.D, Jr, McGraw-Hill ,Hypersonic and high temperature gas dynamics, New York

[5] Anderson J.D,jr,1978 , McGraw Hill, Introduction to flight , New York

[6] Anderson J.D, Jr, McGraw-Hill, Modern compressible flow, New York.

[7] FLUENT USER GUIDE 6.3.

[8] Meir Gerick Bar, Fundamentals of Compressible Fluid Mechanics

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

[10] Santos W.F.N, 2005, Leading edge bluntness effect on aerodynamic heating,Combustion and Propulsion Laboratory National Institute for Space Research 12630-000 Cachoeira Paulista, SP. Brazil.

[11] Wing ,L.D : "Tangent Ogive Nose Aerodynamic eating program:NQLDWO19 (NASA),1988