Available online at http://www.journalcra.com

INTERNATIONAL JOURNAL OF CURRENT RESEARCH

International Journal of Current Research
Vol. 6, Issue, 06, pp.7278-7285, June, 2014

ISSN: 0975-833X

RESEARCHARTICLE

Numerical simulation of fluid flow and aerodynamic characteristic analysis of a semi blunt 9mm bullet

Gurudath K. P., and Dr. Mallikarjunayya C. Math

[1]College of Visvesvaraya Technological University, Mysore, India- 2012
[2]Department of Thermal Power Engineering from Visvesvaraya Technological University, Mysore, India- 2006

ARTICLE INFO

Article History:
Received 08th March, 2014
Received in revised form
25th April, 2014
Accepted 04th May, 2014
Published online 30th June, 2014

Key words:
Co efficient of drag,
Computational fluid dynamics (CFD),
Mach number,
Semi blunt bullet.

ABSTRACT

This work involves the study of semi blunt 9mm bullet when it comes to external ballistics and fluid flow over the bullet by considering speed at different Mach numbers such as subsonic, sonic, supersonic and hypersonic. This work also involves the study of strong shock waves generated on the bullet which results in flow separation and forms the low pressure drag, high temperature, reduction in velocity at the boundaries of the bullet and co-efficient of drag. Bullet surface is created by using CATIA vR20 modeling software. Meshing of the pressure far field is done by using HYPERMESH v10 software, and various aerodynamic constrains are simulated using CFD technique as FLUENT v6.3.2.6 software with needed tools. Thus, boundary layer is simulated numerically using pressure, temperature and velocity contours. The study reveals that at subsonic speed bow shock waves are appeared at the nose of the bullet which is curved in nature. It also indicated that with the increase in Mach number, magnitude of shock waves increases and results in flow separation. It is observed that flow separation occurs at sonic speed. At hypersonic speed, strong entropy layer is formed and is highly curved in nature.

INTRODUCTION

Bullet projectile aerodynamic have been studied in past with different methods such as simple empirical and potential theory methods, computational fluid dynamics (CFD), wind tunnel measurements and shadow graph measurements at indoor firing ranges. The most accurate experimental method used in predicting bullet and projectile aerodynamics are the spark shadow graph images taken at an indoor firing range. Naturally such facilities are expensive and usually reserved for military use (Tomi Honkanen1993). So, it is better to use the computational fluid dynamics. In this method, bullet shape is very important. By using this method, it is possible to improve the geometry of the bullet to enhance its performance. In this work, semi blunt shape is selected to minimize the temperature on the bullet at external ballistics (Maitra1999). Flowing air is considering as a fluid, and aluminum bullet is taken as stationary. It is similar to the wind tunnel measurement in CFD to analysis the fluid flow, and numerical aerodynamic characteristic on bullet surface.

Bullet specification and geometry

In a preprocessing part the 2D geometry of the bullet is drawn in the CATIA vR20 software with accurate dimension 9mm×19mm semi blunt bullet or paraboloid tip bullet.

***Corresponding author: Gurudath K. P.**
College of Visvesvaraya Technological University, Mysore, India- 2012.

Geometric data of the bullet for (Figure 1), diameter d=9.03mm, length l=10.54mm (Tarique Hasan Khan and Sudipta Saha 2013)

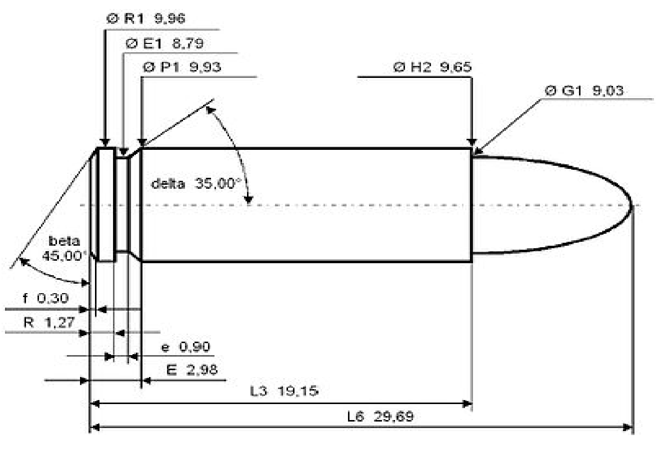

Figure 1.Dimensions of 9mm×19mm semi blunt bullet (3)

Numerical Methodology

In the preprocessing step, the actual geometry of the bullet having (Figure 2) the diameter =9.03mm, and length=10.54mm (Tarique Hasan Khan and Sudipta Saha 2013) is created by using the tools like lines, spline. Geometry is constrained and then it is exported it to HYPERMESH v10 software as a step file.

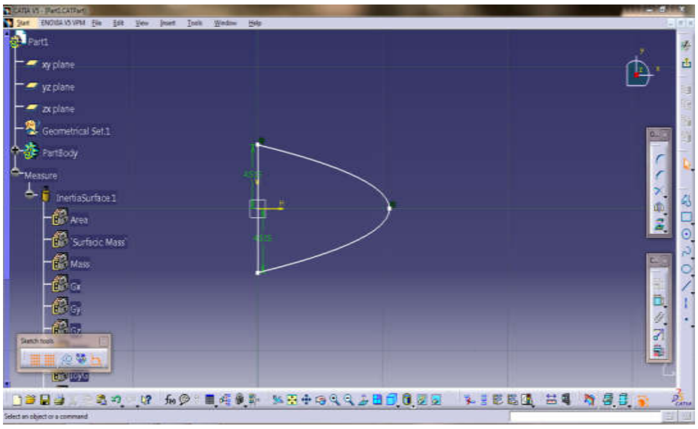

Figure 2. 2D geometry of the semi blunt 9mm bullet in CATIA software

The step file which is created by CATIA is imported to create a pressure far field of area 200×200mm around the bullet (Figure 3). Then, it is meshed as triangular meshing type of element created 16470, using the tool 2D spline and meshing tools. It is then exported to FLUENT v6.3.2.6 software as HMASCII file.

In the post processing part, boundary conditions such as pressure far field inlet and bullet as a stationary wall are selected. To solve for aerodynamic characteristics of bullet the atmospheric pressure (10135 Pa), gravity(Y=9810 m/s2) and temperature (300 K) are selected as inlet conditions. Different Mach numbers (0.6, 1, 3, 5, 8, 11, 14, 17 and 20) are selected to study the effect of fluid over the bullet. Solver used as density based axis symmetry, energy equation and viscous model is standard K-epsilon. Selected air as fluid material having a properties of density (kg/m3) =ideal gas, Cp (j/kg-k) =1006.43, thermal conductivity (w/m-k) =0.0242, viscosity (kg/m-s) =1.7894e-05. Selected aluminium as a solid material having a properties of density (kg/m3) =2719, Cp (j/kg-k) =871, thermal conductivity (w/m-k) =202.4. Considering the above boundary condition to solve the problem up to converged iteration. Finally the solution in 2D model of bullet is simulated from FLUENT for pressure, temperature, velocity contours, co-efficient of drag and velocity vectors are obtained by the simulation.

Post processing analysis and result of semi blunt 9mm bullet

Variation of static pressure with different Mach numbers

Figure 4 to 12 shows the fluid flow behavior pattern across the semi blunt bullet. Initially pressure and velocity contour

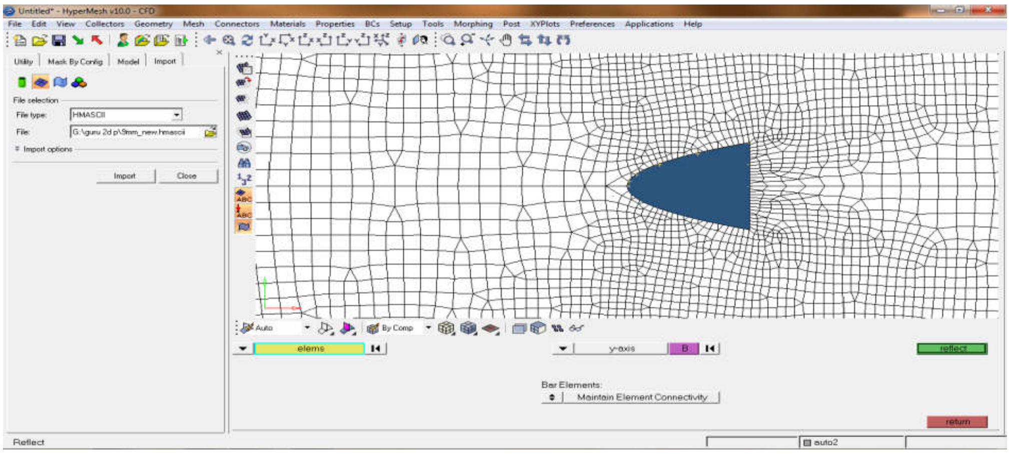

Figure 3. Meshing of pressure far field with bullet in YPERMESH vR20 software

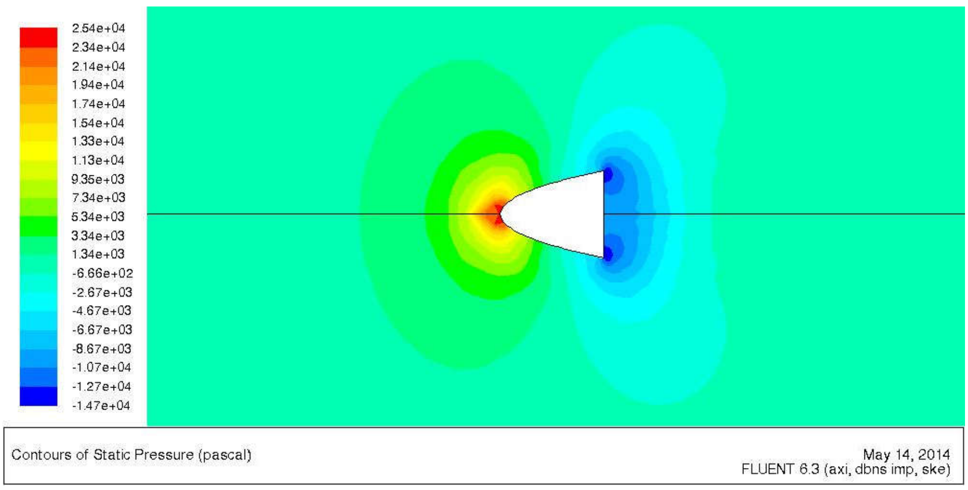

Figure 4. Counter of static pressure at Mach number 0.6

(Figure 4 and 6) shows the stagnation pressure (zero velocity) at the apex of bullet due to no slip condition. Analysis of pressure counterplots at Mach number 0.6 shows that there is an increase in pressure at the tip and normal shock is observed around the front end of bullet whereas very less pressure is found at rare end of bullet. The pressure rises up to Mach number 1. It is also curved in nature because the bow shock is curved in geometry.

At Mach number 3, stagnation pressure gradually shifts to the side surface of the bullet. The same condition is also appeared when the Mach number is 5. If Mach number is increased to 8, the stagnation pressure rise appears only at boundary. Similarly, the same pressure raise trend appears for Mach numbers 11, 14 and 17. There is a drastic pressure rise and high pressure drag at Mach number 20.

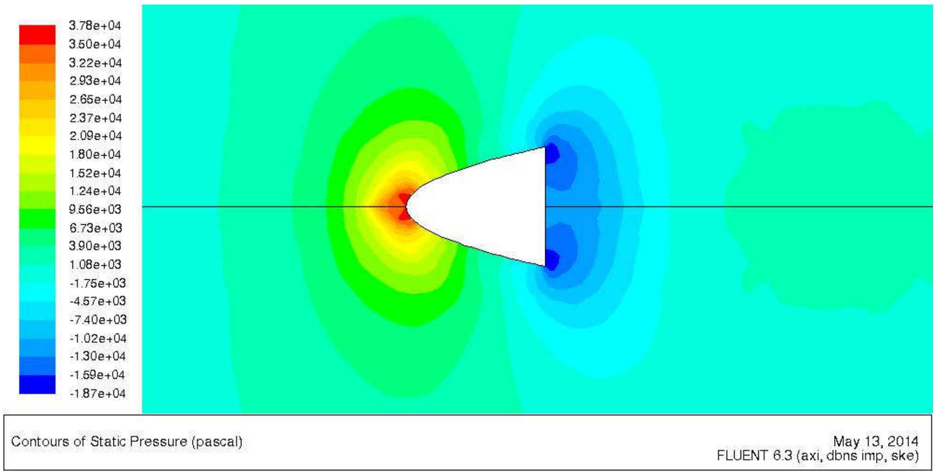

Figure 5. Counter of static pressure at Mach number 1

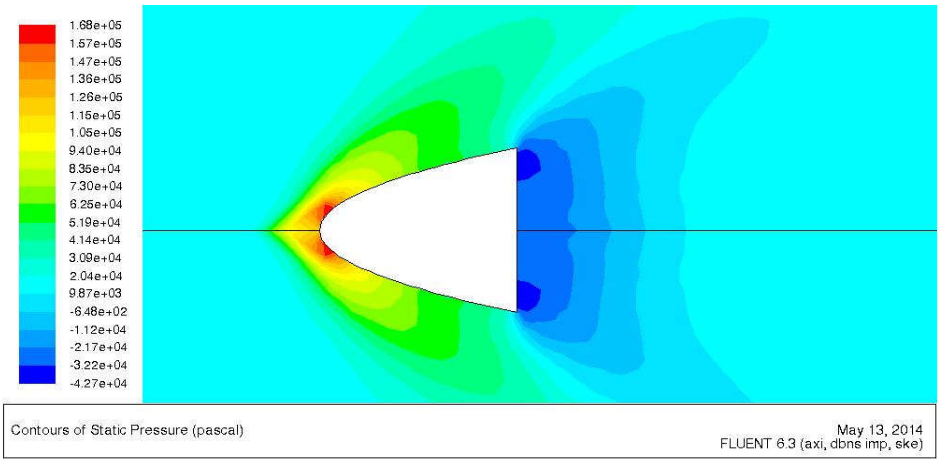

Figure 6. Counter of static pressure at Mach number 3

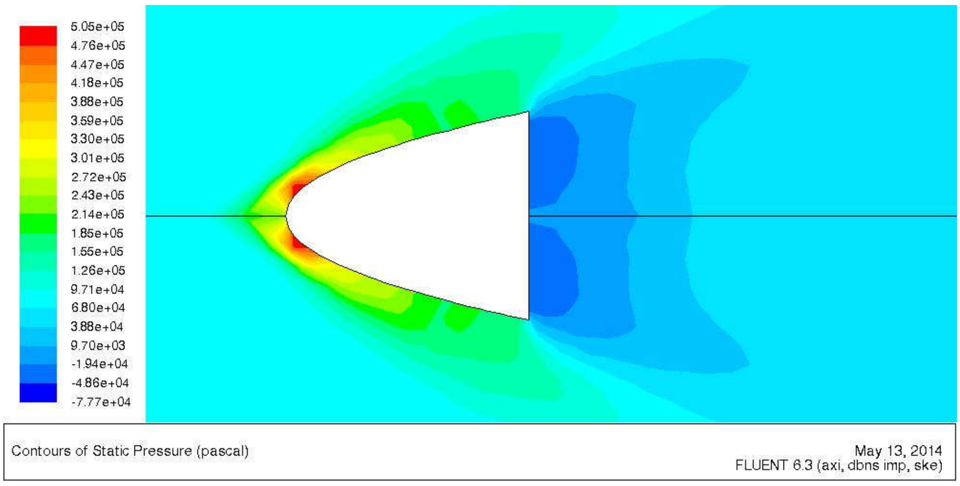

Figure 7. Counter of static pressure at Mach number 5

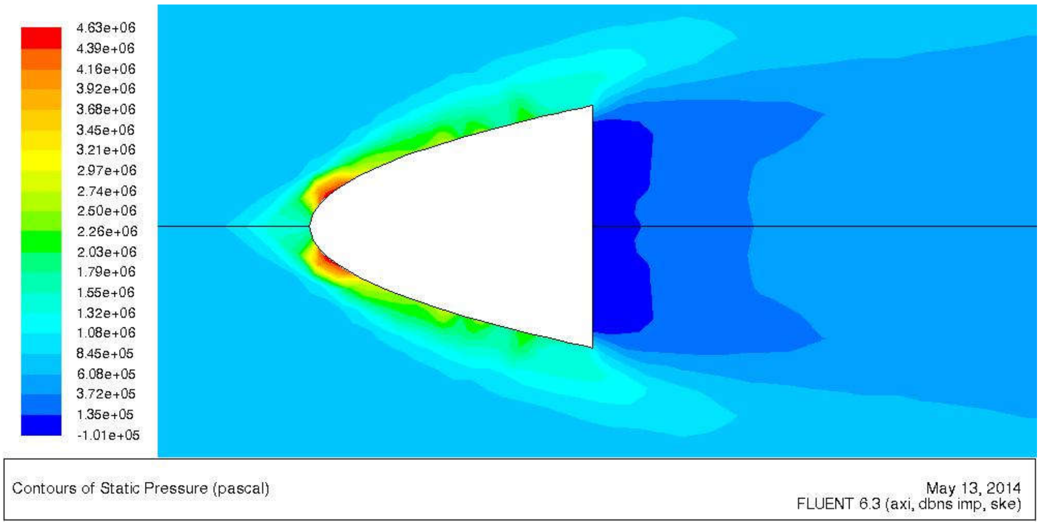

Figure 8. Counter of static pressure at Mach number 8

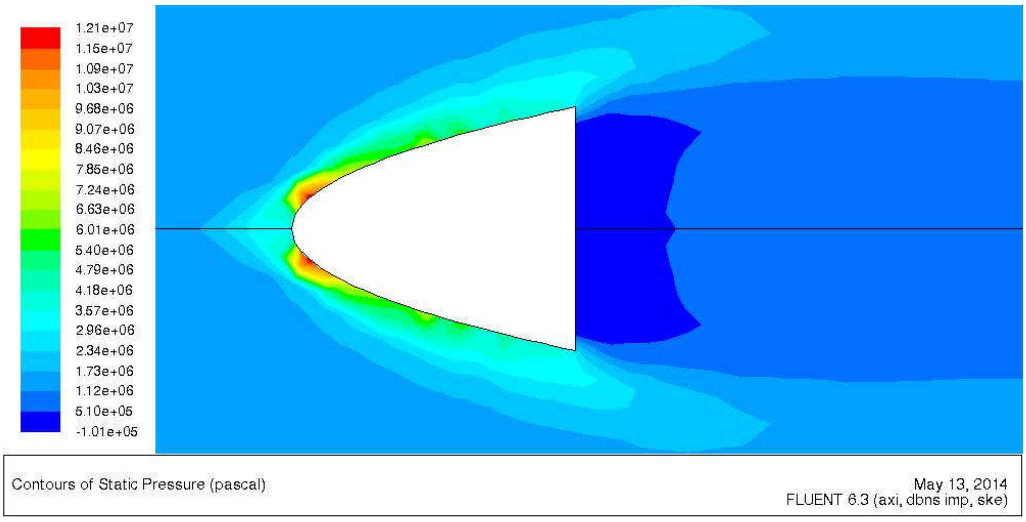

Figure 9. Counter of static pressure at Mach number 11

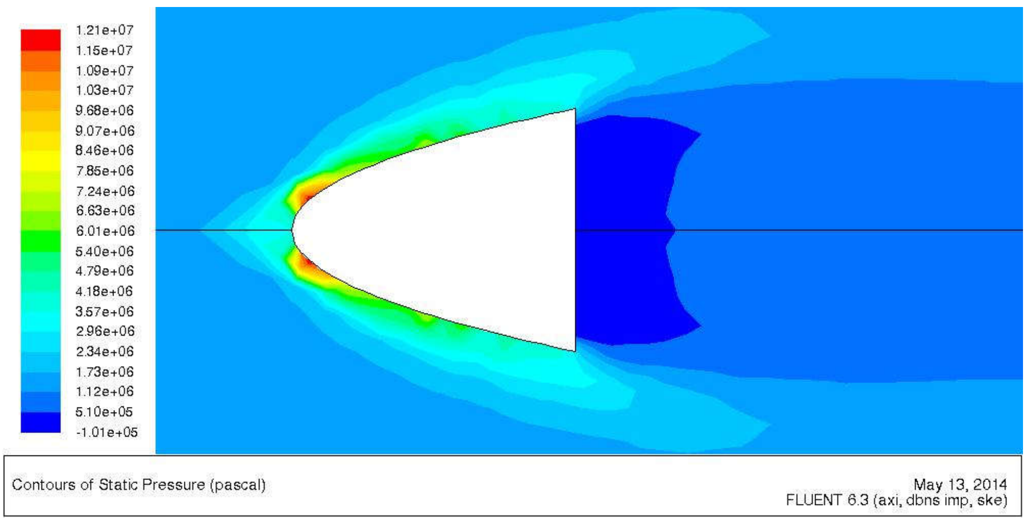

Figure 10. Counter of static pressure at Mach number 14

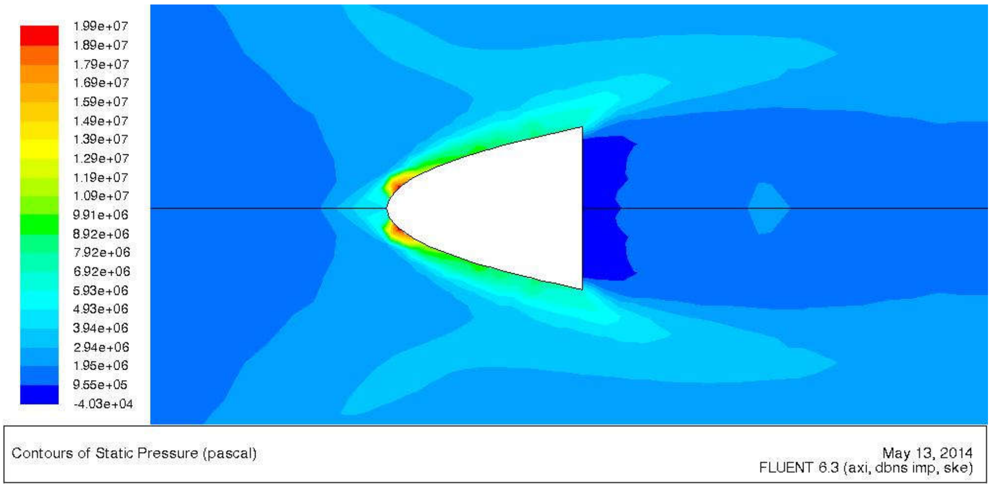

Figure 11. Counter of static pressure at Mach number 17

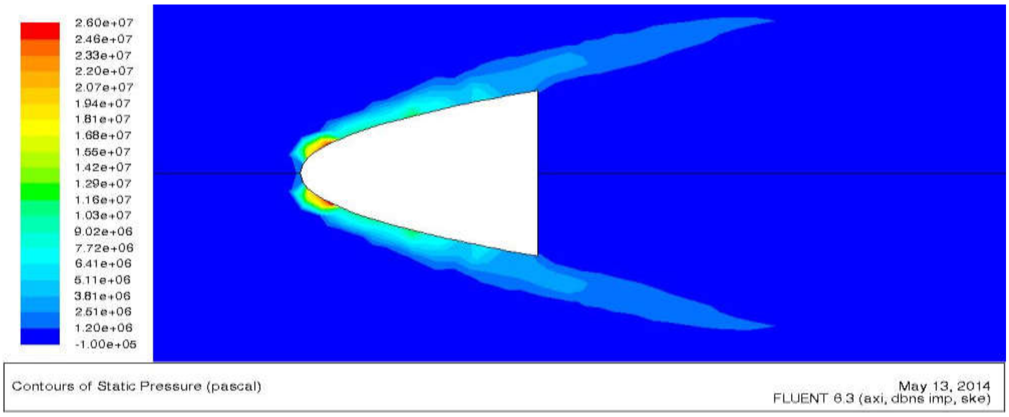

Figure 12. Counter of static pressure at Mach number 20

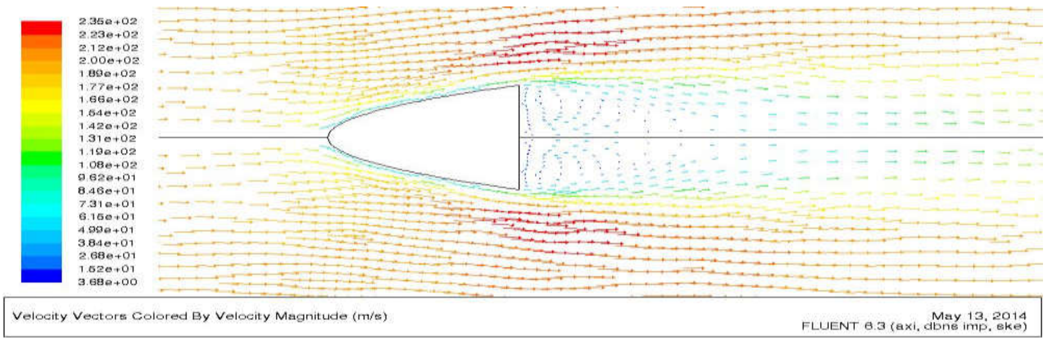

Figure 13. Velocity vector around the bullet at Mach number 0.6

Variation of velocity vector with different Mach numbers

Figures 13 to 21 shows the velocity vector contour plots for Mach numbers 0.6,1,3,5,8,11,14,17, and 20 respectively. Velocity vector plots show that velocity is zero at the apex of the bullet and velocity is increased when fluid is moved away from the surface of the bullet. It is reduced around the surface of bullet. At Mach number 0.6 (subsonic speed), there is a slightly disturbance in the fluid flow at the rare end side of the bullet. At Mach number 1 (sonic speed), velocity is high at the shock wave region and eddy formation is appeared at rare end.Velocity increases rapidly in tip and at boundary of the bullet at Mach number 3, 5, 8, 11 and 14. At Mach number 17 and 20, there is a turbulent flow at the rare end which increases the drag coefficient. Finally observing all the velocity vector contour plots, it is observed that flow velocity pattern is better up to Mach number 5. It is because of the fact that up to Mach Number 5, coefficient of drag is zero. Then, coefficient of drag is increased with the increase in Mach number. It is maximum at Mach number 20.

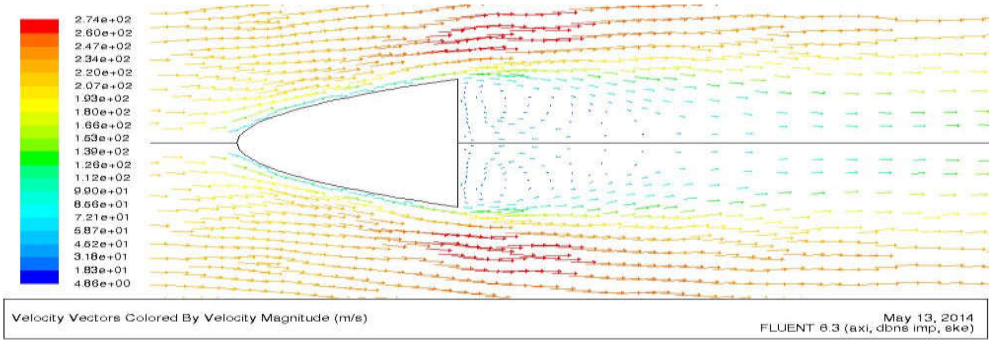

Figure 14. Velocity vector around the bullet at 1 Mach number 1

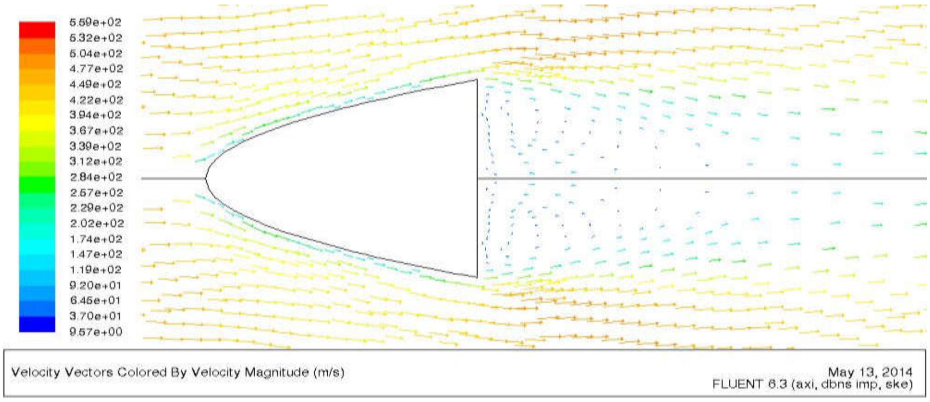

Figure 15. Velocity vector around the bullet at Mach number 3

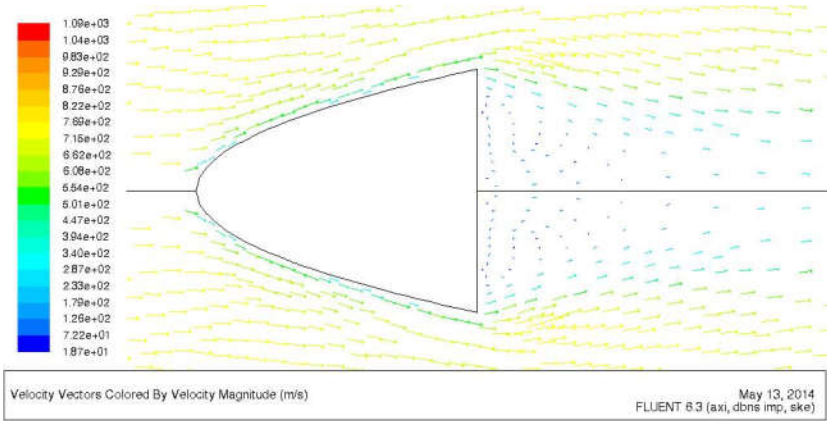

Figure 16. Velocity vector around the bullet at Mach number 5

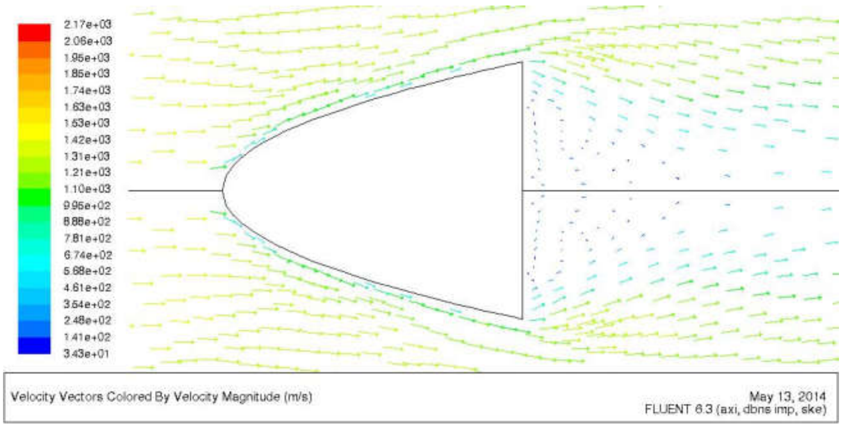

Figure 17. Velocity vector around the bullet at Mach number 8

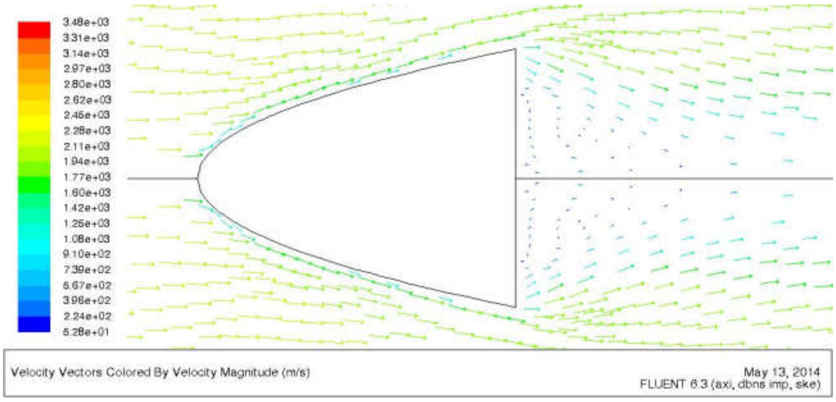

Figure 18. Velocity vector around the bullet at Mach number 11

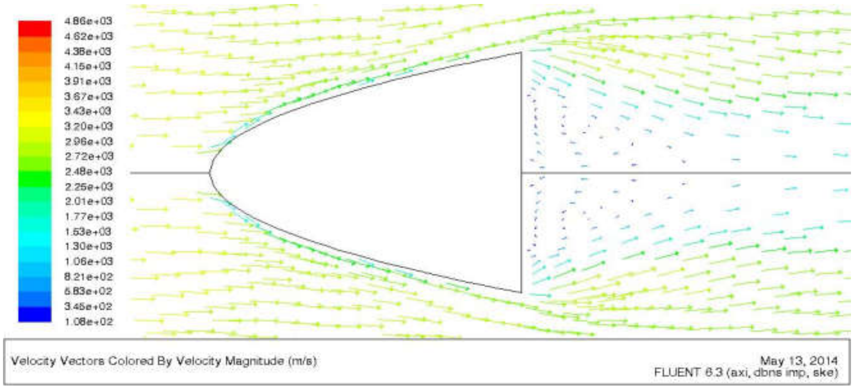

Figure 19. Velocity vector around the bullet at Mach number 14

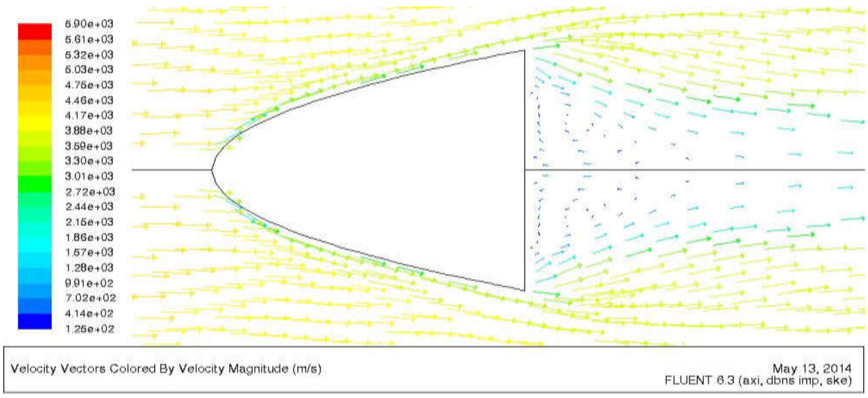

Figure 20. Velocity vector around the bullet at Mach 17

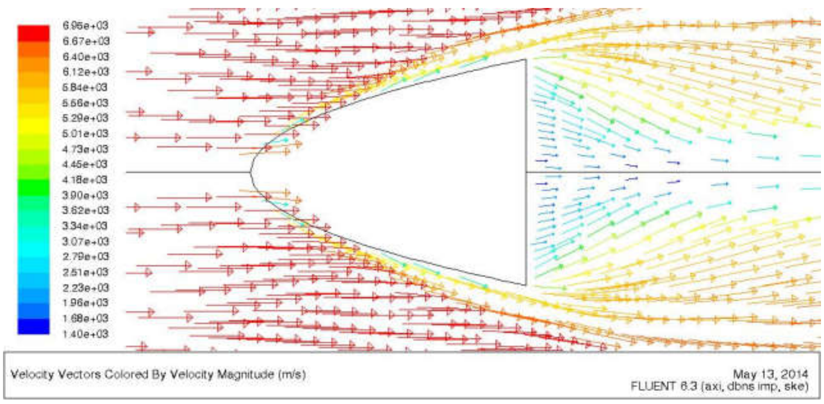

Figure 21. Velocity vector around the bullet at Mach 20

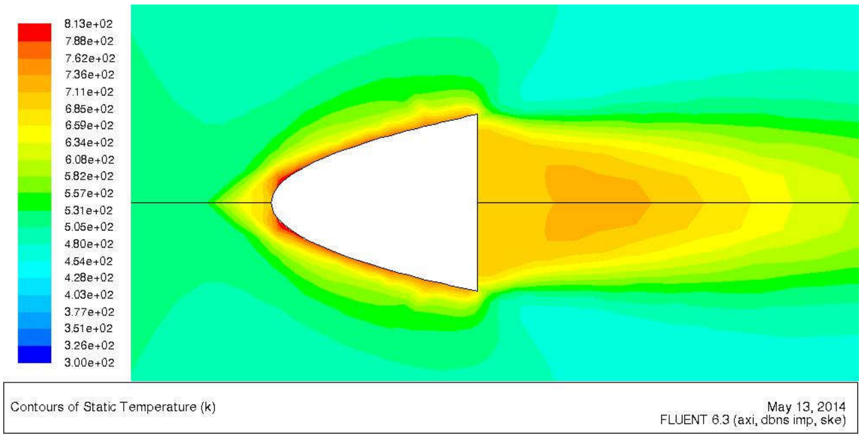

Figure 22. Contour of static temperature at Mach number 5

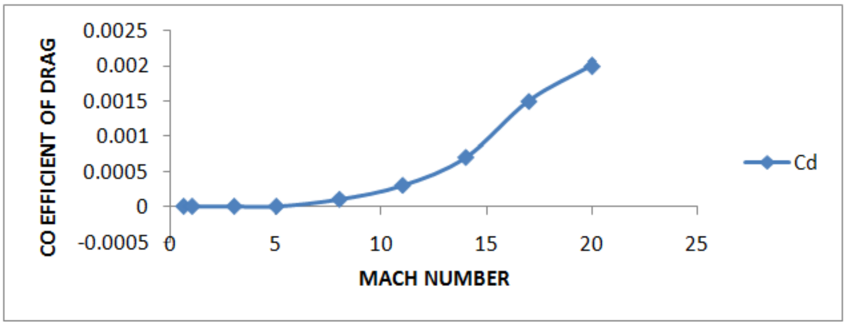

Figure 23. Effect of Mach number on Co efficient of drag

Effect static temperature at Mach Number 5

Figure 22 shows the contour of static temperature at Mach number 5. It is observed that maximum temperature has appeared at the boundary surface of the bullet. Minimum temperature is appeared at the tip of the nose of the bullet. Temperature analysis is done for the bullet at Mach number 5 because it has minimum drag coefficient.

Effect of Mach Number on Coefficient of Drag

Figure 23 shows the effect of different numbers on drag coefficients. Figure 23 indicates that drag coefficient is zero for Mach numbers 0.6, 1, 3, and 5 and then it is increased with the increase in Mach number up to 20. Maximum drag of about 0.002 is found at Mach number 20.

Conclusion

Based on the analysis on semi blunt 9mm bullet, the following conclusions are derived.

1. The analysis is carried out by considering the gravity and viscosity of the fluid on bullet at Mach number 5, air density is appeared more in the entropy layer and air is trapped inside the viscous boundary layer. It results in low temperature and low coefficient of drag.
2. Better performance of semi blunt 9mm bullet and any blunt nosed bodies can be obtained at Mach number 5.

REFERENCES

LongboZRCSC, JunjieKJWPT. The three dimension numerical simulation of aerodynamics of mortar bullet.*Journal of ballistics*, 1999.

Maitra SN. Aerodynamic heating of ballistic missile including the effects of gravity.Mathematics Department, National Defence Academy, Khadakwasla, Pune 411 023, India MS received on 1 February 1999.

Osborne WK, Crane JFW.Flow Field and Pressure Distribution Measurements on Blunt-Nosed Bodies at M=6.8. Ministry of aviation aeronautical research council current papers in 1962.

Robert L, McCoy.The Aerodynamic Characteristics of 7.62mm Match Bullets.Memoramdum report BRL-MR-3733 IN 1988.

Seshimo Y, Fujii M, The supersonic blunt body problem - review and extension.*Journal Aero/Space Sciences*, 25, pp.485-494. August, 1958.

Tarique Hasan Khan, Sudipta Saha. Numerical Simulation and Aerodynamic Characteristic Analysis of a Paraboloid-Tip Bullet", Proceedings of 4th Global Engineering, Science and Technology Conference 27-28 December, 2013, BIAM Foundation, Dhaka, Bangladesh ISBN: 978-1-922069-43-6.

Tomi Honkanen. Numerical Simulation of Bullet Flow Fields.Kul-34.4551 Postgraduate Seminar on Fluid Mechanics, 19.03.1993.

****