DRAG PREDICTION AND VALIDATION OF STANDARD M549, 155mm PROJECTILE

[1]Kiran Torangatti, [2]Dr.Basawaraj

[1]Research Scholar, Department of Aerospace Propulsion and Technology, VTU-CPGS Bangalore -560018,Karnataka, India.
[2]Associate. Professor, Department of Aerospace Propulsion and Technology, VTU-CPGS Bangalore- 560018.Karnataka, India.

Abstract: **Ever-increasing demands for accuracy and range in modern warfare have expedited the optimization of projectile design. The crux of projectile design lies in the understanding of its aerodynamic properties early in the design phase. In this research first investigated the aerodynamic properties of a standard M549, 155mm projectile. Aerodynamic data from wind tunnel and range testing was benchmarked against aerodynamic prediction and semi empirical design codes like MCDRAG, NSWCAP and Aero-Prediction. Further we deal with the prediction of drag by benchmarking the results of standard M549 projectile design and the predicting the boat tail angle effect for different drag coefficient. The detailed study is done and validated to reduce drag and see its effect on the projectile design for both transonic and supersonic speeds.**

Keywords: **Drag reduction, Pressure Drag, Supersonic, Transonic, Total Drag and Viscous Drag.**

I. INTRODUCTION

Drag is the main drawback for flight and to its cost consideration. The flight motions that influence the major forces briefly explained. Aerodynamic forces are the one major force which is further again divided into two categories i.e., lift and drag or (lift force or drag force). With respect to the different flight conditions of flight performances we can calculate the drag by analyzing it. The major work of the flight engineer is to calculate it the drag of flight for various speeds, altitudes and different design configuration and try to analyze, how it can be reduced to increase the performance. It is the main drawback because to go through all that it’s very difficult task forces will be different on different configuration for various different parameters.

For predicting the drag coefficients many research has been undergone, Jubaraj sahu [1] was carried out work on Drag prediction for different projectiles at transonic and supersonic speed. Test is conducted for three different models i.e., SOC configuration, SOCBT configuration and M549 projectile. Prediction of the various drag coefficients is done with different design codes available for the configuration. M. A. Suliman, et al., [2] have carried out the work on, a computational investigation for the 155mm M107 model to reducing the base drag. Three case studies were conducted to investigate the properties of the flow field for Mach numbers at zero angle of attack, with boat tail, with base cavity and base bleed. Sor Wei Lun [3] performed a computational investigation comparing the wind tunnel testing data with computational results with 3D unstructured analysis and tilting of the projectile’s nose to investigate the resulting aerodynamic effect. F. Simon, S. Deck, P. Guillen, [4] investigated the flow around a secant o-give cylinder boat-tail configuration (SOCBT) have been achieved and compared to available experimental data. Both time-averaged and unsteady results are provided to get a better insight into the physics of such flows. The influence of the Mach number and of the rotation is assessed for both static averaged coefficients and fluctuations.

From the related researches it motivated for prediction of drag over the M549 155mm projectile by validating it and see the effect of drag by changing the mesh properties using ANSYS tool and different boundary conditions.

II. BOUNDARY CONDITIONS

Any numerical simulation can consider only part of the real physical domain or system. Truncation of the domain leads to artificial boundaries where we have to prescribe values of certain physical quantity. Improper implementations of the boundary conditions can result in inaccurate simulation of the real system or problem; additionally the stability and convergence speed of the solution scheme can be negatively influenced.
Some of the boundary conditions which are used in solving the real system are,

- Solid wall
- Far-field
- Symmetry
- Co-ordinate cut
- Periodic boundary
- Interference between grid blocks

Far-Field

The numerical implementation of the far field boundary condition has to fulfil basic requirements; first the truncation of the domain should have executable effect on the flow solution as compared to infinite domain. Second is the, any outgoing disturbance must not be reflected back into down flow due to their elliptic nature, subsonic and transonic flow problems are particularly sensitive to the far field boundary condition.

For static pressure;

$$\frac{P}{P_0} = \left[1 + \left(\frac{\gamma-1}{2}\right) M^2\right]^{\frac{\gamma}{\gamma-1}} \qquad (1)$$

Where P_0 = Total pressure = 101325 Pa

P = Static pressure

$\gamma = 1.4$ Constant for air

M = Mach number

For static temperature;

$$\frac{T}{T_0} = 1 + \left(\frac{\gamma-1}{2}\right) M^2 \qquad (2)$$

Where T_0 = Total temprature = 311K

T = Static temperature

$\gamma = 1.4$ Constant for air

M = Mach number

III. METHODOLOGY

In main objective of this project is prediction of drag is numerically simulated for transonic and supersonic speeds and analysed for different influencing factors like

a) Various drag coefficient over missile geometry
b) With respect to different Mach number from transonic to supersonic
c) With zero angle of attack.

Known experimental results predicted by semi empirical codes from the paper Jubaraj sahu [1] are used to benchmark the results for standard M549, 155mm projectile and simulating the problem using CFD code which describes the physics behind the flow.

The tool used for getting accomplished this project work is ANSYS 14.5, which has inbuilt CAD and solver tools that are used for modelling, meshing and post processing as named below.

- Design Modeller – Geometry

- Pre-Meshing tool in workbench
- Solver – FLUENT and for post processing

After benchmarking the result, the tail angle design is changed from M549 and simulated using CFD Code to investigate the effect of drag over the missile body and further case studies are being carried out for other angles and the prediction for drag coefficient is obtained, compared with the remaining respective angles with graphical plots for pressure drag coefficient, viscous drag coefficient, and total drag coefficient.

Design Model

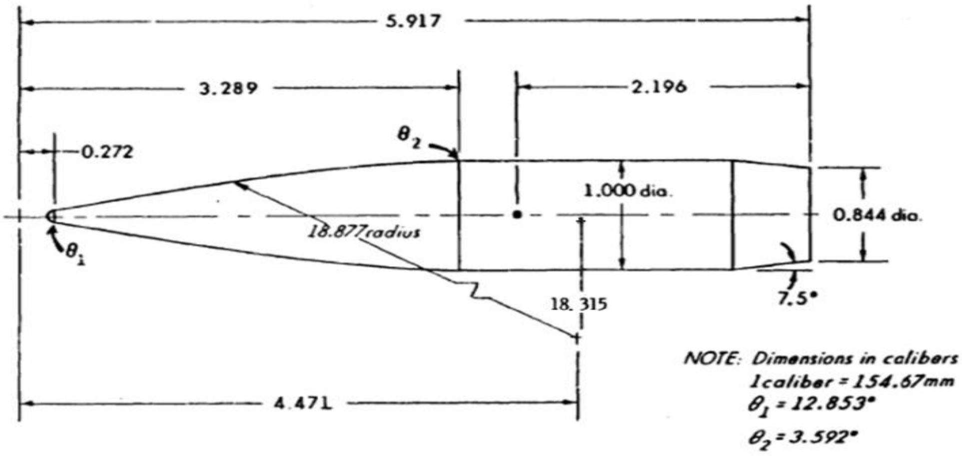


Fig. 1 Computational Model [1]

Fig 1 shows the 2D model geometry of the projectile. With some of the modifications made for computational simulation i.e., o-give cone with a flat nose replaces the o-give nose and the rotating band at the end of tail section is eliminated. The simplified computational model is used for numerical computations and all the dimensions are in calibers.

Mesh Generation

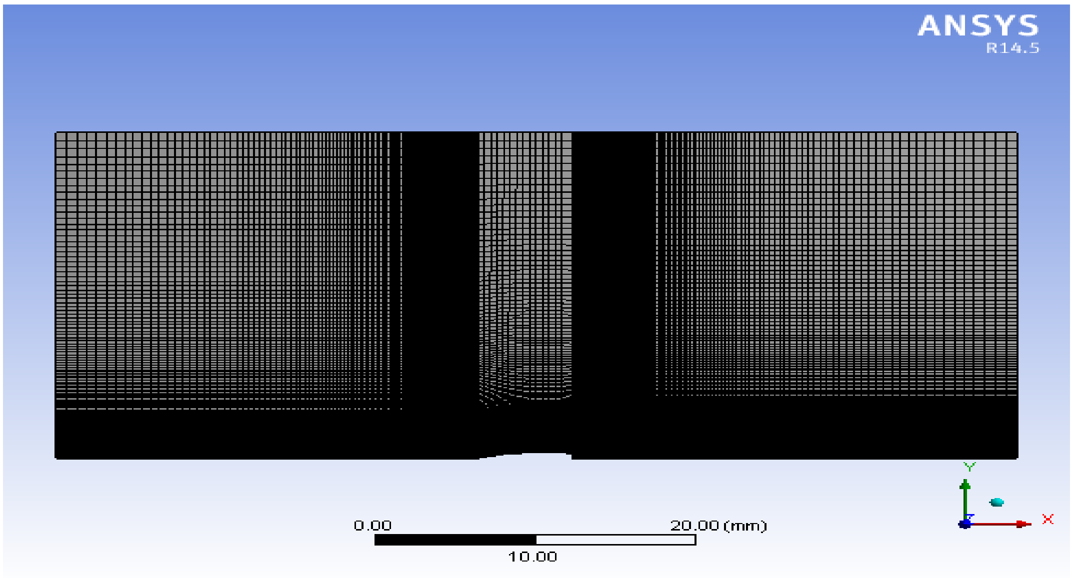


Fig. 2 2D Structured Mesh domain

Fig 2 shows the 2D structured mesh of M549 projectile, with domain size of 30x60 dimensions are in calibers in which the projectile geometry is been subtracted as it is related to prediction of the drag so the flow is over the surface of the missile body and details of the mesh is given below for the domain.

- No. of nodes – 1,10,751
- No. of elements – 1,10,000
- Mesh quality:

Orthogonal quality of mesh ranges from 0 to 1. Where values close to zero corresponds to low quality

Minimum Orthogonal Quality – 0.833129579949384

Maximum Aspect Ratio – 223.679

The following boundary conditions, solving models and schemes are used in post processing procedure using the above mesh details and with quality of the 2D mesh model.

- $k - \varepsilon$ Turbulence model with realizable enhanced wall functions as it is a related to the projectile outer surface and domain top wall surface and the reason behind using this model is that, we have not considered the y+ wall functions as in that case we should use k-ω turbulence model.
- Explicit scheme is used in solving the solution.

IV. RESULTS AND VALIDATION

Various drag coefficients namely, skin friction drag coefficient (viscous drag coefficient) and pressure drag coefficient, as well as total drag coefficient using some semi empirical codes by Jubaraj Sahu [1] predictions. In the present work drag coefficients obtained from ANSYS FLUENT are compared with NSWCAP, MCDRAG predictions by Jubaraj Sahu [1], and are shown in Figure 4.1.5, Figure 4.1.6, and Figure 4.1.7.

For validation of results, input parameters used are as shown in Table 1, the boundary conditions used for inlet, outlet and top wall is the pressure far-field conditions, were we assign static pressure and temperature for the respective Mach number using the far-field formulas, the values of pressure and temperature for different Mach numbers as show in the

Table 1 Shows input boundary condition parameters to predict the drag coefficient.

Mach Number	Pressure In (Pascal)	Temperature in(K)
0.91	59260.750	266.810
1	53528.152	259.166
1.5	27601.240	214.482
2	12949.793	172.777
3	2758.439	111.071

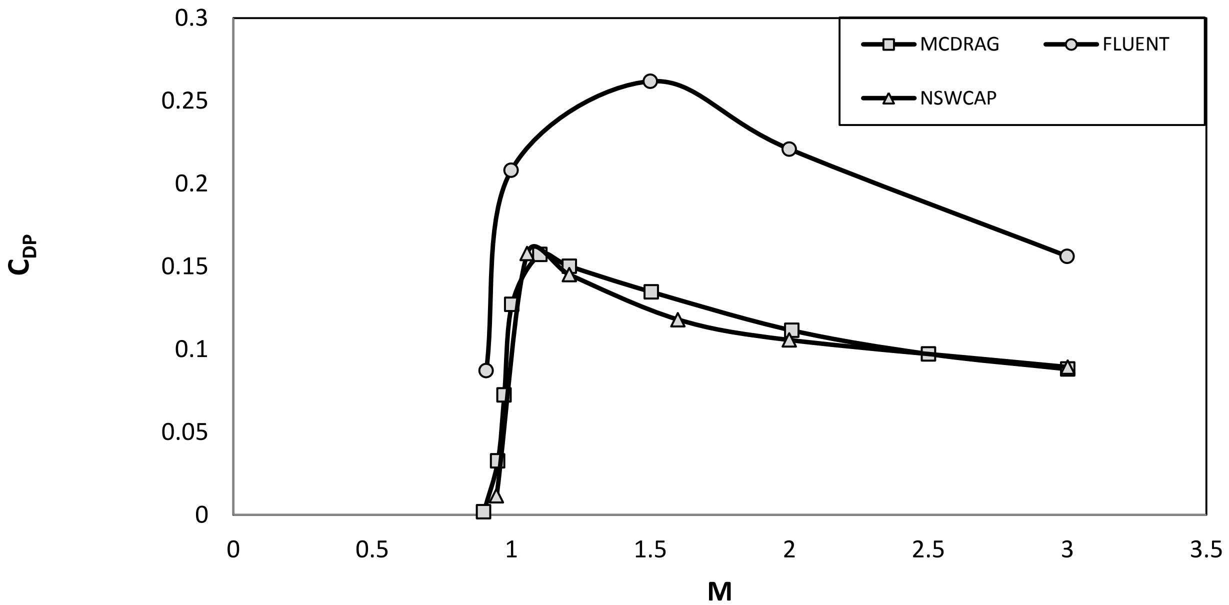


Fig. 3 Pressure Drag Coefficient V/S Mach number

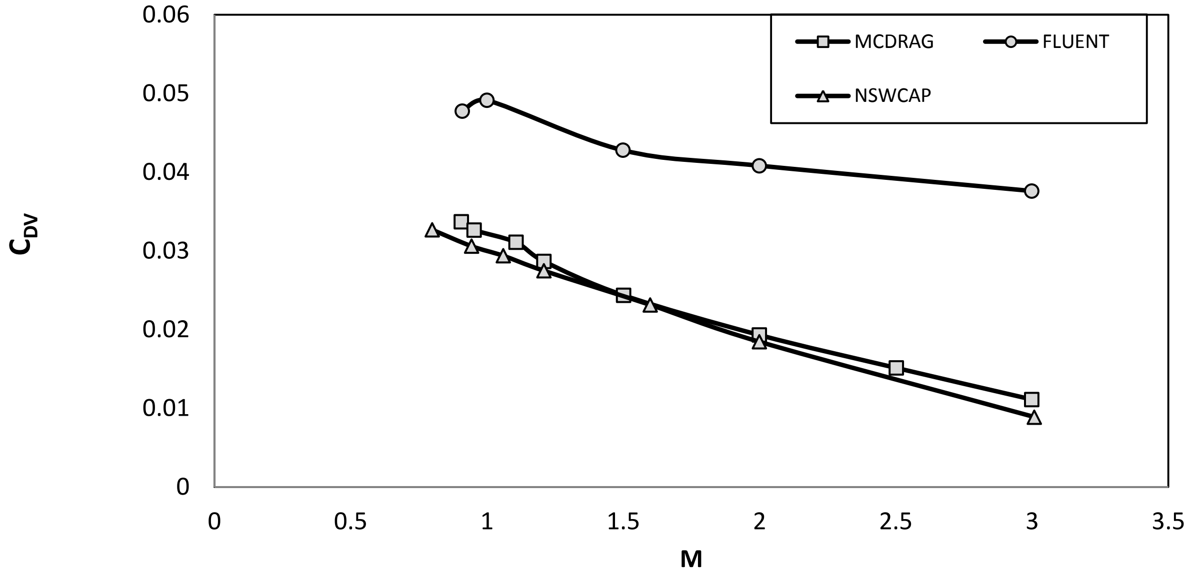


Fig. 4 Viscous Drag Coefficient V/S Mach number

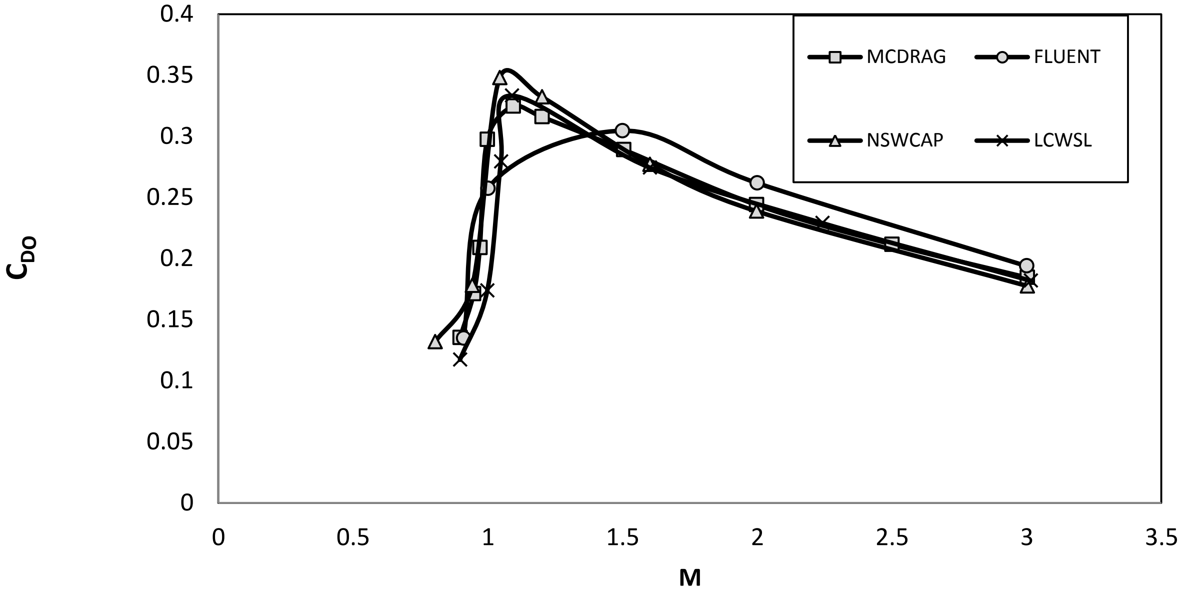


Fig. 5 Total Drag Coefficient V/S Mach number

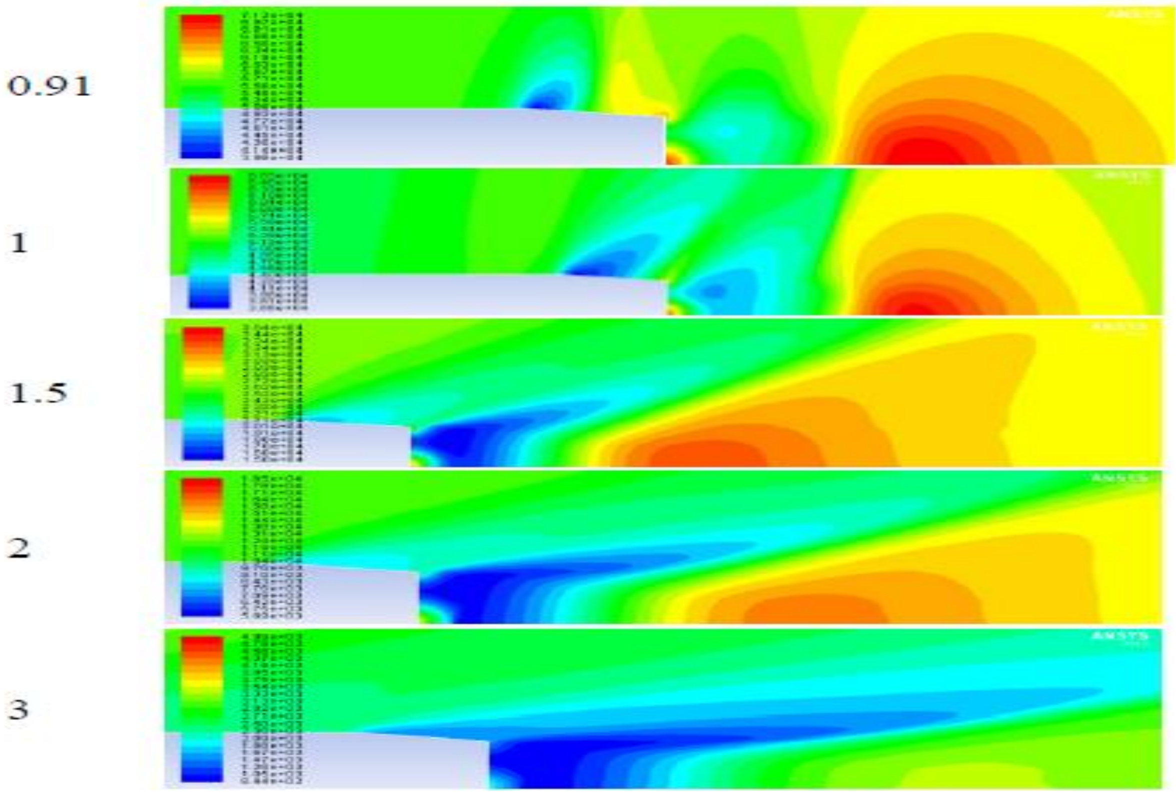
Comparing the results with semi empirical code prediction from J. sahu [1], we can see that in the total drag coefficient there is 28.5% reduction in transonic to some part of supersonic region (i.e., from Mach number 0.91 to 1.5) and around 5% increase in the drag towards supersonic region (i.e., from Mach 2 to 3) as shown in the in the Fig 3, Fig 4 and Fig 5.

Contour Plots

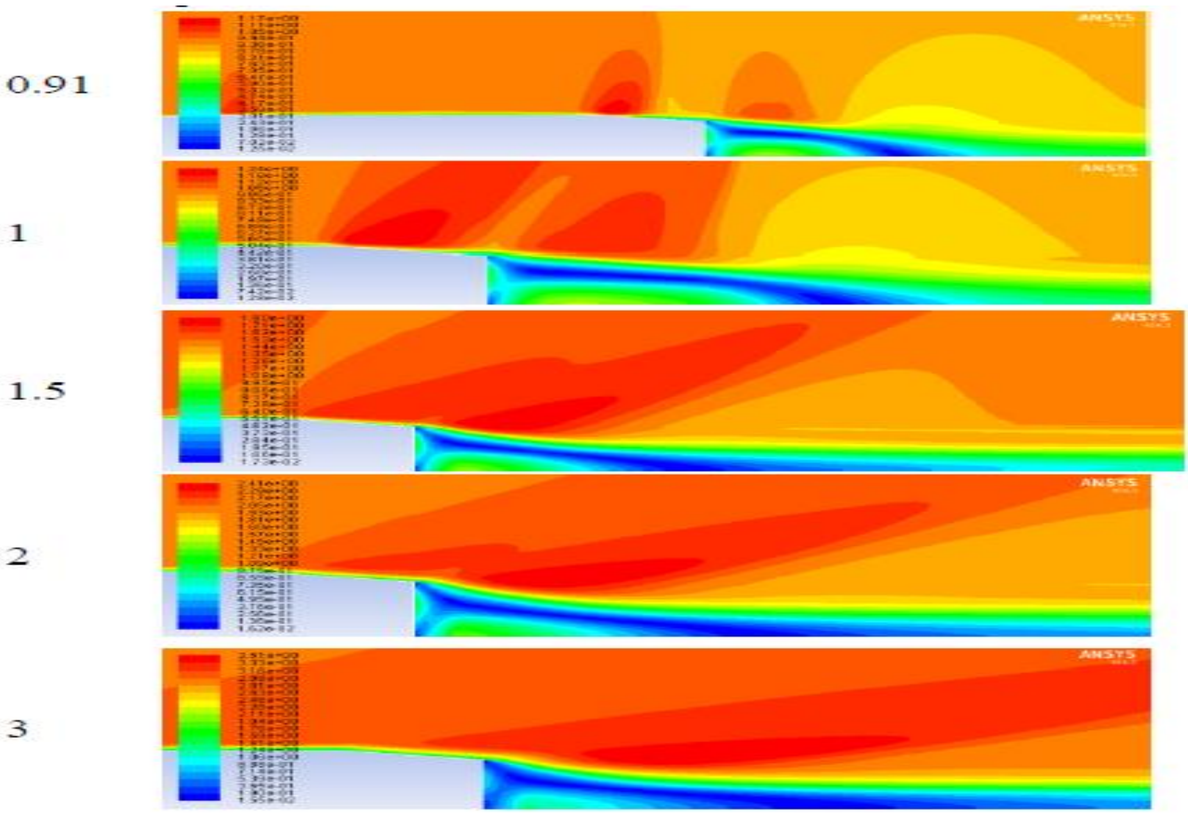
In the below contour plots we clearly see the variation in the static pressure, Mach contour or the velocity vector plots which shows the reduction in the wake region obtained by effect Mach numbers which clearly tells the physics behind the flow were in as the static pressure goes on reducing as the Mach number increases and the flow separation at the tail region goes on reducing which indicates the reduction in the drag coefficients.

Where through the nature of the graph obtained there is increase in the drag in transonic region and it starts reducing as the approaches the supersonic region.

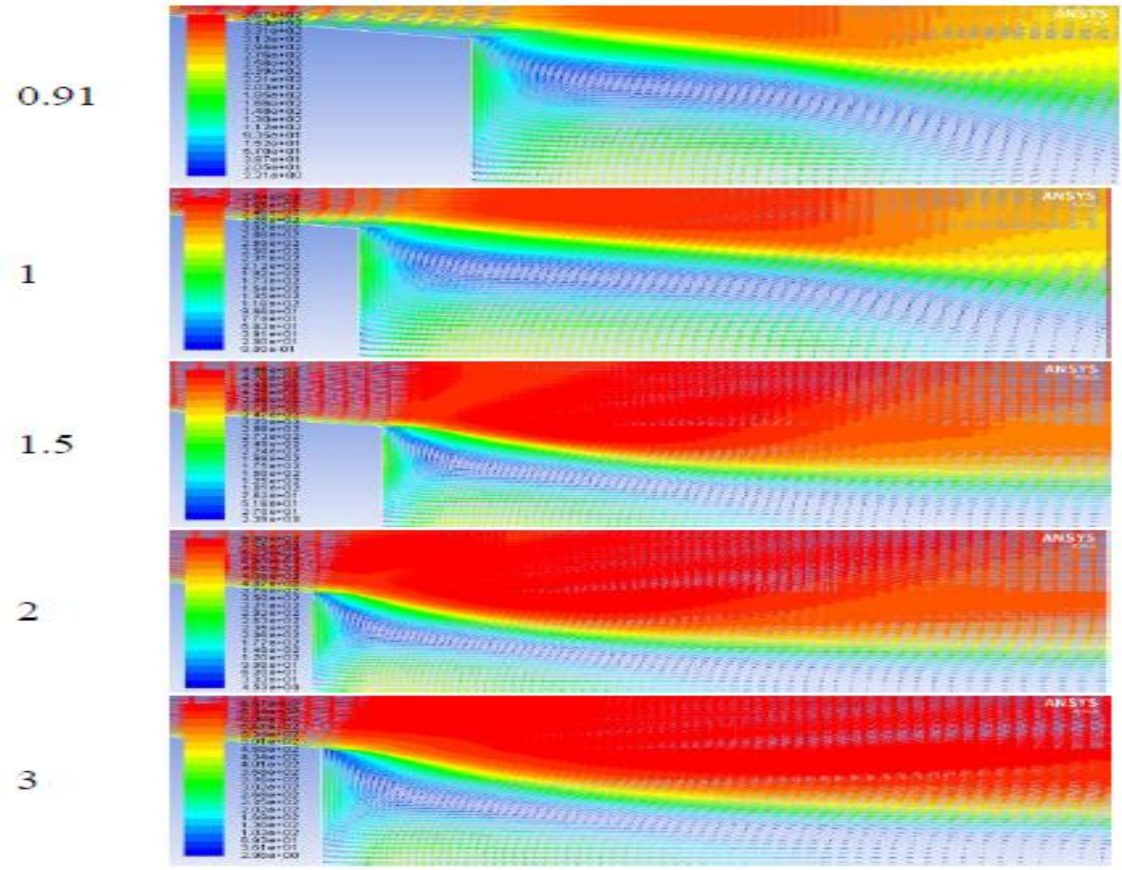
Contour plots of static pressure from Mach 0.91 to 3,



Contour plots for Mach number from Mach 0.91 to 3,



Contour plots for Velocity vectors by velocity magnitude from Mach 0.91 to 3,



V. CONCLUSION

The numerical simulation of the M549 155mm projectile in ANSYS 14.5 tool made easy to predict the drag coefficients. Where in present work we validated the results of the work published by J. Sahu [1] for standard M549, 155m projectile, there was a 28.5% decrease in the drag in transonic to sonic region and around 5% increase towards supersonic region.

According to the physics the flow contour plots for all Mach numbers from 0.91 to 3 explains, the static pressure at base region of projectile tail is reduced during which the velocity of the fluid is increasing, which clearly indicates the reduction in drag coefficients.

REFERENCES

[1] J.Sahu, "Drag Predictions for Projectiles at Transonic and Supersonic Speeds," US Army Ballistics Research Laboratory, Aberdeen Proving Ground, MD, BRL- MR-3523, June 1986.

[2] M. A. Suliman, "Computational Investigation of Base Drag Reduction for a Projectile at Different Flight Regimes," Military Technical College, Kobry Elkobbah, Cairo, Egypt, ASAT- 13, May 26 – 28, 2009.

[3] Sor Wei Lun, "Aerodynamic Validation of Emerging Projectile Configurations," Naval Postgraduate School Monterey, CA 93943-5000, December 2011.

[4] F. Simon, S. Deck, P. Guillen, R. Cayzac, P. Sagaut and A. Merlen, "Rans/Les Simulations of Projectiles with and without Rotation in the Subsonic and Transonic Regimes", 23rd international symposium on ballistics Tarragona, Spain, 16-20 April 2007.

[5] J. D. Anderson, Jr, "Fundamentals of aerodynamics". New York, NY: McGraw-Hill Companies, 2010.

[6] J.D. Anderson, Jr, "COMPUTATIONAL FLUID DYNAMICS," The Basics with Applications, Copyright © 1995 by Mc Graw-Hill, Inc.

[7] M. F. Platzer, "lecture notes for missile aerodynamics," unpublished.