Optimization of Different 2D Nosecones By Computational Fluid Dynamics (CFD) Simulation in Order to Determine Drag.

Gopalakrishnan Dharmaraj
University of Hertfordshire

Abstract- *The study of space plays an important part in the expanding technology of the contemporary period, and it has made a significant contribution to the advancement of humans. The development of rockets via contemporary scientific research and technological innovation has made space travel more effective.*

Through the use of Computational Fluid Dynamics (CFD) modelling, the scope of this project include the optimization of nose cone design employing a variety of forms in order to contend with external forces at supersonic speed. When compared to current rocketry, the older types of rockets were less effective and didn't last as long thanks to improvements in technology and production processes.

Mathematical formulations are employed to create the nose cone for this project, and numerical methods are used in the simulation to conduct an analysis of the nose cone along with the primary parameters.

I. INTRODUCTION

As part of my scientific research, I have created nose cones with a variety of different configurations by making use of mathematical formulations. I then used the computational fluid dynamic approach to study how these nose cones behaved while travelling at supersonic speeds.

Comparisons are made between the different shapes of the nose cone depending on the amount of drag that is generated by each one. The goal is to identify the profile that generates the least amount of drag and then improve the performance of the profile.

In this part of the simulation, both the continuity equation and the energy equation are taken into consideration together with the relevant parameters. In addition to these essential equations, the boundary layer theorem is used in order to do an analysis on the vortex.

II. OBJECTIVES

The purpose of my paper is to design a nose cone with different shapes, then run a numerically based simulation to determine its pressure and drag, and finally compare these parameters to one another in order to determine the lowest possible values. Then, the particular design can be modified with certain measurement values, and also compare the drag and pressure once again for these models in order to choose the best one that is capable of supersonic flight.

III. METHODOLOGY

This assignment involves taking four different forms of nose cone and designing them based on a formulation in the 3D programme solid works. First and foremost, a 2D design is constructed in accordance with the estimated value, and then an extrusion is used to produce the surface. After that, the modelled object is stored in a format that the CFD simulation programme, such as Ansys, would recognize. The imported geometry should then be subdivided in the appropriate manner in order to build a flawless mesh.

Following that, the meshing procedure is carried out as a kind of preprocessing by choosing the suitable Meshing Technique (Method- quadrilaterals and Type of Mesh - Mapped Mesh.) When the Mesh has been generated, the geometry should then be renamed according to the boundary condition.

As a third process, the physics of the geometry, including continuity and the energy equation, should be taken into consideration. Additionally, the type of boundary condition that should be given can be determined by selecting Mach number and pressure values of 2 and 101325, respectively, based on the requirements and conditions. Then, at long last, the setup has to be initiated and computed over a greater number of iterations in order for the simulation's computation to converge on the modelled nose cone shape.

The post-processing stage, which includes visualizing plots, vectors, contours, and other criteria depending on the needs or even the interest, is the very last step, but it is certainly not the least.

VELOCITY CONTOUR PROFILE

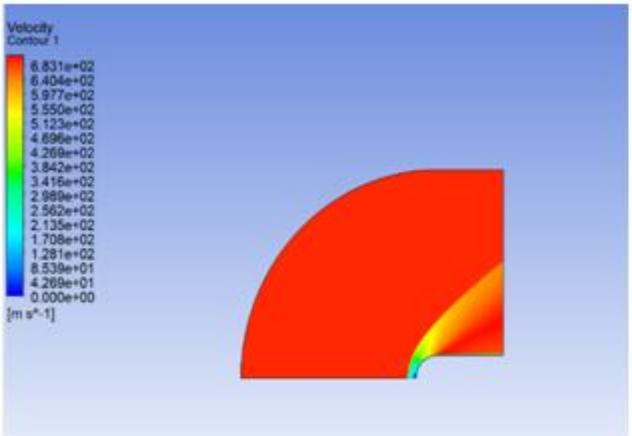

FIG 1: BLUNT BODY

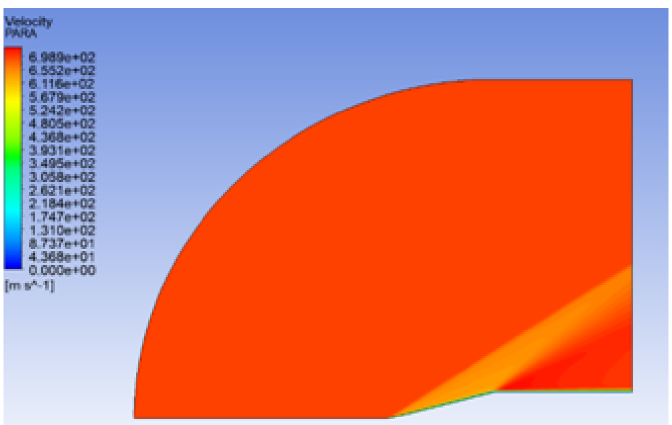

FIG 2: CONIC BODY

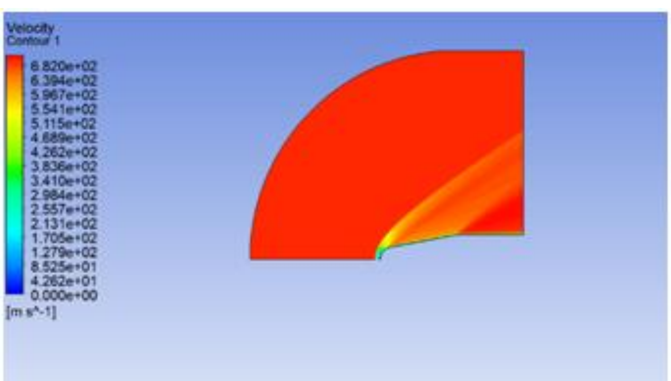

FIG 3: BIUNTED CONIC BODY

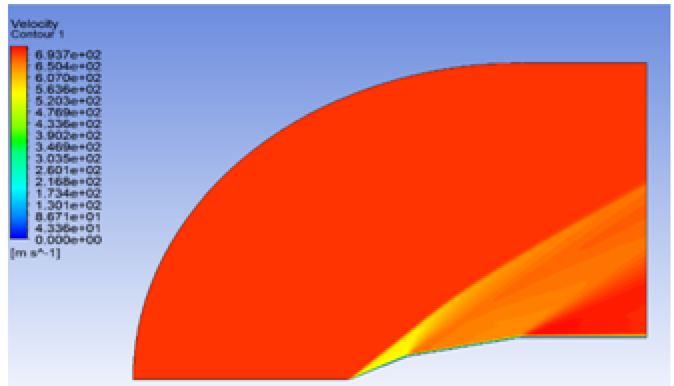

FIG 4: BI CONIC BODY

The selected four nose cone designs are studied with supersonic speed under the velocity profile in order to observe the existence of shocks across the body of the nose cone. However, in order to understand the pattern of Shock, it is required to first understand the phenomena of shock as well as the kind of shock.

SHOCK WAVE:

Any disturbance that travels at a speed greater than the local speed of sound in a medium is said to be a shock wave (often written shockwave) in physics. Shock waves, like regular waves, carry energy and may travel through a medium, but they are distinguished from regular waves by the rapid, almost discontinuous changes they cause in the surrounding pressure, temperature, and density.

TYPES OF SHOCK

1. Moving shock.
2. Detonation wave.
3. Detached shock.
4. Attached shock.
5.
6. However, in accordance with the study, it is necessary to explain in more depth the moving, detached, and attached shock in addition to the oblique and normal shocks.

NORMAL SHOCK:

A "normal shock" is a basic kind of shock wave. They are referred to as "normal" shocks because they travel in a direction perpendicular to the flow. When the flow speed is over the sound barrier, a shock is produced. It's not possible for the molecule to identify an obstruction at such speeds before the speed of sound, thus it has to double back after feeling the obstruction.

As an illustration of a normal shock wave, consider the ignition of propellant from the rocket engine on the ground in the form of smoke. In this scenario, the flow of smoke continues in a straight line as the rocket is lifted up in accordance with Newton's third law, and there is no interruption in the flow of smoke at that speed.

OBLIQUE SHOCK:

In the study of shock waves in a flow field that are still connected to the body, the shock wave that is offset from the flow direction at an arbitrary angle is referred to as an oblique shock.

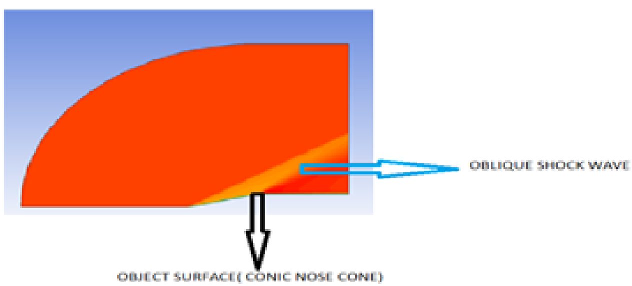

FIG 5: OBLIQUE SHOCK

BOW SHOCK:

When an oblique shock is likely to develop at an angle that cannot stay on the surface, a nonlinear phenomenon known as the shock wave forming a continuous pattern around the body occurs. This pattern is caused by the fact that the shock wave cannot remain on the surface. These kind of shocks are known as bow shocks.

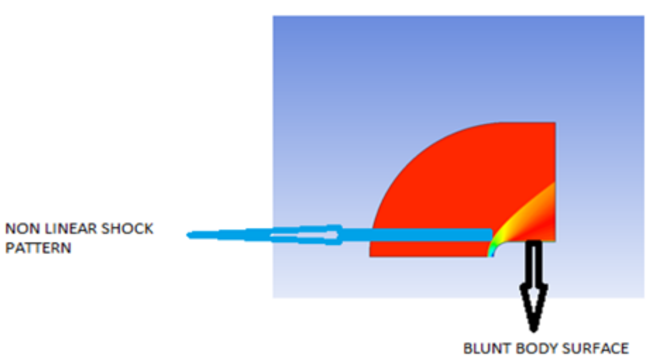

FIG 6: BOW SHOCK

MOVING SHOCK:

The term "moving shock" refers to a shock wave that is travelling through a fluid (typically gaseous) medium at a speed greater than the speed of the fluid itself.

It is crucial to explain the Mach regime and the Reynolds number in order to understand the flow behavior across the body before diving into the pressure contour.

In a manner analogous to shock waves, the phenomena of Mach number coupled with its nature is something that absolutely has to be discussed.

MACH NUMBER:

The ratio of an object's speed to the speed at which sound travels is the Mach number, and it is expressed as a fraction. It's a dimensionless quantity.

$$MACH\ NUMBER = \frac{\text{SPEED OF THE OBJECT}}{\text{SPEED OF THE SOUND}}$$

FORMULA

$$M = \frac{U}{C}$$

SCIENTIFIC NOTATION

M = Mach number.
U = Speed of the Object.
C = Speed of the Sound.
The speed of the sound can be calculated with respective formulae,
C = Sq. root (γ R T)

$$\gamma = \frac{\text{SPECIFIC HEAT OF A GAS AT A CONSTANT PRESSURE}}{\text{HEAT AT A CONSTANT VOLUME}} \quad \textbf{= 1.4 FOR AIR}$$

R = SPECIFIC GAS CONSTANT
T = STATIC AIR TEMPERATURE

It is vital to discuss **Reynolds Number**first in order to prepare for the Mach regime discussion. Along the same lines as the Mach number, the Reynolds number is likewise a dimensionless number, and it is most effectively articulated using formulas.

It is recommended to use formulas when describing the Reynolds number.

FORMULA:

$$Re = \frac{\rho\, u\, l}{\mu}$$

SCIENTIFIC NOTATION:

Re = REYNOLDS NUMBER.
ρ = DENSITY OF FLUIDS.
u = SPEED OF THE OBJECT.
l = CHARACTERISTIC LENGTH (IN CASE PIPE).
μ = FLUID DYNAMIC VISCOSITY.

The Reynolds number is broken down into subcategories according to the many kinds of flows, each of which has its own set of illustrative examples.

CASE 1:

IF Re is less than the 1 the flow is called STREAMLINE FLOW.

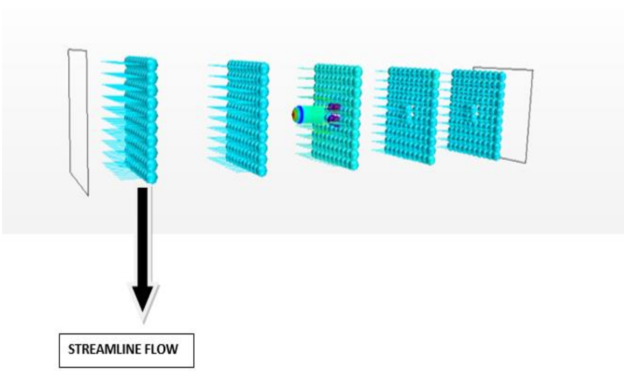

FIG 7: STREAMLINE FLOW

CASE 2:

IF Re is equal to the 1000 then the flow is called TRANSIENT FLOW.

CASE 3:

IF Re is greater than the 1000 the flow is called TURBULENT FLOW.

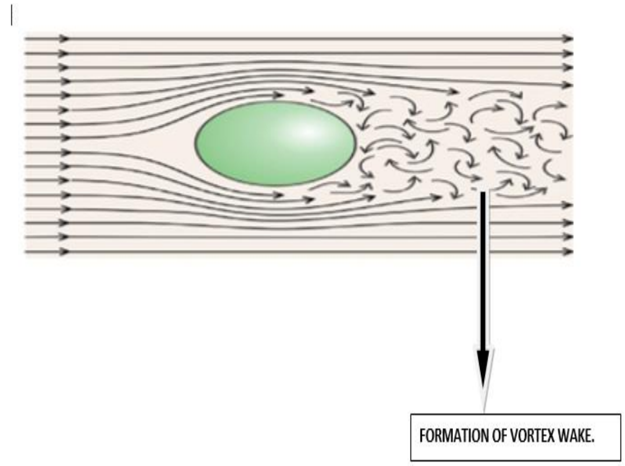

FIG 8: TURBULENT FLOW (courtesy: chemistry libre texts)

MACH REGIME:

The Mach number may be broken down into several categories according on the performance of the speed of the objects, which is referred to as the Mach Regime.

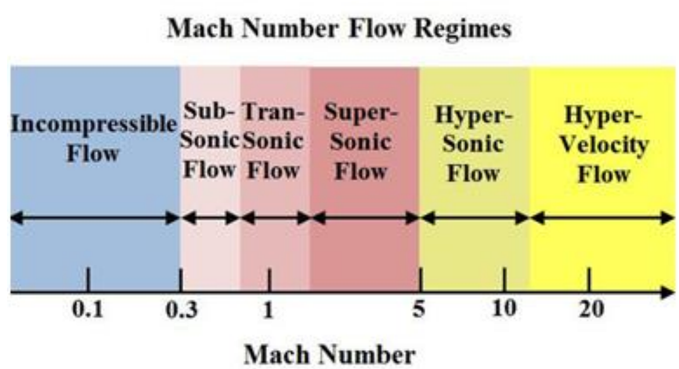

FIG 9: MACH REGIME (Courtesy: Wikimedia Commons)

BOUNDARY LAYER THEOREM:

A streamline flow with a velocity is being sent to the stationary solid body, and the fluid particle that will cling on the surface of that solid body will remain the same velocity as the solid body, whereas it was mentioned before that the body in stationary motion has a velocity of zero, which means that the velocity of the fluid particle will automatically get reduced to zero while it is remaining on the solid body boundary. However, the theory states that the particle will not be in a state of rest. Within a few seconds, the fluid particle moves and shows evidence of velocity gradient, which refers to variations in velocity in relation to the distance. After moving through the layer, the velocity will revert back to its free-flowing state. These velocity modifications will only last for a limited portion of the layer. The phenomenon that occurs when the velocity of particles in a thin layer changes is referred to as boundary layer. Shear stress is produced for a brief period of time inside this boundary layer; however, once the velocity reaches the freestream state, the shear stress will no longer be present and will be equal to zero.

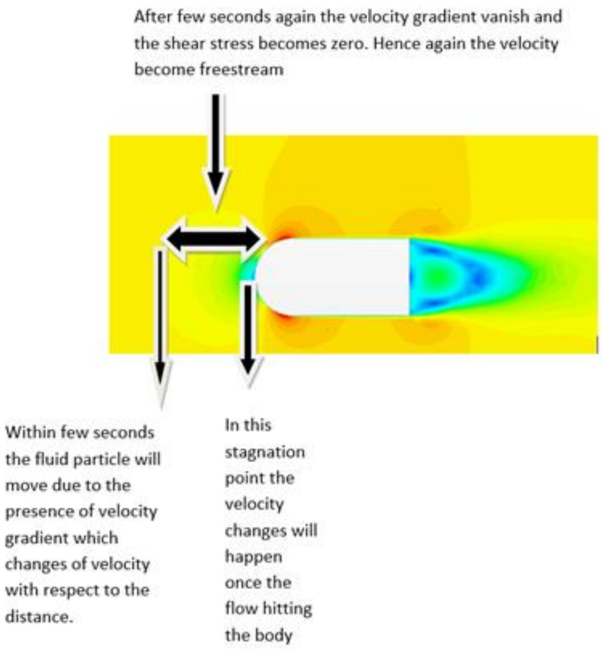

FIG 10: BOUNDARY LAYER THEOREM.

Within the scope of this study, both the pressure and the drag were examined in relation to the plots.

PRESSURE:

The ratio of the force that is delivered to an area on a per-unit basis is the definition of the pressure that is force.

PRESSURE CONTOUR:

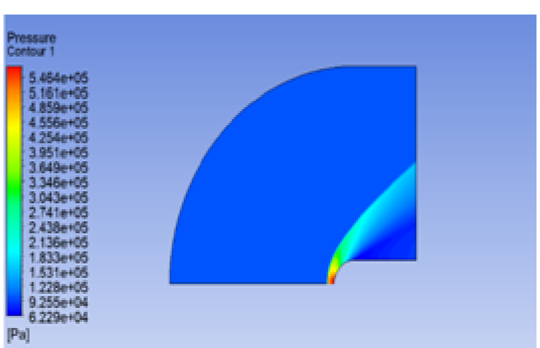

FIG 11: BLUNT BODY

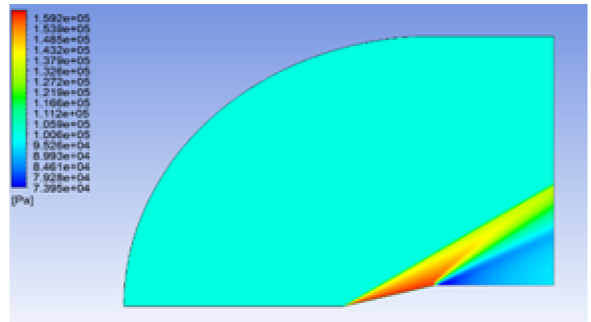

FIG 12: CONIC BODY

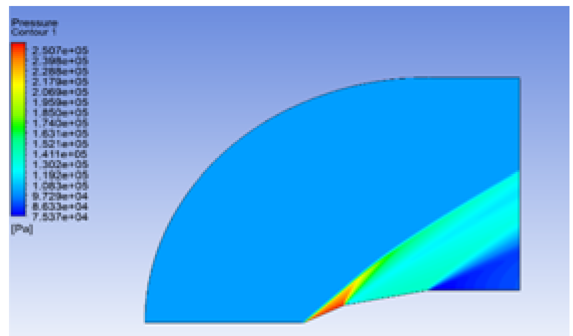

FIG 13: BI CONIC BODY

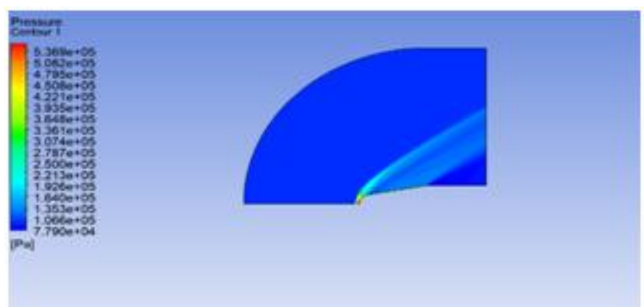

FIG 14: BLUNTED CONIC BODY

Images showing pressure contours reveal varying values in the legend chart as a result of velocity and shape change.

LIFT:

A force will be applied to the surface of the object by the fluid that is flowing around it. A component of this force that operates in a perpendicular direction to the object is called lift. The lift can be measured in Newton and it's called by a formula,

$$L = 0.5\ \rho\ V^2 S C_L$$

SCIENTIFIC NOTATION:

1. L = LIFT.
2. ρ = DENSITY OF FLUIDS.
3. V = VELOCITY OF FLUIDS.

4. S = THE SURFACE AREA OF THE OBJECT.
5. C_L = COEFFICIENT OF LIFT.

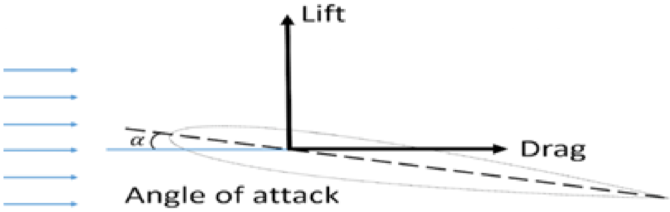

FIG 15: LIFT

According to the data of my research, the lift force is not taken into account since the angle of attack is zero and even though the item was evaluated while it was moving in a stationary position.

DRAG:

A force will be applied to the surface of the object by the fluid that is flowing around it. A component of this force that operates in a parallel direction to the object is called lift. The drag can be measured in Newton and it's called by a formula,

$$D = 0.5\ \rho\ V^2 S C_D$$

SCIENTIFIC NOTATION:

1. D= DRAG.
2. ρ = DENSITY OF FLUIDS.
3. V = VELOCITY OF FLUIDS.
4. S = THE SURFACE AREA OF THE OBJECT.
5. C_D = COEFFICIENT OF DRAG.

It is a well-known fact that an immovable object is not capable of producing positive lift because an approaching stream travelling at a supersonic velocity will attempt to hit the object by pushing it backward. However, in this project, the lift is not given much consideration because the nosecone analysis was done in zero angle of attack with the body in rest. Therefore, the only aspect of this measurement that utilizes simulation is the drag.

S.NO	TYPE	MACH	DRAG
1.	BLUNT BODY	2	40.802518
2.	BLUNT BODY	1.5	20.23814
3.	BLUNT BODY	1	4.7816711
4.	CONIC BODY	2	5.743457
5.	CONIC BODY	1.5	3.7663327
6.	CONIC BODY	1	1.7099029
7.	BLUNTED CONIC BODY	2	16.611415
8.	BLUNTED CONIC BODY	1.5	8.9462534
9.	BLUNTED CONIC BODY	1	2.7621543
10.	BICONIC BODY	2	19.993404
11.	BICONIC BODY	1.5	12.449439
12.	BICONIC BODY	1	3.9865948

Table 1: DRAG OF CONES WITH MODIFIED VELOCITIES.

In the table that can be seen above, the cones are analyzed with various Mach numbers in order to determine the drag at each velocity.

IV. CONCLUSION

To sum up this project's findings, we can say that the conic body achieves favorable results by exhibiting a lower total quantity of drag when compared with other forms; nevertheless, following the conic body, the blunted conic body nose cone may be considered because of its drag values.

APPENDIX:

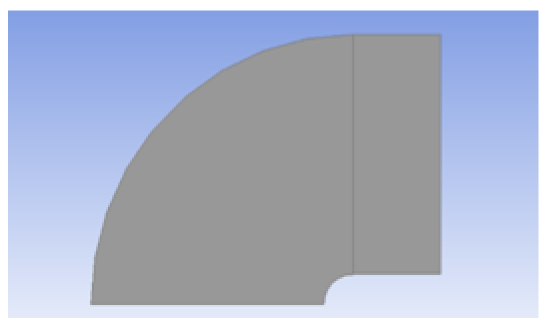

FIG 1.1: BLUNT BODY

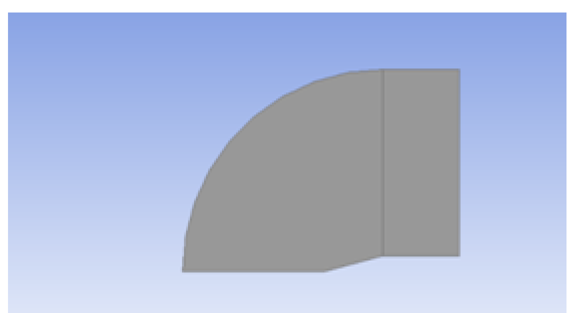

FIG1.2: CONIC BODY

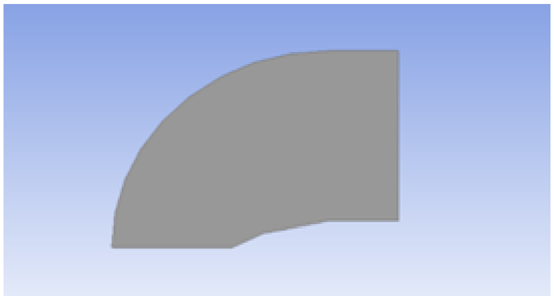

FIG: 1.3: BI CONIC BODY

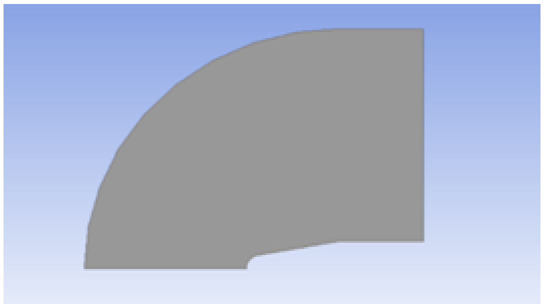

FIG 1.4: BLUNTED CONIC BODY

CALCULATION:

MACH NUMBER

$$MACH\ NUMBER = \frac{\text{SPEED OF THE OBJECT}}{\text{SPEED OF THE SOUND}}$$

SPEED OF THE OBJECT = 2 Mach = 686 m/s.
SPEED OF SOUND = 343m/s (approx.)
C = sq. Root (γ R T)
C = sq. Root (1.4 * 283* 300)
C = 344.7 m/s (exact)

REYNOLDS NUMBER

FORMULA:

$$Re = \frac{\rho u l}{\mu}$$

$\rho = 1.1766\ kg/m^3$
u = 686 m/s
l = characteristic length of pipe but here the object is blunt body so instead of l D is taken to show the diameter of the object.

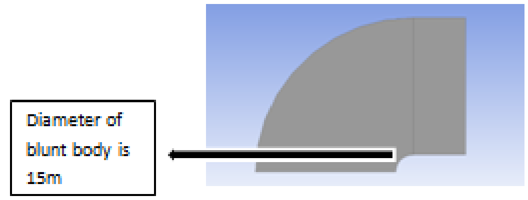

FIG 1.5: BLUNT BODY GEOMETRY

DRAG:

$D = 0.5\ \rho\ V^2 S C_D$

NOSECONE FOREBODY PRESSURE CHART:

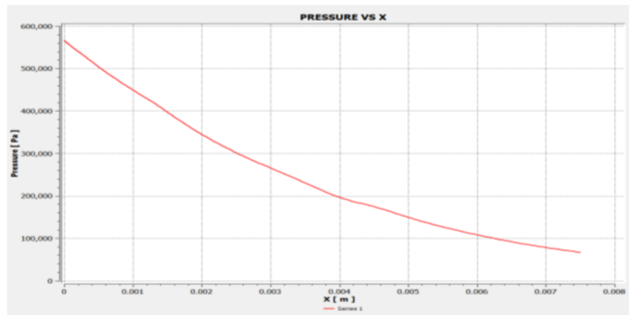

FIG: 1.6: BLUNT BODY

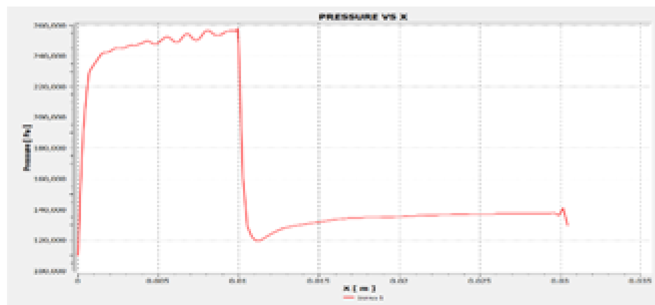

FIG 1.7: BI CONIC BODY

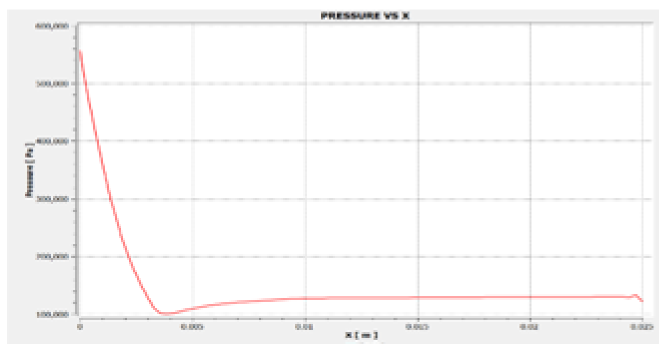

FIG: 1.7: BLUNTED CONIC BODY

FUTURISTIC STUDY

As I indicated in the conclusion, the conic body and blunted conic body exhibits higher decrease in drag. This may be evaluated using a 3D model as a future research to assess that with both drag and lift force to identify a model that has greater efficiency.

REFERENCES

[1] S. R. A. P. R. Spalart, "A one-equation turbulence model for aerodynamic flows," 30th Aerospace Sciences Meeting and Exhibit, Reno. NV, 1992.
[2] S. K. A. S. A. Fedaravičius, "Optimization of the rocket's nose and nozzle design parameters in respect to its aerodynamic characteristics," vol. 14, no. 4, 2012.
[3] G. S. S. S. J. A Sanjay Varma, "CFD analysis of various nose profiles," vol. 3, no. 3, 2016.
[4] D. Stripling, Ogive nose cones, 2012.
[5] N. K. M.Sreenivasul, "Design And structural analysis of missile nose cone," vol. 11, 2017.
[6] M. K. B. Sonia Chalia, "Mathematical modeling of ogive forebodies and nose cones," International Research Journal of Engineering and Technology, p. 4, 2016.
[7] Zhao, Jieliang, Shaoze Yan, Liren Deng, He Huang, and Yueming Liu. "Design and analysis of biomimetic nose cone for morphing of aerospace vehicle." *Journal of Bionic Engineering* 14, no. 2 (2017): 317-326.
[8] Carvalho, Lucas, and Geovanio Claudino. "CFD analysis of drag force for different nose cone design." In *presented at the IX Forum de Pesquisa e Inovacao do Centro de Lancamento da Barreira do Inferno*. 2019.