Computational fluid dynamics analysis on the Convergent Nozzle Design

Ankit Kumar Mishra [1], Amandeep Singh [2], Vinit Goswami[3], Priyanka Sarma[4], Vishlesh Bhimrao Meshram[5], JV Muruga Lal Jeyan [6*]

[1, 2,3,4,5]Undergraduate Student, [6]Research supervisor,
[1, 2, 3, 4, 5, 6] Department of Aerospace Engineering,
[1, 2, 3, 4, 5, 6] Lovely Professional University, Phagwara, India.

Abstract : Now the world is moving closer to the other planet. As we are searching for the new planets for living we had found that rockets are the only way capable to take the peoplefrom earth to the different planets, so scientist and engineers are focusing more on newrocket design which will be more efficient and more powerful in future, for example SpaceXcompany have rocket model SN8 which have ability to take the humans to the mars. Inrockets, the nozzle used for exhaust the gases at high speed which produced by thecombustion of propellants. Today we have different type of rocket nozzle like conical, bellor convergent divergent nozzles. In this paper we will discuss about the convergent anddivergent nozzle design with varying inlet and throat area or throat area ratio and will seethe variation in aerodynamic parameters and analyze all three designs on the ANSYSsoftware which is computational fluid dynamic software. In rockets, convergent divergentshape is mostly used as nozzle which is also known as de-laval nozzle.

Index Terms **- Pressure, Velocity, Dynamic Pressure, Mass flow rate, Drag.**

I. INTRODUCTION

A nozzle is a device designed to control the direction and speed of the fluid flow. The structure of nozzle is often a pipe or tube of varying cross sectional area which is used to direct or control the flow of a fluid, speed, mass, and the pressure of the stream that emerges from them. In a nozzle, the velocity of fluid increases at the expense of its pressure energy. Nozzles are generally categorized into three types- Convergent Nozzle, Divergent Nozzle, and Convergent – Divergent Nozzle [1]. Nozzle was invented by German engineer named Ernst Korting in the year 1878 and Swedish inventor named Gustave de Laval and thus named after him. Based on shape nozzle can be classified as-

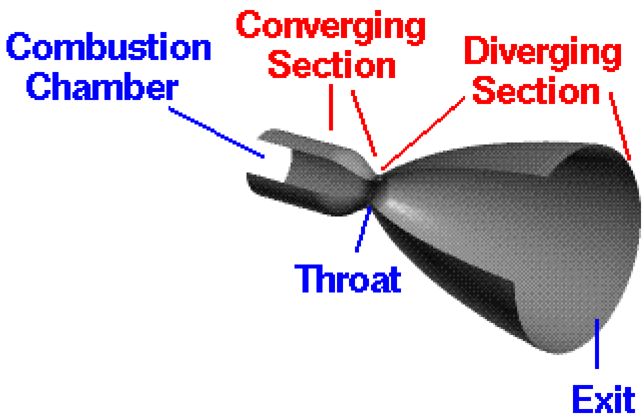
1. Bell Nozzle
2. Conical Nozzle

Convergent-Divergent Nozzle: - A convergent-divergent nozzle [13], also called De Laval nozzle, CD nozzle or con di nozzle, is a tube that is somewhat pinched in the middle so as to accelerate the flow of gas, which is passing through it.

The CD nozzle consists of three major sections: -

1. **The convergent section: -** where the subsonic gas flow will be accelerated thus reducing the pressure and increasing its kinetic energy.
2. **Throat section: -** where it supports the transonic flow.
3. **The divergent section: -** that accelerate the supersonic flow and eventually matching the exit pressure to the outside pressure.

Figure1: Rocket Nozzle Main Section



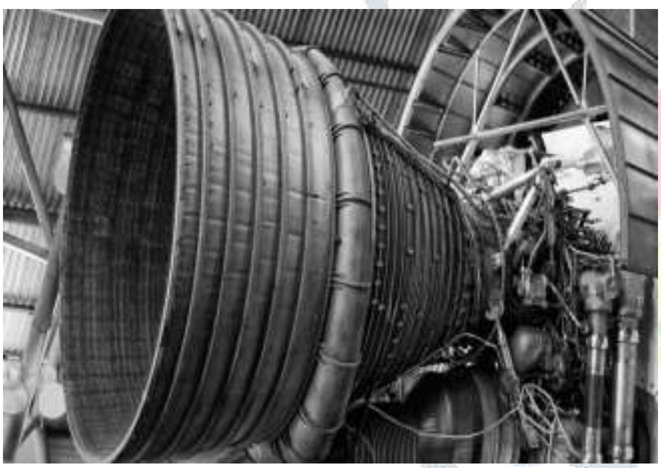
II.

This nozzle is designed for attaining speeds that are greater than speed of sound. The CD nozzle is commonly located at the end of the combustion chamber and it controls the gas expansion at the exhaust to efficiently convert the energy at the combustion into thrust. Its operation relies on the different properties of gases flowing at subsonic and supersonic speeds. The speed of a subsonic flow of gas will increase if the pipe carrying it narrows because the mass flow rate is constant [6]. The gas flow through a de Laval nozzle is isentropic and adiabatic. At subsonic flow the gas is compressible; a small pressure wave will propagate through it. Near the nozzle "throat", where the cross sectional area is a minimum, the gas velocity locally becomes transonic (Mach number = 1.0), a condition called choked flow. As the nozzle cross sectional area increases the gas continues to expand and the gas flow increases to supersonic velocities where a sound wave will not propagate backwards through the gas as viewed in the rest frame of the nozzle (Mach no > 1.0).

Bell Nozzle: - In a bell nozzle combustion gases flow through a throat and then the expansion away from the centerline is contained by the diverging walls of the nozzle up to the exit plane. Bell nozzle has a high angle expansion section (20 to 50 degrees) right behind the nozzle throat; this is followed by a gradual reversal of nozzle contour slope so that at the nozzle exit the divergence angle is small, usually less than a 10 degree half angle. Bells nozzles are a point design with optimum performance at one specific ambient pressure (i.e., altitude). Careful design is needed to achieve desired high altitude performance while avoiding flow separation at the walls of the nozzle near the exit when operating at low altitudes (launch), which can lead to loss of performance and possible structural failure of the nozzle due to dynamic loads. Therefore a compromise altitude must be used for the design point of a bell nozzle [4].

Conical Nozzle: - The conical nozzle was used often in early rocket applications because of its simplicity and ease of construction. The cone gets its name from the fact that the walls diverge at a constant angle. A small angle produces greater thrust, because it maximizes the axial component of exit velocity and produces a high specific impulse. The penalty, however, is a longer and heavier nozzle that is more complex to build. At the other extreme, size and weight are minimized by a large nozzle wall angle. Unfortunately, large angles reduce performance at low altitude because the high ambient pressure causes overexpansion and flow separation [1].

Figure2: Real View Rocket Nozzle



III. PROBLEM STATEMENT

The objective of this paper is to show the variation of aerodynamic parameter with varying the throat area ratio and show the variation on aerodynamic characteristic with different Mach number.

IV. RESEARCH METHODOLOGY

In this paper the rocket nozzle designed by using Creo parametric software. The first step is to create a 2D shape and revolve with respect to the central axis, Last step, convert into 3D model for CFD testing [3]. The commonly used tools in designing to create a model in Creo 4.0 parametric are- Extrude, extrude cut, Revolve, revolve cut Sweep, Swept cut, Fillet, Chamfer, Mirror. CFD Analysis is carried out in three steps i.e.

(i) Pre-processing, geometry, – Designing, meshing, boundary conditions and numerical method.

(ii) Processing – Solving fluid flow governing equations by numerical method till the convergence is reached.

(iii) Post processing – extracting results in terms of graphs, contours which explains the physics of flow and required results. The above three steps are carried out in ANSYS using fluid fluent CFD for designing and meshing with Hybrid grid that is prismatic layer around missile design and unstructured grid. Simulations are carried out using ANSYS fluent a finite volume solver at with inlet conditions. In this analysis we use the automatic mesh generation method because of the complexity of structure [4].

Figure3: Methodology

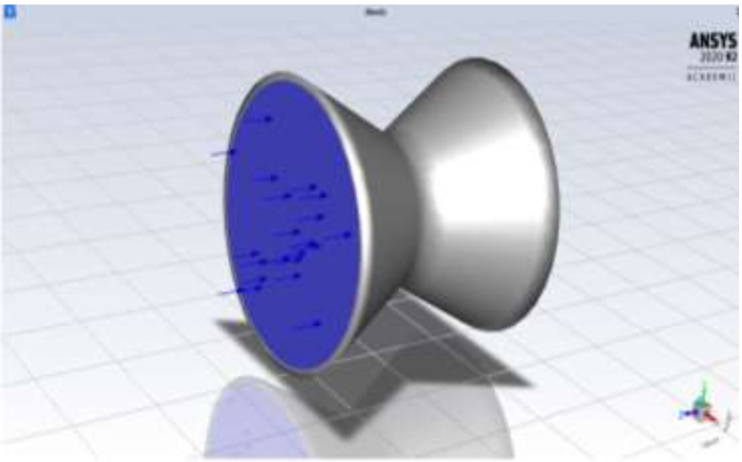
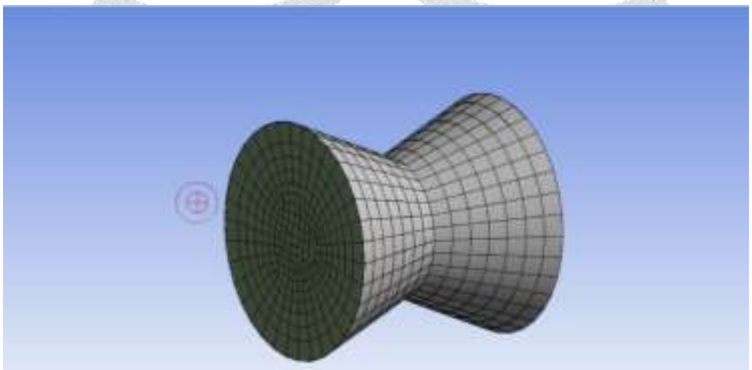
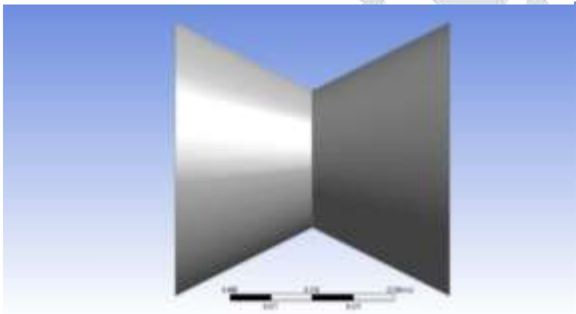


Figure4: Meshing



V. 3-D Nozzle Design

Figure5: inlet: 0.5m, throat: 0.29mFigure6: inlet:0.25m, throat:0.135m



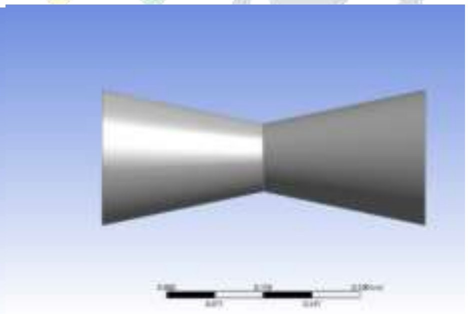
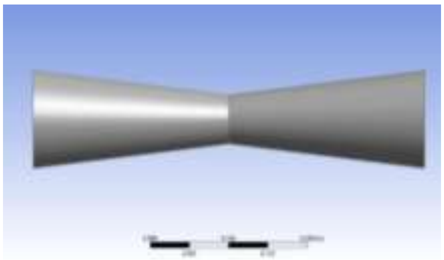


Figure7: inlet:0.125m, throat:0.0625m



IV. INLET AND BOUNDARY CONDITION

Table 4.1: Inlet and boundary condition

SL.NO	Parameter	Value
1	Flow Medium	Air
2	Mach Number	0.3,1.0,1.2,2.0
3	Density	1.225Kg/m^3
4	Turbulent Model	k-omega
5	Kinematic Viscosity	1.7894e-05kgs/m^2
6	Altitude condition	Standard sea level

VI. RESULTS AND DISCUSSION

Table 4.2: Variation of Inlet, throat area with drag

Mach number	Inlet Area(meter)	Throat Area(meter)	Nozzle Area (meter)	Throat area ratio	Mass flow rate	Drag
0.3	0.5	0.29	0.5	0.58	24.39	103.5
0.3	0.25	0.135	0.25	0.54	6.02	5.61
0.3	0.125	0.0625	0.125	0.50	1.47	0.0601

Table 4.3: Variation of Inlet, throat area with drag

Mach number	Inlet Area(meter)	Throat Area(meter)	Nozzle Area (meter)	Throat area ratio	Mass flow rate	Drag
0.3	0.5	0.29	0.5	0.58	24.39	103.5
0.3	0.25	0.135	0.25	0.54	6.02	5.61
0.3	0.125	0.0625	0.125	0.50	1.47	0.0601

Table 4.4: Variation of Inlet, throat area with drag

Mach number	Inlet Area(meter)	Throat Area(meter)	Nozzle Area (meter)	Throat area ratio	Mass flow rate	Drag
0.9	0.5	0.29	0.5	0.58	73.19	900.5
0.9	0.25	0.135	0.25	0.54	18.08	43.4
0.9	0.125	0.0625	0.125	0.50	4.41	1.01

PRESSURE DISTRIBUTION

a. 1st Design Result

Figure8: Mach 0.3

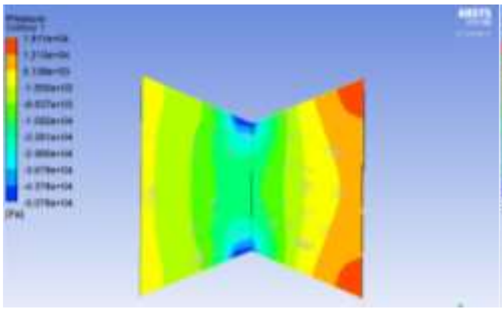


Figure9: Mach 0.6

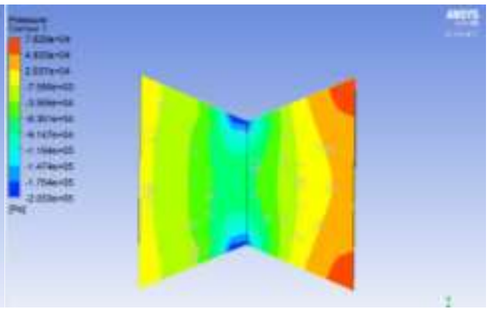
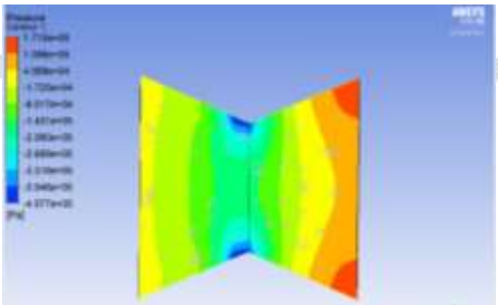


Figure10: Mach 0.9



b. 2nd Design Result

Figure11: Mach 0.3

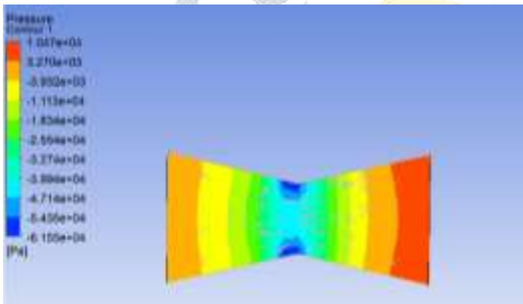


Figure12: Mach 0.6

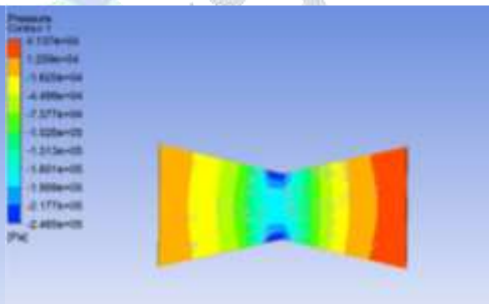
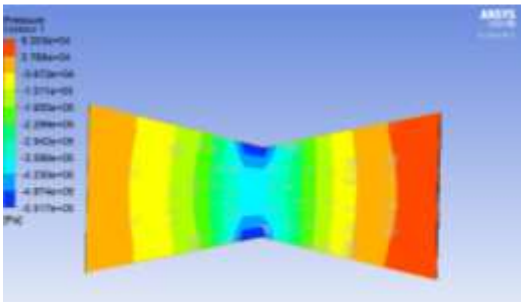


Figure13: Mach 0.9



c. 3rd Design Result

Figure14: Mach 0.3

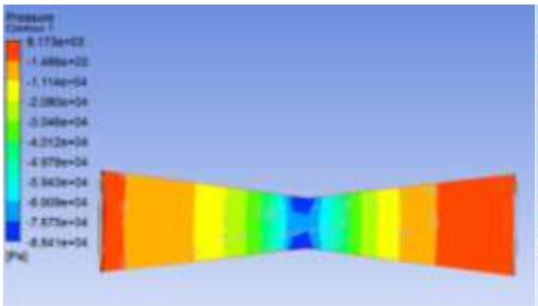


Figure15: Mach 0.6

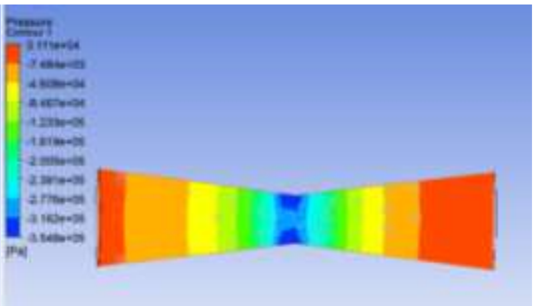
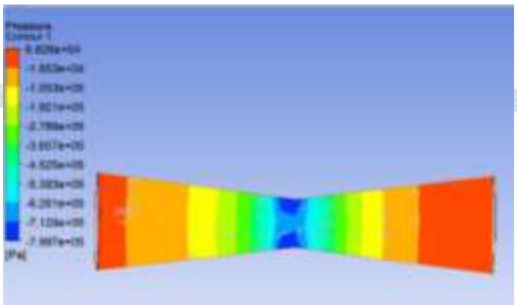


Figure16: Mach 0.9



VELOCITY DISTRIBUTION

a. 1st Design Result

Figure17: Mach 0.3

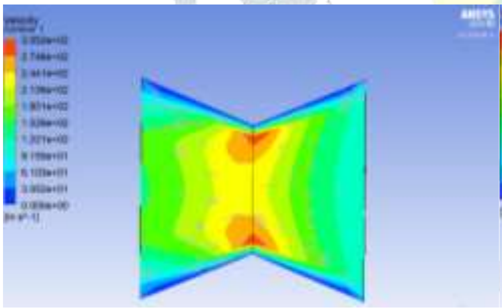


Figure18: Mach 0.6

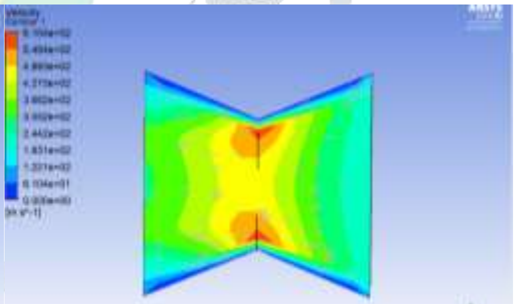
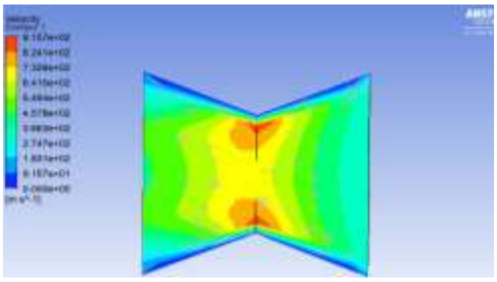


Figure19: Mach 0.9



b. 2nd Design Result

Figure20: Mach 0.3

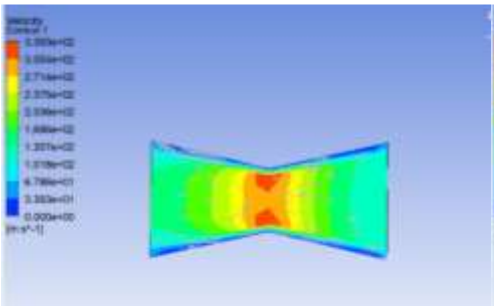


Figure21: Mach 0.6

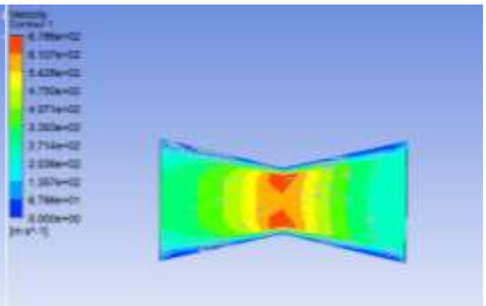
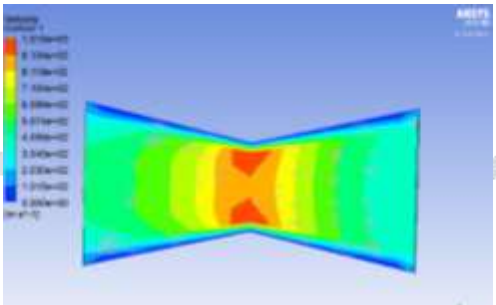


Figure22: Mach 0.9



c. 3rd Design Result

Figure23: Mach 0.3

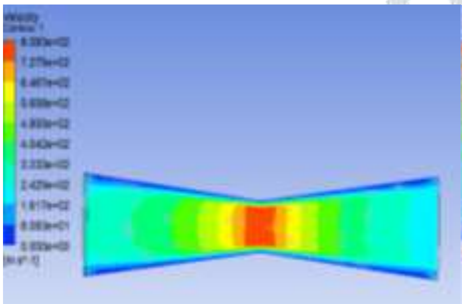


Figure24: Mach 0.6

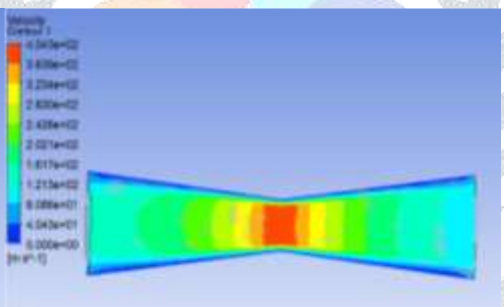
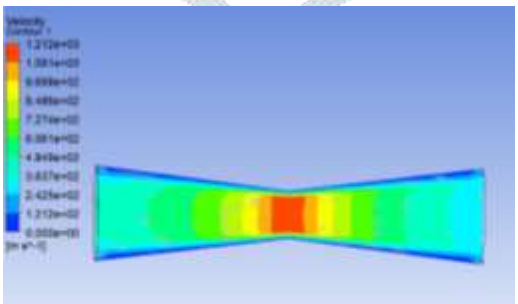


Figure25: Mach 0.9



DYNAMIC PRESSURE

a. 1st Design Result

Figure26: Mach 0.3

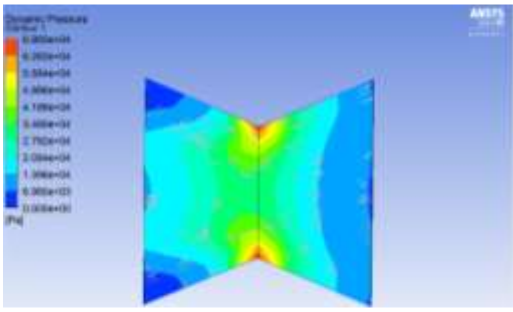


Figure27: Mach 0.6

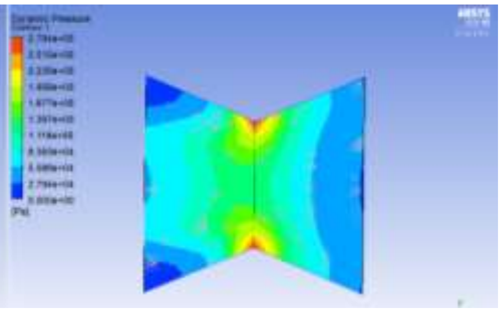
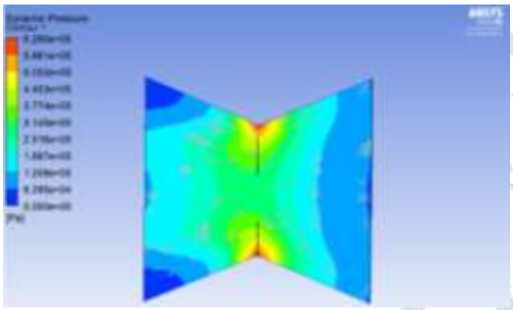


Figure28: Mach 0.3



a. 2nd Design Result

Figure29: Mach 0.3

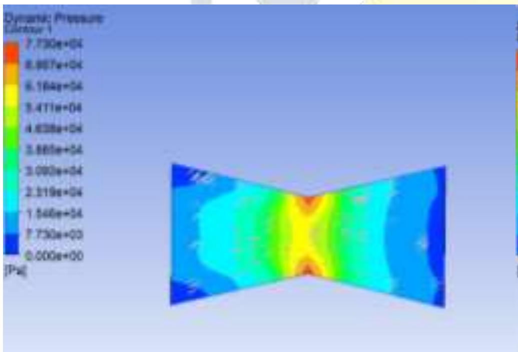


Figure30: Mach 0.6

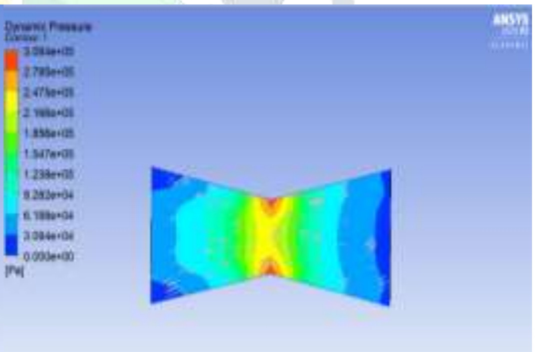
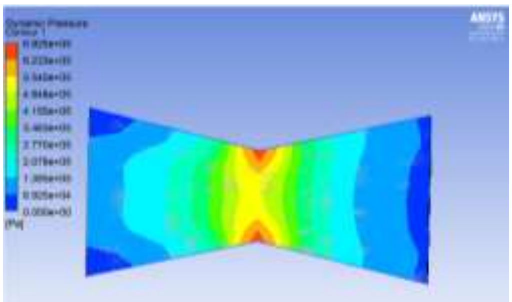


Figure31: Mach 0.9



b. 3rd Design Result

Figure32: Mach 0.3

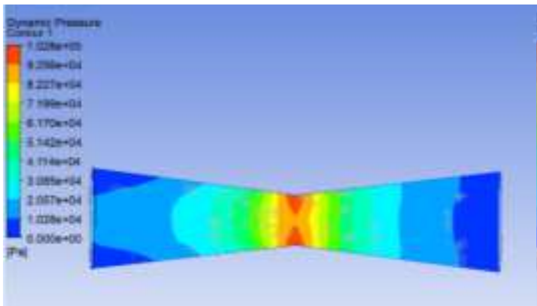


Figure33: Mach 0.6

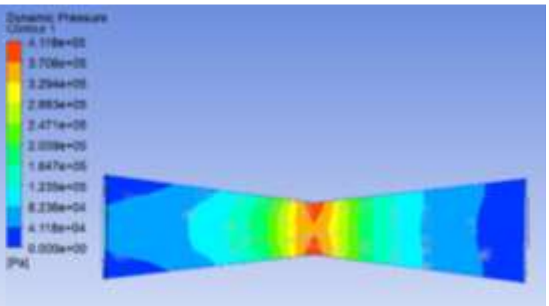
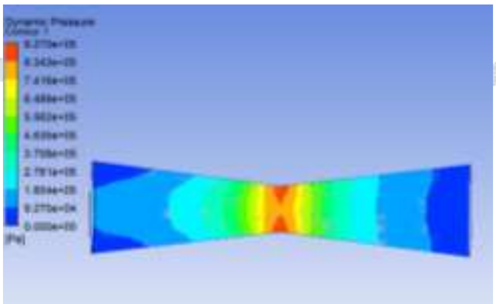


Figure34: Mach 0.9



VI.) CONCLUSION

From results, we conclude that with decreasing the throat area ratio mass flow rate decrease but velocity at throat increase and we analyses the all three design and we see that with decrease in throat area ratio, effect of pressure area increase, dynamic pressure increase and we are easily approaching the supersonic speed . But for heavy rocket we want more mass flow rate because by the thrust equation we see that thrust is depend on the mass low rate so our Last conclusion of this paper is that, first rocket nozzle design is good for heavy weight rocket's because in first design the mass flow rate is greater than our two design as compare.

VII.) REFERENCES

[1] Theodore Stevens and Henry M. Hobart (1906). Steam Turbine Engineering. MacMillan Company. pp. 24– 27.

[2] Robert M. Neilson (1903). The Steam Turbine. Longmans, Green, and Company. pp. 102–103.

[3] Doty, John H. . 1991. "Performance Prediction and Design of Maximum Thrust Planar Supersonic Nozzles Using a Flux-Difference-Splitting Technique", PHD Dissertation. Purdue University.

[4] Doty, John H., Thompson, H. Doyle, and Hoffman, Joe D. 1989."Optimum Thrust Two-Dimensional NASP Nozzle Study", NASP CR1069.

[5] Doty, John H., Thompson, H. Doyle, and Hoffman, Joe D. 1989."Optimum Thrust Airframe Integrated Scramjet Nozzles", Paper #83, Seventh National Aerospace Plane Technology Symposium.

[6]Tu, J., Yeoh, G.H., Liu, C.: "Computational Fluid Dynamics: a Practical Approach", 1st edition, Butterworth-Heinemman, 2008.

[7]Garrett Scaife (2000). From Galaxies to Turbines: Science, Technology, and the Parsons Family. Taylor & Francis Group.p. 197.

[8]Arjun Kundu, Devyanshu Prasad, Sarfraj Ahmed. 2016. "Effect of Exit Diameter on the Performance of Converging-Diverging Annular Nozzle using CFD", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 5, Issue 6.

[9] Emilio E.Paladino Clovis R.Maliska (2011) "Computational modelling of bubbly flows in differential pressure flow meters Flow Measurement and Instrumentation", 22, 309-318.

[10]GuttiRajeswaraRao ,U.S.Ramakanth, A.Lakshman (2013) Flow Analysis in a Convergent-Divergent Nozzle Using CFD (2013) 2347-5188.

[11]R. Sabari Vihar J. V. Muruga Lal Jeyan, K. Sai Priyanka, "A Review on Aerodynamic Parameters, Methodologies and Suppression Techniques Explored in Aircraft Wing Flutter", IJAST, vol. 29, no. 04, pp.
3494 -, Jun. 2020

[12] Balaji R, J V Muruga Lal Jeyan, Vijay Kumar singh "Review on influence of radiating and aerodynamic shock at hypersonic vehicle" Journal of Physics: Conference Series, volume/ issue 1473 march 2020 012004, IOP Publishing

[13] JV Muruga Lal Jeyan, Kavya S Nair and Krishna S Nair "Aerodynamics and flow pattern performance evaluation of off road vehicle for various velocity range and angle of incidence" Journal of Physics: Conference Series, volume/ issue 1473 march 2020 012001, IOP Publishing

[14] J V Muruga lal Jeyan,Krishna S nair , Kavya S nair "The Low Speed Aerodynamic Analysis Of Segmental
Wing Profile "International Journal of Mechanical and Production Engineering Research and Development. Vol. 9, Issue 4, Aug 2019, 1303–1310,1 August 2019,