RESEARCH ARTICLE OPEN ACCESS

Investigation on Divergent Exit Curvature Effect on Nozzle Pressure Ratio of Supersonic Convergent Divergent Nozzle

Shyamshankar.M.B*, Sankar.V**
*(Department of Aeronautical Engineering, Anna University, Chennai)
** (Department of Aeronautical Engineering, Anna University, Chennai)

ABSTRACT
The objective of this project work is to computationally analyze shock waves in the Convergent Divergent (CD) Nozzle. The commercial CFD code Fluent is employed to analyze the compressible flow through the nozzle. The analysis is about NPR (Nozzle Pressure Ratio) i.e., the ratio between exit pressure of the nozzle to ambient pressure. The various models of CD Nozzle are designed and the results are compared. The flow characteristic of shockwave for various design of CD Nozzle is also discussed. The purpose of this project is to investigate supersonic C-D nozzle flow for increasing NPR (Nozzle pressure ratio) through CFD. The imperfect matching between the pressures and ambient pressure and exit pressure leads to the formation of a complicated shock wave structure. Supersonic nozzle flow separation occurs in CD nozzles at NPR values far above their design value that results in shock formation inside the nozzle.
The one-dimensional analysis approximations are not accurate, in reality the flow detaches from the wall and forms a separation region, subsequently the flow downstream becomes non-uniform and unstable. Shock wave affects flow performance of nozzle from NPR value 1.63 for existing geometrical conditions of nozzle. Problem of using this nozzle above 1.63NPR is shock wave at downstream of throat. After shock wave, static pressure increases further downstream of flow. It leads to flow separation and back pressure effects. Back pressure makes nozzle chocked. To investigate this problem, geometry of divergent portion is introduced and analysed through CFD. This is expected in resulting of reduction of flow separation and back pressure effect as well as increase in nozzle working NPR.
***Keywords* -** circular cut-out, elliptical cut-outs, failure load, flat composite panel, mesh size.

I. INTRODUCTION

Convergent nozzles accelerate subsonic fluids. If the nozzle pressure ratio is high enough, then the flow will reach sonic velocity at the narrowest point (i.e. the nozzle throat). In this situation, the nozzle is said to be choked. Increasing the nozzle pressure ratio further will not increase the throat Mach number above one. Downstream (i.e. external to the nozzle) the flow is free to expand to supersonic velocities; however Mach 1 can be a very high speed for a hot gas because the speed of sound varies as the square root of absolute temperature. This fact is used extensively in rocketry where hypersonic flows are required and where propellant mixtures are deliberately chosen to further increase the sonic speed.

A nozzle is often a pipe or tube of varying cross sectional area and it can be used to direct or modify the flow of a fluid (liquid or gas). Nozzles are frequently used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that emerges from them. In nozzle, velocity of fluid increases on the expense of its pressure energy.

A nozzle is a device designed to control the direction or characteristics of a fluid flow (especially to increase velocity) as it exits (or enters) an enclosed chamber or pipe. A gas jet, fluid jet, or hydro jet is a nozzle intended to eject gas or fluid in a coherent stream into a surrounding medium. Gas jets are commonly found in gas stoves, ovens, or barbecues. Gas jets were commonly used for light before the development of electric light. Other types of fluid jets are found in carburettors, where smooth calibrated orifices are used to regulate the flow of fuel into an engine, and in spas. Another specialized jet is the laminar jet. This is a water jet that contains devices to smooth out the pressure and flow, and gives laminar flow, as its name suggests. This gives better results for fountains.

Nozzles used for feeding hot blast into a blast furnace or forge are called tubers. Jet nozzles are also use in large rooms where the distribution of air via ceiling diffusers is not possible or not practical. Diffusers that use jet nozzles are called jet diffuser where it will be arranged in the side wall areas in order to distribute air. When the temperature difference between the supply air and the room air changes, the supply air stream is deflected upwards to supply warm air or deflected downwards to supply cold air.

Frequently, the goal is to increase the kinetic energy of the flowing medium at the expense of its pressure and internal energy. Nozzles can be described as convergent (narrowing down from a wide diameter to a smaller diameter in the direction of the flow) or divergent (expanding from a smaller diameter to a larger one). A de Laval nozzle has a convergent section followed by a divergent section and is often called a convergent-divergent nozzle.

II. LITERATURE SURVEY

Blom Martina has given numerical investigation of 3d compressible duct flow a generic supersonic bump in a duct has been studied. The shock wave over the bump is accelerated, inducing a shock wave in the downstream, interacting with the wall boundary layer and triggering boundary layer separation.

Gustavo Bono et. al presented an adaptive mesh strategy based on nodal re-allocation is presented in this work. This technique is applied to problems involving compressible flows with strong shocks waves, improving the accuracy and efficiency of the numerical solution. The initial mesh is continuously adapted during the solution process keeping, as much as possible, mesh smoothness and local orthogonality using an unconstrained nonlinear optimization method. The adaptive procedure, which is coupled to an edge-based error estimate aiming to equi-distribute the error over the cell edges is the main contribution of this work. The flow is simulated using the Finite Element Method (FEM) with an explicit one-step Taylor- Galerkin scheme, in which an Arbitrary Lagrangean-Eulerian (ALE) description is employed to take into account mesh movement. Finally, to demonstrate the capabilities of the adaptive process, several examples of compressible in-viscid flows are presented.

III. DESIGN OF CD-NOZZLE

Step 1: Inlet Pressure – 17 bar

Exit Pressure – 1.01325 bar (It is nothing but pressure of medium where fluid is going to deliver)

Inlet Temperature – 473K

Mass Flow - 32 Kg/s

Fluid - Air

Step 2: $P^* / P1 = (2/\gamma + 1)\ \gamma/(\gamma-1)$

γ - 1.4 is polytrophic index for air

P^* - Throat Pressure – 9.23 bar

Step 3: $T^* / T1 = (2/\gamma + 1)$

T^* - Throat temperature - 393.78 K

Step 4: Density (throat) = $P^* / R\,T^*$

= 8.08 Kg / m^3

Step 5: V^* Throat Velocity = $(\gamma R T^*)^{1/2}$

= 396.65 m/s

Step 6: A^* Throat Area = mass flow rate / throat density * V^*

= 0.00997 m^2

D^* throat diameter = 113 mm

Step 7: T_1 / T_2 (Exit Temperature) = $(P_1 / P_2$ (exit pressure)$)^{(\gamma-1)/\gamma}$

T_2 = 210 K

Step 8: Density$_2$ = $P_2 / R\,T_2$

= 1.7 Kg / m^3

Step 9: $V_2 = 44.72 * (Cp\ [T_1 - T_2])$

= 726 m/s

Step 10: A_2 = mass flow rate / Density$_2$ * V_2

= 0.049 m2

D_2 = 183.3 mm

Step 11: Length of divergent nozzle = $(D_2 – D^*) / 2 * \tan \alpha_d$

= 105.57 mm

α_d – Divergent angle = 18°

Step 12: Length of convergent nozzle = (d1- d*)/ 2 * tan αc

= 57.94 mm

α_c – Convergent angle = 13°

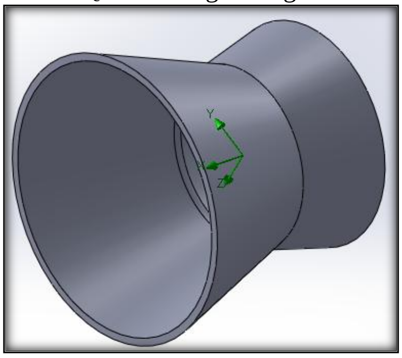


Fig. 1 Original Nozzle

Modified nozzle dimensions are presented below. Convergence portion diameter at end is varied by percentage of outlet diameter.

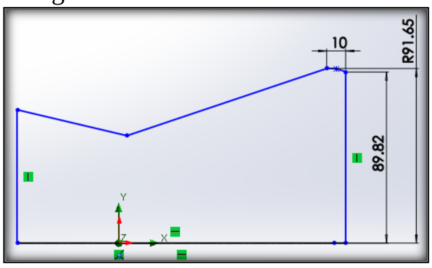


Fig. 2 Modified Nozzle

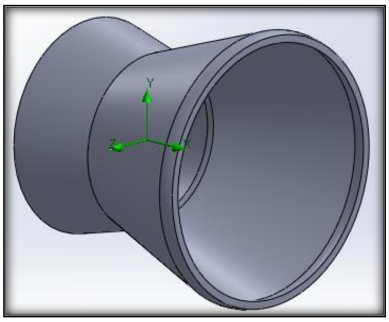


Fig. 3 2% Outer diameter reduction Nozzle

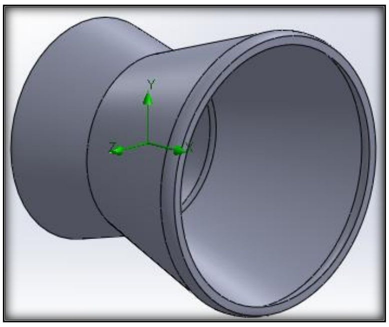


Fig. 4 3% Outer diameter reduction Nozzle

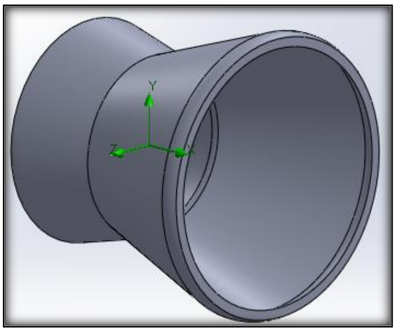


Fig. 5 4% Outer diameter reduction Nozzle

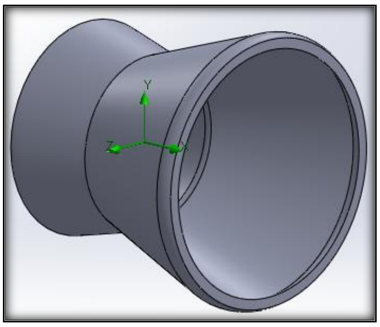


Fig.6 5% Outer diameter reduction Nozzle

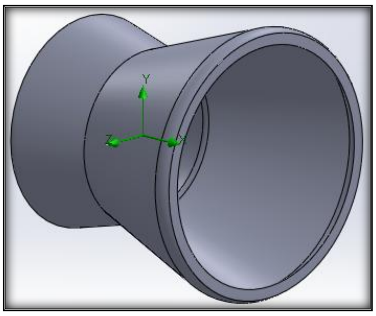


Fig. 7 6% Outer diameter reduction Nozzle

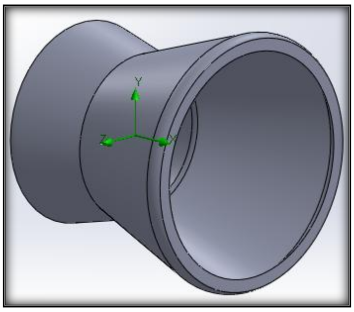


Fig. 8 7.5% Outer diameter reduction Nozzle

Tab.no.1. Dimensions of Modified Nozzles

OD Reduction (%)	Dimensions in mm
2	89.82
3	88.90
4	87.98
5	87.07
6	86.15
7.5	84.77

IV. SIMULATION DETAILS

Analysis Type – Internal Flow Analysis
Solver – COSMOS FLOW WORKS

Meshing

- ✓ Fluid Cells - 8010
- ✓ Solid Cells - 4135
- ✓ Partial Cells – 9311

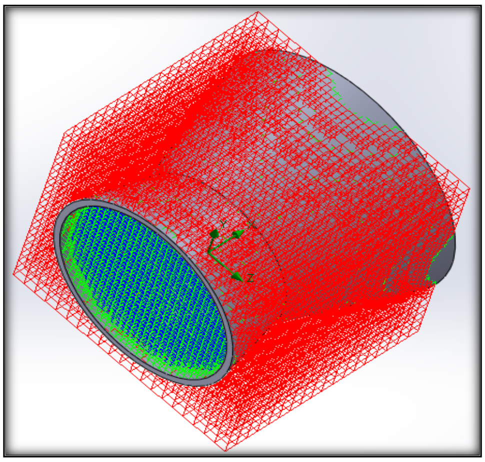


Fig.9 Meshing

4.3 Simulation Results

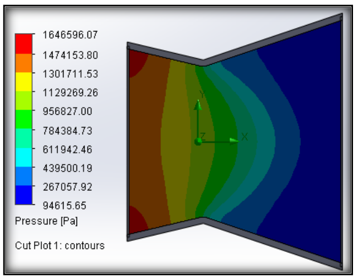


Fig. 10 Inlet – 1700000 Pa and Outlet – 15000 Pa

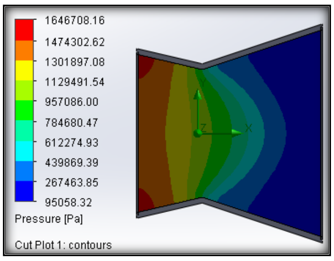


Fig. 11 Inlet – 1700000 Pa and Outlet – 20000 Pa

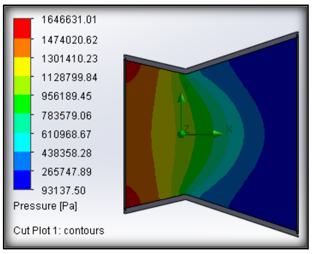


Fig. 12 Inlet – 1700000 Pa and Outlet – 30000 Pa

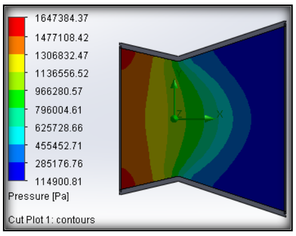


Fig. 13 Inlet – 1700000 Pa and Outlet – 40000 Pa

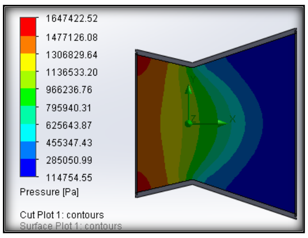


Fig. 14 Inlet – 1700000 Pa and Outlet – 50000 Pa

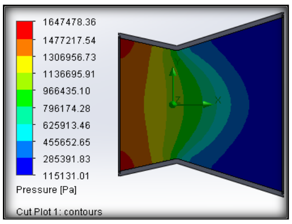


Fig. 15 Inlet – 1700000 Pa and Outlet – 60000 Pa

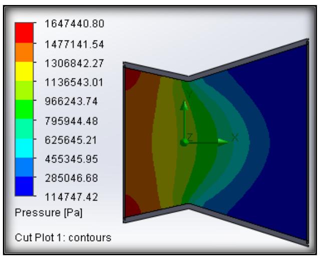


Fig. 16 Inlet – 1700000 Pa and Outlet – 70000 Pa

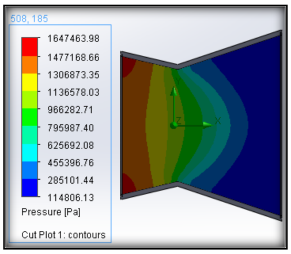


Fig. 17 Inlet – 1700000 Pa and Outlet – 80000 Pa

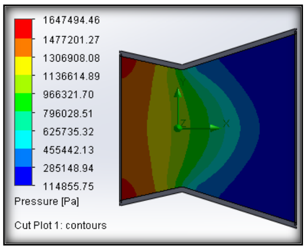


Fig. 18 Inlet – 1700000 Pa and Outlet – 90000 Pa

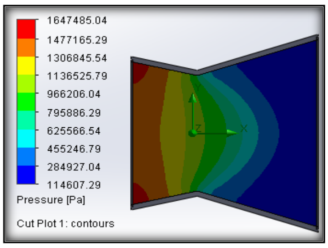


Fig. 19 Inlet – 1700000 Pa and Outlet – 101325 Pa

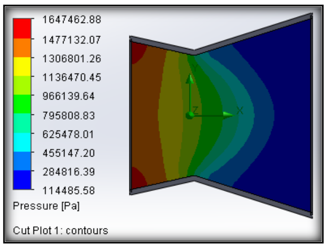


Fig. 20 Inlet – 1700000 Pa and Outlet – 110325 Pa

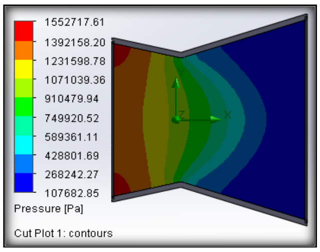


Fig. 21 Inlet – 1600000 Pa and Outlet – 101325 Pa

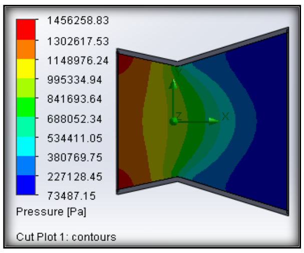


Fig. 22 Inlet – 1500000 Pa and Outlet – 101325 Pa

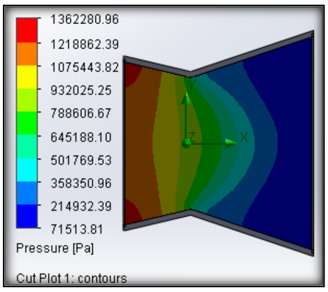


Fig. 23 Inlet – 1400000 Pa and Outlet – 101325 Pa

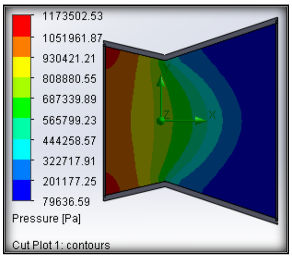


Fig. 24 Inlet – 1200000 Pa and Outlet – 101325 Pa

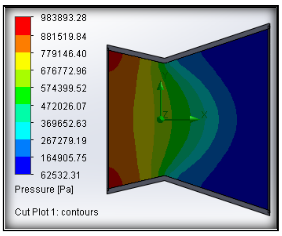


Fig. 25 Inlet – 1000000 Pa and Outlet – 101325 Pa

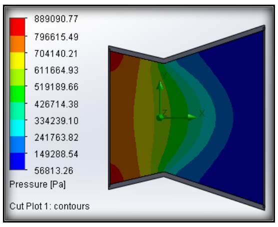


Fig. 26 Inlet – 900000 Pa and Outlet – 101325 Pa

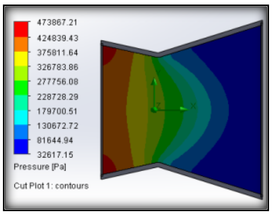


Fig. 27 Inlet – 500000 Pa and Outlet – 101325 Pa

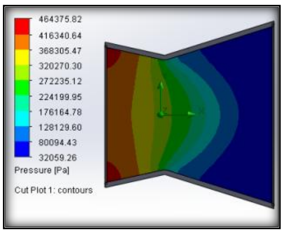


Fig. 28 Inlet – 490000 Pa and Outlet – 101325 Pa

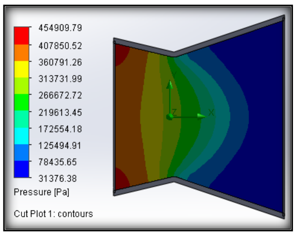


Fig. 29 Inlet – 480000 Pa and Outlet – 101325 Pa

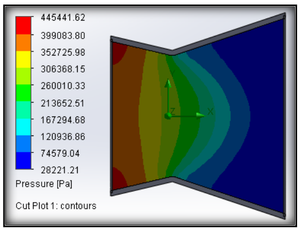


Fig. 30 Inlet – 470000 Pa and Outlet – 101325 Pa

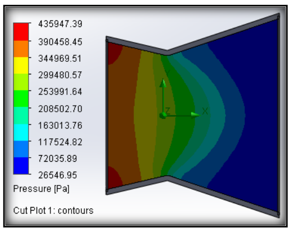


Fig.31 Inlet – 460000 Pa and Outlet – 101325 Pa

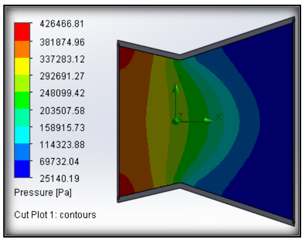


Fig. 32 Inlet – 450000 Pa and Outlet – 101325 Pa

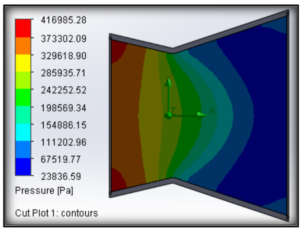


Fig. 33 Inlet – 440000 Pa and Outlet – 101325 Pa

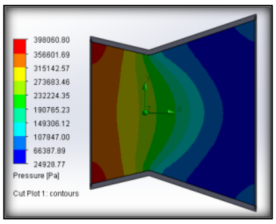


Fig. 34 Inlet – 420000 Pa and Outlet – 101325 Pa

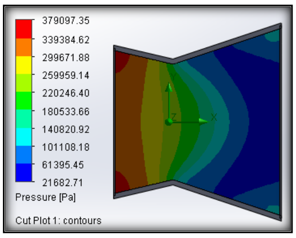


Fig. 35 Inlet – 400000 Pa and Outlet – 101325 Pa

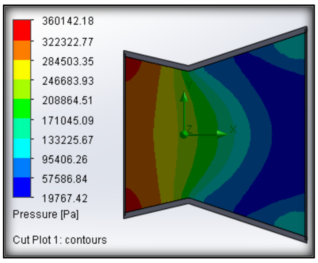


Fig. 36 Inlet – 380000 Pa and Outlet – 101325 Pa

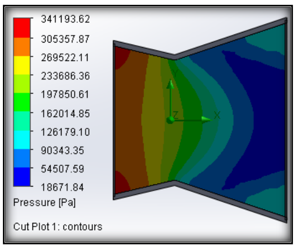


Fig. 37 Inlet – 360000 Pa and Outlet – 101325 Pa

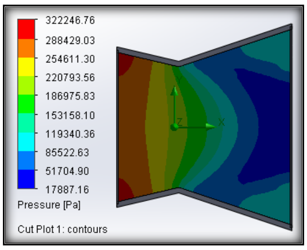


Fig. 38 Inlet – 340000 Pa and Outlet – 101325 Pa

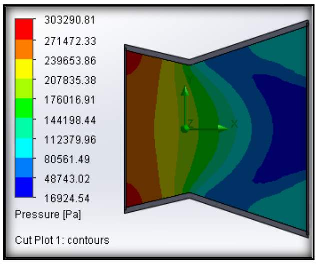


Fig. 39 Inlet – 320000 Pa and Outlet – 101325 Pa

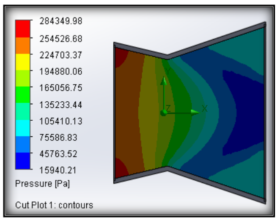


Fig. 40 Inlet – 300000 Pa and Outlet – 101325 Pa

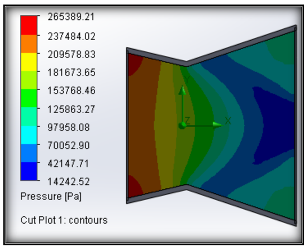


Fig. 41 Inlet – 280000 Pa and Outlet – 101325 Pa

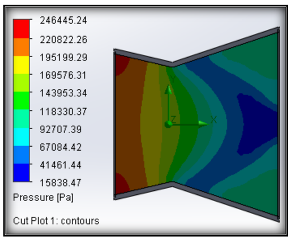


Fig. 42 Inlet – 260000 Pa and Outlet – 101325 Pa

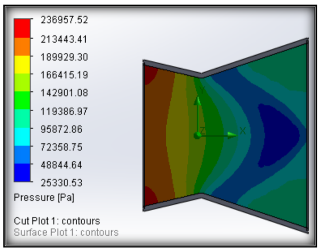


Fig. 43 Inlet – 250000 Pa and Outlet – 101325 Pa

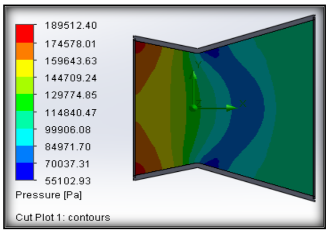


Fig. 44 Inlet – 220000 Pa and Outlet – 101325 Pa

Tab.no.2 Boundaries and Results

Sl.No.	Inlet Pressure (Pa)	Outlet Pressure (Pa)	Ambient Pressure (Pa)	NPR	Mach
1	1700000	152496.37	15000	10.16642467	2.1
2	1700000	152814.38	20000	7.640719	2.1
3	1700000	153489.05	30000	5.116301667	2.1
4	1700000	146265.29	40000	3.65663225	2.16
65	1700000	145583.31	50000	2.9116662	2.16
6	1700000	145249.63	60000	2.420827167	2.16
7	1700000	144649.9	70000	2.066427143	2.17
8	1700000	144347.11	80000	1.804338875	2.17
9	1700000	144145.78	90000	1.601619778	2.17
10	1700000	143955.1	101325	1.420726376	2.18
11	1700000	143758.71	110325	1.303047451	2.18
12	1700000	143945.37	101325	1.420630348	2.18
13	1600000	135671.46	101325	1.338973205	2.18
14	1500000	141061.75	101325	1.392171231	2.04
15	1400000	130620.57	101325	1.289124796	2.06
16	1200000	104498.46	101325	1.031319615	2.16
17	1000000	86076.01	101325	0.84950417	2.18
18	900000	76980.08	101325	0.75973432	2.19
19	500000	41298.43	101325	0.407583814	2.18
20	490000	40504.11	101325	0.399744486	2.18
21	480000	39795.06	101325	0.392746706	2.18
22	470000	43323.61	101325	0.427570787	2.1
23	460000	45012.31	101325	0.44423696	2.05
24	450000	46033.57	101325	0.454316013	2.02

V. RESULT AND CONCLUSION

Designed nozzle is simulated with different inlet and outlet pressures. In CFD, outlet pressure is nothing but pressure of medium where fluid will be delivered, therefore it is ambient pressure. Pressure at nozzle outlet plane is measured after simulation for calculating nozzle pressure ratio. By varying inlet and outlet pressures, wide range of NPR values has been achieved. Below NPR value of 0.454, shockwave comes nearer to nozzle outlet plane and attached to it. Further reduction in NPR leads to movement of shockwave to throat and chocking. Initially, inlet pressure is kept constant outlet pressure is varied. Then, outlet pressure is kept 1.01325 bar and inlet is varied from 17bar to 2.2 bar.Below 4.5 bar of inlet pressure to 1.01325 bar of outlet pressure, shockwave comes at outlet plane of nozzle and comes to throat below 2.2 bar. Nozzle is important component at end in propulsion to get thrust. It is to produce enough mach to achieve desire flight velocity. CD-nozzles are encountered with shockwave problems. It is current researches to reduce shockwave effects at some NPR values to get better efficiency of propulsion system and nozzle. In our nozzle, at inlet pressure of 2.2 bar to 1.01325 bar of outlet pressure, it leads to Mach number of 0.48. Like that, after 4.7 bar to 2.2 bar, Mach number is reduced at outlet due to shockwave. Therefore, we have simulated modified nozzles with boundary conditions of 2.2 bar of inlet pressure to 1.01325bar of outlet pressure. From 17bar to 4.7 bar, Mach number at outlet is within 2.1 to 2.19. This is achievable Mach by nozzle which designed in design chapter. After 4.7bar inlet pressure, it is reduced due to shockwave effects.

Mach is an important parameter for desired flight velocity and CD-nozzle performance, therefore original and modified nozzles are compared by Mach at outlet for 2.2 bar inlet pressure to 1.01325 bar ambient pressure because after 2.2 bar nozzle is chocked. Outer diameter is reduced and a convergence portion is implemented in modified nozzles to reduce shockwave effects. From below results, it is concluded that nozzle end curvature influences slightly on the performance of nozzle at nearby chocking condition. Nozzle will be fixed homogenous geometry, therefore this end curvature effects are required to be analyzed for design point and some other off-design conditions. This is future concern of this project.

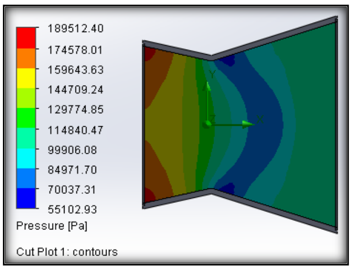


Fig.45 Pressure Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (Original Nozzle)

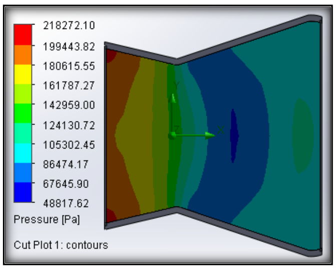


Fig. 46 Pressure Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (2% OD reduction)

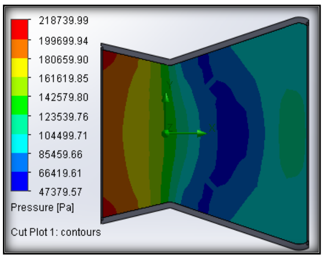


Fig. 47 Pressure Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (3% OD reduction)

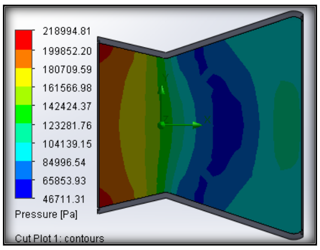


Fig. 48 Pressure Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (4% OD reduction)

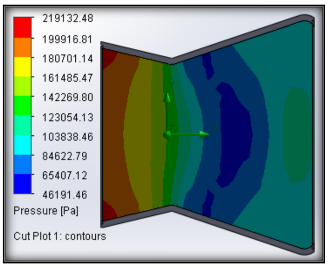


Fig. 49 Pressure Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (5% OD reduction)

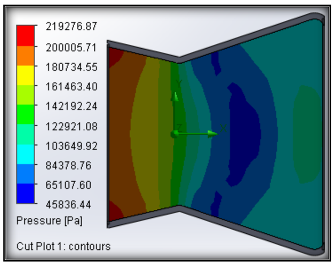


Fig. 50 Pressure Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (6% OD reduction)

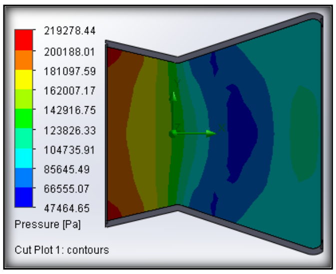


Fig. 51 Pressure Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (7.5% OD reduction)

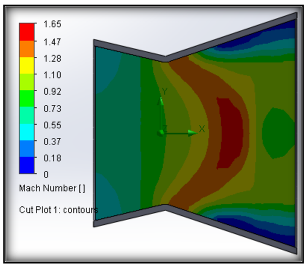


Fig. 52 Pressure Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (Original Nozzle)

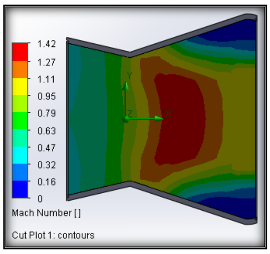


Fig 53 Mach Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (2% OD reduction)

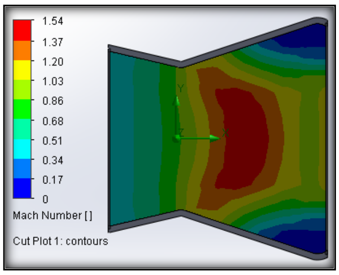


Fig. 54 Mach Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (3% OD reduction)

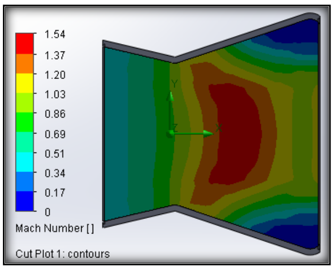


Fig. 55 Mach Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (4% OD reduction)

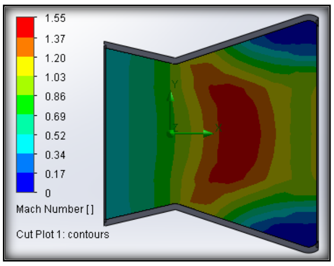


Fig. 56 Mach Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (5% OD reduction)

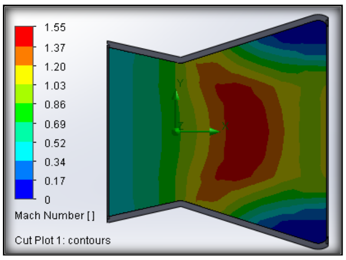


Fig. 57 Mach Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (6% OD reduction)

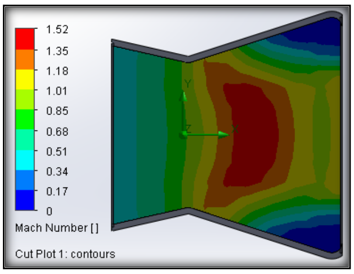
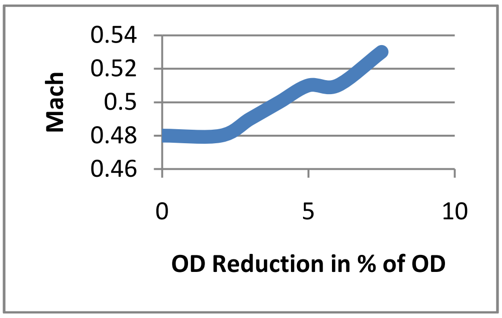


Fig. 58 Mach Plot for Inlet – 220000 Pa and Outlet – 101325 Pa (7.5% OD reduction)

Tab.no.3 Nozzle End Geometry Vs Mach number 2.2bar inlet to 1.01325bar outlet

Type	Mach Number at Outlet Plane
Original Nozzle	0.48
2% OD reduction	0.48
3% OD reduction	0.49
4% OD reduction	0.5
5% OD reduction	0.51
6% OD reduction	0.51
7.5% OD reduction	0.53



Tab.no.4 Nozzle End Geometry Vs Mach number 2.2bar inlet to 1.01325bar outlet

REFERENCES

[1]. Blom M. (2010) 'Numerical Investigation of 3D Compressible Duct Flow', Chalmers University of Technology, Goteborg, Sweden
[2]. Bono G. et. al (2008) 'An Adaptive Mesh Strategy for High Compressible Flows Based on Nodal Re-Allocation', J. of the Braz. Soc. of Mech. Sci. & Eng, No. 3 / 189.