Optimization of Convergent-Divergent Nozzle Design for Enhanced Thrust Generation and Efficiency

[1] Kunal Yele , [2]Arjun Khannade , [3]Preet Gupta ,[4] Yash Vaghmare , [5]Lokesh Aurangabadkar

[1,2,3,4]Students, Mechanical Engineering Department, [5] Assistant Professor, Mechanical Engineering Department
[1]Indore Institute of Science and Technology, Indore, India

Abstract: **Convergent-divergent (CD) nozzles are crucial components in propulsion systems, particularly in aerospace engineering, where they play a significant role in enhancing thrust generation and efficiency of rocket engines and jet propulsion systems. This study focuses on the optimization of CD nozzle design parameters to maximize thrust performance while increasing operational efficiency. Computational fluid dynamics (CFD) simulations and optimization algorithms are employed to explore the complex fluid dynamics within the CD nozzle geometry. The optimization process considers various parameters such as nozzle geometry to achieve desired performance metrics, including thrust and exhaust velocity. Additionally, considerations are made for minimizing shock formation and losses**.

IndexTerms -: **CD Nozzle, CFD, Ansys**

1. INTRODUCTION

A nozzle is a device that increases the velocity of a fluid at the expense of pressure. Nozzle is a part of rocket which is used for the expansion of combustion gases through it and produces thrust. Nozzle is a passage used to transform pressure energy into kinetic energy. During the combustion of fuel, chemical energy is converted into thermal energy and pressure energy. The combustion gases at this stage are at a high pressure and temperature and these gases under such high pressure expand through the nozzle during which the pressure energy is converted into kinetic energy which in turn moves the vehicle in a direction opposite to that the exhaust gases, according to Newton's third law of motion. Two primary functions of nozzle are - First, they must control the engine back pressure to provide the correct and optimum engine performance, which is done by jet area variations. Second, they must efficiently convert potential energy of the exhaust gas to kinetic energy by increasing the exit velocity, which is done by efficiently expanding the exhaust gases to the atmospheric pressure.

Computational Fluid Dynamics (CFD) Simulations utilize CFD simulations using software such as ANSYS to model fluid flow through the nozzle and evaluate performance metrics. The successful optimization of the convergent-divergent nozzle design will result in improved propulsion system efficiency, enhanced thrust generation capabilities, and minimized losses.

The objective of this project is to optimize the design of a convergent-divergent nozzle to enhance its performance in terms of thrust generation, flow velocity, Mach number, and pressure distribution. By systematically varying geometric parameters and conducting optimization analyses, the goal is to identify the most efficient nozzle configuration that achieves the desired performance metrics.

This optimized design has the potential to find applications in various engineering fields, including aerospace propulsion, fluid dynamics research, and industrial processes. Optimizing convergent-divergent nozzle designs is crucial for improving the efficiency and performance of propulsion systems, fluid flow control mechanisms, and related engineering applications. By addressing this engineering challenge, the project contributes to advancements in aerospace engineering, fluid dynamics research, and industrial innovation.

2. LITERATURE SURVEY

1) Arjun Kundu, Devyanshu Prasad and Sarfraj Ahmed worked on the topic of "Effect of Exit Diameter on the Performance of Converging-Diverging Annular Nozzle Using CFD" and there findings are - The result obtained after the CFD analysis shows that smaller exit diameter gives greater mach number compared to the larger diameter for the same inlet and boundary conditions.

2) Mohan Kumar G, Dominic Xavier Fernando and R. Muthu Kumar worked on the topic of "Design and Optimization of De Laval Nozzle to Prevent Shock Induced Flow Separation" and there findings are - For maximum thrust and efficiency without flow separation due to induced shock, the direction of flow of stream through nozzle should be axial. Venkatesh V, C Jaya pal Reddy.

3) Venkatesh V, C Jaya pal Reddy [4] worked on the topic of "Modelling and Simulation of Supersonic nozzle using Computational Fluid Dynamics" and their findings are - Contour nozzle gives a greater mach number at exit compared to conical nozzle because contour nozzle gives maximum expansion ratio.

4) Srinivas M, Suresh A L, G. Harish worked on the topic of "Thrust Enhancement of a Convergent-Divergent Nozzle by Using CFD" and their findings are - Exit velocity and Mach number was increased with increase in divergent angle. Based on the maximum exit velocity obtained, that nozzle geometry is optimized. For still more increase in the exit velocity, radius at the throat has been done on the optimized model and analysis was been carried out. Exit velocity was increased to certain extent and this was taken as final optimized model. Therefore, the final optimized nozzle will reduce the fuel consumption and shock's formation, increases the thrust and exit velocity to a maximum level.

5) Uttam Kumar, Sudhir Singh Rajput, Dr. Praveen Borkar, worked on the topic " CFD analysis and parameter optimization of Divergent Convergent Nozzle" in which the result obtained by the fluent is validated by the experimental result of Mr. Olivera P. Kostić, the result obtained by analysis of DC Nozzle of convergent angle 40° and divergent angle 20° only, after the examination of pressure, Mach number and temperature distribution, its concluded that it shows very good agreement with the experimental result of Mach no with respect to the nozzle axial length

6) Olivera P. Kostić, worked on " CFD Modeling of Supersonic Airflow Generated by 2D Nozzle with and Without an Obstacle at the Exit Section". In which their aim is to establish a reliable calculation methodology, which could be used for the generation and optimizing of new obstacle types and shapes for thrust vectoring, without the need to perform additional, quite expensive and time consuming wind tunnel tests.

7) Pranab Pal, Shailendra Kumar Bohidar, worked on "ANALYSIS AND PARAMETER OPTIMIZATION OF CONVERGENT DIVERGENT NOZZLE BY USING CFD APPROACH" in which The result obtained by the fluent is validated by the experimental result of Mr. Olivera P. Kostic. The phase I consist of result obtained by analysis of DC Nozzle of convergent angle 40° and divergent angle 20° only, after the examination of pressure, mach no.and temperature distribution its concluded that its shows very good agreement with the experimental result of mach no with respect to the nozzle axial length

8) P. Parthiban, M. Robert, T. Ambikapathi, worked on "Design and analysis of rocket engine nozzle by using CFD and optimization of nozzle parameters. "their conclusion was By the proper geometrical design of the nozzle, the exhaust of the propellant gases will be regulated in such a way that maximum effective rocket velocity can be reached. Convergent divergent nozzle is the most commonly used nozzle since in using it the propellant can be heated in combustion chamber.

9) P. Padmanathan, Dr. S. Vaidyanathan, worked on " Computational Analysis of shockwave in convergent divergent nozzle", The objective of this paper is to computationally analyse shock waves in the Convergent Divergent (CD) Nozzle. The commercial CFD code Fluent is employed to analyse the compressible flow through the nozzle. The analysis includes static pressure, temperature, Mach number and density of the flow for different nozzle pressure ratios (NPR i.e., the ratio between exit pressures of the nozzle to ambient pressure). The results are compared with the analytical results of quasi-one dimensional equation. The flow characteristic before and after the shock is discussed.

10) M.H.M Noh, A.H.A Hamid worked on " Numerical investigation of chocked converging diverging nozzles for thruster application" in which seven choked converging-diverging nozzle flow characteristics operating at Reynolds number of 46000 are numerically investigated using a commercial software FLUENT 6.1.22.Computational work is carried out using axisymmetric flow Spalartllmaras model, two dimensional, axisymmetric, coupled solver and linearized using implicit scheme. The nozzles, which are intended to be used for thruster's application, have a throat diameter of 11 mm and an area ratio of 4. Numerical result also reveals that the flow separation phenomenon occurs inside all nozzles, where normal shock forms inside nozzles with divergence angle of 20°, 14° and 10°, and this shock degrades the performance of the nozzle in term of the thrust produced.

11) Johnson, A. and Papamoschou, D., worked on "shock wave motion on the instability of the plume exiting an over expanded, convergent-divergent nozzle." This study evaluates the role of shock wave motion on the instability of the plume exiting an over-expanded, convergent-divergent nozzle. An array of wall pressure transducers was used to track the position of the shock in time, and a Pitot probe was used to obtain simultaneous measurement of the total pressure fluctuations at various points in the jet that emerges from the separation shock. Analysis of the shock motion revealed that the shock wave becomes more unstable as it becomes stronger, as evidenced by an increase in the range of motion and in the frequency of large-scale oscillations. For strong shocks, there is a substantial correlation between shock motion and total pressure fluctuation in the plume. Such correlation is absent for relatively weak shocks. The study indicates that shock motion affects the plume instability if the separation shock is very strong, and that other mechanisms govern the plume instability when the shock is relatively weak.

12) Qun, P. and Tongji, L., worked on "solution of subsonic and transonic flow field in two-dimensional delaval Nozzle with reassigned contraction ratio and expansion ratio and throat wall radius." In which they concluded that A supersonic expansion cannot be achieved within a convergent-divergent nozzle if the flow velocity does not reach that of the sound at the throat. The computation of the flow field characteristics at the throat is thus essential to the nozzle developed thrust value and therefore to the aircraft or rocket it propels. Several approaches were developed in order to describe the transonic expansion, which takes place through the throat of a De-Laval convergent-divergent nozzle. They all allow reaching good results but showing a major shortcoming represented by their inability to describe the transonic flow field for nozzles having a small throat radius. The approach initially developed by Kliegel & Levine uses the velocity series development in terms of the normalized throat radius added to unity instead of solely the normalized throat radius or the traditional small disturbances theory approach. The present investigation carries out the application of these three approaches for different throat radiuses of curvature. The method using the normalized throat radius added to unity shows better results when applied to geometries integrating small throat radiuses.

3. PROBLEM DEFINITION

The performance of a convergent-divergent nozzle is influenced by its geometric parameters, including inlet and outlet dimensions, as well as flow conditions such as pressure, velocity, and Mach number. However, achieving an optimal nozzle design that balances these parameters to maximize thrust while minimizing losses presents a complex engineering challenge.

The objective of this project is to optimize the design of a convergent-divergent nozzle to enhance its performance in terms of thrust generation, flow velocity, Mach number, and pressure distribution. By systematically varying geometric parameters and conducting optimization analyses, the goal is to identify the most efficient nozzle configuration that achieves the desired performance metrics. Defined the geometric parameters of the convergent-divergent nozzle, including inlet and outlet dimensions such as half angle, length, and radius. Specified output parameters to assess nozzle performance, including force on walls, pressure, velocity, and Mach number at both the inlet and outlet sections. Conducted computational fluid dynamics (CFD) simulations using software such as ANSYS to analyze fluid flow through the nozzle and evaluate performance metrics. Utilized optimization techniques, such as direct optimization or genetic algorithms, to systematically vary geometric parameters and optimize nozzle performance. The successful optimization of the convergent-divergent nozzle design will result in enhanced thrust generation, improved flow characteristics, and minimized shock formation and losses.

4. METHODOLOGY

The proposed method begins with problem formulation, where the optimization objectives and constraints are defined. This is followed by the development of a high- fidelity CFD model of the CD nozzle geometry and the simulation of flow through the nozzle under various operating conditions. Design of Experiments (DoE) techniques are then employed to screen the design space and generate initial design points for optimization. Next, optimization algorithms are selected based on their scalability, convergence properties, and computational efficiency. These algorithms are applied to iteratively adjust CD nozzle design parameters to maximize thrust generation and efficiency while satisfying specified constraints. The optimized design is then validated using experimental data or benchmark cases to ensure accuracy and reliability.

In Summary :
- Modelling nozzle geometry and defining boundary conditions for analysis.
- Methodology focused on maximizing thrust and minimizing losses.
- Optimizing design parameters for enhanced nozzle performance.
- Approach tailored to achieve optimal results in nozzle design.
- Modified the design for enhanced output and efficiency.

4.1 CFD Model

In order to create the CFD model of CD nozzle first, we need to create the project schematic in ansys workbench. CFD fluent workbench used for the modelling and analysis of CD nozzle. First of all, we need to create the 2 D sketch of nozzle geometry, we used Ansys's Design Modeler to design the symmetric-half of the Nozzle Geometry and convert it to the 2D plain geometry to perform axisymmetric analysis and the results were mirrored for the other half-geometry.

	Name	Value	Type
⚡	angle_outlet	20 °	Angle
✓	length	200 mm	Length
✓	end_length	20 mm	Length
✓	throat_rad	20 mm	Length
✓	outlet_area	70 mm	Length
⚡	angle_inlet	40 °	Angle
✓	inlet_length	40 mm	Length
✓	outlet_length	50 mm	Length
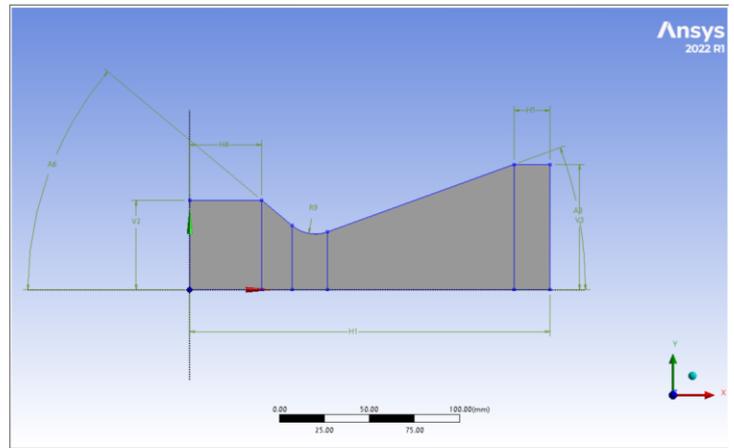
TABLE 1 and FIG 1: Geometry of CD Nozzle (symmetrical half)

4.2　Meshing of Geometry

In order to create the meshing of a geometry, we need to switch the workbench from geometry to meshing. After creating the geometry, it is required to divide the control volume into a smaller number of Nodes and element of finite size, therefore it is called a finite volume method. The method of splitting the Control volume into small finite size volume is known as a meshing of the control volume.

| Details of "Mesh"	
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Element Order	Linear
☐ Element Size	Default (10.595 mm)
Export Format	Standard
Export Preview Surface Mesh	No
Sizing	
Use Adaptive Sizing	No
☐ Growth Rate	Default (1.2)
Mesh Defeaturing	Yes
☐ Defeature Size	Default (5.2974e-002 mm)
Capture Curvature	Yes
☐ Curvature Min Size	Default (0.10595 mm)
☐ Curvature Normal Angle	Default (18.0°)
Capture Proximity	No
Bounding Box Diagonal	211.9 mm
Average Surface Area	2010.7 mm²
Minimum Edge Length	16.885 mm
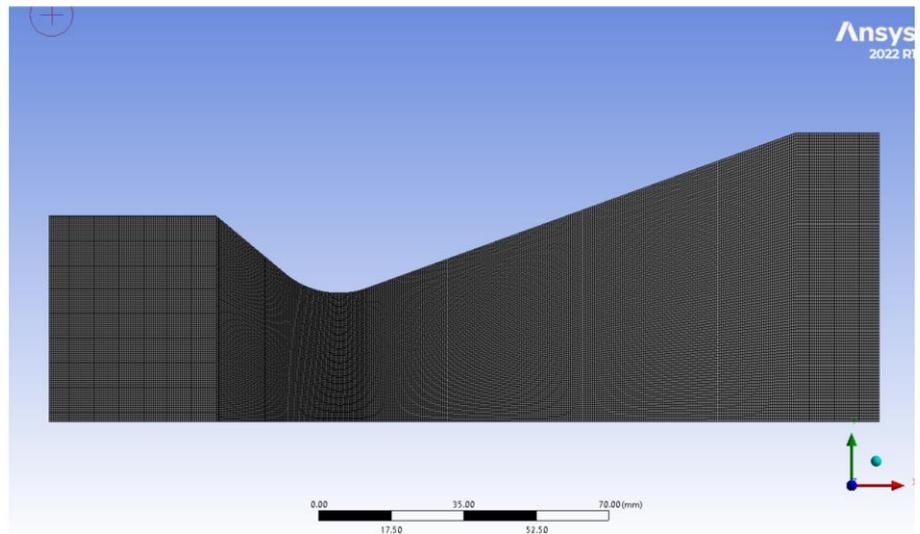
TABLE 2 and FIG 2 : Mesh Geometry of CD Nozzle (symmetrical half)

4.3　Boundary Conditions

The boundary condition provided to the CD Nozzle as per the actual working condition and scaled value given to the nozzle due to the limits of calculation and time. the boundary condition of the nozzle is given in the table below.

BOUNDARY	TYPE
INLET	Gauge total pressure 300000 (Pa), Temp. 300 k
OUTLET	Gauge Pressure
FLUID CONSIDERED	Ideal Gas
FLOW CONSIDERED	INVISID

TABLE 3: BOUNDARY CONDITIONS

5.   RESULTS AND DISSCUSSIONS

5.1   Initial Geometry Analysis

In the first phase we analysed and obtained the results for different parameters such as pressure, velocity and mach number distribution across the Initial Geometry. On the basis of these results and geometrical parameters we calculated the ideal thrust, actual thrust and nozzle efficiency.
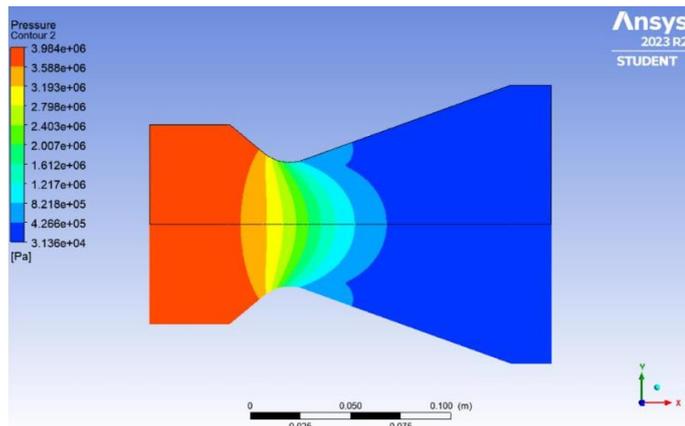

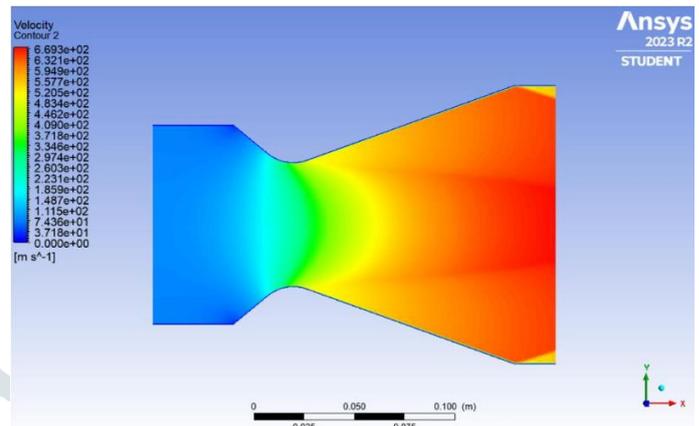**FIG 3 : Pressure Contour of Initial Geometry**

FIG 4 : Velocity Contour of Initial Geometry

Thrust force $(F_T)= \dot{m}(V_e - V_o)$

Here: $\dot{m}$= main flow rate of exhaust gas
V_e = velocity at exit of nozzle
V_o = velocity at inlet of nozzle
$\dot{m} = \rho_e A_e V_e$
ρ_e = density of exhaust gas
A_e = exist area of nozzle

For CASE I :

ρ_e = 0.0203387 kg/m3
R_i (inlet radius) = 50 mm R_o (outlet radius) = 70 mm
R_t (Throat radius) = 31.153 mm L (length of nozzle) = 200 mm
V_o = 83.4459 m/sec V_e = 672.63 m/sec
V_t (Throat velocity) = 122.3m/sec
$\dot{m} = \rho_e A_e V_e = 0.0203387 \times \pi \times (70)2 / 106 \times 672.63$
$\dot{m}$ = .21059 kg/sec
∴ Actual Thrust force (F_T) = 0.21059 [672.63 – 83.4459]
F_T = 124.07 N
Nozzle efficiency (nozzle) = (Actual thrust Force /Ideal thrust force)

(F thrust) ideal = $\dot{m}$ ideal $(V_e - V_o)$
$\dot{m}$ ideal = $\rho \times A_t \times V_t = \rho \times \pi (R_t)2 \times V_t$
= $1.225 \times \pi \times (31.53)2 / 106 \times 122.3$
= 0.4679kg/sec
∴(F thrust) ideal = (652.3 – 81.53) × 0.4679 (F thrust) ideal = 275.67N

∴Nozzle EFFICIENCY= 124.07/275.67
Nozzle EFFICIENCY= .4564 or **45.64%**

Now we performed our first optimization using DesignXplorer which uses genetic algorithm to find the optimal candidate points.

For that we

● Selected outlet half angle, length, and radius as input parameters

● Maximized force(thrust), pressure, velocity, and Mach no. at outlet.

● Used direct optimization method to find candidate points.

Table of Schematic B2: Optimization				
	A	**B**	**C**	**D**
1	Optimization St...			
2	Minimize P9	Goal, Minimize P9 (Default importance)		
3	Minimize P10	Goal, Minimize P10 (Default importance)		
4	Minimize P11	Goal, Minimize P11 (Default importance)		
5	Maximize P12	Goal, Maximize P12 (Default importance)		
6	Minimize P13	Goal, Minimize P13 (Default importance)		
7	Optimization Met...			
8	Adaptive Multiple-Objective	The Adaptive Multiple-Objective method is a variant of the popular NSGA-II (Non -dominated Sorted Genetic Algorithm-II) based on controlled elitism concepts. It supports multiple objectives and constraints and aims at finding the global optimum. It is limited to continuous and manufacturable input parameters.		
9	Configuration	Generate 25 samples initially, 25 samples per iteration and find 3 candidates in a maximum of 4 iterations.		
10	Status	Not Converged.		
11	Candidate Po...			
12		Candidate Point 1	Candidate Point 2	Candidate Point 3
13	P14 - angle_outlet (degree)	9.258	9.1	5.683
14	P15 - end_length (mm)	108.25	117.63	141.53
15	P16 - throat_rad (mm)	23.634	23.591	23.969
16	P9 - massflowrate-op (kg s^-1)	-16.165	-16.106	-14.363
17	P10 - velocityexit-op (N)	-9539.2	-9605.3	-9613.1
18	P11 - velocityexitmach-op (kg s^-1)	-42.559	-43.48	-55.157
19	P12 - pressureinlet-op (kg^2 m^-1 s^-3)	6.3612E+07	6.3384E+07	5.6637E+07
20	P13 - thrust-op (N)	-1288	-1299.9	-518.89

TABLE 4: First Optimization Result

Based on the above result we can deduce that the Candidate Point 2 is the best choice. So we will modify the geometry accordingly.

5.2   Primary Modified Geometry Analysis

In the second phase we analysed and obtained the results for different parameters such as pressure, velocity and mach number distribution across the Primary Modified Geometry. On the basis of these results and geometrical parameters we calculated the ideal thrust, actual thrust and nozzle efficiency.
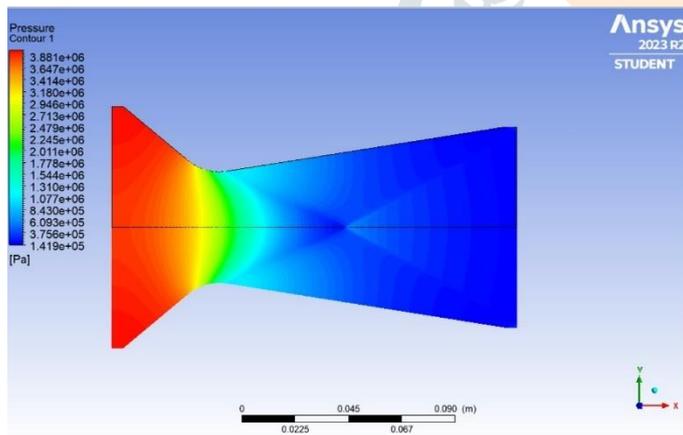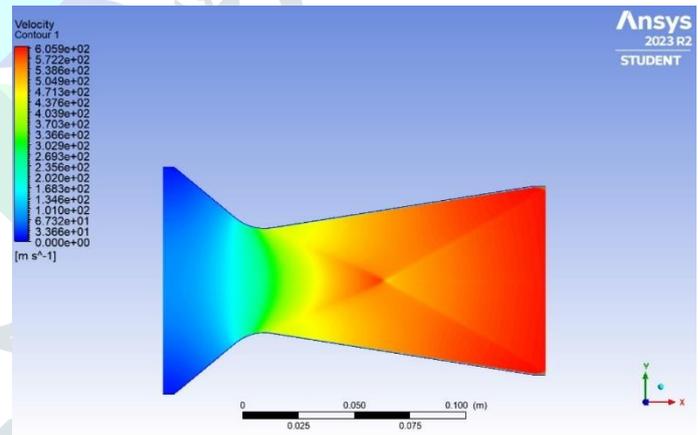

FIG 5 : Pressure Contour of Primary Modified Geometry

FIG 6 : Velocity Contour of Primary Modified Geometry

For CASE II :

Ve = 655.6 m/sec

Vo = 78.41 m/sec Vt = 115 m/sec L= 171.23 mm

Ri = 51.323 mm   , Ro = 62.85 mm , Rt = 23.591 mm

$\dot{m}$=   ρe Ae Ve = 0.0203387 × **π** × (62.85)^2 / 10^6 × 655.6

$\dot{m}$ = 0.165 kg/sec

∴Actual Thrust force (FT)= 0.165 (655.6– 78.

FT = 95.22 N

Nozzle efficiency (nozzle) = (Actual thrust Force /Ideal thrust force)

(F thrust) ideal = $\dot{m}$ ideal (Ve-Vo)

$\dot{m}$ ideal = ρ ×A t × Vt = ρ× **π** (Rt)^2 × Vt

ṁ ideal $=1.225 \times \pi \times (23.591)^2/ 10^6 \times 115= 0.2463$kg/sec

∴(F thrust) ideal $= (655.6– 78.41) \times 0.2463$ (F thrust) ideal $= 142.16$ N

∴Nozzle EFFICIENCY$= 95.2/142.16$ Nozzle EFFICIENCY$=$ .6698 or **66.98%**

As can be observed from the pressure and velocity contours that there is an inherent shock wave formation which is not desired so in order to minimize it, we performed second optimization in which we

- Selected inlet half angle, length, and radius as the input parameters

- Minimized force on inlet walls; Maximized pressure, velocity, and Mach no. at Inlet.

- Used direct optimization to find candidate points.

Table of Schematic B2: Optimization				
	A	B	C	D
1	⊟ Optimization St...			
2	Maximize P12	Goal, Maximize P12 (Default importance)		
3	Minimize P17	Goal, Minimize P17 (Default importance)		
4	Maximize P18	Goal, Maximize P18 (Default importance)		
5	Maximize P19	Goal, Maximize P19 (Default importance)		
6	⊟ Optimization Met...			
7	Adaptive Multiple-Objective	The Adaptive Multiple-Objective method is a variant of the popular NSGA-II (Non-dominated Sorted Genetic Algorithm-II) based on controlled elitism concepts. It supports multiple objectives and constraints and aims at finding the global optimum. I is limited to continuous and manufacturable input parameters.		
8	Configuration	Generate 25 samples initially, 25 samples per iteration and find 3 candidates in a maximum of 4 iterations.		
9	Status	Not Converged.		
10	⊟ Candidate Po...			
11		Candidate Point 1	Candidate Point 2	Candidate Point 3
12	P22 - angle_inlet (degree)	15.065	15.401	15.148
13	P24 - throat_length (mm)	25.6	36.772	51.85
14	P25 - inlet_length (mm)	20.578	21.711	21.864
15	P12 - pressureinlet-op (kg^2 m^-1 s^-3)	⭐⭐ 5.9533E+07	⭐⭐ 6.058E+07	⭐⭐ 6.1234E+07
16	P17 - inletforce-op (N)	✗✗ 3389.9	✗✗ 3917.3	✗✗ 4101
17	P18 - inletvelocity-op (N)	⭐ 1999.2	⭐ 1809.9	⭐ 1682.4
18	P19 - inletmach-op (kg s^-1)	⭐ 5.8352	⭐ 5.2703	⭐ 4.8919

TABLE 5: Second Optimization Result

Based on the above result we can deduce that the Candidate Point 1 is the best choice. So we will modify the geometry accordingly.

5.3   Secondary Modified Geometry Analysis

In the final phase we analysed and obtained the results for different parameters such as pressure, velocity and mach number distribution across the Secondary Modified Geometry. On the basis of these results and geometrical parameters we calculated the ideal thrust, actual thrust and nozzle efficiency.
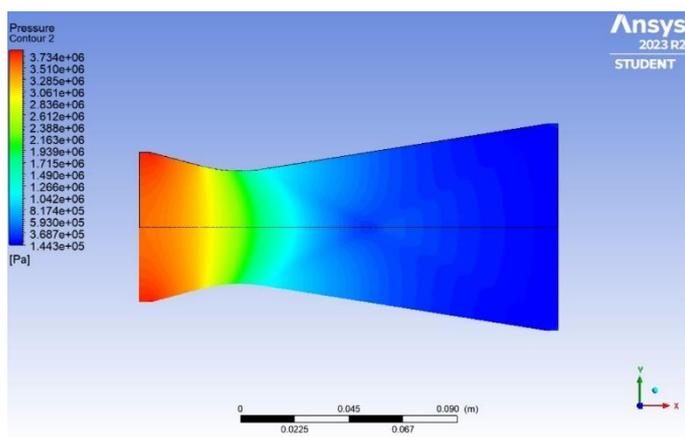

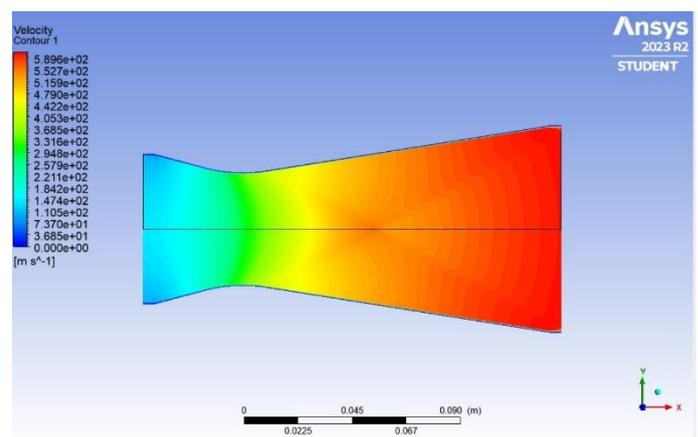**FIG 7 : Pressure Contour of Secondary Modified Geometry**   **FIG 8 : Velocity Contour of Secondary Modified Geometry**

For CASE III :

L = 173.808 mm
Ri = 31.234 mm , Ro = 66.85 mm , Rt = 23.591
Vt = 102.85 m/ sec   ,   Ve = 670.45 m/sec , Vo = 85.41 m/sec
$\dot{m}$ = ρe Ae Ve = 0.0203387 × π × 670.45× (66.85)^2/10^6
$\dot{m}$ = 0.1914 kg/sec

∴Actual Thrust force (FT)= = 0.1914 (670.45 –66.85) = 111.67 N
Nozzle efficiency (nozzle) = (Actual thrust Force /Ideal thrust force)

(F thrust) ideal = $\dot{m}$ideal (Ve-Vo) $\dot{m}$ ideal = ρ ×A t × Vt

$\dot{m}$ ideal =1.225 × π × (23.591)^2 / 10^6 × 102.85
$\dot{m}$ ideal   = 0.2202 kg/sec

∴(F thrust) ideal = (670.45 – 85.41) × 0.2202
(F thrust) ideal = 128.87 N

∴Nozzle EFFICIENCY= 111.97/128.87
Nozzle EFFICIENCY= .8688or **86.88%**

Hence the Shock wave is minimized and efficiency increased.

The Observations are Summarised in the below table:

Case	Inlet Radius (Ri)	Outlet Radius (Ro)	Throat Radius (Rt)	Length (L)	Inlet Half Angle	Outlet Half Angle	Inlet Velocity (Vo)	Outlet Velocity (Ve)	Throat Velocity (Vt)	Main Flow Rate (m)	Actual Thrust Force (FT)	Ideal Thrust Force (FT ideal)	Nozzle Efficiency (%)
Case I	50 mm	70 mm	31.153 mm	200 mm	40°	20°	83.4459 m/s	672.63 m/s	122.3 m/s	0.21059 kg/s	124.07 N	275.67 N	45.64%
Case II	51.323 mm	62.85 mm	23.591 mm	171.23 mm	40°	9.1°	78.41 m/s	655.6 m/s	115 m/s	0.165 kg/s	95.22 N	142.16 N	66.98%
Case III	31.234 mm	66.85 mm	23.591 mm	173.808 mm	15.065°	9.1°	85.41 m/s	670.45 m/s	102.85 m/s	0.1914 kg/s	111.67 N	128.87 N	86.88%

TABLE 6: Result Observation

5.4   Comparative Analysis
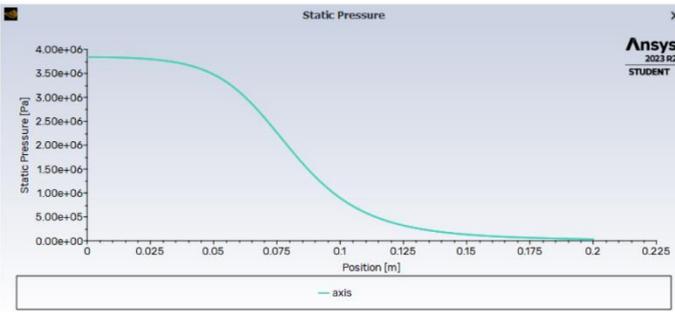
5.4.1 Pressure Plot

FIG 9 : Pressure Plot of Initial Geometry
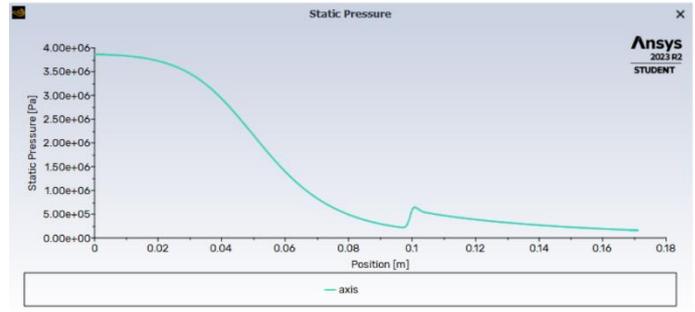

FIG 10 :Pressure Plot of Primary Modified Geometry
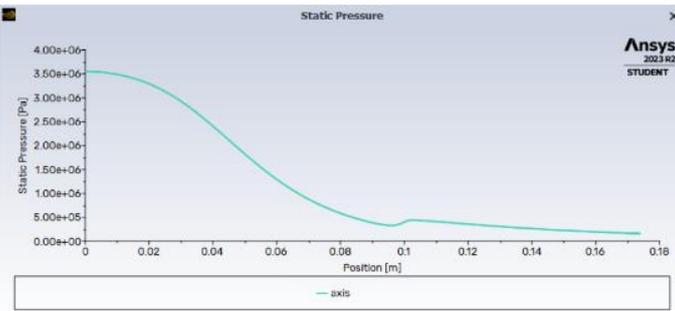

FIG 11 : Pressure Plot of Secondary Modified Geometry
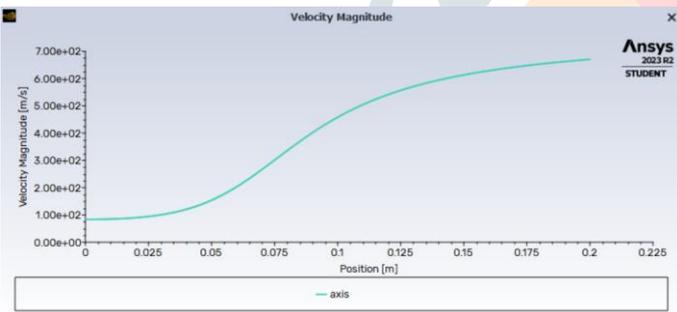
5.4.2 Velocity Plot

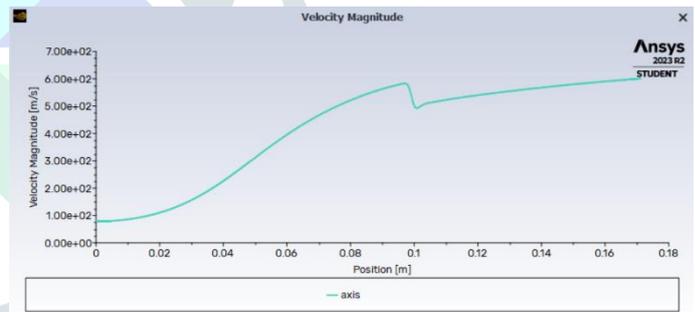
FIG 12 :Velocity Plot of Initial Geometry

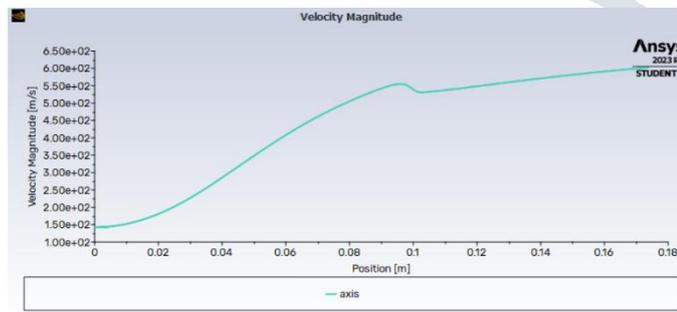
FIG 13 :Velocity Plot of Primary Modified Geometry

FIG 14 :Velocity Plot of Secondary Modified Geometry
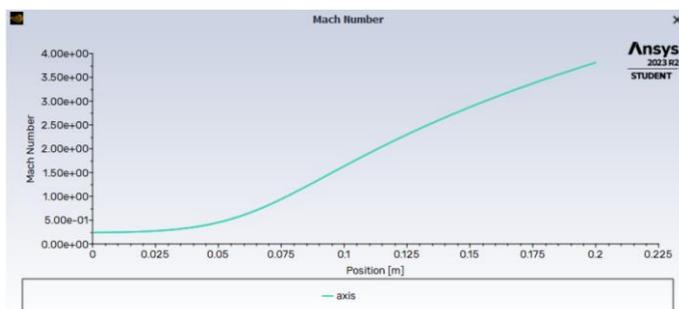
5.4.3 Mach Number Distribution

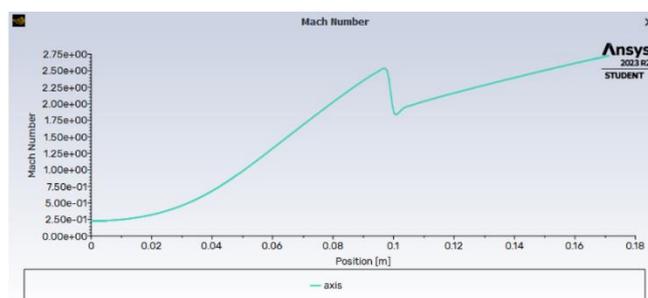**FIG 15 :Mach Number Distribution of Initial Geometry**

FIG 16 : Mach Number Distribution of Primary Modified Geometry
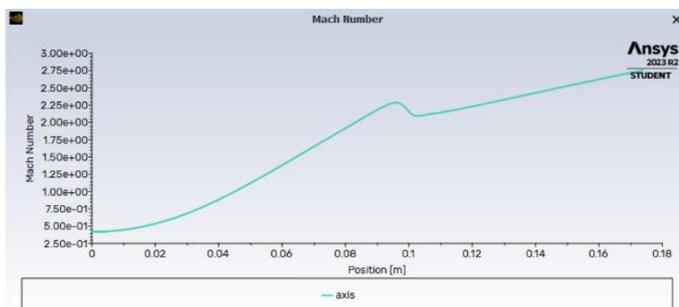

FIG 17 : Mach Number Distribution of Secondary Modified Geometry

6. CONCLUSION

The analysis of the convergent-divergent nozzle design was conducted through three cases, each representing variations in geometric parameters and their effects on nozzle performance.

- ❖ Case I - resulted in an actual thrust force of 124.07 N, with an ideal thrust force of 275.67 N, leading to a nozzle efficiency of 45.64%.
- ❖ Case II - Though there was a shock formation exhibited an actual thrust force of 95.22 N, closer to the ideal thrust force of 142.16 N, resulting in a higher nozzle efficiency of 66.98%.
- ❖ Case III - showcased further enhancement in nozzle performance, achieving an actual thrust force of 111.67 N, closer to the ideal thrust force of 128.87 N, resulting in the highest nozzle efficiency of 86.88% and the shock was minimized.

The systematic variation of geometric parameters allowed for the optimization of nozzle performance, with Case III demonstrating the highest efficiency and hence it also can be concluded that with manipulating the outlet geometrical parameters can be specifically effective in maximizing thrust and exit velocity while manipulating the inlet can be effective in shock wave and losses minimization. These findings underscore the significance of geometric optimization in achieving desired thrust generation and efficiency and minimization of shock formation and losses in convergent-divergent nozzle designs.

7. FUTURE SCOPE

Further refinement of the CFD model and optimization algorithms to enhance accuracy, reliability, and computational efficiency.

Exploration of advanced optimization techniques, such as machine learning and evolutionary algorithms, to tackle complex design optimization problems.

Investigation of multi-objective optimization approaches to simultaneously optimize conflicting objectives, such as thrust and fuel consumption.

Application of the proposed methodology to real-world aerospace propulsion systems and validation through experimental testing and validation campaigns.

Integration of uncertainty quantification and sensitivity analysis methods to assess the robustness of optimized designs to variations in input parameters and operating conditions.

REFERENCES

1) Arjun Kundu, Devyanshu Prasad and Sarfraj Ahmed , "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, June 2016.

2) Mohan Kumar G, Dominic Xavier Fernando and R. Muthu Kumar, "Design and Optimization of De Laval Nozzle to Prevent Shock Induced Flow Separation". Advances in Aerospace Science and Applications, ISSN 2277-3223 Volume 3, Number 2, 2013, pp. 119-124.

3) Venkatesh V, C Jaya pal Reddy, "Modelling and Simulation of Supersonic nozzle using Computational Fluid Dynamics" International Journal of Novel Research in Interdisciplinary Studies, ISSN: 2394-9716, Vol. 2, Issue 6, Nov-Dec 2015, pp. 16-27.

4) Srinivas M, Suresh A L, G. Harish worked on the topic of "Thrust Enhancement of a Convergent-Divergent Nozzle by Using CFD". International Journal for Research in Applied Science & Engineering Technology (IJRASET) ISSN: 2321-9653; Volume 5 Issue X1, November 2017

5) Uttam Kumar, Sudhir Singh Rajput, Dr. Praveen Borkar, " CFD analysis and parameter optimization of Divergent Convergent Nozzle". International Journal of Advance Research and Development; Volume 3, Issue 10, October 2018

6) Olivera P. Kostić," CFD Modeling of Supersonic Airflow Generated by 2D Nozzle with and Without an Obstacle at the Exit Section". FME Transactions, (2015) 43.

7) Pranab Pal, Shailendra Kumar Bohidar, "ANALYSIS AND PARAMETER OPTIMIZATION OF CONVERGENT DIVERGENT NOZZLE BY USING CFD APPROACH" International Research Journal of Modernization in Engineering Technology and Science (Peer-Reviewed, Open Access, Fully Refereed International Journal) Volume:06/Issue:04/April-2024

8) P. Parthiban, M. Robert, T. Ambikapathi, "Design and analysis of rocket engine nozzle by using CFD and optimization of nozzle parameters. "International journal of engineering research, Vol.3,Issue 5,2015.

9) P. Padmanathan, Dr. S. Vaidyanathan, " Computational Analysis of shockwave in convergent divergent nozzle", IJERA, ISSN: 2248-9622, 2012

10) M.H.M Noh, A.H.A Hamid " Numerical investigation of chocked converging diverging nozzles for thruster application"., IIUM Engineering Journal,2011

11) Johnson, A. and Papamoschou, D., "shock wave motion on the instability of the plume exiting an over expanded, convergent-divergent nozzle." IIUM Engineering journal,2011

12) Qun, P. and Tongji, L., "solution of subsonic and transonic flow field in two-dimensional delaval Nozzle with reassigned contraction ratio and expansion ratio and throat wall radius." , IJERA, ISSN: 2276 9523, 2011