Design And Analysis of a Rocket C-D Nozzle

Rajit Tiwari
210818

ME670,
2024-2025 1st sem.
20/11/2024

LIST OF FIGURES

LIST OF TABLES

Abstract
In this paper the study of convergent divergent nozzle is done in the supersonic regime with the use of Ansys Fluent software. The flow within the nozzle is modeled as choked flow, ensuring accurate simulation of high-speed gas dynamics. The material and methods used for the additive manufacturing of rocket nozzle are also discussed. The results obtained are discussed using the temperature, pressure and velocity contours obtained from the Ansys Fluent software. These findings provide valuable information for the design, optimization, and manufacturing of high-performance rocket nozzles.

Keywords: Computational Fluid Dynamics, ANSYS Fluent,Convergent-Divergent, Additive manufacturing, Ideal gas

1. Introduction and Background

A nozzle is a device that is used to increase the kinetic energy of the fluid on the expanse of static enthalpy without having any moving parts. Nozzles are generally used to control the rate of flow, speed, direction and pressure of fluid passing through them. In 1890 a Swedish engineer Karl Gustaf de Laval developed a convergent divergent nozzle that has the capacity to increase the steam jet to supersonic speed. These nozzles are called de Laval nozzle and are used in rocket propulsion.

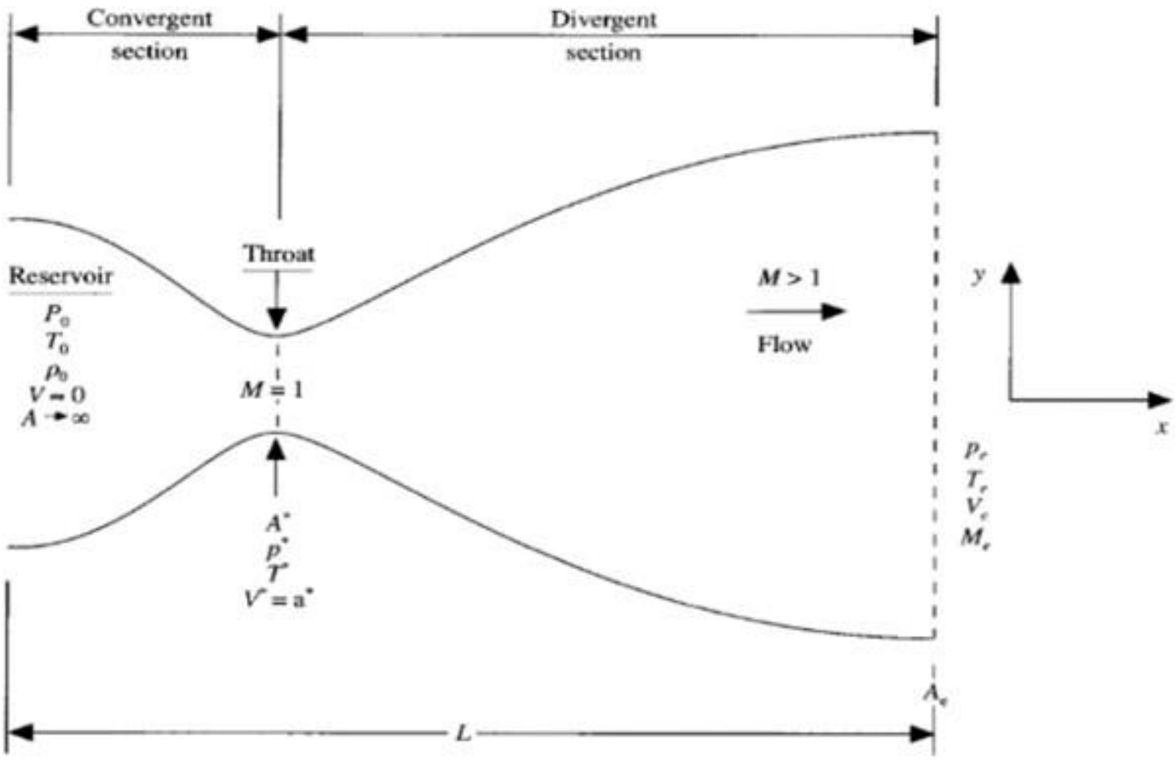

Fig 1.1 De Lavel nozzle (Jayprakash et al, 2022)

The flow through the convergent divergent nozzle is compressible as it deals with higher Mach numbers in the divergent section(M>1). The flow is considered as adiabatic and isentropic through the nozzle neglecting the friction. The study of pressure, velocity, temperature etc. within the nozzle is important for maximizing the thrust and efficiency of rocket propulsion.

Additive manufacturing of rocket nozzle

Manufacturing rocket nozzles is a tedious task. Manufacturing nozzles becomes difficult when we try to optimize the design by adding cooling channels and reducing weight. Additive manufacturing (AM), also known as 3D printing, is revolutionizing the production of rocket nozzles by offering several advantages over traditional manufacturing methods. Here's a breakdown of how additive manufacturing is being used to create these critical components:

(a) AM allows for the creation of intricate and optimized designs that would be difficult or impossible to achieve with traditional manufacturing techniques. This enables the design of nozzles with internal cooling channels and other features that improve performance and reduce weight.

(b) By creating parts with optimized lattice structures and other design features, AM can significantly reduce the weight of rocket nozzles. This leads to increased payload capacity and fuel efficiency.

(c) AM enables the integration of multiple components into a single part, reducing assembly time and potential points of failure. This can be particularly beneficial for rocket nozzles, where injectors, cooling channels, and other components can be combined into a single structure.

(d) AM significantly reduces the time required to produce prototypes and test new designs. This allows for rapid iteration and optimization of rocket nozzle designs.
With these benefits various additive manufacturing techniques can be used to manufacture the rocket nozzle as follows.

(a) **Direct metal laser sintering (DMLS)**: This technique uses a high-powered laser to melt and fuse metal powder layer by layer, building up the rocket nozzle structure. It is well-suited for high-temperature materials like Inconel and titanium alloys, which are commonly used in rocket nozzles.
(b) **Electron beam melting (EBM)**: Similar to DMLS, EBM uses a high-energy electron beam to melt and fuse metal powder. It is particularly effective for producing large and complex rocket nozzle components.
(c) **Binder Jetting:** This technique uses a binder to selectively bond ceramic or metal powder particles, creating a green part that is subsequently sintered to achieve the final density and strength. Binder jetting is well-suited for producing large-scale rocket nozzles with intricate internal structures.

At present different type of materials are used for the manufacturing of rocket nozzle including metals, composites and ceramics. Different metal alloys such as aluminium alloy, titanium alloy, nickel alloy, copper alloy etc are used for their low weight, high thermal conductivity, corrosion resistance, high melting temperature, high temperature strength, fatigue resistance properties. Ceremic silicon carbide (SiC) offers high temperature resistance, excellent thermal shock resistance, and low thermal expansion. It is often used for the throat and nozzle cone of rocket engines. Carbon-carbon composite materials exhibit exceptional thermal shock resistance, high-temperature strength, and low density. They are commonly used for high-performance rocket nozzles.

There are some limitations to the additive manufacturing of nozzles such as post processing requirements, material limitations etc. Instead, additive manufacturing is playing a vital role in the manufacturing of rocket nozzles and other aerospace components.

2. Problem Definition

In this paper, a comparative study is done for the convergent divergent nozzle with theoretical values to the computational values using the CFD software Ansys Fluent. As nozzles are used in various industries including aerospace and power plant turbines, so it is important to study the fluid flow behavior inside the nozzle. The variation of physical properties of fluid such as pressure, temperature, enthalpy, Mach number etc. is studied for the choked flow in the nozzle in the subsonic, sonic and supersonic regions.

Table 2.1-Problem setup

General	Solver type: Density-based 2D Space: Axi-symmetric
Models	Energy equation: On
	Viscous model: Standard k-e model, realizable, enhanced wall treatment
Boundary conditions	Inlet Pressure = 3bar Inlet Temperature = 300K Walls are assumed to be adiabatic.

3. Materials and Methods

3.1 Methodology

Rocket thrust equation

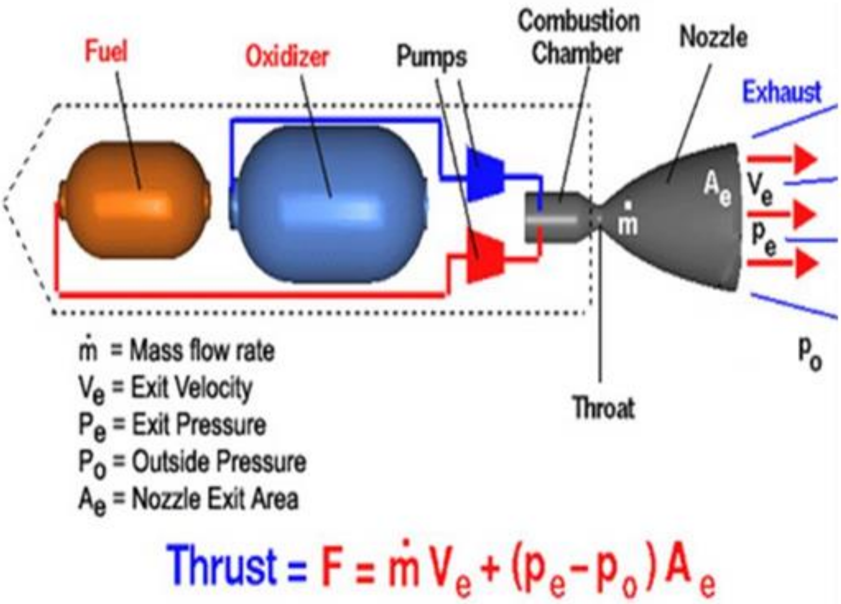

Fig 3.1 Rocket thrust equation (Munipally Prathibha et al, 2015)

Nozzles are used to accelerate hot exhaust to produce thrust as described by Newton's third law of motion. The amount of thrust produced by the engine depends on the mass flow rate through the engine, the exit velocity of the flow, and the pressure at the exit of the engine. The value of these three flow variables is all determined by the nozzle design.

Here thrust is maximum when the exit pressure is equal to the ambient pressure, so it is important to find the exit pressure variation for the nozzle.
The shape of the nozzle is crucial in various applications, for example in controlling the flow, speed, direction and pressure. In rocket engines the shape of the nozzle decides the efficiency and performance of the engine. The nozzle used for study in this paper is De Lavel nozzle with the following dimensions.

Table 3.1 Dimensions of nozzle (Mahima Arhanth et al, 2021)

Dimensions of the Nozzle	
Inlet diameter	25mm
Exit diameter	35mm
Throat diameter	10mm
Length of nozzle	75mm

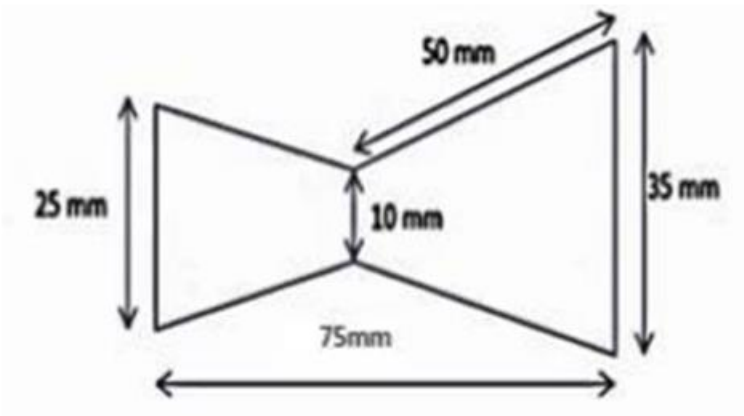

Fig 3.2 Side view of nozzle (Mahima Arhanth et al, 2021)

3.2 Governing Equations

The flow through a nozzle is assumed to be adiabatic and isentropic flow. The flow is governed by the steady state continuity and steady state energy equations. The flow is assumed as one dimensional for theoretical purposes.

The continuity equation is simplified as,

$$\rho_1 A_1 V_1 = \rho_2 A_2 V_2$$

Energy equation is simplified as,

$$h_i + \frac{V_i^2}{2} = h_e + \frac{V_e^2}{2}$$

i.e. stagnation enthalpy is constant along the length of the nozzle.

Now for choked flow the following equations can be derived using the above equations.

$$\frac{A_x}{A_{th}} = \left(\frac{T_{th}}{T_x}\right)^{\frac{1}{\gamma-1}} \times \left(\frac{\sqrt{\gamma R T_{th}}}{V_x}\right)$$

$$C_p T_{th} + \frac{V_{th}^2}{2} = C_p T_x + \frac{V_x^2}{2}$$

Where A_x is area of cross section, T_x is the temperature and V_x is the velocity at a distance x from inlet in axial direction. A_{th}, T_{th} *and* V_{th} are area, temperature and velocity at throat of nozzle respectively. C_p is the heat capacity of ideal gas at constant pressure.

Solving the above equations simultaneously will give us the value of temperature and velocity at the required section of the nozzle.

4. Results and Discussion

The material used in the study is the ideal gas which has following properties

Materials	Density: ideal gas Cp = 1880J/kg K Adiabatic index(γ) = 1.19 Viscosity= 8.983×10^{-5}Pa. s Thermal conductivity = 0.0142 W/Mk Mean molecular mass = 27.7 g/mol

For the choked flow the throat temperature, inlet temperature and throat pressure, inlet pressure are related as

$$\frac{T_{th}}{T_i} = \frac{2}{\gamma + 1}$$

and

$$\frac{P_{th}}{P_i} = \left(\frac{2}{\gamma + 1}\right)^{\frac{\gamma}{\gamma - 1}}$$

Using the above equation, the theoretical values of temperature, pressure and velocity can be calculated and then compared with the values obtained from the Ansys Fluent. The comparison analysis shows that the theoretical values are slightly lower than the values obtained from Ansys Fluent. This is because in theoretical analysis we neglect the frictional losses, boundary layer, shock waves, drag etc. but Fluent will take care of all these things as it directly solves the differential equations involved.

Table 4.1 Solution (Jayprakash et al, 2022)

Solution controls	Courant number = 5
Solution initialization	Compute from: Inlet
Run calculation	Check case No. of iterations: 2000 Click Calculation

Preprocessing of the nozzle is done in the Ansys fluent. Two dimensional and double precision settings are used while reading the mesh. Mesh was checked for errors and boundary conditions are applied. And steady state solution is run for 2000 iterations and following residuals are obtained.

The solution converged in 330 iterations with residuals less than 1e-03.

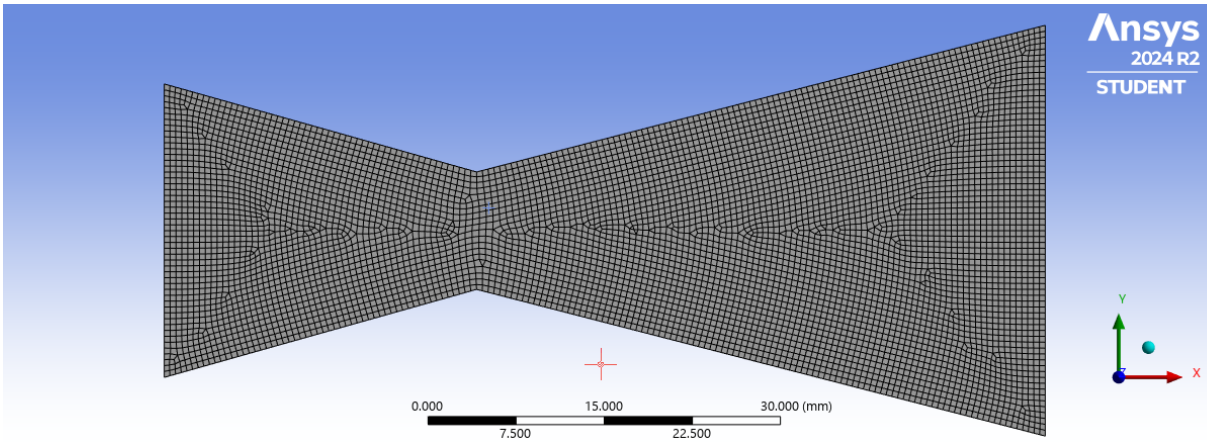

Fig 4.1 Discretised mesh of the nozzle domain

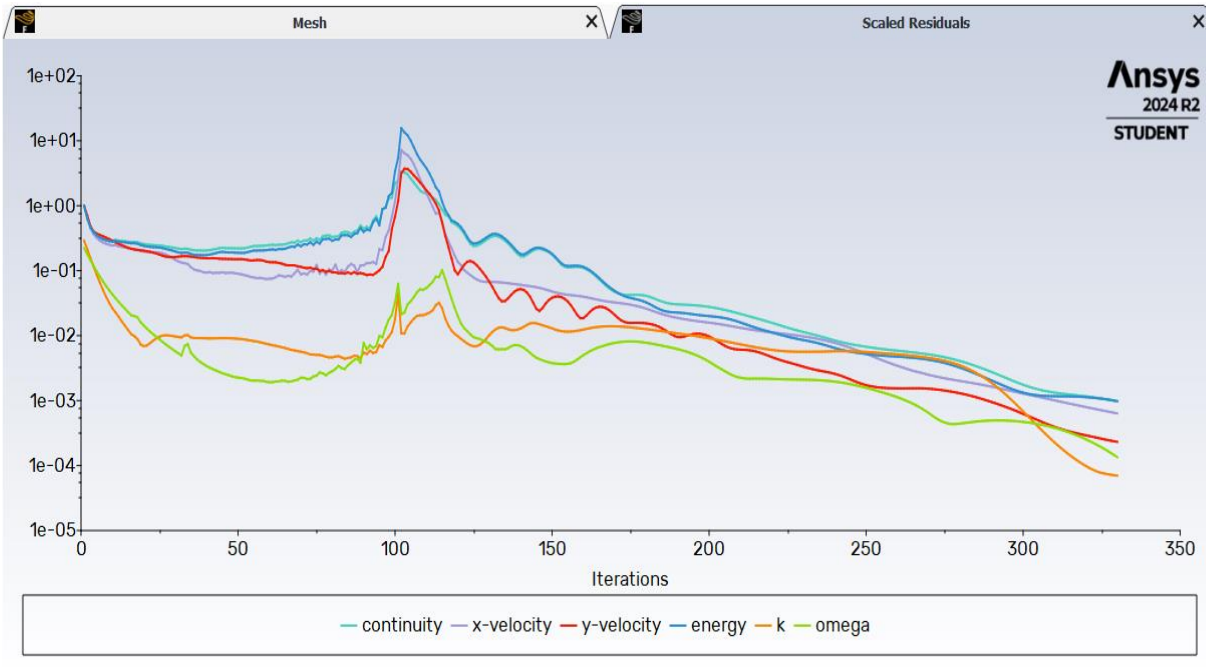

Fig 4.2 Scaled residuals

The following contours of velocity, temperature and pressure are obtained from the Ansys Fluent.

Velocity Contour

The velocity of gas goes on increasing from the nozzle inlet to the exit because the flow is chocked at the throat of the nozzle as can be seen from the contour.

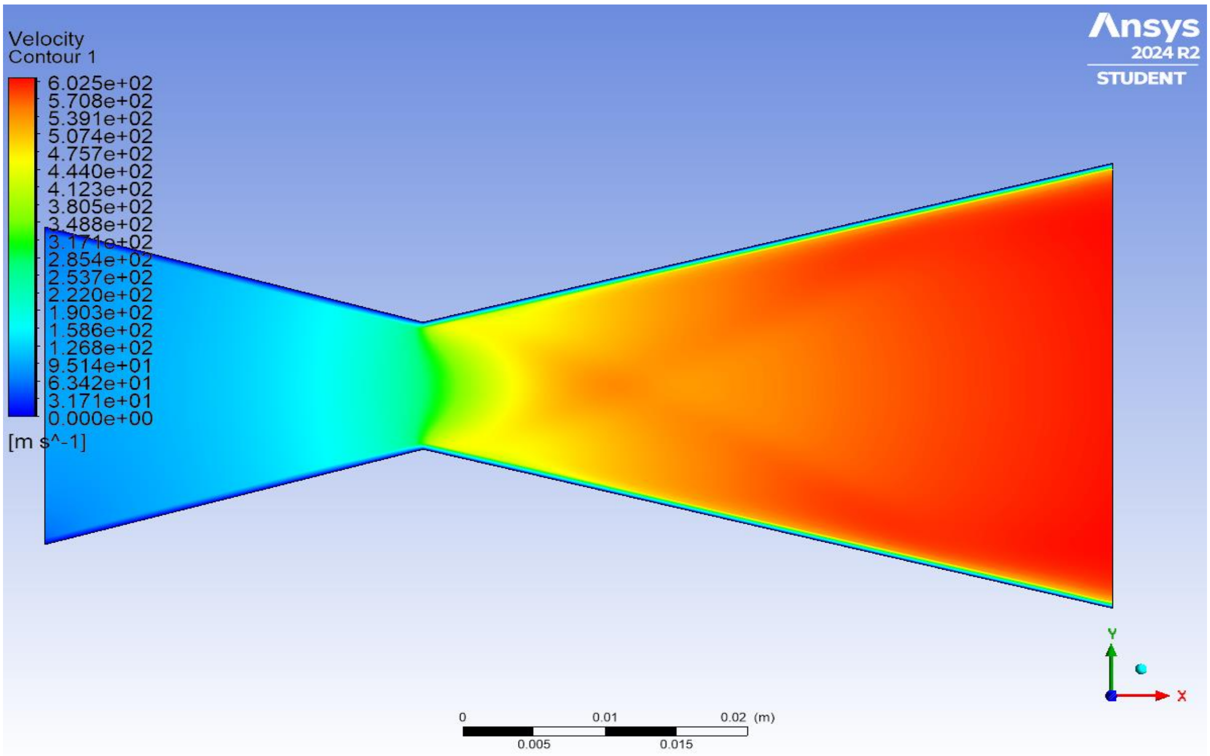

Fig 4.3 Contour of velocity magnitude(m/s)

Pressure contours

Pressure goes on decreasing from the inlet to the exit. As from the thrust equation, thrust will be maximum when exit pressure is very less.

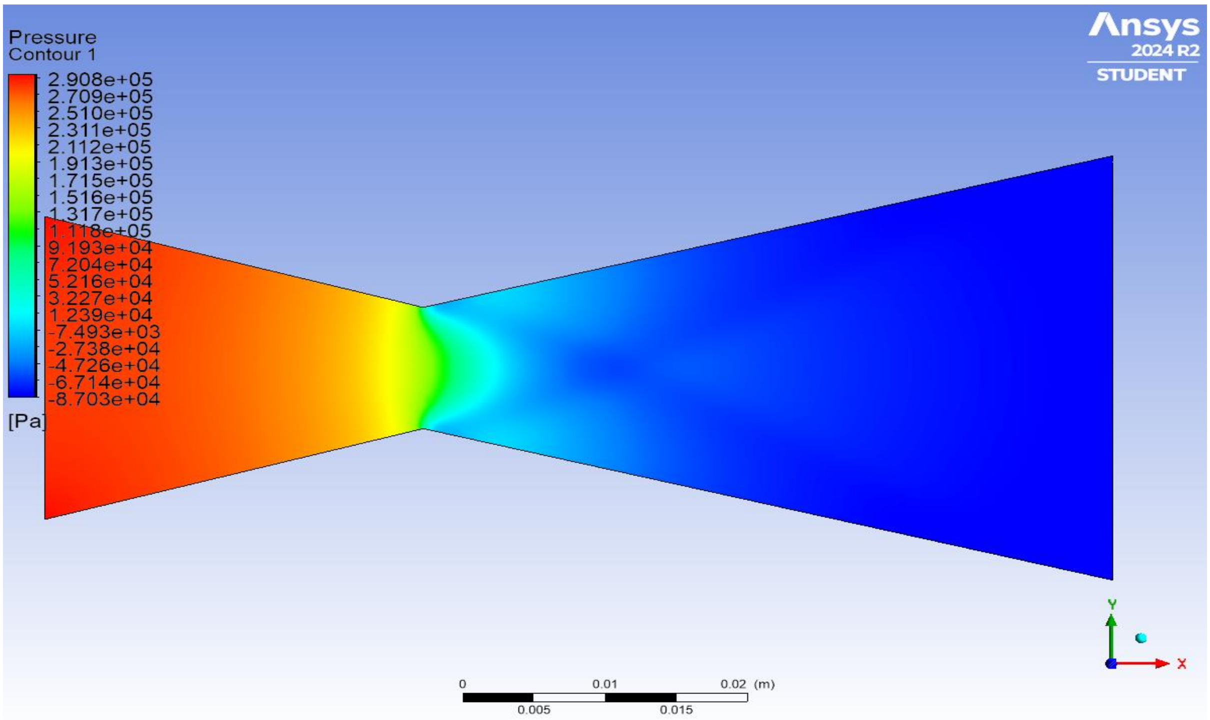

Fig 4.4 Contours of static pressure (Pa)

Temperature contour

The temperature is maximum at the inlet and minimum at the outlet. The temperature profile is necessary for designing the nozzle cooling system and selecting the material for the nozzle design.

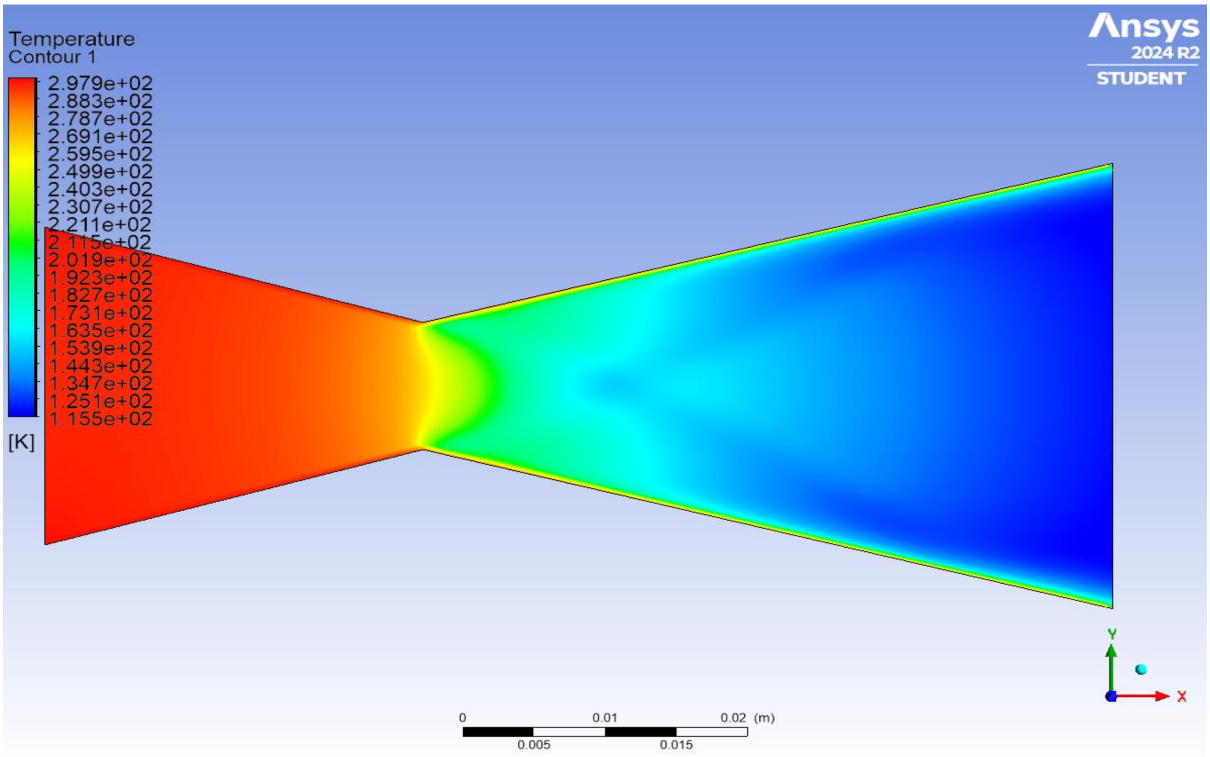

Fig 4.5 Contour of static temperature (K)

Density contour

The density of gas varies the same as pressure and temperature i.e. keep on decreasing from inlet to exit.

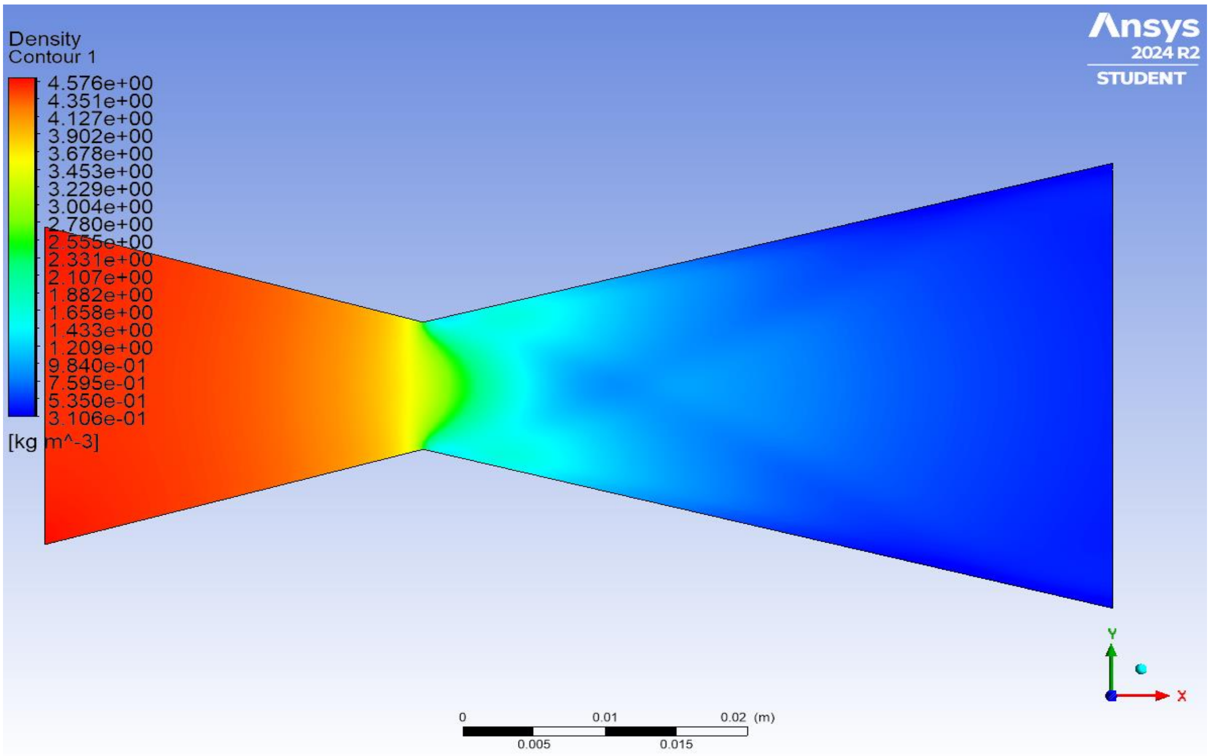

Fig 4.6 Contour of density (Kg/m^3)

Velocity vectors

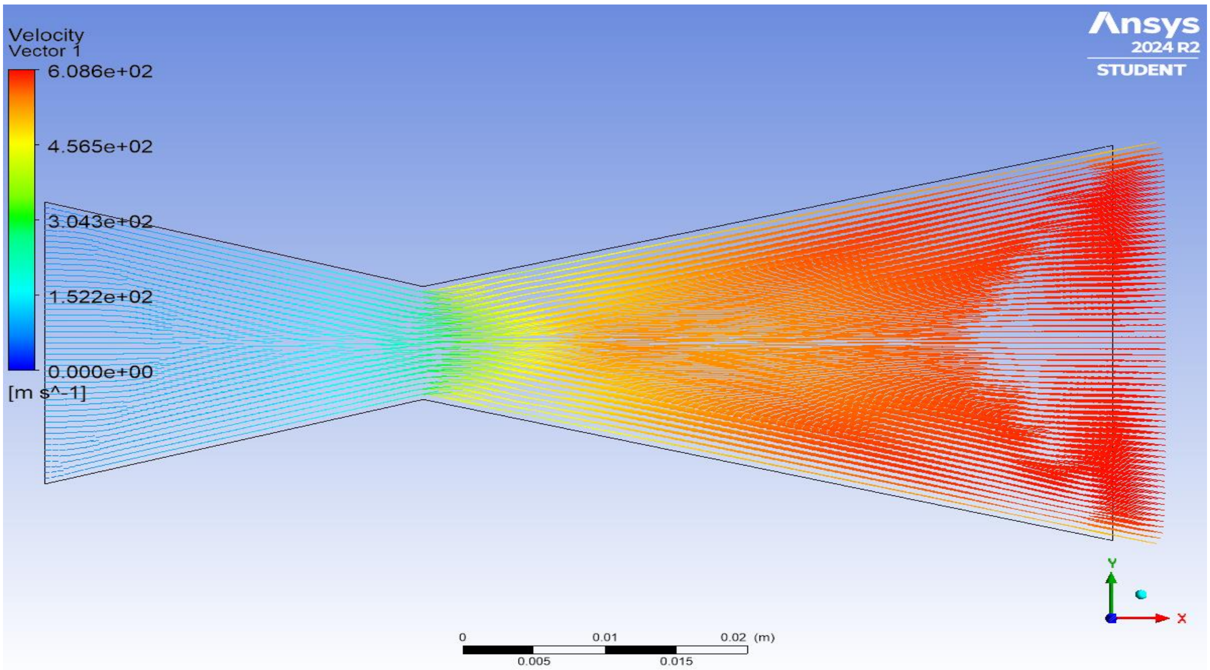

Fig 4.7 Velocity vectors within the nozzle

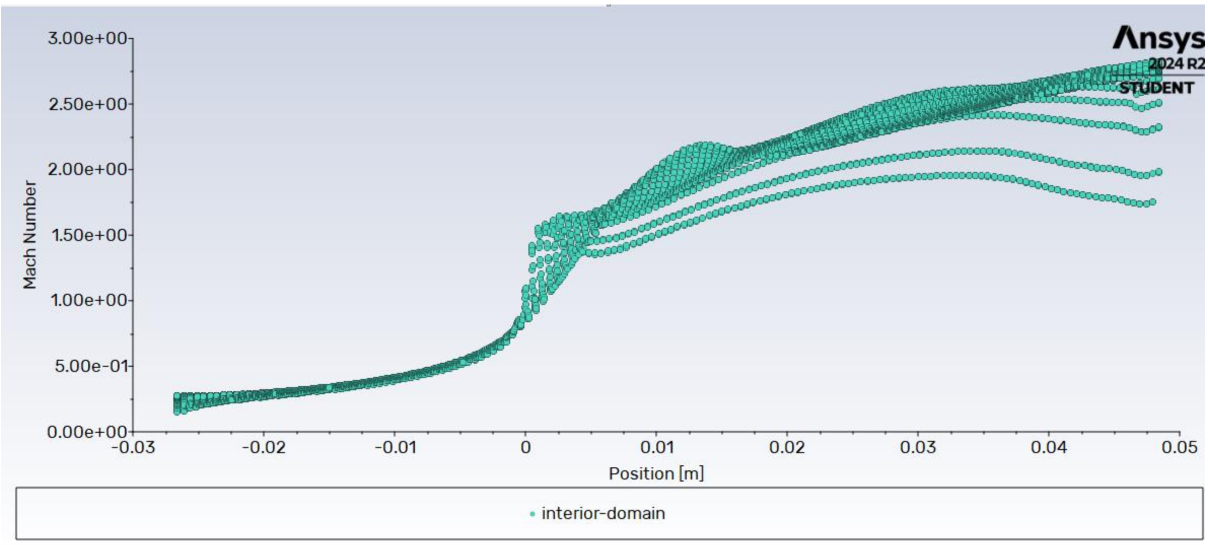

Fig 4.8 Mach number within the nozzle domain

5. Conclusion

The findings in this paper can be summarized in the following bullet points.

- The flow through the nozzle is chocked flow i.e. Mach number at the throat is 1.
- The temperature and pressure decreases continuously in the nozzle with maximum at inlet and minimum at exit.
- The velocity of gas increases continuously within the nozzle with maximum at exit and minimum at inlet.
- Theoretical values are slightly smaller than the values obtained from Ansys Fluent as the software considers frictional losses, drag, boundary layer effect etc.
- The study is done for sample temperature and pressure which does not represent the actual rocket conditions but the results are similar so it just may be a matter of scaling.

Areas of further study

- Study of the structural analysis of the nozzle can done for determining the thermal stresses and strains.
- study of the temperature of the nozzle can be done containing conformal cooling channels or regenerative colling.
- Further study can be done using different nozzles with different angles of convergence and divergence.

References

[1] Mahima Arhanth, Kushal Jagadish, K M Vinod Kumar, Naveen H E (2021). Analysis of Rocket Nozzle with 1 and 4 Inlets, e-ISSN: 2395-0056.

[2] Jayaprakash, P., Dhinarakaran, D., & Das, D. (2022). Design and analysis of a rocket C-D nozzle. International Journal of Health Sciences, 6(S5), 3545–3559.

[3] Pardhasaradhi Natta, V.Ranjith Kumar, Dr.Y.V.Hanumantha Rao, (2012). Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics, Vol. 2, pp.1226-1235.

[4] https://www.grc.nasa.gov/www/k-12/airplane/nozzled.html

[5] https://www.metal-am.com/articles/advancing-rocket-propulsion-through-3d-printing-novel-surface-finishing-technologies-and-public-private-partnerships/

[6]https://www.youtube.com/watch?v=P_jdywkbRQ4&list=PLYN_aHIvN7CHdK8haWpaM2NN-0-U6tfjg

[7] Munipally Prathibha, M. Satyanarayana Gupta, Simhachalam Naidu, (2015). CFD Analysis on a Different Advanced Rocket Nozzles, ISSN: 2249-8958.