PROJECT REPORT – NOZZLE ANALYSIS (GAS TURBINES & JET PROPULSION)

A Report Submitted for the Course ME430

BACHELOR OF TECHNOLOGY

MECHANICAL ENGINEERING

MEMBERS:

VAMSHIK M (201ME364)

ANANNYA RAO (201ME209)

AMOGH NAYAK (201ME109)

DEPARTMENT OF MECHANICAL ENGINEERING

NATIONAL INSTITUTE OF TECHNOLOGY KARNATAKA

SURATHKAL, MANGALORE-575 025

NOVEMBER, 2023

Table of contents:

1. Introduction

This project addresses the analysis of propulsion nozzles within the context of the gas turbines and propulsion course. Nozzles, integral to high-speed jet-based propulsion systems, play a crucial role in accelerating fluid while minimizing energy loss. Operating at speeds surpassing the speed of sound introduces challenges like choking and shocks. While analytical solutions offer insights into average nozzle variation, finite element CFD evaluation provides a more nuanced understanding of geometry-based effects, such as slope variation and throat characteristics.

Most hand calculations for nozzle systems typically focus on inlet, throat, and area, assuming linear variation. To address these limitations, a robust solution is essential. This report serves as a compilation, exploring the fundamentals of nozzle dynamics, its governing ODEs, and attempting to outline a CFD methodology. Importantly, the intention is not to introduce a novel method for calculating nozzle effects, but rather to leverage computational nozzle datasets for real-time optimization of nozzle geometry.

Furthermore, the evolution of rocket nozzle components is characterized by continual development aimed at enhancing efficiency and performance. Primarily tasked with directing and accelerating combustion-generated gases, nozzles contribute to thrust force. This research scrutinizes the design of nozzles, analyzing parameters such as Mach velocity, temperature, and pressure. Evaluating nozzle efficiency under specific temperature conditions involves a Computational Fluid Dynamics (CFD) simulation using ANSYS Fluent, encompassing the entire design process within Spaceclaim. Then a comparative analysis of elementary equations and CFD was done to validate the CFD simulations with ANSYS.

In essence, this project not only delves into the intricate world of nozzle dynamics but also places a strong emphasis on the validation of CFD results using ANSYS, ensuring reliability and accuracy in the analysis of propulsion nozzles.

2. Literature review

2.1 Propulsive nozzle design

The rocket nozzle comprises three key components: a converging section, a throat, and a diverging section. Initially, the combustion exhaust gas enters the converging section, where it moves at subsonic speeds. As it progresses through this section, the gas accelerates due to the reduction in cross-sectional area. To achieve supersonic speeds, the gas must traverse a region of minimum cross-sectional area known as the throat. Subsequently, the supersonic gas expands through the diverging section and exits the nozzle, with the expansion causing further acceleration.

The nozzle encompasses several key features that collectively define its pivotal functionality within the propulsion system:

1. **Thrust Generation**: The primary function of the nozzle is to generate thrust, a critical aspect in the propulsion process.
2. **Conversion of Thermal to Kinetic Energy**: The nozzle plays a crucial role in converting the thermal energy derived from the hot chamber gases into kinetic energy. This kinetic energy is then precisely directed along the axis of the nozzle.
3. **Guidance of Combustion Exhaust Gases**: Combustion exhaust gases are skillfully directed into the throat region of the nozzle, marking a strategic step in the propulsion sequence.
4. **Compression in the Throat Region**: The throat, featuring a smaller cross-sectional area compared to the rest of the engine, induces compression of gases to achieve elevated pressure levels.
5. **Gradual Increase in Cross-Sectional Area:** The nozzle systematically expands in cross-sectional area, facilitating the controlled expansion of gases. This expansion exerts force against the nozzle walls, ultimately creating the desired thrust.
6. **Mathematical Optimization for Efficient Gas Expansion:** Mathematically, the nozzle is designed to efficiently expand gases, with the primary objective of maximizing exit velocity.
7. **Significance of Expansion Area Ratio:** The expansion area ratio emerges as a critical parameter in nozzle design. By keeping other variables constant, particularly chamber pressure, there exists a singular ratio that optimizes nozzle performance for a given altitude or ambient pressure.

However, it's important to consider that a rocket operates across various altitudes, necessitating an understanding of its trajectory to select an expansion ratio that maximizes performance over a range of ambient pressures.

As a result, variable expansion ratio nozzles are preferred for space travel, allowing for optimal performance across diverse ambient pressures during the rocket's trajectory.
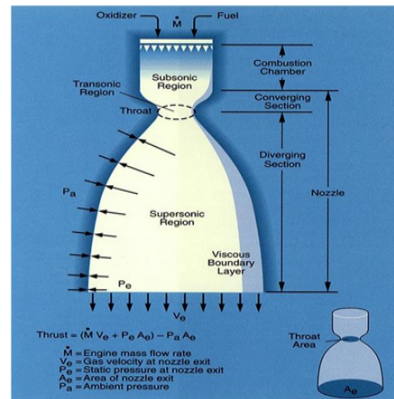

Fig 1 : Geometry of bell shaped nozzle

2.2 Different types of nozzle

They could be based on either speed or shape:
Based on speed
         1. Spray nozzles
         2. Ramjet
Based on shape
         3. Conical
         4. Bell
         5. Annular

2.2.1 Conical :

The "cone" nozzle, initially employed in early rocket applications for its simplicity and ease of construction, earned its name from the consistent divergence of its walls at a constant angle. This design, featuring a smaller angle, is strategically chosen to maximize thrust by emphasizing the axial component of exit velocity, consequently achieving a high specific impulse. However, this advantage comes at the cost of a longer and heavier nozzle, introducing complexity to the manufacturing process. Conversely, opting for a larger nozzle wall angle minimizes both size and weight, offering a more streamlined design. Yet, this approach presents a trade-off, as large angles can result in reduced performance at low altitudes. The challenges stem

from overexpansion and flow separation induced by the elevated ambient pressure, a phenomenon quantified by the key metric of divergence loss in this context.

2.2.2 Bell Nozzle

The concept of this nozzle was initially explored at the Jet Propulsion Laboratory in 1949, gaining notable attention in the late 1960s when Rocketdyne patented it. This particular nozzle design has garnered renewed interest in recent years in both the United States and Europe. The design features a distinctive inner base nozzle, a sectional wall, and an outer nozzle extension.

One of the noteworthy characteristics of this nozzle concept is its altitude adaptation, primarily facilitated by the wall in section. At lower flow altitudes, controlled and symmetrical flow separation occurs at this specific section of the wall, resulting in a reduced effective area ratio. As the altitude increases, the nozzle flow remains attached to the wall until the exit plane, allowing for the utilization of the full geometrical area ratio. This higher area ratio contributes to improved vacuum performance. However, it is essential to note that bell nozzles introduce additional performance losses despite the enhanced vacuum performance achieved.

2.2.3 Annular Nozzle

Annular nozzles, also known as plug or altitude-compensating nozzles, although less commonly employed due to increased complexity, are regarded as theoretically superior in certain aspects. The annular design of these nozzles involves combustion occurring along a ring or annulus around the base, with the term "plug" referring to the central body that disrupts flow in the central portion, marking a departure from conventional nozzle configurations. The primary advantage of annular nozzles lies in their altitude-compensation capabilities, allowing them to adapt effectively to varying altitudes.

A crucial parameter for characterizing annular nozzles is the annular diameter ratio, represented as Dplug / Dthroat. This ratio serves as a metric for comparing the geometry of annular nozzles with other plug nozzle shapes. There are two main types of annular nozzles that have been developed: those that expand exhaust outward and those that expand exhaust inward. Despite their lower prevalence in practical usage, annular nozzles are esteemed for their theoretical efficiency, providing altitude compensation and flexibility in adapting to diverse altitude conditions. The distinction in expansion methods, whether outward or inward, contributes significantly to the development of various annular nozzle designs.

2.3 Incompressible Flow

In the study of incompressible nozzle flow, the governing equations revolve around the principles of fluid dynamics, emphasizing the conservation of mass and the simplified nature of density within the nozzle.

2.3.1 Governing Equations for Incompressible Flow

Continuity Equation:

The continuity equation for incompressible flow is expressed as:

$$\nabla \cdot V = 0$$

Where:

1. ∇ represents the divergence operator.
2. V is the velocity vector.
3. Navier-Stokes Equation:

The Navier-Stokes equation in the absence of compressibility effects simplifies to:

$$\rho \left(\frac{\partial \mathbf{V}}{\partial t} + (\mathbf{V} \cdot \nabla)\mathbf{V} \right) = -\nabla p + \mu \nabla^2 \mathbf{V}$$

Where:

1. ρ is the fluid density.
2. V is the velocity vector.
3. t represents time.
4. p is the pressure.
5. μ is the dynamic viscosity.

2.3.2 Analyzing the Incompressible Flow Equations

Continuity Equation Implications:

The continuity equation ensures that the divergence of the velocity field is zero, emphasizing the conservation of mass within the nozzle. Since the density does not change, this flow equation reduces to the given equation.

$$A_1 v_1 = A_2 v_2$$

This equation implies that velocity and area are inversely proportional and the acceleration can be calculated via very simple calculations.

2.3.3 Advantages & Limitations of Incompressible Nozzles:

Advantages:

1. Simplified analysis: With constant density, equations governing incompressible flow are often simpler and more manageable.
2. Applicability to low-speed flow: Incompressible nozzles are well-suited for scenarios where flow velocities are significantly below the speed of sound.

Limitations:

1. Incompressible nozzle models are ineffective at high velocities where the assumption of constant density becomes invalid.
2. Shock waves, characteristic of supersonic and hypersonic flows, are not considered in incompressible nozzle analyses, limiting their applicability to subsonic regimes.
3. The models exhibit constraints when applied to gases, particularly in scenarios where compressibility effects are significant.
4. The critical choking phenomenon in high-speed flows is often overlooked in incompressible nozzle analyses.

2.4 Compressible Flow in a Nozzle

Compressible flow pertains to how fluids behave when subjected to conditions where variations in pressure and density notably impact their properties. Unlike incompressible flow, compressible flow recognizes that changes in pressure induce consequential shifts in fluid density. This distinction is particularly relevant in scenarios where compressibility becomes a critical factor, such as at high speeds or in aerodynamic applications.

2.4.1 Characteristics & Factors

In the realm of compressible flow within a nozzle, several critical characteristics and factors come into play:

1. Density Variations: Compressible flow accounts for changes in fluid density, a crucial factor in high-speed scenarios.
2. Mach Phenomena: The occurrence of Mach waves becomes significant as fluid velocities approach and exceed the speed of sound, introducing complexities in nozzle behavior.
3. Temperature Effects: Compressible flow brings temperature variations into

consideration, influencing the compressibility factor and overall flow dynamics.

2.4.2 Mach Number

The Mach number is a dimensionless parameter that characterizes the speed of a fluid flow relative to the speed of sound in the same fluid.

Mach (M) = Vf / Vs

1. Vf →Velocity of fluid
2. Vs→Velocity of Sound at same pressure and temperature

Velocity of sound is generally a function of temperature or density of air in the same space for a given type of gas.

$$v_{sound} = \sqrt{\frac{\gamma R T}{M}}$$ where
γ = adiabatic constant
R = gas constant
M = molecular mass of gas
T = absolute temperature

The Mach number (M) serves as a pivotal parameter characterizing compressible flow. It is the ratio of fluid velocity to the speed of sound, categorizing flow regimes as subsonic (M<1), transonic (M≈1), supersonic (M>1), and hypersonic (M≫1).

2.5 Flow Parameters and Equations in Compressible Nozzle Analysis

Within the context of compressible flow in a nozzle, understanding the interplay of key parameters and equations is crucial for insightful analysis:

2.5.1. Equation Derivation:

The relationship is given as:

$$(1 - M^2)\frac{dV}{V} = -\frac{dA}{A}$$

It is derived from the following equations:

1. **Continuity Equation**: Expressing mass conservation, it ensures that the product of cross-sectional area (a) and velocity (v) remains consistent along the nozzle.

$$\dot{m} = \rho.V.A$$

2. **Momentum Equation**: Incorporating the effects of varying velocity and pressure, this equation contributes to the derivation, offering insights into the dynamic behavior of compressible flow.

$$\rho V\, dV = -dp$$

3. **Isentropic Flow Equations**: Considering adiabatic and reversible processes, these equations capture the entropy and temperature changes, providing a foundation for understanding compressible flow characteristics.

$$\frac{dp}{p} = \gamma \frac{d\rho}{\rho}$$

2.5.2. Interpretation:

Breaking down the derived equation:

1. da/a: Represents the relative change in cross-sectional area along the nozzle.
2. $(M^2 - 1)$: Quantifies the impact of Mach number (M) on the flow dynamics.
3. dv/v: Reflects the variation in velocity, indicating the influence of compressibility on the fluid motion.

Significance:

This relationship unveils how alterations in area, Mach number, and velocity collectively dictate the behavior of compressible flow in a nozzle. It serves as a pivotal tool for engineers and researchers to predict and optimize nozzle performance under varying conditions.

2.6 Choking Phenomena

2.6.1 Definition

Choking occurs when the flow through a nozzle reaches sonic conditions $(M=1)$, and further reductions in downstream pressure do not increase the mass flow rate. It is a critical point where the flow becomes "choked" or constrained. Typically, choking is prevalent at the throat of a converging-diverging nozzle, emphasizing its occurrence at the point of maximum constriction in the nozzle geometry. At this specific location, the flow reaches its sonic limit, and any attempts to further reduce pressure result in no increase in the mass flow rate. The throat's role as the focal point of choking underscores its significance in understanding and analyzing the behavior of compressible flows through nozzles.

2.6.2 Pressure Equation

The pressure equation at the throat of a converging-diverging nozzle can be expressed as:

$$\frac{p_0}{p} = \left(1 + \frac{\gamma-1}{2}M^2\right)^{\frac{\gamma}{\gamma-1}}$$

This equation illuminates the pressure ratio (p_0/p) at the throat, emphasizing the influence of the Mach number (M) and the specific heat ratio (γ).
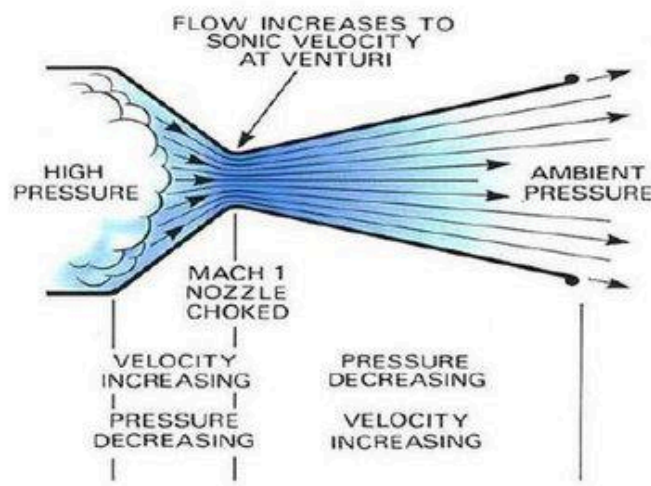

Fig: Choking Phenomena

3. Methodology

3.3 Elementary Approach

As per literature review, the governing equations can be solved with respect to the parameters at the choke/throat region. This elementary approach assumes an isentropic fluid following the relation:

$$PV^\gamma = constant$$

Dynamic pressure is derived as:

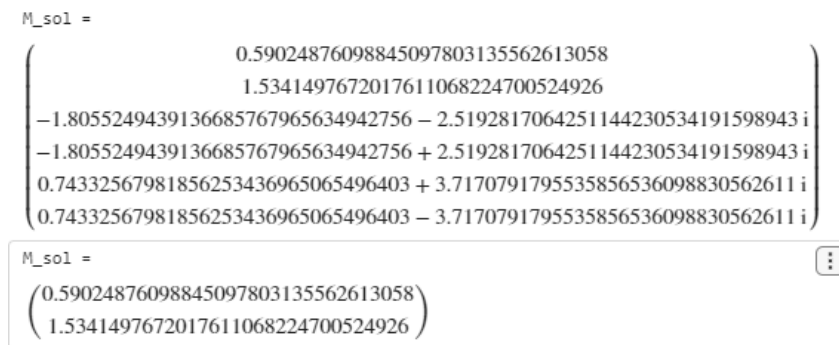$$q = \frac{\rho v^2}{2} = \frac{\gamma p M^2}{2}$$

Using the conservation of mass, momentum, and energy and the definition of total enthalpy in the flow, we can derive the following relations for area, pressure, density and temperature w.r.t. total conditions. Here * refers to the throat while 't' refers to the total value.

$$\frac{A}{A^*} = \left(1 + M^2 \tfrac{\gamma-1}{2}\right)^{\frac{\gamma+1}{\gamma-1} \cdot \frac{1}{2}} \frac{\left(\frac{\gamma+1}{2}\right)^{-\frac{\gamma+1}{\gamma-1} \cdot \frac{1}{2}}}{M}$$

$$\frac{p}{p_t} = \left(1 + M^2 \tfrac{\gamma-1}{2}\right)^{-\frac{\gamma}{\gamma-1}}$$

$$\frac{T}{T_t} = \left(1 + M^2 \tfrac{\gamma-1}{2}\right)^{-1}$$

$$\frac{\rho}{\rho_t} = \left(1 + M^2 \tfrac{\gamma-1}{2}\right)^{-\frac{1}{\gamma-1}}$$

Since Mach number is required for all other parameters, the area equation is rewritten as a symbolic equation and the value is solved for in MATLAB using the symbolic toolbox library. Since two solutions can be formed, one solution corresponds to the converging section and the other corresponds to the diverging section. For $\gamma=1.4$ and A/A* as rA, the equation can be written as:

$$f(M, rA) =$$

$$\left(\frac{M^2}{5} + 1\right)^3 - \frac{216\,M\,rA}{125}$$

The equation can be solved using the vpa solve method to get Mach number for any given area change. The actual equation when solved gives 6 solutions among which only 2 are real in general, where one solution is less than 1 and another greater than 1 to denote the converging and diverging sections.

```
M_sol =
```

$$\begin{pmatrix} 0.59024876098845097803135562613058 \\ 1.5341497672017611068224700524926 \\ -1.8055249439136685767965634942756 - 2.5192817064251144230534191598943\,i \\ -1.8055249439136685767965634942756 + 2.5192817064251144230534191598943\,i \\ 0.74332567981856253436965065496403 + 3.7170791795535856536098830562611\,i \\ 0.74332567981856253436965065496403 - 3.7170791795535856536098830562611\,i \end{pmatrix}$$

```
M_sol =
```

$$\begin{pmatrix} 0.59024876098845097803135562613058 \\ 1.5341497672017611068224700524926 \end{pmatrix}$$

<p style="text-align:center; color:blue">Fig: Numeric Solution in MATLAB using vpasolve function</p>

The results and its variation is discussed in the results section.

Code:

```
syms M A f(M)

g = 1.4;
A1 = 0.1; %Throat
A= 0.33;
c = ((g+1)/(g-1))*0.5;

f(M) = (1 + M*M*(g-1)*0.5)^c - M*(A/A1)*(((g+1)/2)^c)

M_sol=vpasolve(f,M)

M_sol = M_sol(imag(M_sol)==0);

x=-0.5:0.02:0.5;
M_arr = x*0;
A_arr = 0.1 + x.*x

for i = 1:length(x)

    A = A_arr(i);
    f(M) = (1 + M*M*(g-1)*0.5)^c - M*(A/A1)*(((g+1)/2)^c);
    M_sol = vpasolve(f,M);
    M_sol = M_sol(imag(M_sol)==0);
```

```
 if x(i)<0
   M_arr(i) = M_sol(M_sol<1);
   else
   Y = M_sol(M_sol>=1);
   M_arr(i) = Y(1);
   end
end

rho_r = (1 + (M_arr.^2)*((g-1)/2)).^(1/(1-g));
T_r = (1 + (M_arr.^2)*((g-1)/2)).^-1;
P_r = (1 + (M_arr.^2)*((g-1)/2)).^(g/(1-g));
```

3.2 Discrete Analysis of Mach Variation (1D Quasi static analysis)

In the discrete analysis of Mach variation within the quasi-1D flow method, governing equations for mass conservation and momentum in the x-direction are employed and discretized using the finite volume method. The conservation of mass ensures constant mass flow through each element, while the momentum equation considers both velocity and pressure gradients along the x-direction. The Mach number, a dimensionless quantity expressing the ratio of fluid velocity to local speed of sound, is calculated at each control volume face. This is solved within Ansys fluent and is commonly used to verify the final solutions from the 2D CFD.

3.1.1. Governing Equations & Discretization:

Conservation of Mass:

The governing equation for the conservation of mass along the quasi-1D flow can be expressed as:

$$\frac{\partial(\rho A)}{\partial x} = 0$$

This equation ensures that the mass flow rate through each element remains constant. To discretize this equation, we use the finite volume method, where the mass flow rate at the faces of each control volume is related to the density (ρ) and cross-sectional area (A):

$$\frac{\rho_{i+1/2} A_{i+1/2} - \rho_{i-1/2} A_{i-1/2}}{\Delta x} = 0$$

Here, $p_{i+1/2}$ & $p_{i-1/2}$ represent the density at the faces between control volumes i and i+1 and i and i−1 respectively, and Δx is the distance between control volume centers.

Conservation of Momentum (x-direction):

The governing equation for the conservation of momentum in the x-direction is:

$$\frac{\partial(\rho u A)}{\partial x} + \frac{\partial P}{\partial x} = 0$$

To discretize this equation, we again use the finite volume method. The momentum equation is discretized as follows:

$$\frac{\rho_{i+1/2}u_{i+1/2}A_{i+1/2} - \rho_{i-1/2}u_{i-1/2}A_{i-1/2}}{\Delta x} + \frac{P_{i+1} - P_i}{\Delta x} = 0$$

This equation accounts for the change in momentum due to both velocity gradients and pressure gradients along the x-direction.

Mach Number:

The Mach number (M) is a dimensionless quantity representing the ratio of the fluid velocity (u) to the local speed of sound (a):

$$M_{i+1/2} = \frac{u_{i+1/2}}{\sqrt{\gamma \frac{P_{i+1}}{\rho_{i+1/2}}}}$$

In the context of the discrete analysis, $M_{i+1/2}$ is calculated based on the velocity, density, and pressure at the faces of each control volume. This equation provides a measure of the flow speed relative to the speed of sound, offering insight into compressibility effects. These governing equations and their discretizations form the foundation for the quasi-1D flow analysis with discrete elements. However, this model requires custom modeling of the functions and numerical methods while only providing 1D equations. A 2D model that is available as part of the ANSYS toolbox is readily available as discussed in the next section.

3.3 Numerical 2-Dimensional CFD Analysis

A 2D analysis is necessary to observe the nuances of localized effects of geometric variations. A 1D model says nothing about temperature variations at the surface, pressure shock variation radially and more.
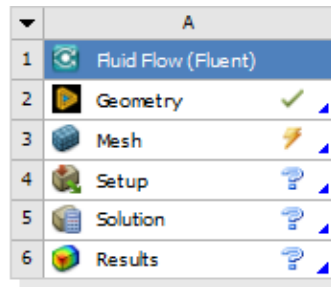
3.3.1 Ansys Fluent Choice

Ansys Fluent is chosen as the Computational Fluid Dynamics (CFD) software for its user-friendly interface and robust documentation. This section outlines the methodology employed for a 2-dimensional CFD analysis using Ansys Fluent. Designed to solve complex fluid flow problems, Fluent offers a comprehensive suite of tools for engineers and researchers engaged in aerodynamics, heat transfer, combustion, and other fluid-related analyses.

Key Features:

1. **User-Friendly Interface**: Fluent boasts an intuitive interface, facilitating ease of use for both novice and experienced users.
2. **Multiphysics Simulations**: Beyond fluid dynamics, Fluent enables multiphysics simulations, incorporating heat transfer, chemical reactions, and other phenomena into analyses.
3. **Meshing Capabilities**: Robust meshing tools ensure the generation of high-quality meshes suitable for various simulation scenarios.
4. **Solver Flexibility**: Fluent supports a range of solvers, including density-based, pressure-based, and segregated solvers, adapting to diverse flow regimes.
5. **Pre- and Post-Processing Tools**: Extensive pre-processing features assist in model setup, while post-processing tools offer in-depth analysis and visualization of simulation results.
6. **Parallel Processing**: Efficient parallel processing capabilities expedite complex simulations, enhancing computational performance.
7. **Turbulence Modeling:** Fluent incorporates various turbulence models to accurately capture turbulent flows in different applications.
8. **Comprehensive Physics Models:** The software provides a wide array of physics models, including compressible flows, chemical reactions, and radiation, ensuring a comprehensive simulation environment.

Alternatives:

1. OpenFOAM

2. COMSOL Multiphysics
3. Hyperworks Fluid Sim

3.3.2 Geometric Modeling Using Ansys Fluent:

SpaceClaim Modeler:

In 2-dimensional CFD analysis using Ansys Fluent, the geometric modeling process is streamlined through the use of SpaceClaim Modeler. This tool proves essential for establishing the boundary of the nozzle by employing a curve equation. This equation allows a method to use optimization toolboxes to determine an optimum curve shape for best performance. Spline and Arc methods for geometric modeling can also be used similarly.
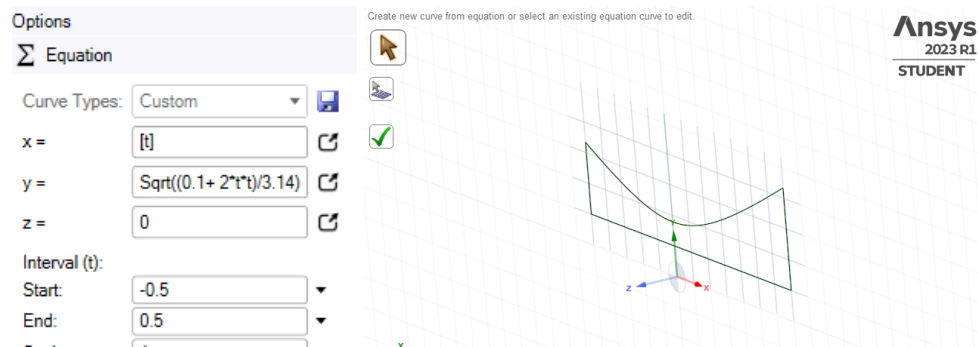
Fig: Spaceclaim modeler with equation editor - Ansys Student

Workflow:

1. **Function Representation**: SpaceClaim Modeler enables the utilization of a function to define the boundary of the nozzle. The curve equation, in this case, is expressed as Area=x^2 +1.
2. **Modeling Range:** The defined boundary is constrained within the range of -0.5m to +0.5m, this is just for the test simulation focus.
3. **Symmetry Considerations:** Due to the cylindrical cross-section of the nozzle, a symmetric solver option is employed. This decision allows modeling only half of the geometry, optimizing computational efficiency.

3.3.3 FEM Meshing

Quadrilateral Meshing for Precision:

In the numerical 2-dimensional CFD analysis workflow using Ansys Fluent, the meshing phase plays a pivotal role in ensuring accuracy and computational efficiency.

The chosen meshing strategy involves the implementation of a quadrilateral mesh for its simplicity and compatibility with Finite Difference Methods (FDM) and Finite Volume Methods (FVM).
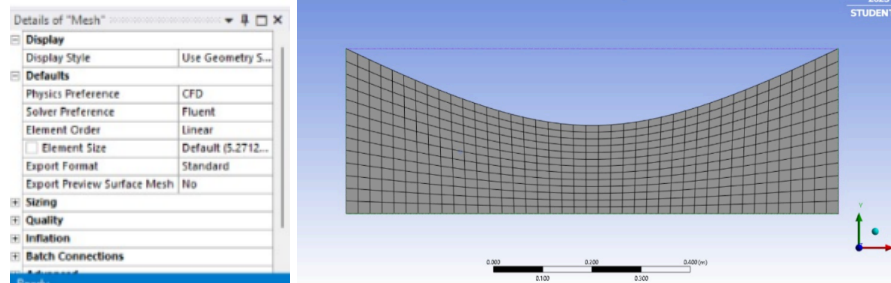

Fig: Quadrilateral Meshing in Ansys

Mesh Type:

The utilization of a quad-based mesh ensures simplicity in the numerical solution process. Quadrilateral elements are favored for their ease of handling in both FDM and FVM methodologies.

Mesh Size Specification:

A maximum mesh size of 0.02 meters is implemented, to balance between computational efficiency and precision. This size specification aligns with the scale of the nozzle geometry, capturing relevant details.

3.3.4 Physics Setup

As part of the numerical 2-dimensional CFD analysis using Ansys Fluent, the physics setup phase involves configuring essential parameters to accurately represent the fluid flow within the nozzle. This step is crucial for obtaining reliable simulation results that align with real-world scenarios.
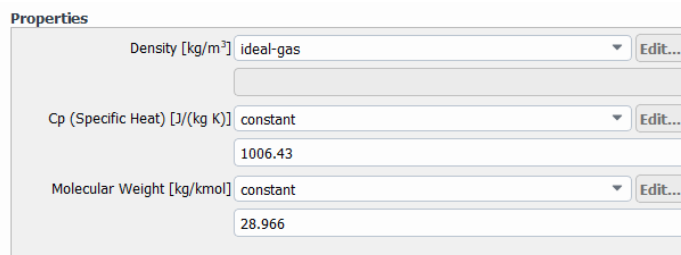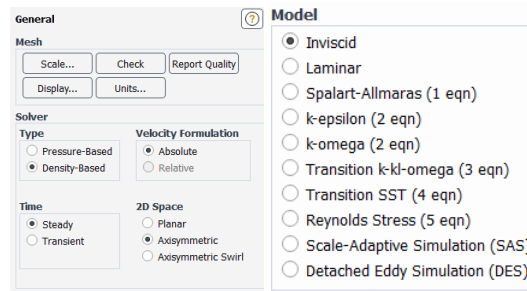
Fig: Images representing physics setup

Physics Setup Elements:

1. **Steady State Configuration**: The choice of a steady-state configuration is motivated by the rapid achievement of a steady flow state. While ignoring initial transient effects, it enhances computational efficiency.
2. **Density-Based Solver**: Ansys Fluent density-based solver is employed, particularly suitable for compressible flows where density variations play a significant role. It allows iterative updates for precise representation of pressure and temperature.
3. **Inviscid Flow Assumption**: Given the high-speed nature of the flow within the nozzle, the physics setup assumes an inviscid flow. This assumption is grounded in the negligible impact of viscous effects, a common characteristic in supersonic flows.
4. **Energy Equation Activation**: The activation of the energy equation is essential for simulations involving heat transfer. This feature allows the consideration of variations in temperature and internal energy.
5. **Ideal Gas Density Model**: To simplify density calculations based on pressure and temperature, the ideal gas density model is applied. This model proves suitable for compressible flows, especially in scenarios with lower gas densities.

The physics setup phase establishes the foundation for a thorough understanding of fluid behavior within the nozzle. By tailoring the simulation parameters to match the compressible and high-speed nature of the flow, Ansys Fluent ensures that the ensuing numerical analysis accurately reflects the physical realities of the nozzle's operation. This meticulous physics setup is crucial for obtaining meaningful insights into the fluid dynamics under consideration.

3.3.5 Boundary Conditions and Solution Criteria

Properly defined boundary conditions mimic real-world scenarios, allowing for a comprehensive understanding of fluid behavior within the nozzle.
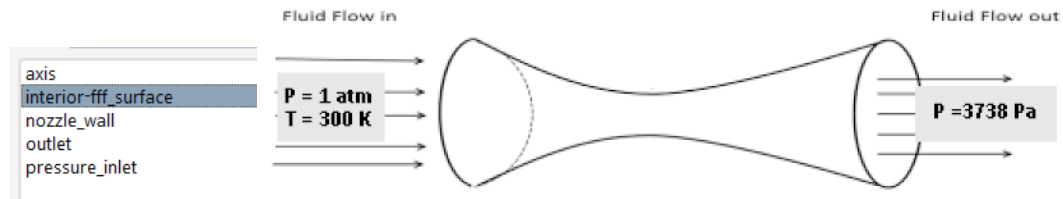
Fig: Boundary Conditions established in the model

1. **Axis Definition**: The designation of the axis as a reference is essential for the axisymmetric solver. This boundary condition sets the foundation for simulating rotational symmetry within the nozzle geometry.
2. **Interior Surface Specification:** Ansys Fluent automatically identifies and defines the interior flow region with the "interior-fff_surface" boundary. This recognition is crucial for accurate computation within the computational domain.
3. **Nozzle Wall Configuration**: The nozzle wall is specified as a rigid and inviscid boundary condition. This choice aligns with the characteristics of high-speed flows, where viscous effects are often negligible, simplifying the simulation process.
4. **Pressure Outlet Definition:** The outlet is configured as a "Pressure Outlet," where the gauge pressure is set to 3700 pascal. This boundary condition represents the exit conditions of the nozzle and influences the overall flow dynamics.
5. **Pressure Inlet Configuration:** The pressure inlet is characterized by a "Pressure Inlet" type. Initial conditions, including gauge pressure and supersonic gauge pressure, are set to 1 atm and 0.9 atm, respectively. The inlet temperature is specified as 300K.
6. **Operating Pressure Assignment**: To ensure consistency in pressure calculations, the reference pressure is set to zero under "Operating Pressure" conditions. This step aligns gauge and absolute pressures, facilitating meaningful pressure differentials across the domain.
7. **Reference Values and Convergence Criteria:** Reference values such as pressure coefficients are solved with respect to the inlet conditions. An overall convergence criterion of 1×10^{-6} has been set, ensuring the simulation reaches a stable and accurate solution.

The primary variables include velocity, temperature, and pressure. Each of these parameters plays a pivotal role in characterizing the behavior of the fluid as it traverses through the nozzle geometry.

4. Results and Analysis

The results from the CFD and elementary analysis are detailed below. Comparisons of the model are primarily done to validate the order and variation but not the actual value itself.

4.1 Elementary Analysis

As per the elementary non-dimensional equations mentioned in section 3.3, the results of the model as plotted in MATLAB utilizing the Symbolic Math toolbox is given below. $A = 0.1 + x^2$ is the considered equation, two variations of this equation are considered. One normally and one offsetted to the left by 0.2. It shall be denoted as curve A & curve B respectively.
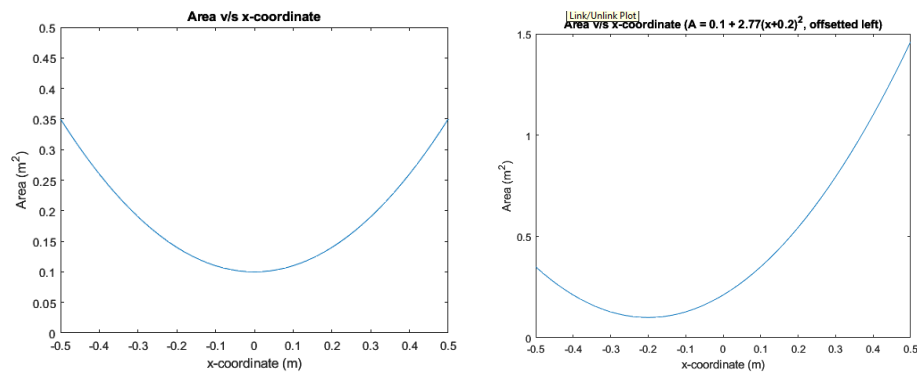
Fig: Area v/s x-coordinate (1+x^2) - Curve A & Curve B

4.1.1 Mach Number Variation

Using the same code, Mach number variation is computed and plotted against x-coordinate. This model can be quickly reiterated by simply changing the equation for Area calculation.
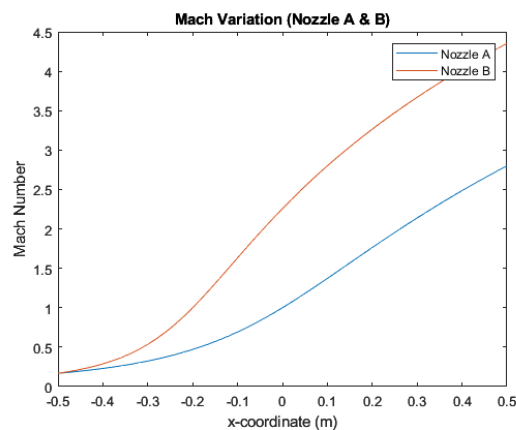
Fig: Mach Variation with x-coordinate of nozzle (A & B)

4.1.2 Thermodynamic Properties Variation

Using the isentropic Mach relations, the temperature, pressure and density variation with Area is also plotted as shown:
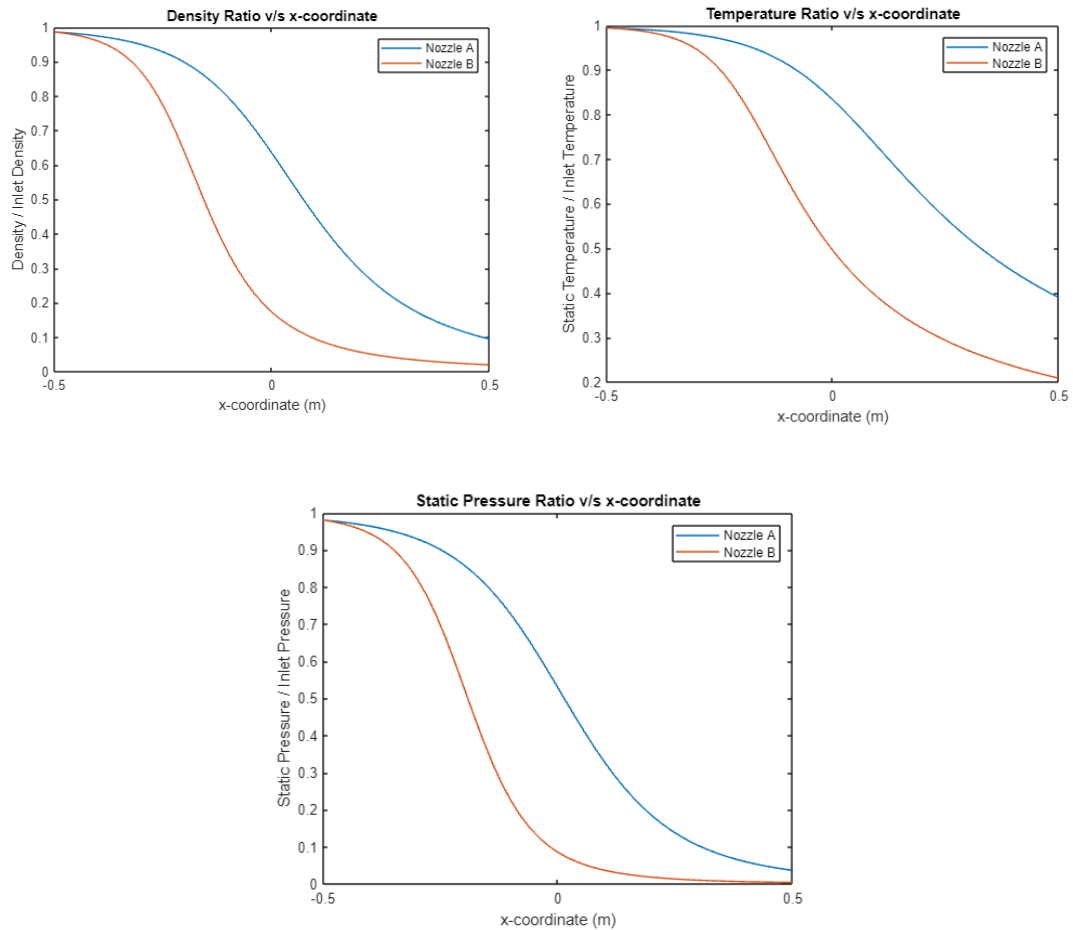
Fig: Graphs plotted showing variation of Density, Pressure and Temperature with x-coordinate

4.2 CFD Analysis - ANSYS Fluent

For CFD Analysis, nozzle A was chosen for validation and hence the following results were obtained. The results were obtained as contour plots and vector/quiver plots.

4.2.1 Velocity Variation:

The above graphics represent Mach number and absolute velocity variation over the nozzle space.
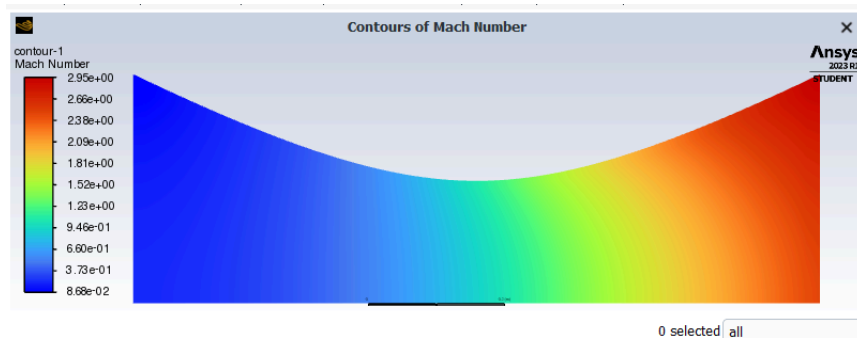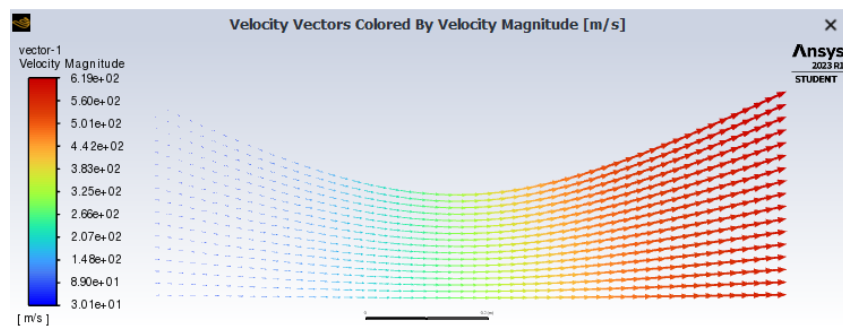
Fig: Mach Number Contour Plot

Fig: Velocity Magnitude Contour Plot

While this graphic represents only a cross section, it does solve over the entire axisymmetric space. It is noted that Mach number is equal to 1 at the throat.

4.2.2 Thermodynamic Properties Variation (CFD)

The CFD Solver also solves for Pressure, Density and Temperature variations. Here the static variations are graphed. The graphics are shown below:
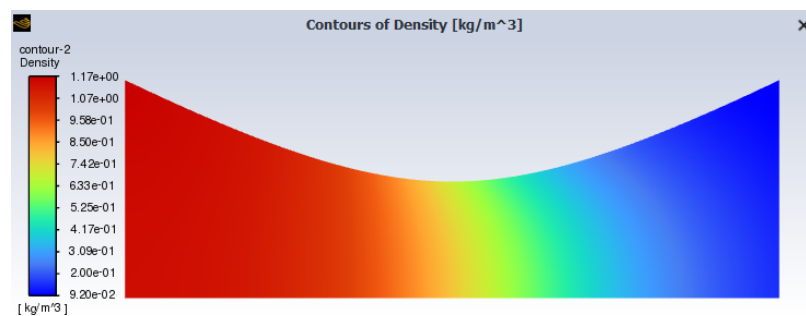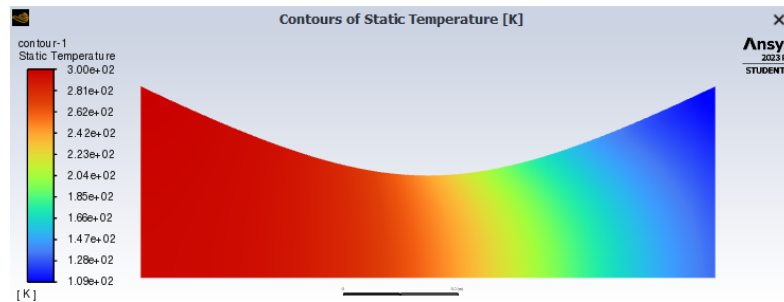

Fig: Density Contour Plot
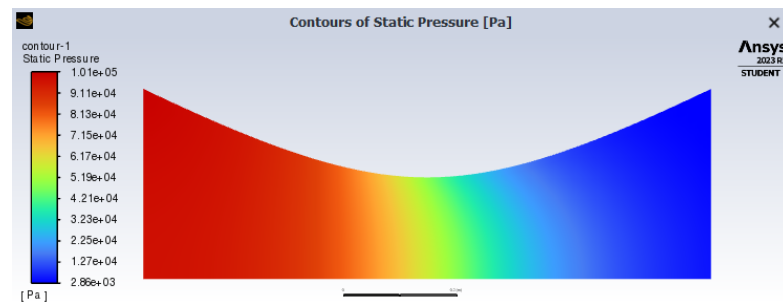
Fig: Static Temperature Contour Plot

Fig: Static Pressure Contour Plot

4.3 Validation of 2D CFD v/s Elementary Model

To corroborate the ideal gas assumption in the CFD analysis, data extracted along the axis from Ansys Fluent is utilized, referencing the x-coordinate. A comparative analysis is then conducted with an elementary model to validate the results. The thermodynamic values of the elementary model are scaled by the inlet values to obtain their absolute counterparts.
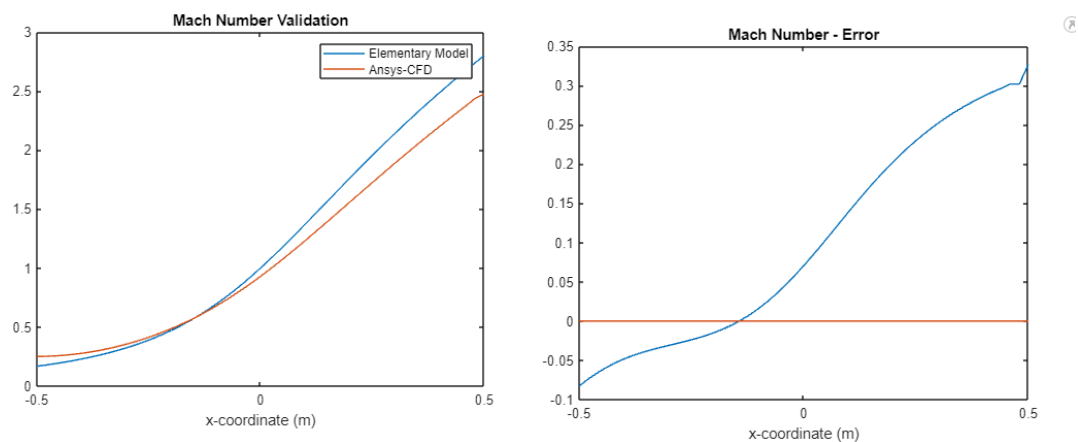
4.3.1 Mach Number

Fig: Mach Number Validation (Elementary v/s Ansys CFD), Error Plot

4.3.2 Thermodynamic Variations

The absolute temperature, pressure and density value difference is shown below.
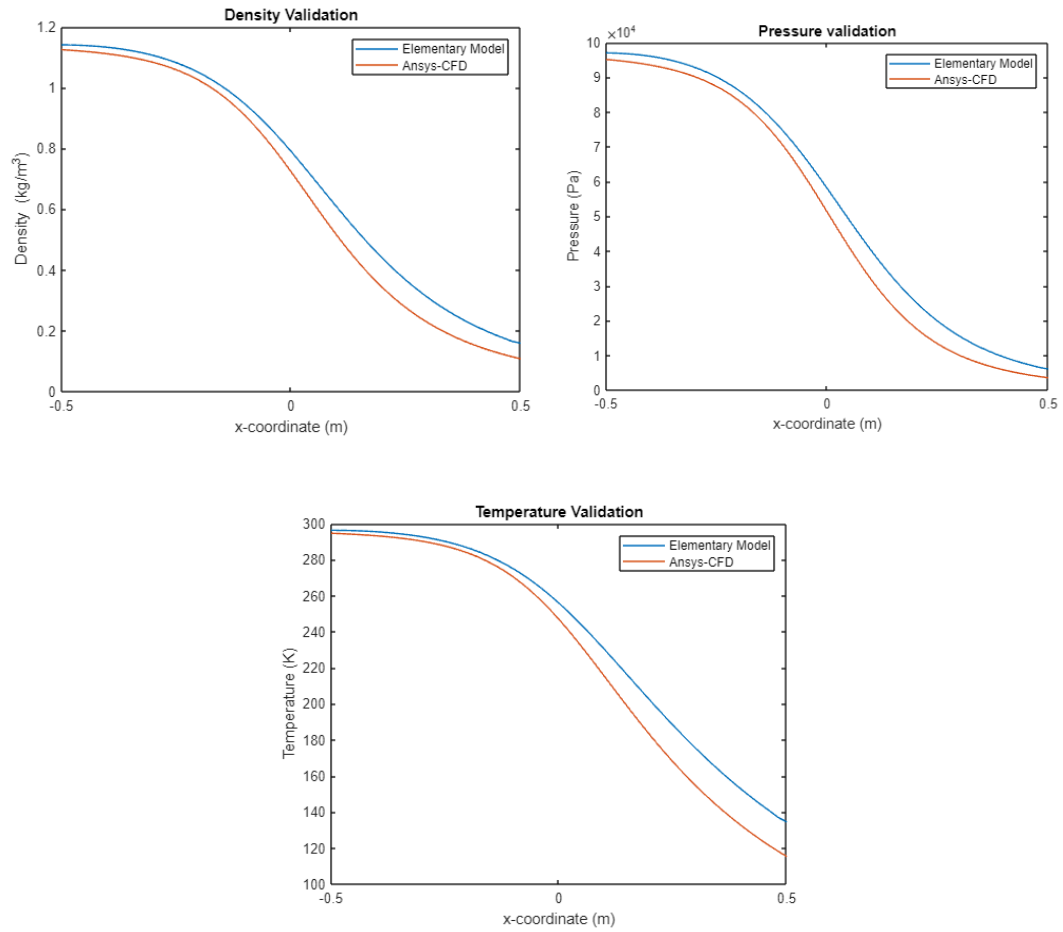

Fig: Thermodynamic Validation (Elementary v/s Ansys CFD)

5. Conclusions

5.1 Inferences

1. **Throat Mach Number Insight**: The consistent attainment of a Mach number of 1 at the nozzle throat is a critical observation, highlighting its importance in the acceleration process.

2. **Isentropic Relations Reliability:** The robust performance of isentropic relations in CFD emphasizes their reliability, making them essential tools for inviscid, ideal gas scenarios.

3. **Mach Number's Shape Dependence**: The Mach number emerges as a shape-dependent phenomenon, allowing for scalability among identical shapes and versatile applications.

4. **Scaling of Thermodynamic Ratios**: Pressure, density, and temperature ratios demonstrate scalability by shape, relying on Mach number, particularly applicable in ideal gas scenarios with a constant gamma.

5. **Insights from Radial Variation in CFD Plots**: CFD plots unveil the radial variation of thermodynamic properties, providing crucial insights for optimal physical design.

6. **Impact of Diverging Section Slope**: Higher slopes in the diverging sections enhance acceleration but require careful mass flow rate management to prevent shocks induced by excessive density drops.

7. **Critical Pressure Drop at Outlet Consideration**: The pressure drop at the outlet is a vital consideration, especially avoiding values lower than external pressure to prevent shocks, as exemplified in the Nozzle B design, particularly suitable for rocket applications where external pressure decreases significantly at higher altitudes.

5.2 Importance & Applications:

1. **Adjustment Mechanisms:** The derived data from CFD models offers insights for adjusting nozzle dimensions in response to changing inlet and outlet conditions in jet mechanisms.

2. **Automated CFD Optimization**: Utilizing a vast dataset generated by automated CFD, nozzle designs for jets and rockets can be systematically optimized for enhanced performance.

3. **Thrust Control in Missiles:** The understanding gained can contribute to the

effective thrust control of missiles, ensuring optimal performance in various conditions.

4. **Live Failure Monitoring:** The ability to observe critical pressure and temperature in real-time allows for effective monitoring of material conditions, preventing potential failures.

6. References

1. "Gas Turbine Theory" H.I.H. Saravanamuttoo, G.F.C. Rogers, H. Cohen, P.V. Straznicky , A.C. Nix; Pearson, 7th Edition.
2. CFD Analysis on Different Advanced Rocket Nozzles by Munipally Prathibha, M. Satyanarayana Gupta, Simhachalam Naidu.
3. Critical Designing and Flow Analysis of Various Nozzles using CFD Analysis by Prapti Joshi, Tarun Gandhi, Sabiha Parveen, Dept of Aerospace Engg Chandigarh University Gharuan, Mohali-140413.
4. NASA Glenn Research Center. (n.d.). Isentropic Processes and Isentropic Flow Equations. NASA.
https://www.grc.nasa.gov/www/k-12/airplane/isentrop.html
5. MathWorks. (2022, June 22). Next Step Suggestions for Symbolic Workflows in Live Editor. MathWorks.
https://in.mathworks.com/videos/next-step-suggestions-for-symbolic-workflows-in-live-editor-1624395800411.html