Study of Propelling Nozzle Systems

Vamshik M (201ME364) | Anannya Rao - 201ME209 | Amogh Nayak (201ME109)

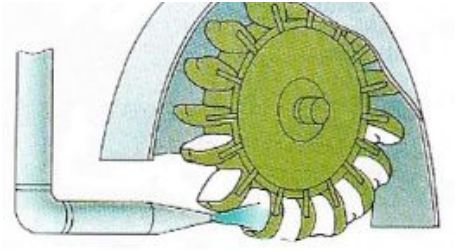
Understanding Nozzles

What is a Nozzle?

- Nozzles are fundamental components designed to control and accelerate the flow of fluids.
- Their primary purpose is to convert the potential energy of a fluid into kinetic energy, achieving specific objectives in various applications.

Diverse Applications:

- Nozzles find applications in a wide array of industries, from **aerospace** to **manufacturing**, where controlling and directing fluid flow is essential.
- Notable applications include propulsion systems, **fuel injection, spray systems, and more.**



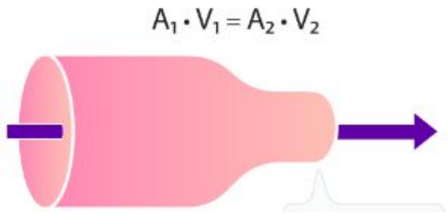
Critical Components in Propulsion:

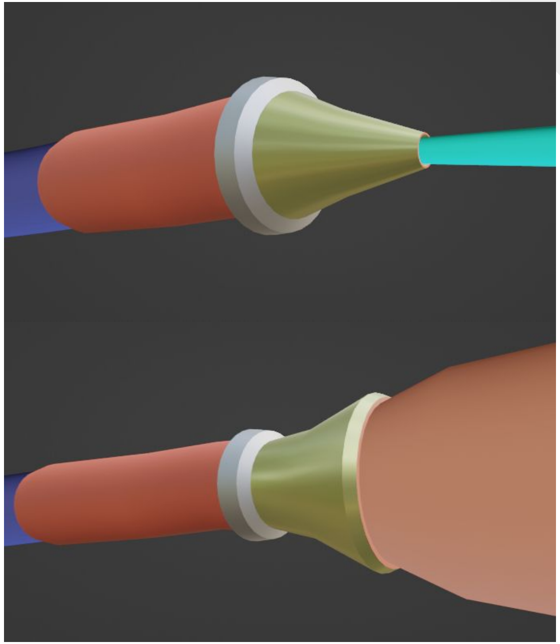
- In the realm of propulsion, nozzles play a crucial role in optimizing thrust.
- They are integral to the performance of jet engines, rocket motors, and other propulsion systems, enabling controlled and efficient fluid acceleration.

Incompressible Flow

Définition:

- In incompressible flow, Change in Pressure → No change in density
- Applicable to most liquids and also gases at low speeds (much below Mach 1)





Advantages of Incompressible Nozzles:

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

Limitations: 1. Inapplicability at High Speeds 2. No Consideration of Shock Waves 3. Limited Applicability to Gases 4. Choking Phenomenon Ignored

Compressible Flow

Définition:

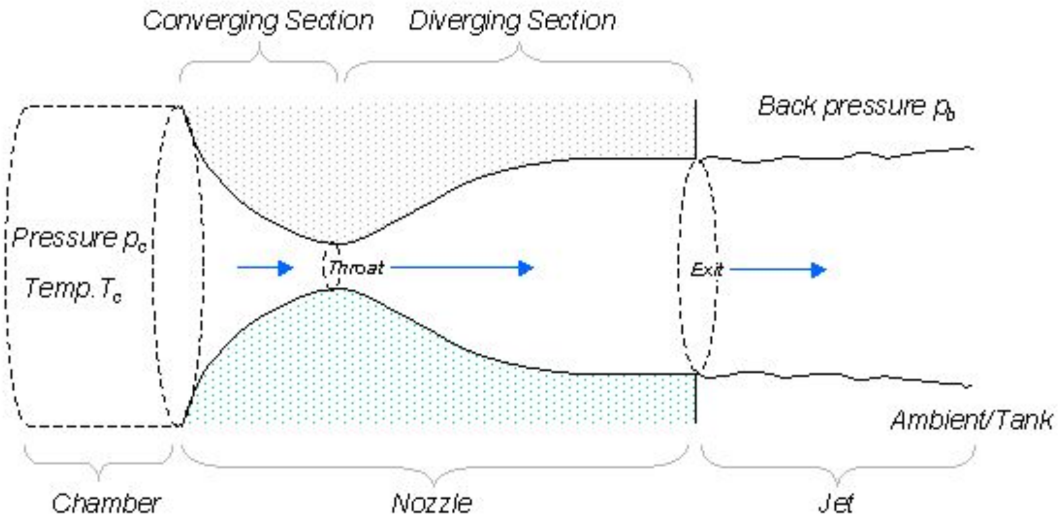
- Compressible flow refers to the behavior of fluids under conditions where changes in pressure and density significantly impact the fluid's properties

Key Characteristics of Flow

- Speed of Sound
- Mach Number

Factors:

- Density effects
- Compressibility
- Heat Capacity
- Températures
- Back Pressures



Compressible Flow – Mach Number

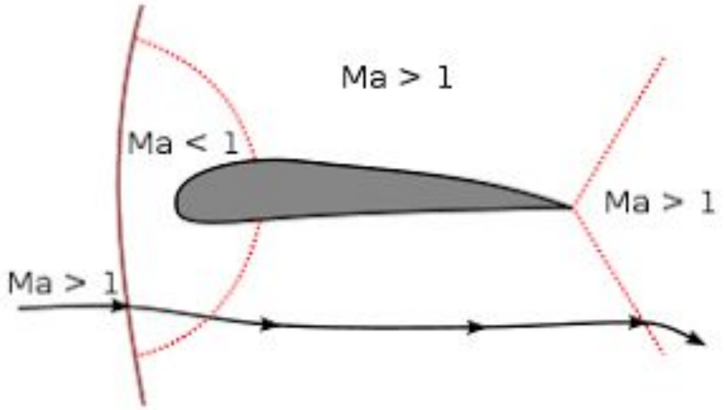
Définition:

- 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
  - Vf →Velocity of fluid
  - Vs→Velocity of Sound at same pressure and temperature

Velocity of Sound

$$v_{sound} = \sqrt{\frac{\gamma R T}{M}} \quad \text{where}$$

γ = adiabatic constant
R = gas constant
M = molecular mass of gas
T = absolute temperature



Catégories:

- Subsonic (M<1)
- Transonic (M=1)
- Supersonic (M>1)
- Hypersonic (M>>1)

Converging – Diverging Nozzle – 01

Continuity Equation:

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

$$\frac{d\rho}{\rho} + \frac{dV}{V} + \frac{dA}{A} = 0$$

Momentum Conservation:

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

Isentropic Flow:

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

Governing Equation

Combining the Isentropic flow Equation, Mach number & Momentum conservation, we obtain:

$$-M^2 \frac{dV}{V} = \frac{d\rho}{\rho}$$

By obtaining density differential from mass continuity Equation:

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

This is the governing equation for a compressible flow system without any external force

Unveiling the Choking Phenomenon

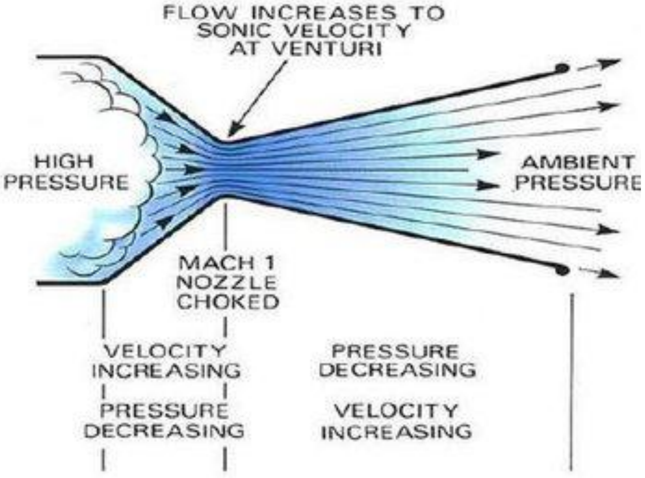
Occurrence:

- 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 represents a critical point in nozzle flow where the flow becomes "choked" or constrained.
- Choking typically occurs at the throat of a converging-diverging nozzle.
- The throat is the point of maximum constriction in the nozzle geometry.

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

Critical Pressure ratio (Exit:Stagnation):

$$\frac{P_e}{P_0} = \left(\frac{2}{\gamma+1}\right)^{\frac{\gamma}{\gamma-1}}$$



Implications:

- Hence further reduction in area has no effect on velocity
- As per the governing equation, at this point for an increase in V, **the area must be increased.**

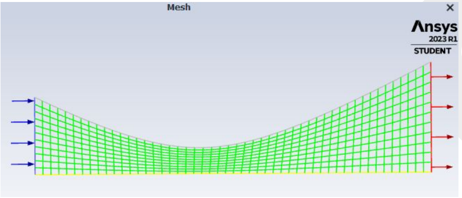
Now, we shall observe this phenomenon via numerical simulation.....

Numerical Analysis

For numerical analysis, various software have been made, for our case, we shall be using **Ansys - Fluent** due to its simplicity and ready documentation.

Essentials of Numerical Analysis (CFD)

Governing Equations:



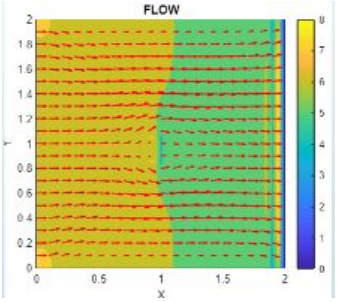
- Continuity Equation
- Momentum Conservation (Usually Navier Stokes compressible form)
- Boundary condition equations (Viscous effects)

Discretization:

$\frac{\partial u}{\partial x}$	$\frac{u^n_{i,j} - u^n_{i-1,j}}{\Delta x}$
$\frac{\partial u}{\partial t}$	$\frac{u^{n+1}_{i,j} - u^n_{i,j}}{\Delta t}$
$\frac{\partial^2 u}{\partial x^2}$	$\frac{u^n_{i+1,j} - 2u^n_{i,j} + u^n_{i-1,j}}{\Delta x^2}$

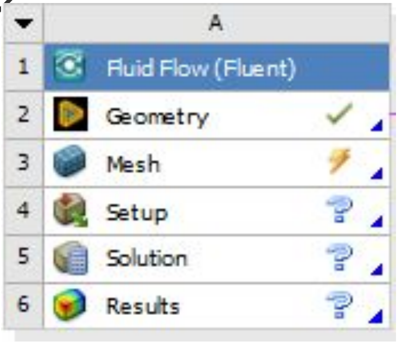
- The above governing equations are usually in differential form which cannot be analytically solved in the computer
- So discrete elements are made and the equations are applied in their crude-simplified form.

Solving:



- Various numerical methods like finite difference, finite volume etc.
- Quantitative methods like Lattice Boltzmann Method, Discontinuous Gallerkin Method are also employed.

Numerical Analysis Steps



Geometry Modeling
Establishing the boundary of the nozzle via curve equation

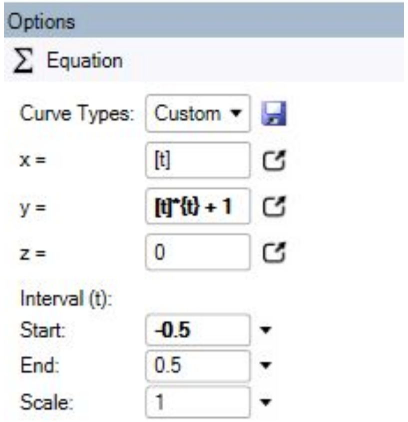
Meshing
Establishing a good quality mesh (discrete elements for solving the model)

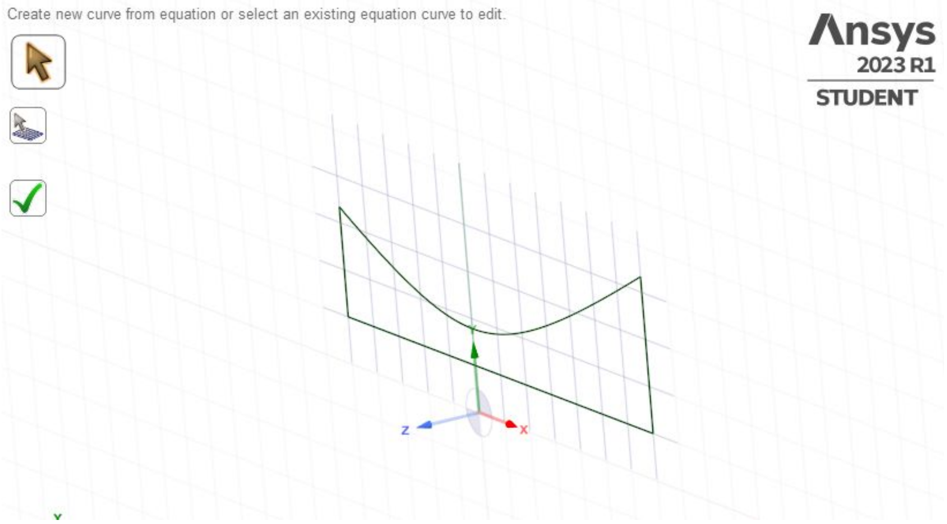
Physics setup & Solving
The appropriate boundary conditions, materials and flow parameters need to specified

Geometry – Modeling

SpaceClaim Modeler:

- Spaceclaim modeler was used as it provides the ability to utilize a function to represent a curve boundary.
- The boundary was written as Area = x^2 + 1 (y = sqrt((x^2 + 1)/pi))
- This model was written within -0.5 to +0.5m
- Due to cylindrical cross-section, only half of the geometry is modelled as a symmetric solver option exists.

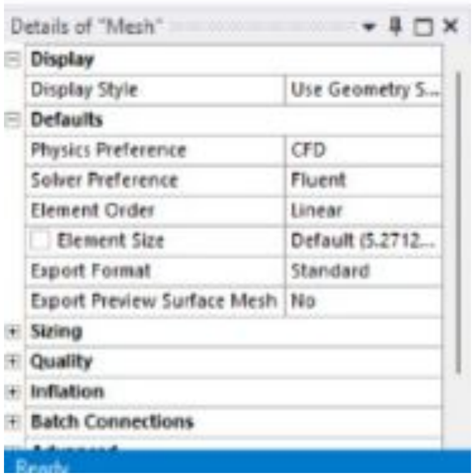


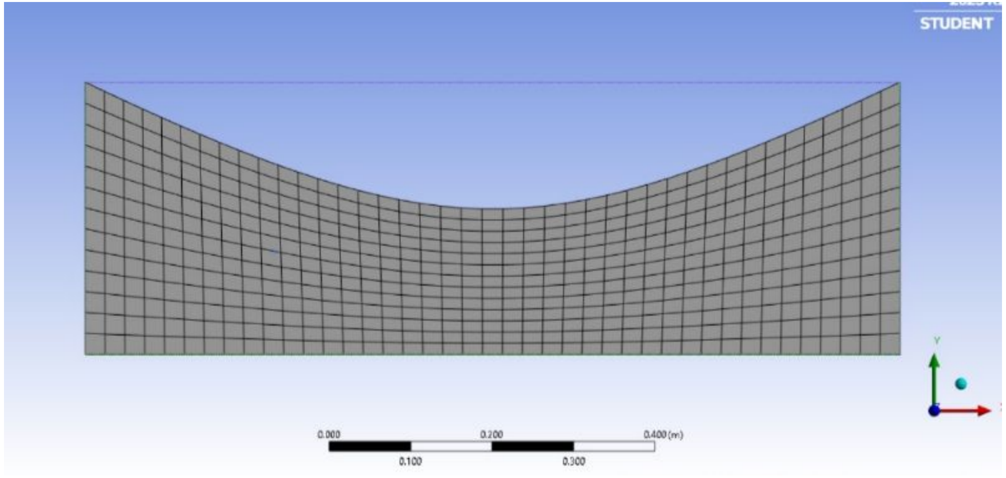


Meshing

Quadrilateral Meshing:

- A quad based mesh was utilized for it’s simplicity in FDM and FVM methods.
- A maximum mesh size of 0.02 m is used.

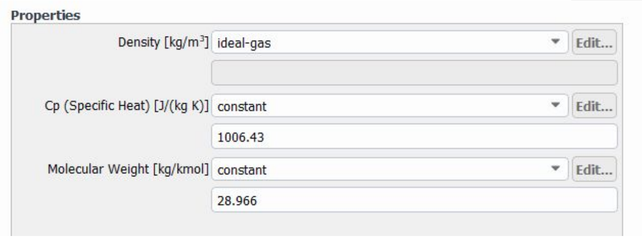


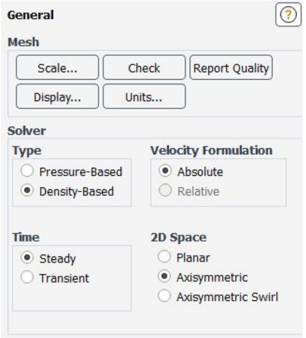


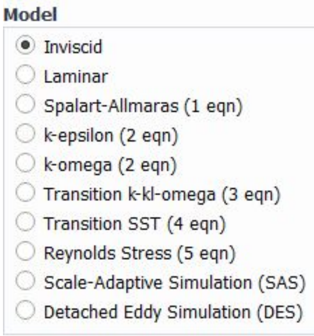
Physics Settings

Settings:

- **Steady State:** The flow achieves steady state in very short duration of time. Also this increasing computation time by ignoring initial effects.
- **Density-Based Solver**: Appropriate for compressible flows where density changes significantly, allowing iterative updates for accurate pressure and temperature representation.
- **Inviscid Flow**: Efficient for scenarios with negligible viscous effects, common in high-speed aerodynamics and compressible flows.
- **Energy Equation Enabled:** Essential for simulations involving heat transfer or combustion, accounting for variations in temperature and internal energy.
- **Ideal Gas Density Model**: Simplifies density calculations based on pressure and temperature, suitable for compressible flows, especially at lower gas densities.







Applications:

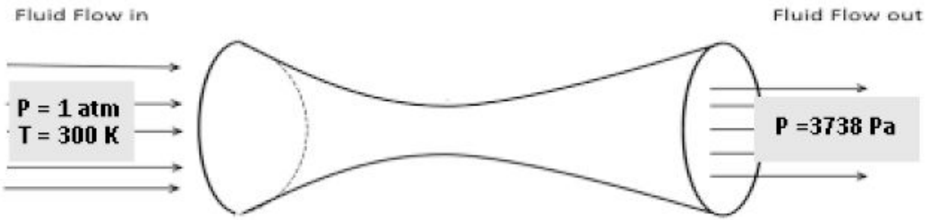
- Aerospace applications for high speed aerodynamics
- Propulsion Systems where compressibility is dominant
- Supersonic, Hypersonic Flow systems

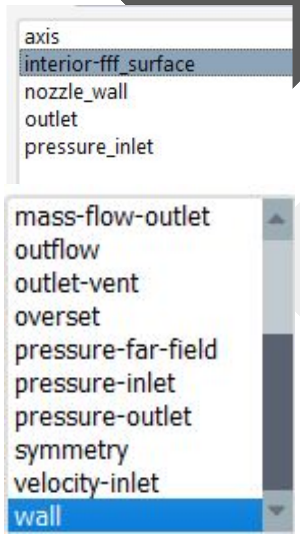
Boundary Conditions

Within Fluent, geometrical boundaries need to identified and named with appropriate pre-defined names, so that the solver can use them to run the analysis, the named selections are described below:

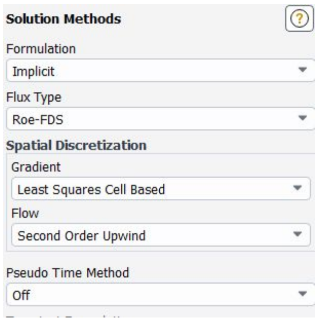
- **Axis:** This has been set to the type "Axis" and is used for the axisymmetric solver reference
- **interior-fff_surface:** This is automatically set by Fluent to describe the inner flow region
- **nozzle_wall**: Has been set to a wall (here rigid - inviscid)
- **outlet**: "Pressure outlet" where the gauge pressure has been set to 3700 pascal.
- **pressure_inlet**: Here, the type is "pressure inlet" and gauge pressure is set as 1 atm and supersonic gauge pressure as 0.9 atm (Velocity-pressure drop). Temperature is set to 300K
- **Operating_pressure**: Since all the pressures are set in absolute terms, this value is set to zero.

Reference Values like pressure coefficients are solved with respect to the inlet here. Also an overall convergence criteria of 10^-6 has been set.





List of boundary conditions



Solution Method

Solutions

Velocity

Velocity will be visualized in terms of Mach number for better understanding

Pressure

Variation of pressure along the axis shall be observed

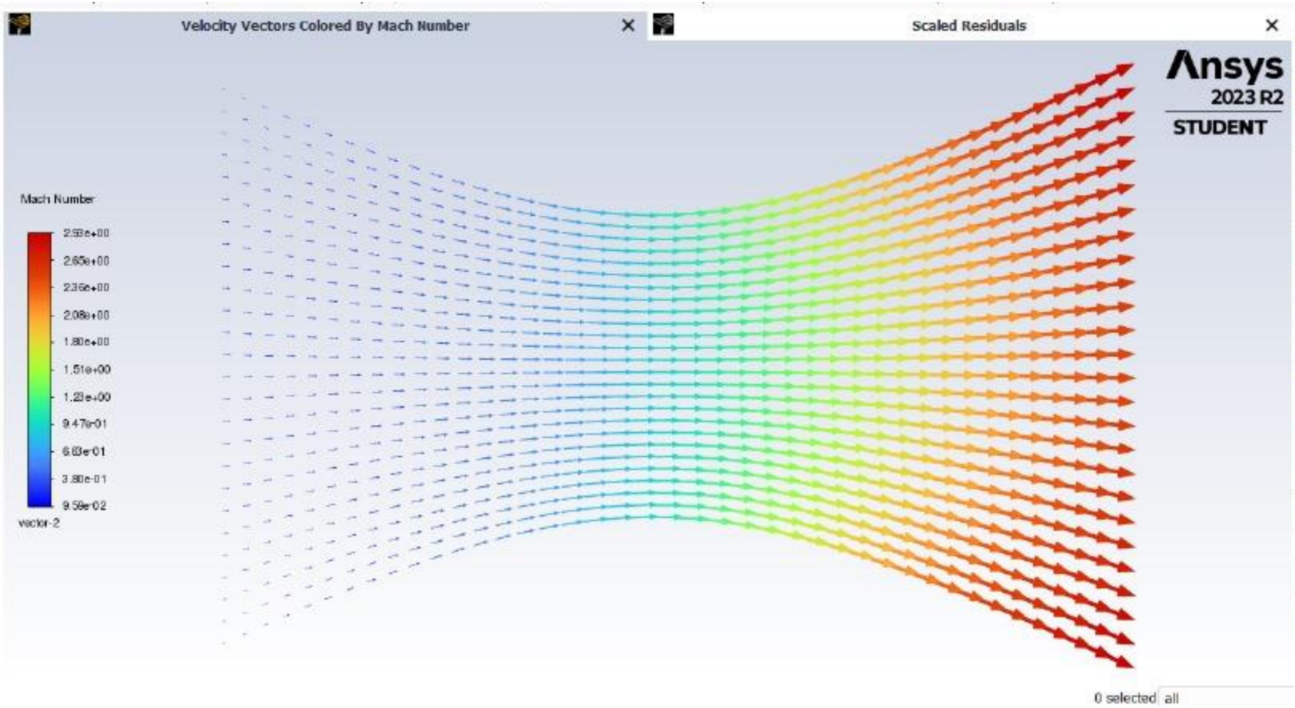
Température

Temperature along the axis shall be monitored

Velocity

Why Analyze?

- **Optimizing Thrust:** Different Mach number regimes affect the efficiency of the nozzle, flow characteristics helps engineers design nozzles that achieve maximum thrust for specific operating conditions.
- **Choked Flow Prediction:** Simulating the Mach number is crucial for predicting choked flow conditions in a converging-diverging nozzle, helps in designing nozzles to operate efficiently and avoid limitations of choked



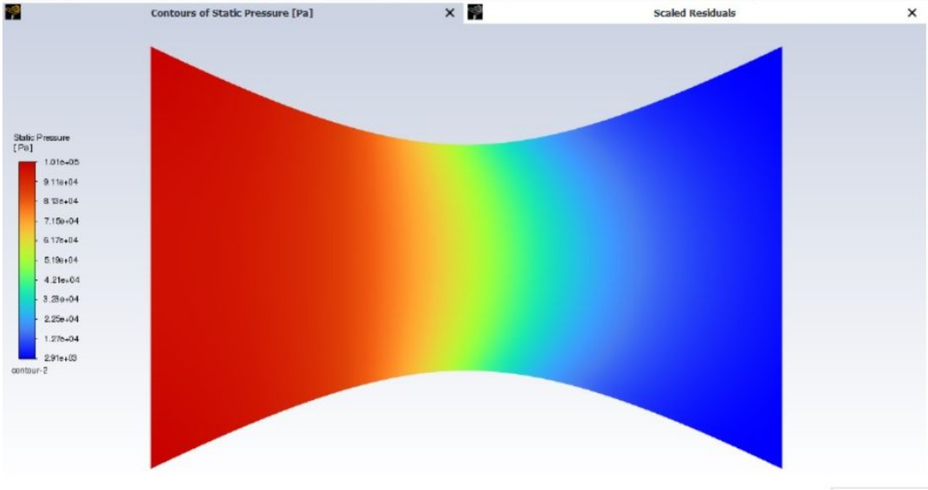
- **As shown, the flow accelerates from M = 0.09 to M = 2.7**
- **The throat area has a Mach number of 1 as predicted.**

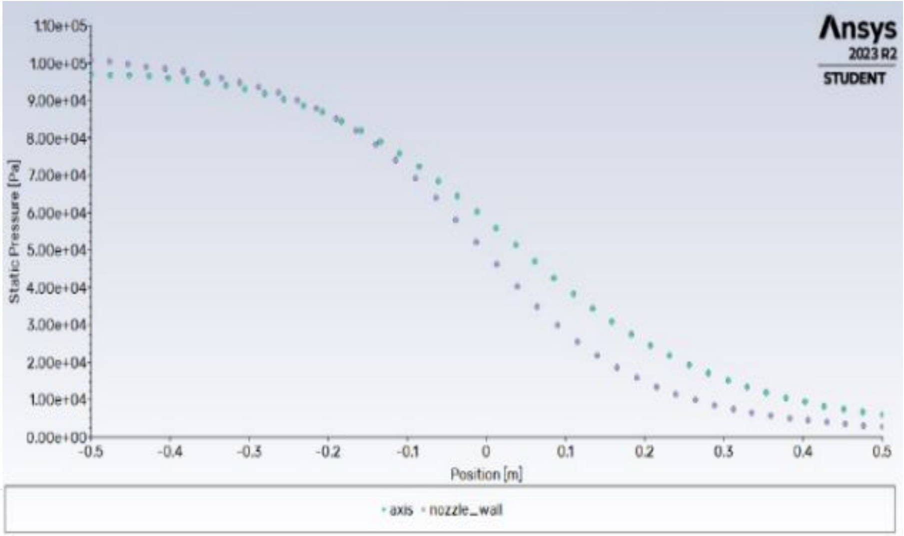
Note: It is important to note that this numerical analysis was done via generic Navier stokes equations but still attains similar solutions.

Pressure

Why Analyze?

- **Shock Wave Prediction**: Pressure analysis is crucial for predicting and managing shock waves within the nozzle, ensuring structural integrity and optimizing performance.
- **Pressure Field Validation**: Pressure analysis serves as a key tool for validating nozzle design by comparing simulation results with experimental data, ensuring accurate representation of the pressure field.
- **Structural Considerations**: Examining pressure is essential for evaluating the structural integrity of the nozzle, especially in high-pressure environments, guiding design decisions to meet safety and reliability.

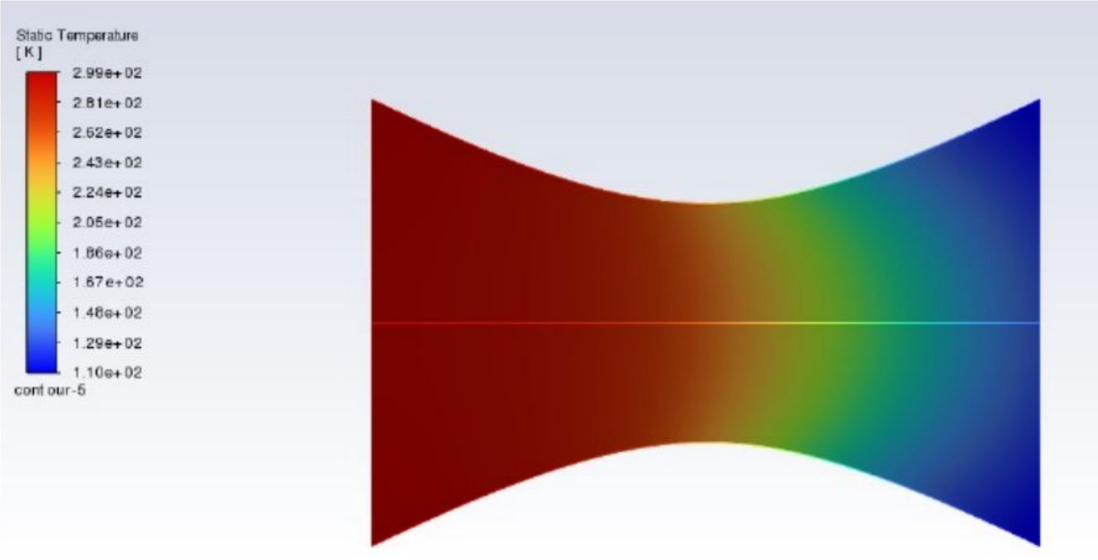




Température

Why Analyze?

- **Combustion Efficiency**: Analyzing temperature is critical for optimizing combustion efficiency in propulsion systems, ensuring the temperature profile supports efficient fuel combustion.
- **Material Durability:** Temperature analysis is essential for evaluating material durability in nozzles, helping engineers select materials that can withstand high-temperature conditions and maintain structural integrity.
- **Thermal Stress Prediction:** Understanding temperature variations aids in predicting thermal stresses within the nozzle, allowing for design adjustments to mitigate potential issues related to thermal expansion and contraction.



Scope of CFD Analysis for nozzles

01.

Complex stage nozzles

Analysis of multi stage nozzles or complex shapes

02.

Live Failure Monitoring

Onboard computer can track possibility of shock waves or temperature spikes.

03.

Optimization of Design

The geometry/curve coefficients can be optimized for maximum thrust given an inlet state

04.

On-board Nozzle Control

A database model developed can be used for controller programming

05.

Thrust Control for Rockets

Rocket nozzles are generally controlled in order to maintain a thrust value

THANK YOU