A Report
On

Gas Expansion in Compressible Flow: Pressure Variation Analysis

Submitted in partial fulfilment of the requirements
of the degree of

Bachelor of Technology

Submitted by

Amandeep Singh -(22117020)
Amar Choudhary -(22117021)
Mahanti Ajay Babu -(22117079)
Karan Kumar Singh -(21117057)

Under the guidance of

Prof. Ankit Bansal

Department of Mechanical Engineering Indian Institute of Technology Roorkee 2025

Acknowledgement

We want to thank our teachers, Prof. Ankit Bansal, for their amazing support on our project about " Gas Expansion in Compressible Flow: Pressure Variation Analysis." Their helpful suggestions really guided us in creating this report.

We are very grateful for their patience and constant encouragement, which improved our work a lot. Through this project, we learned a great deal about Gas Expansion in Compressible Flow.

Finishing this report makes us feel proud. We also want to thank our friends and classmates, who gave us great feedback and ideas that helped make our project better.

Finally, we thank God for giving us the strength and determination to complete this project successfully.

Table of Contents

1. Introduction

1.1 Background and Objectives

Compressible flow through nozzles is a fundamental aspect of many engineering applications, including propulsion systems, power generation, and process industries. The behavior of gas as it expands through a divergent nozzle is of particular interest due to its complex thermodynamic and fluid dynamic characteristics. This project focuses on analyzing the pressure variation and flow behavior of a gas with a molecular weight of 10 expanding through a divergent nozzle.

The primary objectives of this study are:

- To determine the pressure distribution along the nozzle in both transient and steady-state conditions
- To analyze velocity profiles throughout the nozzle
- To understand the relationship between nozzle geometry and flow characteristics
- To validate theoretical concepts of compressible flow with computational simulation results

1.2 Theoretical Framework

In a divergent nozzle, as the cross-sectional area increases, the velocity of a subsonic flow decreases while the pressure increases, following the principles of the conservation of mass, momentum, and energy. For supersonic flow, the opposite occurs: velocity increases and pressure decreases with increasing area. The behavior of the gas in the nozzle is governed by the principles of compressible flow, including:

- Continuity equation: $\rho_1 A_1 v_1 = \rho_2 A_2 v_2$
- Energy conservation
- Momentum conservation
- Equation of state for ideal gases: $P = \rho RT/M$

Where ρ is density, A is cross-sectional area, v is velocity, P is pressure, R is the universal gas constant, T is temperature, and M is molecular weight.

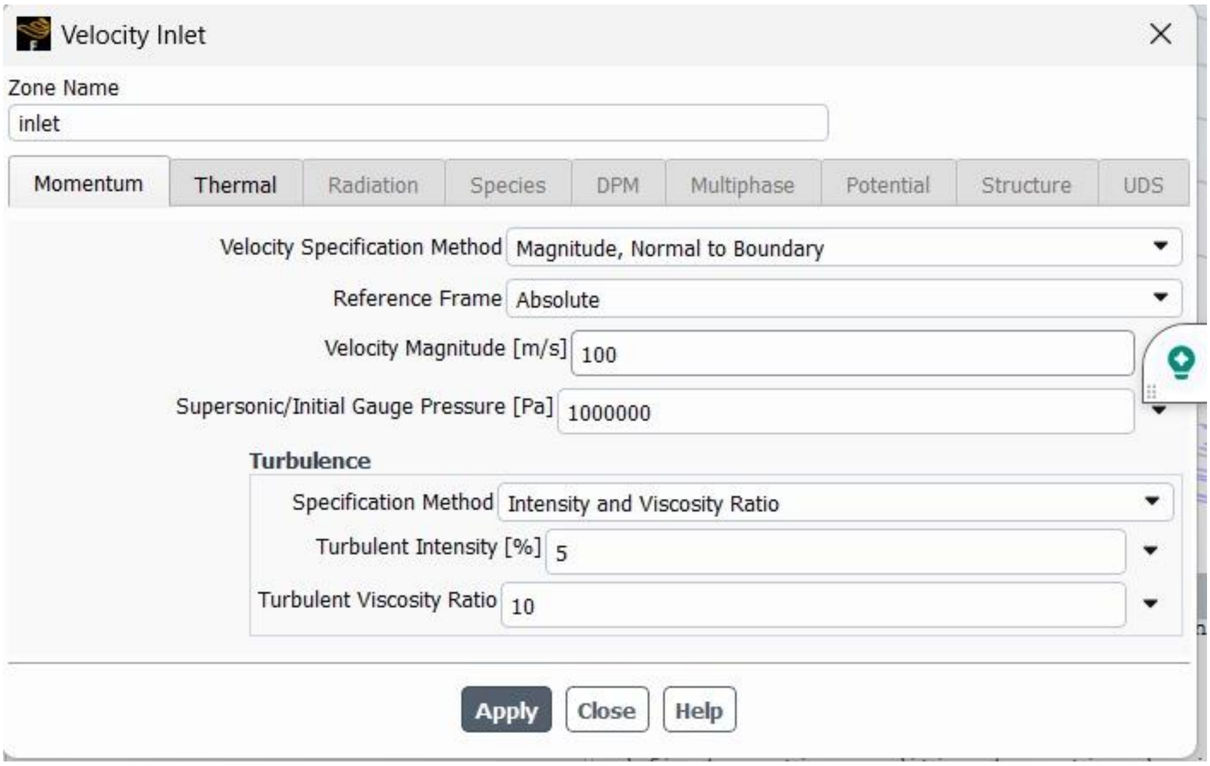
2. Methodology

2.1 Geometric Modeling

The geometry of the divergent nozzle was created using ANSYS SpaceClaim. As shown in Figure 1, the nozzle has the following dimensions:

- Total length (H2): 80 mm
- Inlet diameter (V1): 30 mm
- Outlet diameter (V3): 60 mm

The nozzle was designed as a 2D axisymmetric model to reduce computational requirements while maintaining accuracy. The geometry was created by defining the axial length and radial dimensions, forming a half-profile that is revolved around the central axis to create the complete 3D nozzle.



2.2 Mesh Generation

The computational domain was discretized using a structured mesh with a uniform element size of 0.01 mm. This fine mesh was chosen to capture the complex flow phenomena accurately, particularly near the walls and in regions of high-pressure gradients. The mesh quality was verified to ensure orthogonality, aspect ratio, and skewness were within acceptable limits for accurate CFD calculations.

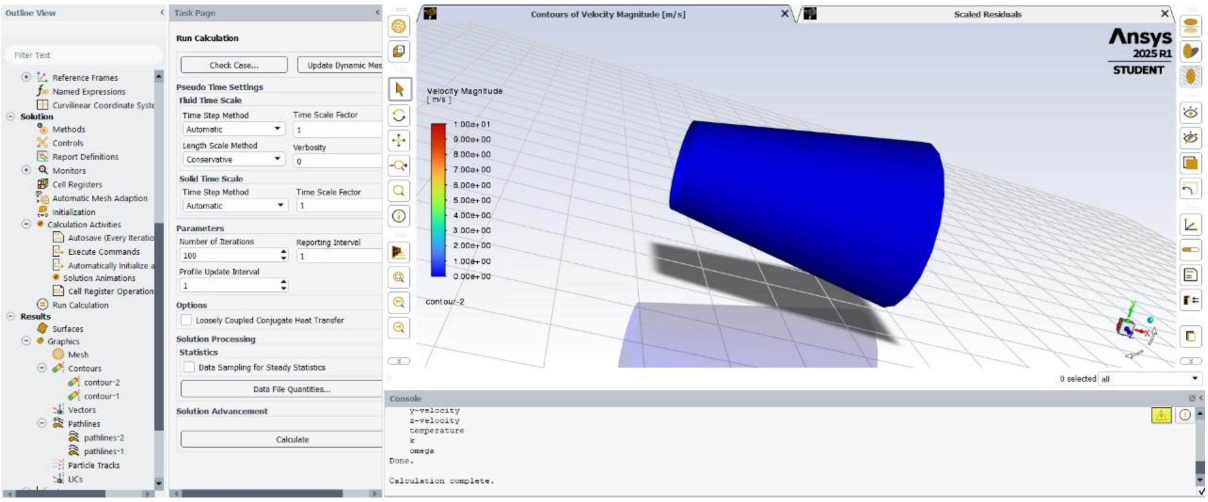
The mesh was refined in critical regions, particularly near the inlet and along the diverging section, to capture boundary layer effects and flow acceleration accurately. Inflation layers were added near the walls to resolve the boundary layer properly.

2.3 Material Properties and Boundary Conditions

Material Properties

As shown in Figure 2, the working fluid was defined with the following properties:

- Material: Gas with molecular weight of 10 kg/kmol
- Density model: Ideal gas law
- Specific heat capacity (Cp): 1006.43 J/(kg·K) (constant)

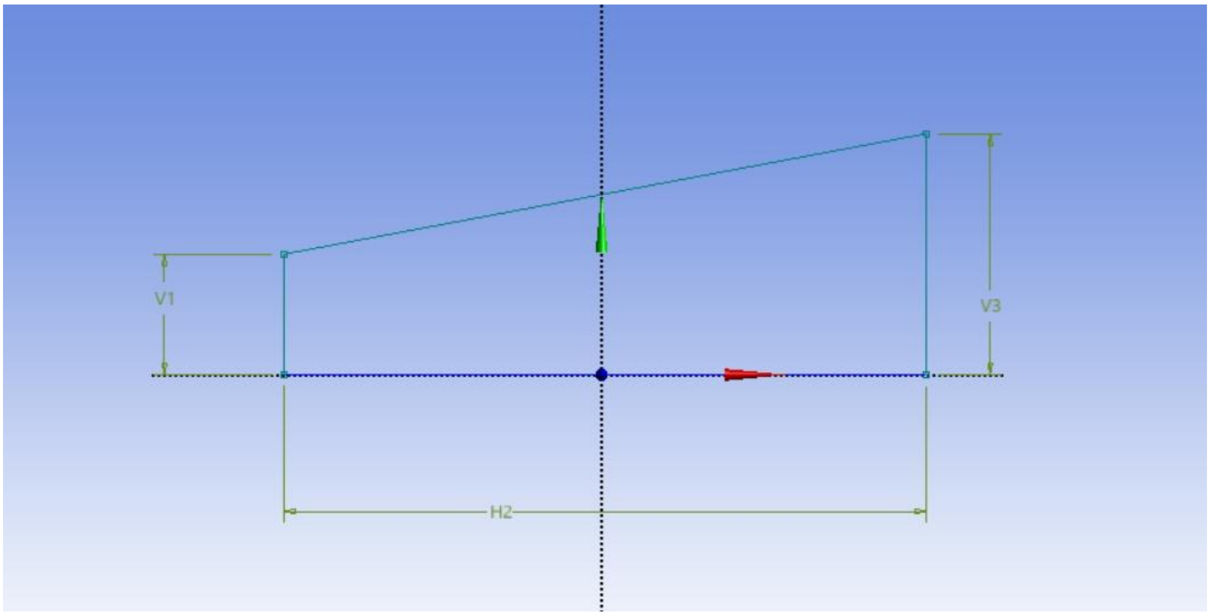
- Thermal conductivity: 0.0242 W/(m·K) (constant)
- Viscosity: 1.7894×10^{-5} kg/(m·s) (constant)
- 

The nozzle wall material was defined as steel, though its thermal properties were not critical for this simulation as the focus was on the flow characteristics rather than heat transfer.

Boundary Conditions

Inlet boundary conditions (as shown in Figure 3):

- Velocity inlet with a magnitude of 100 m/s normal to the boundary
- Initial gauge pressure: 1,000,000 Pa (10 bar)
- Turbulence specification method: Intensity and viscosity ratio
- Turbulent intensity: 5%
- Turbulent viscosity ratio: 10

- 

Outlet boundary conditions:

- Pressure outlet with ambient pressure (0 gauge pressure)
- Backflow turbulent intensity: 5%
- Backflow turbulent viscosity ratio: 10

Wall boundary conditions:

- No-slip condition at the walls
- Adiabatic walls (no heat transfer)

3. Solution Setup and Processing

3.1 Solver Settings

The simulation was configured with the following solver settings:

- Pressure-based solver (as the flow is compressible but not highly supersonic)
- Axisymmetric model
- Steady-state and transient analyses were conducted separately
- Energy equation enabled to account for temperature variations
- k-ε turbulence model with standard wall functions

3.2 Solution Methods

The following numerical schemes were employed:

- Pressure-velocity coupling: SIMPLE algorithm
- Gradient discretization: Least Squares Cell Based
- Pressure discretization: Second Order
- Momentum discretization: Second Order Upwind
- Turbulent kinetic energy and dissipation rate: Second Order Upwind
- Energy: Second Order Upwind

3.3 Solution Controls and Initialization

The solution controls were set as follows:

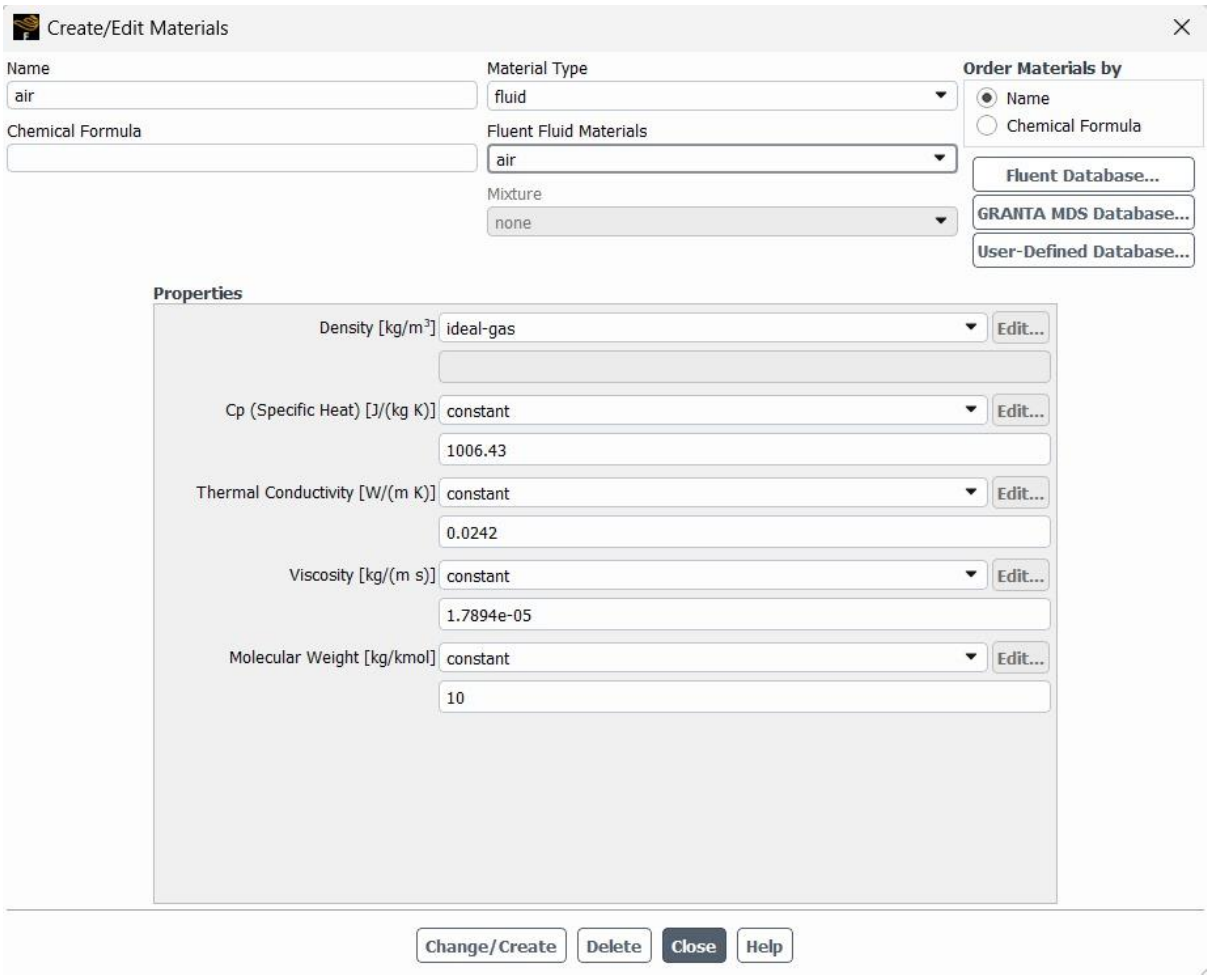
- Pressure: 0.3 (under-relaxation factor)
- Momentum: 0.7
- Turbulent kinetic energy: 0.8
- Turbulent dissipation rate: 0.8
- Turbulent viscosity: 1.0
- Energy: 1.0

The solution was initialized using the hybrid initialization method, which intelligently solves the Laplace equation to produce a velocity and pressure field suitable for starting the calculation.

3.4 Calculation Execution

As shown in Figure 4, the calculation settings included:

- Number of iterations for steady-state: 100
- Reporting interval: 1
- Profile update interval: 1

- 

For the transient analysis:

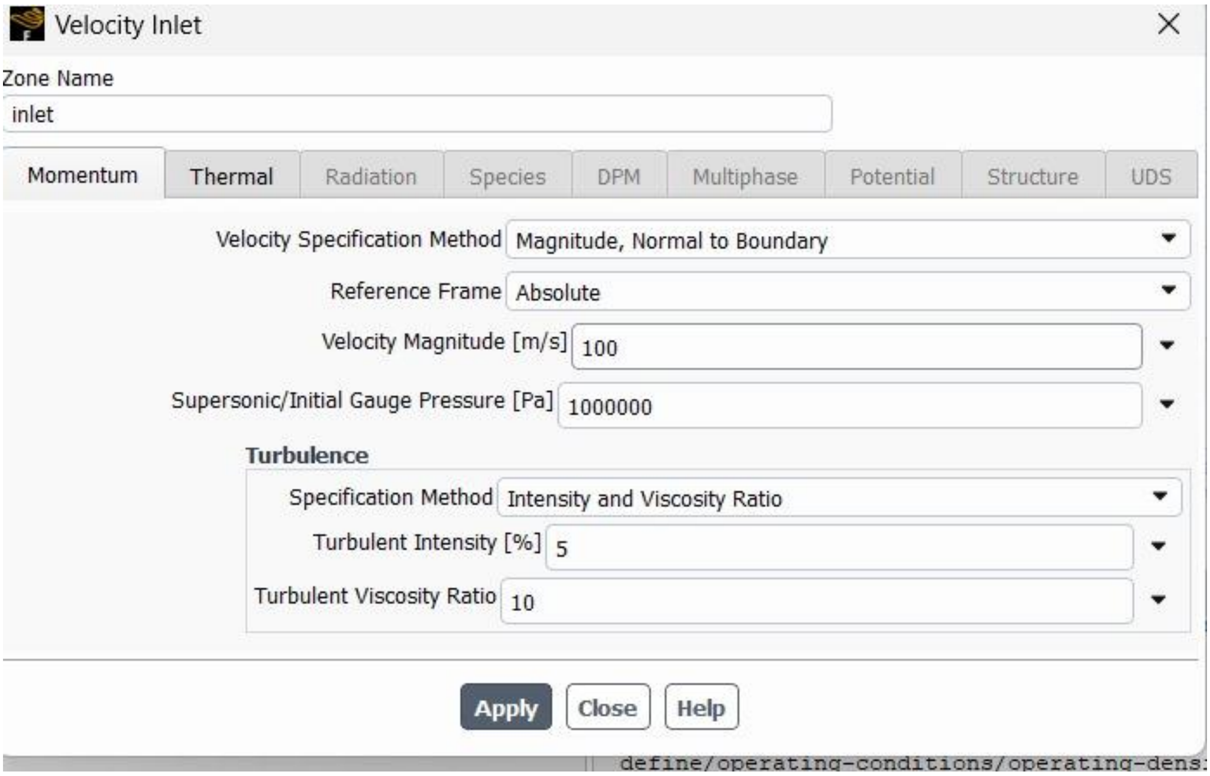
- Time step size: Automatic (based on Courant number)
- Maximum iterations per time step: 20
- Total simulation time: Sufficient to reach fully developed flow

The simulation was monitored for convergence based on residuals of continuity, velocity components, energy, and turbulence parameters, with a convergence criterion of 10^{-3} for all equations except energy (10^{-6}).

4. Results and Discussion

4.1 Velocity Distribution

Figure 5 shows the velocity contour plot for the steady-state solution. The velocity magnitude ranges from 0.4118 m/s to 10 m/s, with a distinct pattern of velocity distribution throughout the nozzle.



The velocity contour exhibits the expected behavior for subsonic flow in a divergent nozzle, with the velocity decreasing as the cross-sectional area increases. The highest velocity (red region) is observed at the inlet, where the diameter is smallest (30 mm). As the gas expands through the diverging section, the velocity gradually decreases, indicated by the transition from red to yellow, green, and finally blue regions.

The velocity distribution shows a core region of higher velocity along the centerline of the nozzle, with lower velocities near the walls due to the no-slip boundary condition. This creates a boundary layer effect where velocity gradients are steeper near the walls.

4.2 Pressure Distribution

The pressure distribution along the centerline of the nozzle was analyzed in both steady-state and transient conditions. For subsonic flow in a divergent nozzle, the pressure is expected to increase as the velocity decreases, following the principles of Bernoulli's equation for compressible flow.

In the steady-state analysis, the pressure increases from inlet (10 bar) as the gas expands through the diverging section. This pressure recovery is a characteristic feature of subsonic flow in divergent nozzles and is consistent with theoretical expectations.

For the transient analysis, pressure variations were monitored at a reference point at the center of the nozzle. The pressure at this point stabilizes after an initial transient period, eventually approaching the steady-state value. The initial pressure fluctuations are due to the propagation of pressure waves within the nozzle as the flow develops from the initial conditions.

4.3 Flow Pattern Analysis

The flow pattern within the nozzle shows several important characteristics:

1. The expanding cross-section causes the flow to decelerate in a subsonic regime, converting kinetic energy to pressure energy.
2. Near the walls, the boundary layer development is evident from the velocity gradients. The boundary layer thickens as the flow moves downstream, affecting the effective flow area.
3. The core flow remains relatively uniform, with minimal disturbances or recirculation zones, indicating a well-designed nozzle geometry without flow separation.
4. The gradual expansion angle (from 30 mm to 60 mm over 80 mm length) promotes smooth flow expansion without significant losses due to sudden expansion or flow separation.

5. Parametric Analysis and Validation

5.1 Effect of Inlet Velocity and Pressure

The initial inlet velocity of 100 m/s and pressure of 10 bar create specific flow characteristics within the nozzle. For subsonic flow (Mach number < 1), increasing the inlet velocity would result in a correspondingly higher velocity throughout the nozzle but would maintain the same general pattern of velocity decrease through the diverging section.

The inlet pressure directly influences the pressure distribution throughout the nozzle. With the current setting of 10 bar, the pressure recovery in the divergent section follows the expected theoretical pattern for subsonic flow.

5.2 Comparison with Analytical Solutions

The simulation results can be validated against analytical solutions for isentropic flow through divergent nozzles. For subsonic flow, the relationship between velocity, pressure, and area can be expressed as:

For subsonic flow ($M < 1$):

- As area increases, velocity decreases and pressure increases

The simulation results conform to these theoretical expectations, providing confidence in the accuracy of the CFD analysis.

5.3 Mesh Sensitivity Analysis

Although not explicitly shown in the provided images, a mesh sensitivity analysis would typically be performed to ensure the solution is independent of the mesh size. The current mesh size of 0.01 mm is quite fine and likely provides adequate resolution for capturing the flow phenomena accurately.

6. Conclusions and Recommendations

6.1 Summary of Findings

The CFD analysis of gas expansion in a divergent nozzle has provided valuable insights into the flow behavior and pressure distribution:

1. The velocity distribution follows the expected pattern for subsonic flow in a divergent nozzle, with velocity decreasing as the cross-sectional area increases.
2. The pressure distribution shows pressure recovery in the diverging section, consistent with theoretical expectations for subsonic flow.
3. The flow remains well-behaved throughout the nozzle, without significant separation or recirculation zones, indicating an efficient nozzle design.
4. The transient analysis shows the development of the flow field from initial conditions to a steady-state solution, providing insights into the dynamic behavior of the system.

6.2 Engineering Implications

The findings from this CFD analysis have several engineering implications:

1. For applications requiring pressure recovery, such as diffusers or exhaust systems, the divergent nozzle geometry used in this study provides efficient conversion of kinetic energy to pressure energy.
2. The gradual expansion angle used (from 30 mm to 60 mm over 80 mm) promotes smooth flow expansion without significant losses, which could be a design guideline for similar applications.
3. The relationship between nozzle geometry and flow characteristics demonstrated in this study can inform the design of more efficient nozzles for specific applications.

6.3 Recommendations for Future Work

Based on the current analysis, the following recommendations are made for future work:

1. Extend the analysis to include different expansion ratios and nozzle lengths to optimize the design for specific applications.
2. Investigate the effects of varying the molecular weight of the gas to understand its impact on flow behavior and pressure recovery.
3. Include heat transfer effects by considering non-adiabatic walls, which would be relevant for applications involving hot gases or prolonged operation.
4. Perform a more detailed analysis of the boundary layer development and its effect on the effective flow area and pressure losses.
5. Extend the study to supersonic flow conditions to analyze shock formation and propagation in divergent nozzles.

References

1. Anderson, J.D. (2011). Fundamentals of Aerodynamics. McGraw-Hill Education.
2. ANSYS Inc. (2024). ANSYS Fluent User's Guide, Release 2025 R1.
3. White, F.M. (2016). Fluid Mechanics. McGraw-Hill Education.
4. Zucrow, M.J., & Hoffman, J.D. (1976). Gas Dynamics. Wiley.

5. Shapiro, A.H. (1953). The Dynamics and Thermodynamics of Compressible Fluid Flow. Wiley.