

Structural Analysis of a Converging -Diverging Rocket Nozzle under Internal Pressure using Finite Element Method

Pranav Mohan
B.E Aerospace Engineering
BMS College Of Engineering

Abstract - Rocket nozzles play a critical role in propulsion systems by converting high-pressure combustion gases into high-velocity exhaust to generate thrust. Due to the extreme internal pressures experienced during operation, the structural integrity of the nozzle is an important aspect of propulsion system design.

In this study, a structural analysis of a converging–diverging rocket nozzle is performed using the finite element method. The geometry of the nozzle was modelled and analysed using ANSYS Mechanical to evaluate the deformation and stress distribution under internal pressure loading.

A static structural analysis was conducted by applying a fixed support at the mounting flange to represent attachment to the propulsion system, while an internal pressure of 2 MPa was applied along the inner surface of the nozzle to simulate combustion chamber pressure conditions.

The model was discretized using an automatically generated finite element mesh, enabling numerical evaluation of the structural response. Results obtained from the simulation indicate that maximum stress concentrations occur near the throat and transition regions of the nozzle geometry, while deformation is observed primarily toward the nozzle exit section.

The study demonstrates the effectiveness of finite element analysis in predicting the structural behaviour of rocket propulsion components and provides insights into stress distribution under operational loading conditions. Such analyses are essential in the preliminary design phase of aerospace structures to ensure structural reliability and performance.

Keywords - Rocket Nozzle , Finite Element Analysis , ANSYS Mechanical , Structural Analysis

Internal Pressure , Aerospace Structures

1. INTRODUCTION

Rocket propulsion systems rely on the efficient conversion of thermal energy generated during combustion into directed kinetic energy that produces thrust. One of the most critical components responsible for this energy conversion is the rocket nozzle. A rocket nozzle accelerates high-temperature, high-pressure combustion gases through a converging–diverging passage, allowing the gases to expand and exit at extremely high velocities. This process converts the internal energy of the gas into kinetic energy, which ultimately generates thrust according to Newton’s third law of motion.

Rocket nozzles typically operate under extremely demanding conditions, including high internal pressures, elevated temperatures, and strong mechanical loads. The structural integrity of the nozzle is therefore essential for safe and efficient propulsion system operation. During engine operation, the nozzle walls experience significant stresses caused by internal combustion pressure as well as thermal gradients resulting from hot exhaust gases. These stresses can lead to deformation, material fatigue, or even structural failure if the design is not properly evaluated.

To ensure the reliability and safety of nozzle structures, engineers commonly perform structural analysis using Finite Element Analysis (FEA) techniques. Finite Element Analysis is a computational method that divides complex geometries into smaller discrete elements, allowing engineers to evaluate stress distribution, deformation, and structural behaviour under various loading conditions. By applying appropriate boundary conditions and loads, FEA enables detailed investigation of how a component will respond during real operating conditions.

In modern engineering practice, advanced simulation software such as ANSYS Mechanical is widely used to perform these analyses. ANSYS provides powerful numerical tools that allow engineers to model complex geometries, generate computational meshes,

apply loads and constraints, and compute structural responses such as deformation, stress, and strain. These simulations help identify critical regions where stresses may concentrate, allowing engineers to optimize designs and improve structural reliability before physical testing.

In this project, a static structural analysis of a rocket nozzle is performed using ANSYS Mechanical.

The nozzle geometry consists of a converging–diverging profile with a mounting flange, which represents a simplified rocket propulsion nozzle configuration. Internal pressure is applied to the inner surface of the nozzle to simulate the mechanical loading caused by combustion gases during engine operation. A fixed support condition is applied at the flange region to represent the nozzle attachment to the engine structure.

The objective of this analysis is to evaluate the stress distribution and deformation behaviour of the nozzle when subjected to an internal pressure load of 2 MPa.

The results obtained from the simulation provide insight into how the structure responds to pressure loading and help identify regions that may experience higher stress concentrations. Such analyses are important in the preliminary design and evaluation of rocket propulsion components, as they allow engineers to assess structural performance and ensure the reliability of the system.

Through this study, the effectiveness of finite element methods in analysing aerospace components is demonstrated, highlighting how computational tools can assist engineers in designing safer and more efficient propulsion systems.

2. LITERATURE REVIEW

Previous studies have demonstrated the importance of structural integrity in rocket nozzle design under combined thermal and pressure loads.

Finite element analysis has been widely used to evaluate stress concentrations and deformation behaviour in converging–diverging nozzles.

However, simplified structural models under internal pressure provide useful preliminary insights during early design stages.

3. LIMITATIONS OF STUDY

The present analysis considers only static structural loading under uniform internal pressure.

Thermal effects, material nonlinearities, and transient loading conditions have not been included.

In practical rocket nozzle applications, these factors significantly influence structural performance and should be considered in future work.

4. OBJECTIVE

The objective of this project is to perform a static structural analysis of a converging–diverging rocket nozzle under internal pressure using the finite element method.

The analysis aims to determine the stress distribution and deformation characteristics of the nozzle structure when subjected to pressure loading conditions representative of rocket engine operation.

5. SOFTWARE USED

The structural analysis of the rocket nozzle was carried out using ANSYS Mechanical 2026 R1 (Student Version), a widely used engineering simulation software for finite element analysis (FEA). ANSYS provides advanced computational tools for modelling complex geometries, generating meshes, applying boundary conditions, and solving for structural responses such as stress and deformation.

The nozzle geometry was initially created using SolidWorks, a computer-aided design (CAD) software, and then imported into ANSYS Mechanical for further analysis. The integration between CAD and FEA tools allows accurate representation of the geometry and efficient simulation setup.

A static structural analysis was performed to evaluate the behaviour of the nozzle under internal pressure loading. The simulation involved discretizing the geometry into finite elements, applying material properties, defining boundary conditions, and solving for stress distribution and deformation.

The use of ANSYS Mechanical enables precise prediction of structural performance, making it a powerful tool for preliminary design and validation of aerospace components.

6. GEOMETRY DESCRIPTION

The geometry used for this analysis represents a converging–diverging rocket nozzle attached to a mounting flange. The nozzle geometry consists of a converging section followed by a diverging exit section, which is typical for propulsion systems designed to accelerate exhaust gases to high velocities. The flange region is included in the model to represent the structural interface where the nozzle would be mounted to the combustion chamber or supporting structure.

The geometry was created in SolidWorks and imported into ANSYS Mechanical for structural evaluation. The internal surface of the nozzle is subjected to pressure loading to simulate the effect of high-pressure combustion gases acting on the nozzle walls during operation.

Units: mm

Geometry type: 3D solid model

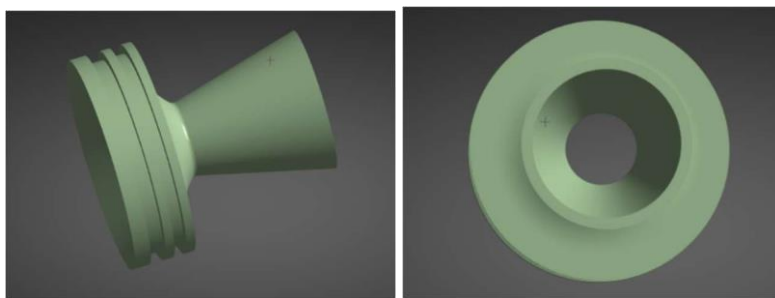


Fig 1

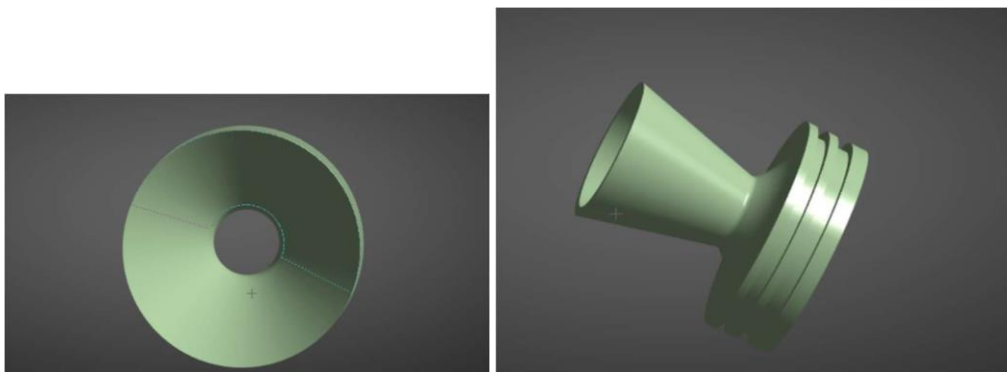


Fig 2

7. MATERIAL PROPERTIES

For the structural analysis, the nozzle was assigned the material properties of structural steel from the ANSYS material library. Structural steel is commonly used in engineering applications due to its high strength and durability.

The key material properties used in the simulation include:

- Young's Modulus: 200 GPa Poisson's Ratio: 0.3
- Density: 7850 kg/m³
- Yield Strength: ~250 MPa

These material properties allow the simulation to accurately predict stress distribution and deformation behaviour under the applied loading conditions.

8. BOUNDARY CONDITIONS

Fixed Support –

Applied on the flange surface to represent mounting to the engine structure.

Internal Pressure -

Pressure applied on the internal surface of the nozzle.

Magnitude: 2 MPa

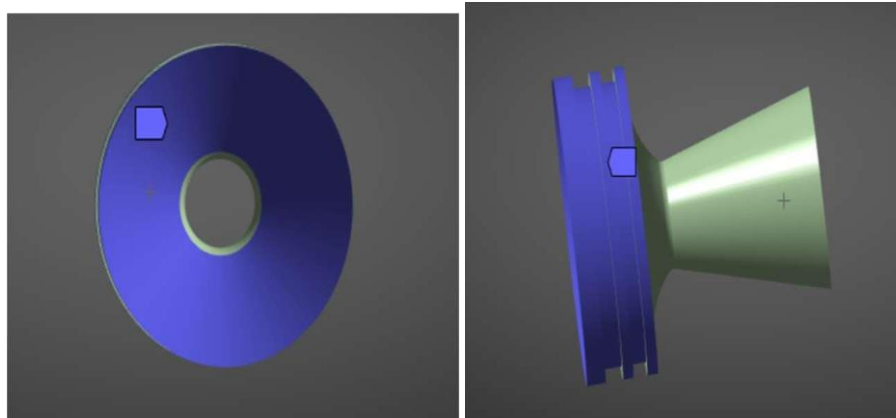


Figure 3: Fixed support applied on the flange surface

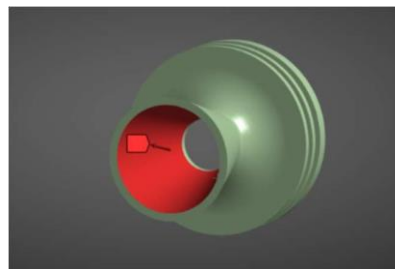


Figure 4: Internal pressure applied on the inner surface of the nozzle

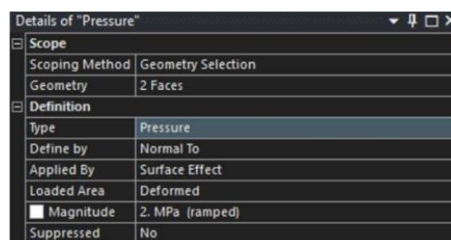


Fig 5

Figure 5 gives the scope and definition of pressure .

9. MESH DETAILS

The model was discretized using tetrahedral elements. Mesh refinement was applied automatically by the ANSYS meshing algorithm.

To perform the finite element analysis, the geometry was discretized into smaller elements using the ANSYS meshing tool. A fine tetrahedral mesh was generated across the nozzle geometry to ensure accurate representation of stress distribution and deformation.

The mesh consists of triangular surface elements and volumetric elements throughout the structure, allowing the solver to accurately calculate the structural response. A finer mesh helps capture stress concentrations in regions where geometry changes occur, such as near the flange connection and along the nozzle wall.

Proper mesh quality is essential for achieving reliable simulation results, and the generated mesh provided sufficient resolution for the structural analysis performed in this study.

The mesh was generated automatically using tetrahedral elements with sufficient refinement to capture stress concentrations near geometric transitions. The mesh density was chosen to balance computational efficiency with solution accuracy.

A detailed mesh convergence study was not performed; however, sufficient mesh refinement was applied to accurately capture stress concentrations in critical regions of the nozzle geometry.

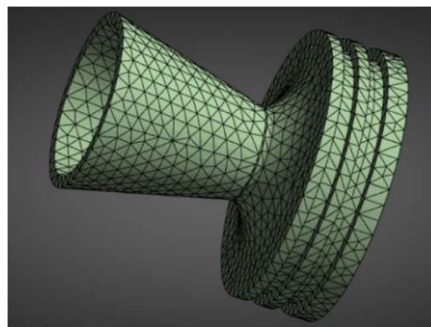


Figure 6: Finite element mesh generated for the nozzle geometry

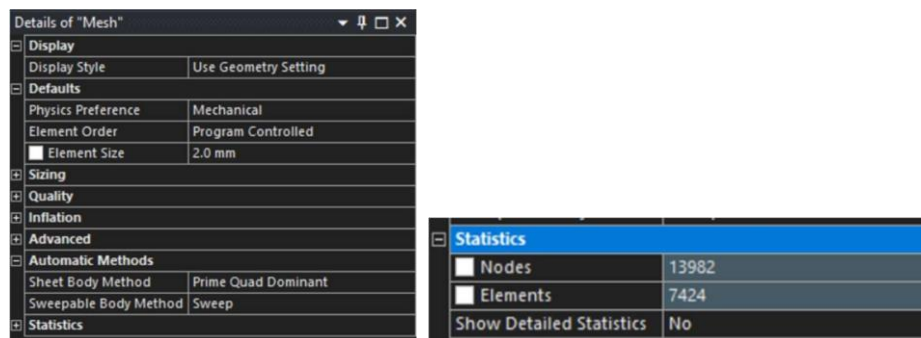


Fig 7

The details of the mesh are in figure 7.

10. GOVERNING EQUATIONS

The structural behaviour of the nozzle is governed by the equations of linear elasticity.

The stress-strain relationship is given by Hooke's law, and failure is evaluated using the von-Mises yield criterion.

$$\sigma_v = \sqrt{\left(\frac{1}{2}[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2]\right)}$$

where σ_1 , σ_2 , and σ_3 are the principal stresses.

11. RESULTS AND DISCUSSION

▪ Total Deformation

The total deformation result shows the displacement of the nozzle structure under internal pressure. Maximum deformation occurs near the nozzle exit due to pressure loading.

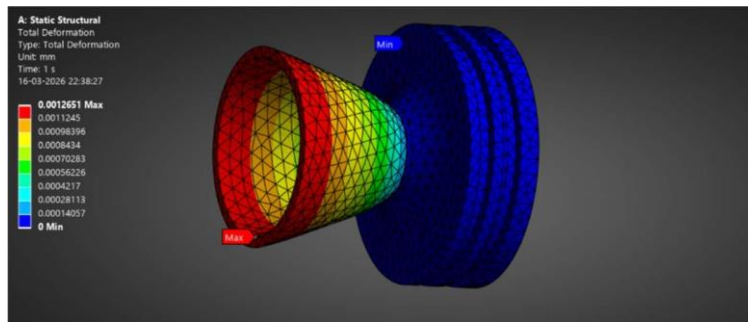


Figure 8: Total deformation distribution under internal pressure

Tabular Data				
	Time [s]	<input checked="" type="checkbox"/> Minimum [mm]	<input checked="" type="checkbox"/> Maximum [mm]	<input checked="" type="checkbox"/> Average [mm]
1	1.	0.	1.2651e-003	2.665e-004

Fig 9

The total deformation plot illustrates the displacement experienced by the nozzle structure under the applied internal pressure. The simulation results indicate that the maximum deformation observed is approximately 0.001265 mm, which occurs near the nozzle exit region where the structure experiences the greatest flexibility.

The deformation gradually decreases toward the flange region due to the fixed boundary condition applied at the mounting interface. The extremely small deformation value indicates that the nozzle structure remains highly rigid under the applied pressure loading.

This minimal displacement suggests that the structural stiffness of the nozzle is sufficient to maintain its shape and performance under the specified operating conditions.

The maximum deformation of 0.001265 mm is negligible compared to the overall nozzle dimensions, indicating high structural stiffness and minimal geometric distortion under pressure loading.

• Equivalent (Von-Mises) Stress Analysis

The equivalent stress distribution indicates the regions experiencing the highest stress. The highest stress occurs near the throat and flange transition region due to geometric discontinuities.

Factor of Safety (FOS) = Yield Strength / Maximum Equivalent Stress

$$\text{FOS} = 250 \text{ MPa} / 20.49 \text{ MPa} \approx 12$$

The equivalent stress distribution of the rocket nozzle was obtained using a static structural analysis in ANSYS. The von-Mises stress criterion was used to evaluate the structural integrity of the nozzle under an applied internal pressure of 2 MPa. The simulation results indicate that the maximum equivalent stress observed is 20.489 MPa, which occurs near the junction between the nozzle section and the mounting flange region, as indicated by the red zone in the stress contour plot. This region experiences higher stress due to the geometric transition and the constraint imposed by the fixed support.

The stress gradually decreases along the nozzle surface away from the constrained region, as shown by the transition from yellow to green and blue contours. The minimum stress value is approximately 6.9×10^{-9} MPa, occurring in regions far from the load concentration and structural constraints. The stress distribution pattern indicates that the majority of the nozzle structure experiences relatively low stress levels under the applied loading condition.

Considering a typical structural steel material with a yield strength of approximately 250 MPa, the maximum stress obtained from the simulation is significantly lower than the allowable limit. This results in a factor of safety of approximately 12, indicating that the nozzle structure is capable of safely withstanding the applied internal pressure without risk of yielding or structural failure.

Overall, the stress analysis confirms that the designed nozzle geometry maintains structural integrity under the specified loading conditions and demonstrates adequate strength for the applied pressure environment.

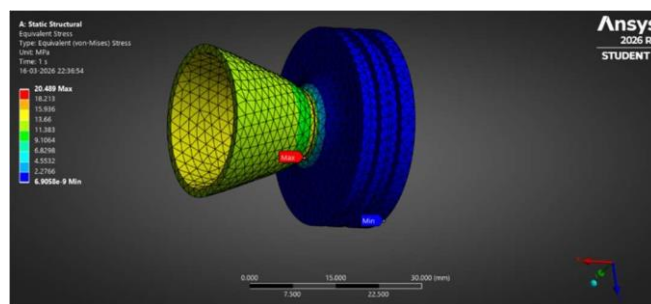


Figure 10: Equivalent von-Mises stress distribution

Factor of Safety Evaluation-

The factor of safety provides an indication of how safely the structure operates under the applied loading conditions. It is defined as the ratio of the material yield strength to the maximum equivalent stress obtained from the structural analysis.

The stress concentration near the throat region is attributed to geometric discontinuity and localized constraint effects.

For the present study, structural steel with an approximate yield strength of 250 MPa was used as the nozzle material.

The maximum equivalent stress obtained from the simulation was 20.489 MPa. Using these values, the factor of safety can be calculated as:

$$FOS = \frac{250}{20.489} = 12$$

12. CONCLUSION

A static structural analysis of a converging–diverging rocket nozzle was successfully performed using finite element analysis in ANSYS Mechanical.

The simulation evaluated the structural behaviour of the nozzle under an internal pressure load of 2 MPa applied along the inner surface.

The results indicate that the maximum equivalent stress is approximately 20.49 MPa, which is significantly lower than the yield strength of structural steel (~250 MPa). Additionally, the maximum deformation is extremely small at approximately 0.001265 mm, indicating high structural stiffness.

Based on these results, the calculated factor of safety is approximately 12, confirming that the nozzle structure can safely withstand the applied pressure without risk of yielding or structural failure. The analysis demonstrates that the current nozzle design is structurally sound and capable of operating safely under the specified loading conditions.

The results validate the effectiveness of finite element analysis as a reliable tool for preliminary structural evaluation of aerospace components.

13. FUTURE WORK

Future work may include coupled thermal-structural analysis, transient pressure loading, and optimization of nozzle geometry for weight reduction while maintaining structural integrity.