RESEARCH ARTICLE OPEN ACCESS

DESIGN AND STRUCTURAL ANALYSIS OF CONVERGENT–DIVERGENT NOZZLE IN ROCKET ENGINE USING CFD

R. Nallendran[1], A. Nihal Ahamed[2], A. Ragavendira[3]

Department of Aeronautical Engineering, Parisutham Institute of Technology and Science, Thanjavur, India

Email: nallendran2005@email.com, nihalahamed323@email.com, rockeragavendira@email.com

Abstract:

A convergent-divergent (CD) nozzle plays a critical role in rocket propulsion by converting thermal energy into kinetic energy to generate thrust. In this work, a conical CD nozzle is designed and analyzed using CFD and structural simulations. The nozzle is modeled in CATIA and analyzed using ANSYS under operating conditions of inlet pressure 300 psi and outlet pressure 14.7 psi. The results show supersonic flow generation with Mach number reaching 2.57 at the exit. Structural analysis indicates safe stress levels with minimal deformation. Analytical and simulation results are found to be in close agreement, validating the design approach.

Keywords—Nozzle, CFD, Mach number, Structural analysis, Rocket propulsion

1. INTRODUCTION

A nozzle is a device used to control fluid flow characteristics such as velocity and pressure. In rocket engines, the convergent–divergent nozzle is essential for accelerating gases to supersonic speeds. The convergent section accelerates subsonic flow, while the divergent section expands and further accelerates the flow beyond Mach 1.

The principle of operation is based on energy conversion where pressure and thermal energy are transformed into kinetic energy. When the flow reaches the throat, it becomes sonic, and further expansion leads to supersonic velocities. This mechanism is crucial in aerospace propulsion systems.

The present study focuses on designing a conical nozzle and analyzing its performance using CFD and structural methods. The goal is to validate theoretical calculations with simulation results.

2. LITERATURE REVIEW

Several researchers have contributed to nozzle design and analysis:

Pandey and Yadav (2010) performed CFD analysis showing improved performance with multi-inlet nozzles. Surya Narayana et al. (2016) highlighted the role of CD nozzles in converting thermal energy into kinetic energy.

Solomon et al. (2020) used mathematical methods to optimize nozzle geometry and concluded that expansion ratio significantly affects performance. Hari Krishnan et al. (2020) demonstrated that optimized geometries reduce stress concentration.

Shaik Muzib (2019) emphasized nozzle design for reaction control systems. Kurtcebe (2022) discussed the importance of avoiding shock formation in supersonic nozzles. Baxi et al. (2021) used CFD to validate nozzle performance under real conditions.

Overall, literature confirms that nozzle geometry, material, and operating conditions strongly influence performance and structural integrity.

3. METHODOLOGY

The methodology followed includes:

- Literature study of CD nozzle behavior
- Manual design calculations

- CAD modeling using CATIA
- CFD simulation using ANSYS Fluent
- Structural analysis using ANSYS Mechanical
- Comparison of analytical and simulation results

The workflow ensures accurate evaluation of both fluid flow and structural performance.

4. DESIGN AND CALCULATIONS

The nozzle design is based on isentropic flow relations.

Throat area is calculated using:

$$A = \frac{\dot{m}}{}\sqrt{\frac{RT_t}{}}$$

Key design parameters:

- Inlet pressure = 300 psi
- Exit pressure = 14.7 psi
- Throat diameter = 0.238 in
- Exit diameter = 0.455 in
- Inlet diameter = 1.2 in
- Temperature at throat = 5650 R

Wall thickness is calculated using stress relation:

$$= \frac{}{}$$

Final wall thickness selected = 0.09375 in for safety.

The nozzle geometry is optimized to ensure smooth flow expansion and avoid shock formation.

5. CFD ANALYSIS

CFD analysis is performed using ANSYS Fluent under ideal gas assumptions. The simulation results provide detailed insights into pressure, temperature, and Mach number variation along the nozzle.

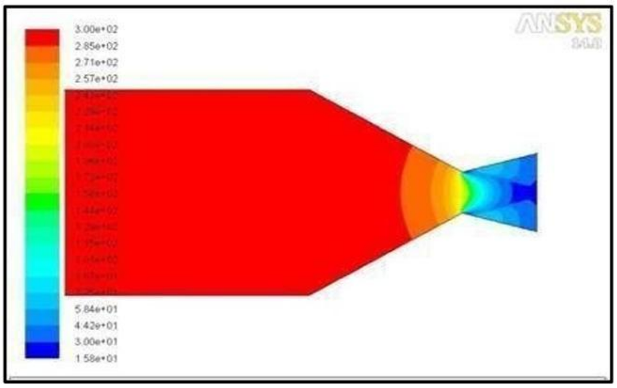

Fig. 1: Pressure contour of nozzle

Pressure decreases from 300 psi at inlet to atmospheric pressure at exit. A sharp drop is observed after the throat.

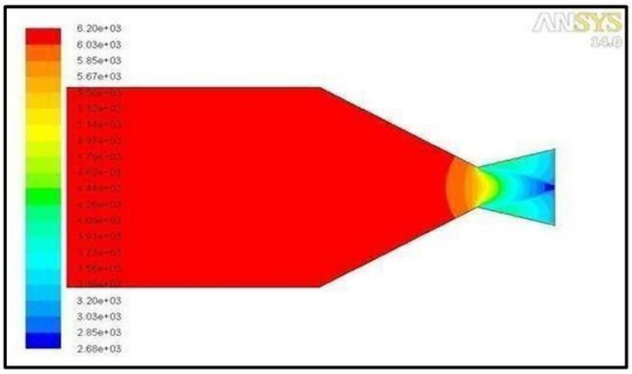

Fig. 2: Temperature distribution

Temperature decreases along the nozzle length due to expansion and energy conversion.

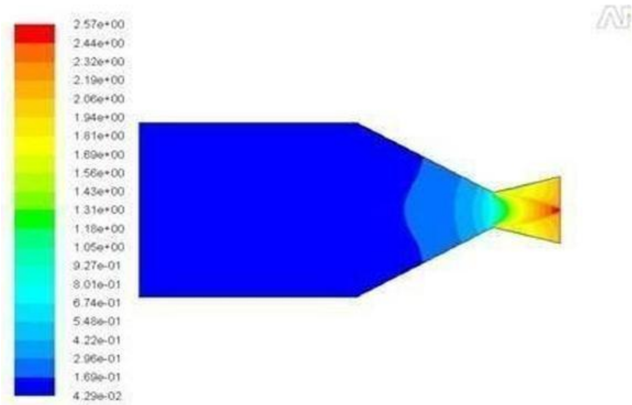

Fig. 3: Mach number variation

- Convergent section: Subsonic ($M \approx 0.04$)

- Throat: Sonic (M ≈ 1)
- Exit: Supersonic (M ≈ 2.57)

The results confirm proper CD nozzle behavior with efficient acceleration of flow.

The pressure decreases from inlet to exit, while temperature reduces due to expansion. The Mach number increases from subsonic to supersonic values, reaching approximately 2.57 at the nozzle exit.

6. STRUCTURAL ANALYSIS

Structural analysis is conducted to evaluate stress and deformation under operating conditions. The results indicate that maximum stress occurs near the throat region due to high pressure concentration.

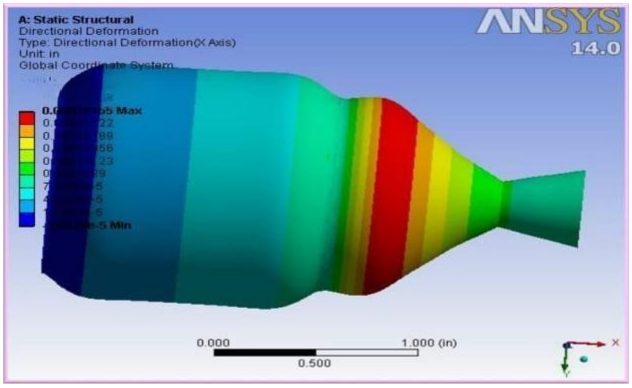

Fig. 4: Total deformation of nozzle

The deformation is minimal and within allowable limits, confirming that the selected material is suitable for the design. Key observations:

- Maximum stress ≈ 263 psi near throat
- Minimum stress at exit region
- Maximum deformation ≈ 0.000309 in

The nozzle material (Titanium alloy) shows sufficient strength to withstand operating conditions.

7. RESULTS AND DISCUSSION

A comparison between analytical and simulation results is presented below:

Parameter	Analytical	Simulation
Pressure (psi)	169	171
Temperature (R)	5650	5500
Mach Number	1.0	1.05

The results show close agreement, validating the accuracy of CFD simulation and design equations.

The slight variation is due to numerical approximations and boundary condition assumptions.

8. CONCLUSION

A convergent–divergent nozzle was successfully designed and analyzed using CFD and structural methods. The results confirm that the nozzle efficiently converts thermal energy into kinetic energy, achieving supersonic flow at the exit. Structural analysis indicates that the design is safe under given operating conditions with minimal deformation. The comparison between analytical and simulation results shows good agreement. This study demonstrates that CFD tools are effective for predicting nozzle performance and can be used for further optimization in aerospace applications.

REFERENCES

- K. M. Pandey, S. K. Yadav, “CFD Analysis of Rocket Nozzle,” 2010
- K. P. S. Surya Narayana, “Simulation of CD Nozzle,” 2016
- Gelu Solomon, “Design and Analysis of Rocket Nozzle,” 2020
- R. Hari Krishnan, “Optimized Nozzle Design,” 2020
- Shaik Muzib, “Solid Propellant Nozzle Design,” 2019
- Dr. Cemil Kortleve, “Supersonic Nozzle Design,” 2022
- Preet Baxi et al., “Bell Nozzle CFD Analysis,” 2021
- P. Chinmai et al., “Nozzle Performance Study,” 2018