

Design And Analysis Of Bell Rocket Nozzle Using Cfd

Abinesh P, Ashwinth B, Balaji A, Kamalesh S , Puviyarasan M

Department of Mechanical Engineering, Panimalar Engineering College ,

Chennai – 600 123, India

{ ashwinthb2004@gmail.com , abinesh21032004@gmail.com , balaji2752005@gmail.com ,
muthupuvi@gmail.com }

Abstract

Rocket nozzles are critical components in rocket propulsion systems because they convert thermal energy produced in the combustion chamber into kinetic energy to generate thrust. The bell-shaped rocket nozzle is widely used in modern propulsion systems due to its ability to provide high efficiency with reduced structural length. This study focuses on the design and computational analysis of a bell rocket nozzle using theoretical calculations and Computational Fluid Dynamics (CFD). The nozzle geometry was designed using CATIA software and analyzed using ANSYS Fluent. The CFD simulation was used to determine flow properties such as pressure, temperature, velocity, and Mach number distribution throughout the nozzle. Theoretical results were compared with CFD results to evaluate the accuracy of the model. The results indicate that the bell nozzle effectively accelerates exhaust gases from subsonic to supersonic speeds, resulting in optimized thrust generation.

Keywords: Bell Nozzle, Rocket Propulsion, CFD, Supersonic Flow, ANSYS Fluent

I. INTRODUCTION

Rocket propulsion systems depend heavily on the efficient conversion of energy generated inside the combustion chamber into thrust. The nozzle is the component responsible for accelerating exhaust gases and directing them to produce forward thrust according to Newton's third law of motion. Rocket nozzles are designed to control the velocity, pressure, and direction of gas flow. Among different types of nozzles, the convergent–divergent nozzle (De Laval nozzle) is widely used in rocket engines to achieve supersonic flow.

The bell-shaped nozzle is a modified form of the convergent–divergent nozzle that improves performance by reducing nozzle length while maintaining high thrust efficiency. Compared to conical nozzles, bell nozzles reduce weight and material usage, making them suitable for aerospace applications. In addition, bell nozzles provide better expansion of exhaust gases, which increases thrust efficiency in rocket engines.

With the increasing demand for satellite launches and space exploration, optimization of rocket propulsion systems has become essential. Computational Fluid Dynamics (CFD) allows researchers to analyze fluid flow behavior in rocket nozzles without performing expensive experimental testing. This study aims to design a bell rocket nozzle and analyze its performance using CFD simulation techniques.

2. Literature Review

Several researchers have studied the design and analysis of rocket nozzles to improve propulsion efficiency. Pandey (2010) performed CFD analysis of rocket nozzles with multiple inlets to understand pressure distribution and flow characteristics. The results showed that CFD simulations can accurately predict nozzle performance.

Biju Kuttan et al. (2013) optimized the divergent angle of rocket engine nozzles using computational analysis. Their research demonstrated that nozzle geometry significantly influences thrust generation and fluid flow

characteristics. Narayana and Reddy (2016) performed CFD simulations on convergent–divergent nozzles and investigated parameters such as pressure, temperature, and velocity variations.

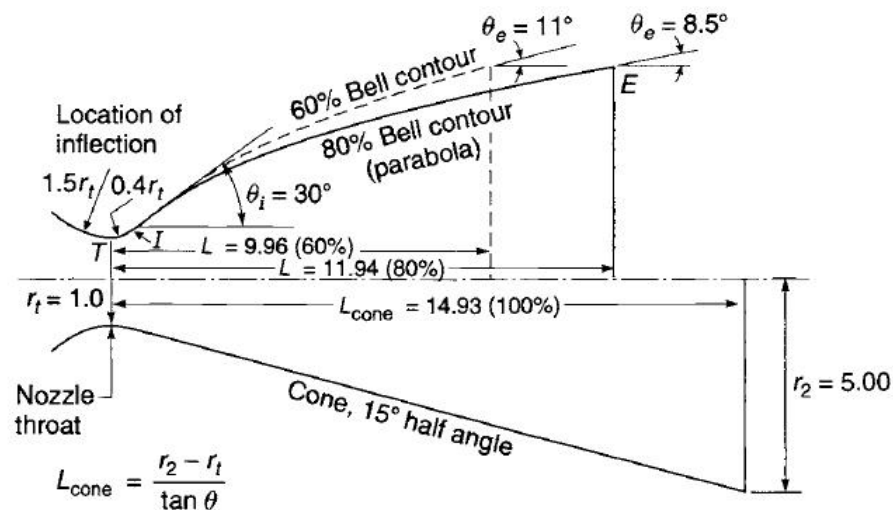
Solomon and Alemayehu (2020) studied rocket nozzle performance using theoretical calculations and CFD analysis. Their work highlighted the importance of analyzing thermodynamic properties and flow behavior inside the nozzle. These studies indicate that CFD tools are highly effective for predicting and optimizing rocket nozzle performance.

3. Methodology

The methodology used in this study includes design, modeling, meshing, simulation, and result analysis. Initially, the nozzle design parameters were calculated using compressible flow equations. The nozzle geometry was then modeled using CATIA software. The 3D model was imported into ANSYS Fluent for CFD analysis.

The simulation process involved generating a mesh for the nozzle geometry, defining boundary conditions, and running iterative simulations to obtain stable results. Flow parameters such as density, Mach number, pressure, temperature, and velocity were evaluated across the nozzle domain. The CFD results were then compared with theoretical calculations.

3.1 Design Approach



The nozzle is designed using isentropic flow relations based on given operating conditions such as chamber pressure and temperature.

3.2 Theoretical Analysis

Mathematical equations governing compressible flow are used to calculate:

Mach number

Pressure ratio

Temperature ratio

Exit velocity

3.3 Modeling

The geometry is created using CATIA software based on calculated dimensions.

3.4 Meshing

The model is discretized into finite volumes using ANSYS meshing tools.

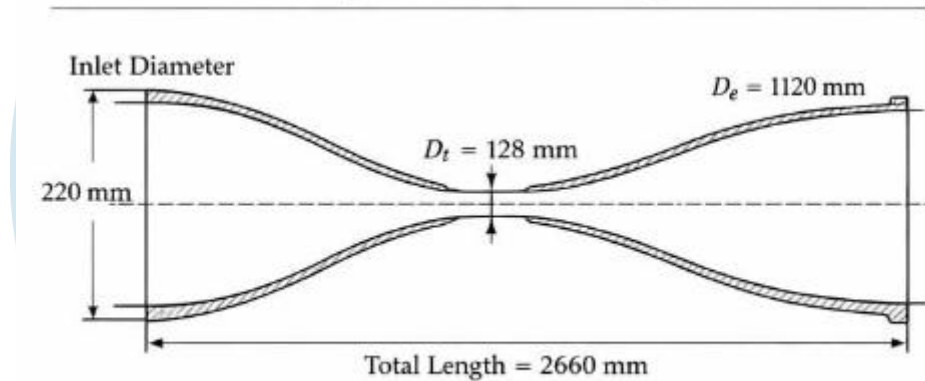
3.5 CFD Simulation

The flow is simulated using ANSYS Fluent with appropriate boundary conditions.

3.6 Validation

Results from CFD are compared with theoretical values.

4. Nozzle Design Parameters



Parameter	Value
Total Nozzle Length	2660 mm
Inlet Diameter	220 mm
Throat Diameter	128 mm
Exit Diameter	1120 mm

The area ratio of the nozzle determines the exit Mach number. Using the design parameters, the calculated area ratio resulted in an exit Mach number of approximately 4.69, indicating supersonic flow at the nozzle exit.

5. THEORITICAL CALCULATIONS

The calculations are performed using the Rocket Thrust equations. The working fluid is taken as air. Ideal combustion chamber pressure is taken as 20 MPa. Ideal combustion chamber temperature is taken as 3500K.

We consider,

Combustion Chamber Pressure (P_c) = 20 MPa,

Combustion Chamber Temperature (T_c) = 3500K

Specific heat ratio (γ) = 1.2

Gas Constant (R) = 0.287 KJ/kg-K

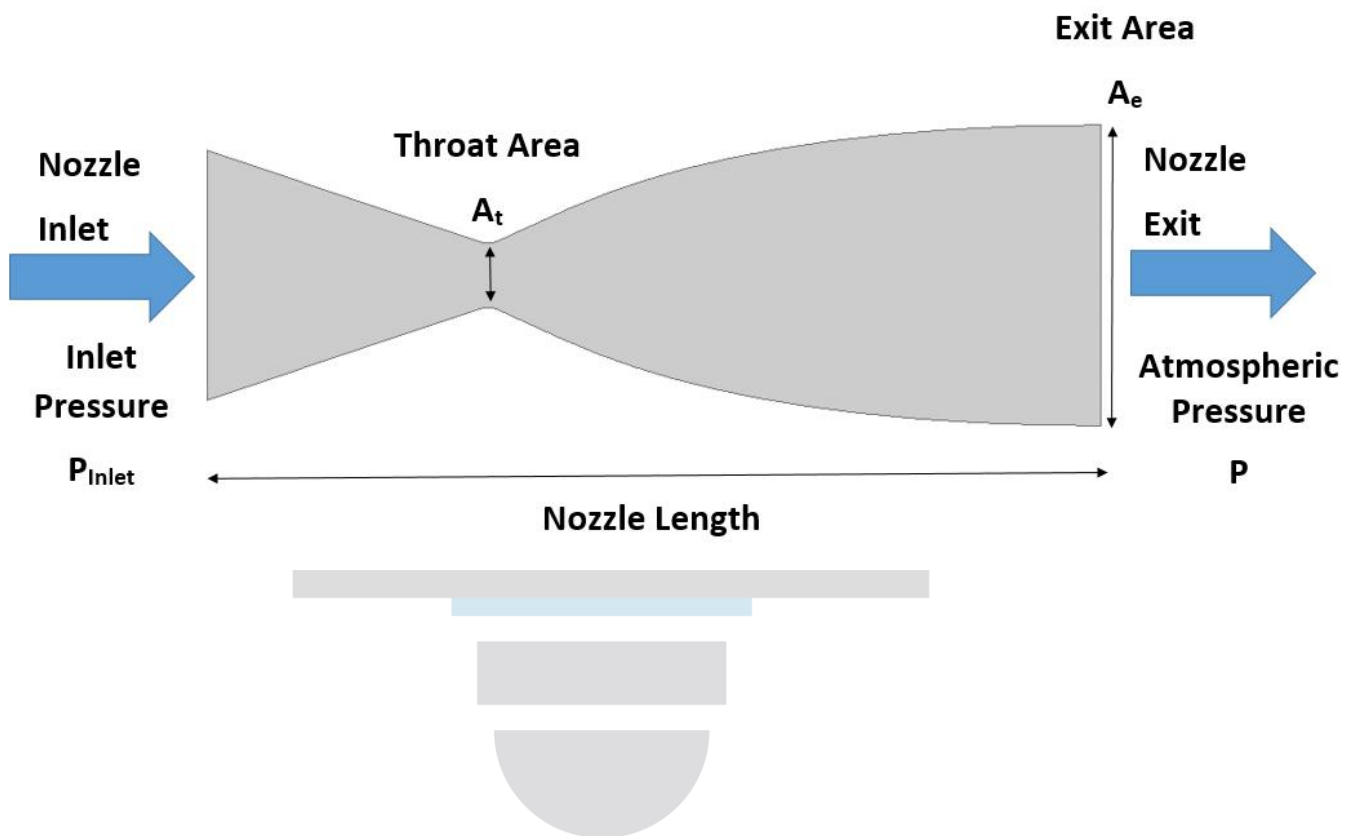
6. CFD Analysis

The CFD simulation was carried out using ANSYS Fluent. A density-based solver was used because the flow inside rocket nozzles involves compressible high-speed gas dynamics. The inlet pressure was set to 20 MPa and the inlet temperature was assumed to be 3500 K.

The mesh was generated using structured elements to improve numerical accuracy. After meshing, boundary conditions such as inlet pressure, outlet conditions, and wall boundaries were applied. The simulation was run for multiple iterations until convergence was achieved.

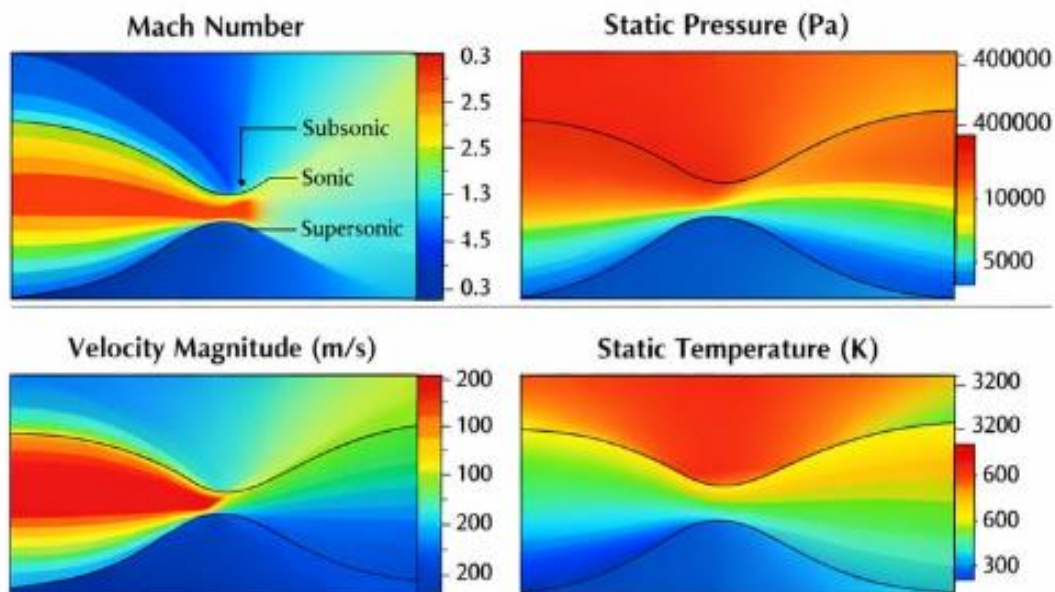
7. Boundary Condition

The mesh was checked in fluent software as already discussed. And then the next process is carried out which is giving boundary conditions. The values mentioned in the below table are referred and given as boundary condition.



7. Results and Discussion

General	Solver type: Density-based
Models	Energy equation: On
	Viscous model: Inviscid
Materials	Density: ideal gas
	$C_p = 1880 \text{ J/kg K}$
	Mean molecular mass = 27.7 g/mol
Boundary conditions	Inlet Pressure = 20MPa
	Inlet Temperature = 3500K
Solution initialization	Compute from: Inlet
Run calculation	No. of iterations: 200



PARAMETER	THEORITICAL	CFD
Static Pressure at Throat	11.28 MPa	11.6 MPa`
Static Pressure at Exit	0.0186 Mpa	0.0814 MPa
Static Temperature at Throat	3181.81 K	3250 K
Static Temperature at Exit	1093.88 K	1160 K

The CFD results show that the flow velocity increases significantly from the convergent section toward the divergent section of the nozzle. The Mach number increases from subsonic at the inlet to sonic at the throat and becomes supersonic at the nozzle exit. At the same time, pressure and temperature gradually decrease along the nozzle length.

The CFD results were compared with theoretical calculations. The differences between theoretical and simulation results were small and mainly due to factors such as shock waves, frictional forces, and boundary layer effects considered in CFD simulations but neglected in theoretical analysis.

8. CONCLUSION

This paper presented HSAMS, an IoT-based hierarchical solar array monitoring system delivering panel-level fault detection at This study presented the design and CFD analysis of a bell rocket nozzle. The nozzle geometry was modeled using CATIA and analyzed using ANSYS Fluent. The results confirmed that the bell nozzle effectively accelerates exhaust gases to supersonic speeds, which improves thrust performance.

The comparison between theoretical calculations and CFD results showed good agreement, validating the design approach. The study also demonstrated that CFD tools are powerful for analyzing fluid flow in rocket propulsion systems. Future research can focus on optimizing nozzle geometry and investigating thermal stresses and material performance under high temperature conditions.

ACKNOWLEDGMENT

The authors thank Dr. M. Puviyarasan, Professor, Department of Mechanical Engineering, Panimalar Engineering College, for his invaluable guidance and supervision throughout this project. The authors also thank the Management, Principal Dr. P. Mani, and Head of Department Dr. L. Karthikeyan of Panimalar Engineering College for providing laboratory facilities and institutional support. This work was carried out as part of the B.E. Final Year Project, Academic Year 2025–2026.

REFERENCES

1. K. M. Pandey, "CFD analysis of rocket nozzle with four inlets," *International Journal of Chemical Engineering and Applications*, 2010.
2. K. M. Pandey and S. K. Yadav, "CFD analysis of a rocket nozzle with two inlets," *Journal of Environmental Research and Development*, 2010.
3. B. Kuttan et al., "Optimization of divergent angle of rocket engine nozzle using CFD," *International Journal of Engineering and Science*, 2013.
4. K. P. S. Narayana and K. S. Reddy, "Simulation of convergent–divergent rocket nozzle using CFD analysis," *IOSR Journal of Mechanical and Civil Engineering*, 2016.
5. G. Solomon and Y. Alemayehu, "Design and analysis of rocket nozzle," *IOSR Journal of Engineering*, 2020.
6. J. D. Anderson Jr., *Modern Compressible Flow with Historical Perspective*, 3rd ed. New York: McGraw-Hill, 2003.
7. G. P. Sutton and O. Biblarz, *Rocket Propulsion Elements*, 9th ed. Hoboken, NJ: Wiley, 2017.
8. H. W. Liepmann and A. Roshko, *Elements of Gasdynamics*. New York: Dover Publications, 2001.
9. F. M. White, *Fluid Mechanics*, 8th ed. New York: McGraw-Hill, 2016.
10. C. Hirsch, *Numerical Computation of Internal and External Flows*. Oxford: Butterworth-Heinemann, 2007.
11. S. P. Vanka, "Computational fluid dynamics for engineering applications," *Annual Review of Fluid Mechanics*, vol. 28, pp. 113–144, 1996.
12. ANSYS Inc., *ANSYS Fluent Theory Guide*, 2023.
13. R. D. Zucker and O. Biblarz, *Fundamentals of Gas Dynamics*. New York: Wiley, 2002.
14. M. J. Zucrow and J. D. Hoffman, *Gas Dynamics*, vols. I & II. New York: Wiley, 1976.
15. H. Schlichting and K. Gersten, *Boundary Layer Theory*, 8th ed. Berlin: Springer, 2000.
16. NASA Glenn Research Center, "Rocket nozzle design and performance." [Online]. Available: <https://www.grc.nasa.gov>
17. S. M. Yahya, *Fundamentals of Compressible Flow*, 5th ed. New Delhi: New Age International, 2010.