DESIGN AND ANALYSIS OF ROCKET NOZZLES FOR ENHANCED THRUST EFFICIENCY AND REDUCE STRUCTURAL MASS

By

Safwan Qamrul

ID: 21107002

Program: BSME

Shohag Alam Apon

ID: 21207021

Program: BSME

DEPARTMENT OF MECHANICAL ENGINEERING

COLLEGE OF ENGINEERING AND TECHNOLOGY

IUBAT—INTERNATIONAL UNIVERSITY OF BUSINESS AGRICULTURE AND TECHNOLOGY

SUMMER, 2025

DESIGN AND ANALYSIS OF ROCKET NOZZLES FOR ENHANCED THRUST EFFICIENCY AND REDUCE STRUCTURAL MASS

By

Safwan Qamrul

ID: 21107002

Program: BSME

Shohag Alam Apon

ID: 21207021

Program: BSME

A THESIS SUBMITTED IN PARTIAL FULFILMENT OF THE REQUIREMENTS FOR THE DEGREE,

BACHELOR OF SCIENCE IN MECHANICAL ENGINEERING (BSME)

DEPARTMENT OF MECHANICAL ENGINEERING

COLLEGE OF ENGINEERING AND TECHNOLOGY

IUBAT—INTERNATIONAL UNIVERSITY OF BUSINESS AGRICULTURE AND TECHNOLOGY

DECLARATION

This thesis has been prepared after twelve months of research on "Design and Analysis of Rocket Nozzles for Enhanced Thrust Efficiency and Reduced Structural Mass". The thesis is solely for the academic requirement of the course MEC 488 and has not been submitted in part or full elsewhere for any other degree, reward, or any other purpose. we do solemnly declare that all and every right in the copyright of this thesis belongs to IUBAT-International University of Business Agriculture and Technology. Any reproduction or use in any form or by any means whatsoever is prohibited without the written consent of IUBAT.

Safwan Qamrul

ID: 21107002

Shohag Alam Apon

ID:21207021

ACKNOWLEDGMENT

First of all, we would like to express our gratitude to our honourable supervisor Arif Md. Shahed Iqubal, Associate Professor, Department of Mechanical Engineering for his guidance and contribution to our work. Our thesis is the result of his support and effort.

We sincerely acknowledge the contributions of Professor and Chair Dr. Dewan Mohammad Nuruzzaman, Associate Professor and Coordinator Dr. A. K. M. Parvez Iqbal, Department of Mechanical Engineering at IUBAT-International University of Business Agriculture and Technology for continuous support to come up with a thesis.

We would like to express our gratitude to the jury for their thoughtful comments which helped us to enrich our thesis project. We also acknowledge the contributions of our faculties at IUBAT-International University of Business Agriculture and Technology for their cooperation, support, and motivation that enabled us to complete our thesis.

Special thanks to former student Md. Motiur Rahman, whose contributions helped make this thesis turn into a reality.

We would also like to thank our parents, family members, friends and relatives for their cooperation, affection, and help throughout attending this study. Their support in hard times and motivation to keep doing hard work constantly, helped us a lot to come up with this good work.

Safwan Qamrul
ID: 21107002

Shohag Alam Apon
ID: 21207021

DESIGN AND ANALYSIS OF ROCKET NOZZLES FOR ENHANCED THRUST EFFICIENCY AND REDUCE STRUCTURAL MASS

Candidates

Safwan Qamrul
ID: 21107002

Shohag Alam Apon
ID: 21207021

Supervisor

Arif Md. Shahed Iqubal
Department of Mechanical Engineering

ABSTRACT

Rocket nozzles are the most essential part of a rocket engine because they control how the hot gases from combustion convert into thrust. The overall efficiency of the rocket, how much it can carry, and even the cost of the mission depends a lot on this part. Conical nozzles are simple and easy to make, but they are usually heavier and lose some energy in the process. Bell-shaped nozzles, on the other hand, are shorter and lighter, and they tend to give better efficiency, although they take more effort to design and optimize.

In our study, we wanted to see how much difference the design really makes. We started by testing a known reference nozzle in ANSYS Fluent so that we could check if our simulations were giving trustworthy results. Once we saw that the results matched well enough, we moved on to designing our own bell-shaped nozzle. For that, we used basic equations from isentropic flow and expansion ratios, and then we improved the design step by step through simulations.

What we found was quite encouraging. The bell-shaped nozzle gave almost the same thrust and specific impulse as the conical one, but it was about 32% shorter in length. That difference might sound small, but in aerospace it really matters because it means less weight, less material, and smoother gas flow overall. To put it simply, this shows that with the right design, a bell nozzle can do the same job as a conical nozzle while being lighter and more efficient. We believe this kind of work can help make rockets more cost-effective and practical in the future.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLES

LIST OF ABBREVIATIONS

CFD	Computational Fluid Dynamics
ISP	Specific Impulse
ISA	International Standard Atmosphere
NPR	Net Pressure Ratio
$k–\varepsilon$	Turbulent Kinetic Energy – Turbulent Dissipation Rate
$k–\omega$	Turbulent Kinetic Energy – Specific Dissipation Rate
SST	Shear Stress Transport ($k–\omega$ based hybrid model)

LIST OF SYMBOLS

Symbol	Definition	Unit
A	Area	m^2
Ae	Exit area of nozzle	m^2
A*	Throat area of nozzle	m^2
ε (Ae/A*)	Expansion ratio	–
$\dot{m}$	Mass Flow Rate	kg/s
M	Mach Number	–
P	Static Pressure	Pa
P_0	Total (Stagnation) Pressure	Pa
Pe	Exit Pressure	Pa
Pa	Ambient Pressure	Pa
T	Static Temperature	K
T_0	Total (Stagnation) Temperature	K
Te	Exit Temperature	K
u	Flow Velocity	m/s
ue	Exit Velocity	m/s
F	Thrust	N
Isp	Specific Impulse	s
NPR	Net Pressure Ratio (P_0/Pe)	–
γ	Specific Heat Ratio	–
R	Gas Constant	J/kg·K
ρ	Density	kg/m^3
θn	Initial Expansion Angle	Degree
θe	Nozzle Exit Angle	Degree
Lf	Nozzle Length Factor	–
Y+	Non-dimensional wall distance	–

CHAPTER 1: INTRODUCTION

1.1 Research Background

Rockets have always been important for space travel, putting satellites into orbit, and even in defense work. At the center of all this is the rocket nozzle. It might look like just a pipe, but it does the real job of pushing the rocket forward. The nozzle takes the hot, high-pressure gases from the engine and lets them expand into a very fast jet, which creates thrust. If the nozzle isn't designed properly, the rocket can waste energy. That means it might carry less weight, travel less distance, or even fail its mission.

In the old days, engineers had to figure out nozzle designs by building them and running tests. That was slow and very expensive. Nowadays, things are different. With computers and software like Computational Fluid Dynamics (CFD), it's possible to test many designs without building them first. This saves both time and money, and the results are much more reliable than before.

But there is still a problem. A rocket doesn't stay in one place — it starts at sea level and goes all the way into space. The air pressure changes a lot during that time. At ground level, the outside pressure is high, and at high altitude, it is almost nothing. The same nozzle has to work well in all those conditions. If it doesn't, you can get things like shock waves or poor expansion inside the nozzle, which wastes energy. So, making a nozzle that stays efficient across different altitudes is a big challenge.

That's why this study was done. We wanted to design and improve a nozzle that gives strong thrust, stays efficient, and works smoothly at different heights. To make sure our method was correct, we first tested a reference nozzle design that was already published and trusted. We recreated it on ANSYS Fluent and checked if our results matched the data. After that, we designed our own nozzle using equations from isentropic flow and expansion ratios. Then we improved it step by step with simulations until it gave better results.

In short, this research is about combining theory with simulation to show a simple but reliable way of designing rocket nozzles. The idea is not just for academics but also for real aerospace projects, where things like saving weight, using less material, and making rockets more efficient are always important.

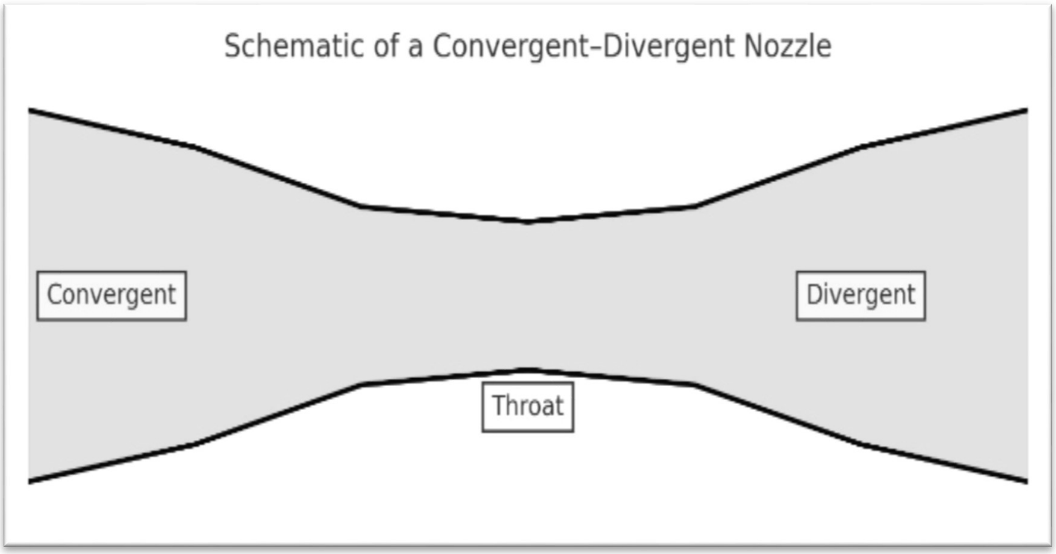

Fig:1.1 Schematic Figure of a convergent – divergent Nozzle

1.2 Problem Statement and Significance

The nozzle of a rocket engine is not just another part — it's the one that really controls how much power the rocket gets from burning fuel. If the nozzle does not work properly, the rocket loses energy, carries less load, and wastes fuel. That's why nozzle design is such a big deal in propulsion.

The conical nozzle is an older and simpler design. It is easy to build and has been used a lot. But it also has problems. Because of its wider angle, some of the gas does not push straight out but spreads sideways. That wastes energy. Also, the conical nozzle is longer and heavier, and in rockets, extra weight is always bad. Every bit of weight adds cost and reduces performance.

The bell-shaped nozzle was made to fix these issues. It is shorter, lighter, and better at keeping the gas flow in the right direction. On paper, it looks like the better design. But the truth is, unless we compare it carefully with the conical nozzle under the same conditions, we cannot say for sure how much better it really is.

For a long time, people used simple one-dimensional equations to guess how nozzles would behave. These models are fine for a rough idea, but they ignore real effects like shocks, gas separation, and viscosity. With CFD, we can now study nozzles in much more detail. Still, there are not many works that directly put the theory, the conical nozzle, and the bell nozzle side by side.

Because of that, engineers often have to make a choice without full evidence. The conical nozzle is simple but heavy. The bell nozzle is efficient but harder to design. The real difference between them, in numbers, is not always clear.

This research is meant to close that gap. We want to show a proper, fair comparison between conical and bell nozzles using both theory and CFD. The aim is simple: prove why the bell nozzle saves weight and improves efficiency. In rockets, even a small saving or a slight boost in thrust can make a huge difference in cost and success.

1.3 Objectives

The primary objective of this research is to **design, validate, and optimize a rocket nozzle** capable of delivering high thrust and specific impulse across varying altitude conditions, using a combination of theoretical calculations, computational fluid dynamics (CFD), and performance optimization techniques.

Specific Objectives

1. **Validation of Simulation Parameters**
    - Reproduce a reference rocket nozzle design from a peer-reviewed publication using ANSYS Fluent.
    - Conduct mesh independence studies to ensure numerical accuracy and reliability of the CFD results.
    - Compare CFD outcomes with published results for thrust, specific impulse, and pressure distribution to confirm the validity of chosen parameters.
2. **Nozzle Design Using Theoretical Principles**
    - Apply isentropic flow relations and compressible gas dynamics equations to calculate geometric parameters such as throat radius, exit radius, expansion ratio, and nozzle length.
    - Use these theoretical results to generate an initial nozzle model for CFD analysis.
3. **Performance Simulation Across Altitudes**
    - Simulate nozzle operation at multiple altitudes to evaluate variations in thrust, specific impulse, exhaust velocity, and pressure distribution.

- Analyze the influence of ambient pressure changes on nozzle efficiency.

4. **Optimization of Nozzle Geometry**
   - Modify divergence angles, expansion ratios, and length-to-diameter ratios to achieve maximum thrust and specific impulse.
   - Perform side-by-side comparisons between the baseline and optimized designs to quantify performance improvements.
5. **Documentation and Knowledge Contribution**
   - Provide a comprehensive methodology integrating theory, CFD validation, and optimization into a replicable framework.
   - Contribute practical design insights that can be applied in academic research and industrial aerospace projects.

.

1.4 Organization of the Report

This thesis is organized into seven chapters. Chapter 1 introduces the research background and objectives of this study. Chapter 2 provides an extensive literature review on rocket nozzle design, including comparisons between conical and bell-shaped nozzles, and the role of computational fluid dynamics (CFD) in nozzle optimization. Chapter 3 describes the methodology, including the theoretical foundations, governing equations, CFD simulation setup, and optimization process used in this study. Chapter 4 presents the results and discussion of both theoretical calculations and CFD simulations for conical and bell-shaped nozzles, including performance comparisons. Chapter 5 validates the CFD model through mesh independence studies and turbulence model comparisons. Chapter 6 discusses the socio-environmental impacts of the research, with a focus on sustainable aerospace propulsion. Finally, Chapter 7 concludes the thesis and provides recommendations for future research and practical implementation.

1.5 Limitations of the Report

- This report doesn't contain any experimental data.
- It is highly based on theoretical and numerical results.

CHAPTER 2: LITERATURE REVIEW

The optimization and design of rocket nozzles has been a subject of sustained research for decades, as nozzle geometry directly influences thrust, specific impulse, and overall propulsion efficiency. This chapter reviews foundational studies and recent developments in nozzle design, with emphasis on conical, bell-shaped, and alternative nozzle configurations. The review also highlights the role of computational fluid dynamics (CFD) and analytical methods in advancing nozzle research.

2.1 Fundamentals of Rocket Nozzle Design

Rocket nozzles serve as critical components that accelerate exhaust gases to supersonic velocities, thereby generating thrust. Early developments in nozzle theory were based on the convergent–divergent (de Laval) nozzle, which remains the foundation for modern nozzle design. Billheimer (1968) presented one of the earliest optimization frameworks, emphasizing computational methods for solid rocket design and showing that nozzle contour selection directly affects thrust efficiency.

Theoretical formulations such as quasi-one-dimensional (1D) flow models continue to be valuable for predicting nozzle performance. However, these models often require refinement to account for real fluid effects, viscous losses, and non-ideal expansion phenomena. Classical textbooks, such as Anderson (2017) and Sutton & Biblarz (2017), provide the theoretical basis for analyzing compressible flow in nozzles, including Mach number variation, expansion ratios, and performance parameters.

2.2 Conical and Bell Nozzles

Conical nozzles are among the simplest designs due to their geometric ease of manufacture. However, their divergence angle leads to performance losses from non-axial exhaust momentum. Patil et al. (2020) confirmed that while conical nozzles provide reliable performance, they are longer and heavier than optimized designs, reducing their efficiency in launch applications.

Bell nozzles, developed using Rao's method in the late 1950s, introduced a shortened contour with reduced divergence losses. Fernandes et al. (2023) emphasized that bell nozzles strike a balance between performance and structural efficiency by reducing nozzle length while maintaining thrust nearly equivalent to an ideal expansion contour. Saputra and Andria (2021) further demonstrated via CFD that bell contours achieve higher specific impulse and reduced weight compared to conical nozzles in equivalent operating conditions.

The central distinction lies in geometry: while both designs achieve similar thrust and specific impulse values, bell nozzles offer superior efficiency because of their compact size, leading to material and weight savings without significant performance penalties. Adde (2020) similarly concluded that contour optimization plays a critical role in aligning CFD results with quasi-1D theoretical predictions.

2.3 Advanced Nozzle Configurations

Beyond conventional conical and bell geometries, several advanced nozzle types have been-proposed:

- Aerospike Nozzles: Kumar et al. (2017) highlighted the altitude-compensating nature of aerospike nozzles, which adjust their effective area ratio with changing ambient pressure, minimizing performance losses across altitudes. Despite superior theoretical efficiency, complexity in cooling and fabrication has limited their adoption.
- Dual-Bell and Expansion-Deflection Nozzles: Fernandes et al. (2023) reviewed dual-bell and expansion-deflection designs, which provide altitude adaptability by modifying effective area ratio mid-flight. These designs have shown potential for improved performance in variable atmospheric conditions.
- Supersonic and Hypersonic Flow Considerations: Silva and Brójo (2025) reviewed nozzle design under supersonic and hypersonic regimes, using the Method of Characteristics (MoC) to optimize contours for reduced divergence and boundary losses.

These studies underline that while advanced nozzles can outperform bell designs theoretically, manufacturability, weight, and thermal management remain barriers to widespread use.

Comparison of Conical, Bell, and Advanced Rocket Nozzles

Nozzle Type	Key Characteristics	Advantages	Limitations	Supporting Studies
Conical Nozzle	Straight-wall divergence; half-angle ~15°–20°	Simple geometry, easy to manufacture, reliable baseline	Longer length, heavier, higher divergence losses	Patil et al. (2020); Saputra & Andria (2021)
Bell (Contour) Nozzle	Rao's optimized contour with curved profile	Shorter (~25–35% reduction), reduced divergence losses, near-theoretical performance	Slightly more complex to design/manufacture	Saputra & Andria (2021); Fernandes et al. (2023)
Dual-Bell Nozzle	Two expansion sections with inflection contour	Altitude adaptability, reduced off-design losses	Structural complexity, switching shock control	Fernandes et al. (2023)
Expansion-Deflection Nozzle	Flow expands against center body before exiting	High altitude adaptability, compact	Thermal stress, cooling challenges	Fernandes et al. (2023)
Aerospike Nozzle	Central spike replaces divergent section	Self-compensate with altitude, theoretically highest efficiency	Cooling complexity, manufacturing difficulty	Kumar et al. (2017)
Supersonic/Hypersonic Optimized	Contour shaped with Method of Characteristics (MoC)	Reduced shock/divergence losses at high Mach	Highly case-specific, complex optimization	Silva & Brójo (2025)

Table 2.1: Comparison of Conical, Bell, and Advanced Rocket Nozzles.

2.4 CFD and Numerical Methods in Nozzle Research

The increasing reliability of CFD has transformed nozzle research from empirical testing to computational optimization. Patil et al. (2020) validated CFD models against analytical predictions, showing strong agreement in thrust and Isp trends. Similarly, Fernandes et al. (2023) proposed surrogate-based optimization combining Method of Characteristics with CFD corrections, enabling fast and reliable shape optimization.

Saputra and Andria (2021) emphasized that CFD simulations of nozzle flow structures, including separation and shock patterns, are essential for understanding real nozzle performance. Sheikh and Kumar (2024) also examined thermal-structural interactions in nozzles with ablative protection systems, demonstrating how material response affects performance and durability. Such computational insights allow comparison between theoretical quasi-1D flow predictions and actual nozzle behavior, which is central to the present thesis.

2.5 Literature Summery & Synthesis

From the literature, it is evident that:

- Conical and bell nozzles remain widely studied due to their simplicity and applicability.
- Bell nozzles consistently demonstrate near-theoretical performance with reduced length and weight penalties compared to conical nozzles.
- Advanced designs (aerospike, dual-bell) offer altitude adaptability but face practical implementation challenges.
- CFD has become indispensable in nozzle research, bridging theoretical quasi-1D models with experimental performance validation.

The present research builds upon this foundation by directly comparing quasi-1D predictions, conical nozzle CFD results, and bell nozzle CFD results to determine whether performance equivalence exists between designs, and to evaluate the efficiency gains achieved by shorter bell geometries.

CHAPTER 3: METHODOLOGY

3.1 Introduction

The methodology adopted in this research follows a structured process aimed at developing, validating, and optimizing a rocket nozzle for improved performance. The workflow ensures that both theoretical formulations and computational approaches are integrated into a unified framework. The main steps are:

1. Formula Selection – Fundamental equations of compressible flow, gas dynamics, and thermodynamics were identified from standard references (Cengel & Boles, 2015; Anderson, 2017; Sutton & Biblarz, 2017).
2. Code Generation – A Python code was developed to implement the selected formulas and generate theoretical graphs for thrust, specific impulse, pressure, and temperature variations with altitude.
3. Initial Design – Using analytical calculations (isentropic flow relations, expansion ratio, area–Mach relations), a baseline nozzle was designed.
4. Simulation Test – The baseline design was simulated in ANSYS Fluent and validated against theoretical results. A mesh-independence study was performed to ensure accuracy.
5. Bell Shaped Nozzle Design– Geometric modifications (throat area, expansion ratio, contour shaping) were applied to improve thrust and specific impulse.
6. Simulation Test – The optimized nozzle was analyzed under varying altitude conditions using CFD, with contour plots generated for pressure, velocity, Mach number, and temperature distributions.
7. Comparison – Theoretical results, baseline CFD data, and optimized design results were compared to evaluate performance improvements.

This systematic approach integrates theoretical analysis, coding, simulation, and validation into a single coherent methodology for nozzle design.

The overall methodological process followed in this study is illustrated in Fig. 3.1. The flowchart summarizes the sequential steps starting from formula selection, code generation, initial design, mesh generation, validation, optimization, and final comparison. This structured approach ensured that both conical and bell nozzles were evaluated consistently and effectively against quasi-one-dimensional predictions.

Methodology flowchart

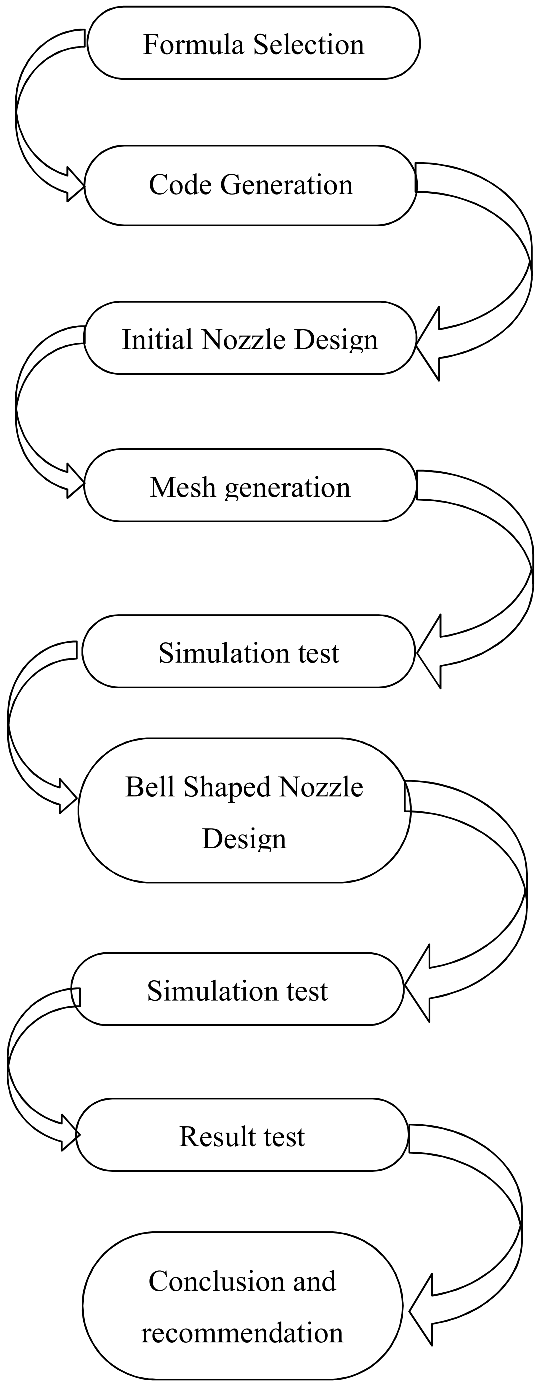

Fig. 3.1: Methodology flowchart illustrating the sequential research process used in this study

3.2 Theory (Aero/Thermodynamic)

3.2.1 Thermodynamic Systems

A thermodynamic system is defined as "a quantity of matter or a region in space chosen for study, separated from its surroundings by boundaries which may be fixed or movable" (Cengel & Boles, 2015). In rocket nozzle studies, the nozzle is modeled as a control volume where enthalpy is converted into kinetic energy under isentropic assumptions.

Types of processes include Isochoric (constant volume), Isobaric (constant pressure), Isothermal (constant temperature), Adiabatic (no heat transfer), and Isentropic (adiabatic and reversible).

3.2.2 Mach Number

Mach number is defined as "the ratio of the velocity of the flow to the local speed of sound, serving as the most fundamental parameter in compressible flow" (Anderson, 2017).

Equation:

$$M = \frac{V}{a} = \frac{V}{\sqrt{\gamma RT}}$$

Flow regimes: Subsonic ($M < 1$), Sonic ($M = 1$), Supersonic ($M > 1$), Hypersonic ($M > 5$).

3.2.3 Shock Waves

Shock waves are "thin regions in a supersonic flow across which flow properties such as pressure, temperature, and density change almost discontinuously" (Anderson, 2017).

Types include: Normal shock (perpendicular to flow, reducing Mach to subsonic), Oblique shock (inclined, downstream flow may remain supersonic), and Expansion waves (Prandtl–Meyer expansion).

3.2.4 One-Dimensional Gas Dynamics

One-dimensional gas dynamics "simplifies compressible flow analysis by assuming properties vary only along one spatial dimension, making it ideal for nozzle flow calculations" (Sutton & Biblarz, 2017).

Key relations:

$$\frac{T_0}{T} = 1 + \frac{\gamma - 1}{2} M^2$$

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

$$\varepsilon(M) = \frac{A_e}{A_*}$$

3.2.5 International Standard Atmosphere (ISA)

ISA provides standard reference conditions for pressure, temperature, and density as a function of altitude (Anderson, 2005). At sea level: Temperature = 288.15 K, Pressure = 101325 Pa, Density = 1.225 kg/m^3.

3.2.6 Thrust (Rocket Nozzle)

Thrust is "the reaction force generated when high-pressure, high-velocity gases are expelled through a nozzle, producing propulsion" (Sutton & Biblarz, 2017).
Equation: $F = \dot{m}\, u_e + (P_e - P_a)\, A_e$.
Expansion conditions: Under-expanded ($P_e > P_a$), Over-expanded ($P_e < P_a$), Properly expanded ($P_e = P_a$).

3.2.7 Specific Impulse (Isp)

Specific impulse is "the total impulse per unit weight of propellant, representing the efficiency of a rocket engine" (Sutton & Biblarz, 2017).
Equation:

$$I_{sp} = \frac{F}{\dot{m} g_0}$$

The design and performance evaluation of a convergent–divergent (CD) rocket nozzle rely heavily on one-dimensional compressible flow relations, derived from fundamental gas dynamics and thermodynamics. The following equations are adopted in this research:

3.2.8 Isentropic Relations

For an ideal isentropic process:

$$\frac{T_0}{T} = 1 + \frac{\gamma - 1}{2} M^2$$

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

$$\frac{\rho_0}{\rho} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\frac{1}{\gamma-1}}$$

3.2.9 Area–Mach Relation

$$\varepsilon(M) = \frac{1}{M}\left[\frac{2}{\gamma + 1}\left(1 + \frac{\gamma - 1}{2} M^2\right)\right]^{\frac{\gamma+1}{2(\gamma-1)}}$$

3.2.10 Thrust Equation

$F = \dot{m}\, u_e + (P_e - P_a)\, A_e$

3.2.11 Specific Impulse

$$I_{sp} = \frac{F}{\dot{m} g_0}$$

3.2.12 Net Pressure Ratio (NPR)

$$NPR = \frac{P_0}{P_e}$$

with $20 \leq NPR \leq 60$ in this study.

3.3 Code Generation

To efficiently process these equations, a Python code was developed. Inputs included chamber conditions, throat area, and exit pressure. Outputs included mass flow rate, expansion ratio, exit Mach number, exit velocity, thrust, and specific impulse. This automation allowed direct comparison between theoretical and CFD results.

3.4 Selection of Design Parameters

The nozzle designs were based on a chamber pressure of $P_0 = 2.0 \times 10^6$ Pa and a chamber temperature of $T_0 = 3000$ K. The exit pressure was targeted at $P_e \approx 5.0 \times 10^4$ Pa, corresponding to a net pressure ratio (NPR) ≈ 40 for the quasi-1D baseline case. Gas properties were taken as $\gamma = 1.4$ and $R = 287$ J/kg·K, representing an ideal diatomic gas model.

Two nozzle geometries were analyzed:

- Conical Nozzle (Table 4.1):
  • Inlet radius: 0.018 m
  • Throat radius: 0.01262 m
  • Exit radius: 0.02667 m
  • Convergence length: 0.00932 m
  • Divergence length: 0.05247 m
  • Total length: 0.06579 m
  • Half angles: 30° convergence, 15° divergence

- Bell-Shaped Nozzle (Table 4.2):
  • Expansion ratio: 4.47
  • Inlet radius: 0.018 m
  • Throat radius: 0.01262 m
  • Exit radius: 0.02667 m
  • Convergence length: 0.00932 m
  • Divergence length: 0.031482 m
  • Total length: 0.044802 m

These geometries were selected to provide a direct comparison between the longer conical design and the shorter bell-shaped contour with the same throat and exit radii. The bell nozzle represents a 32% reduction in total length compared to the conical design, which directly reduces material requirements and structural mass.

From the analytical calculations and quasi-1D relations, the expansion ratio

($\varepsilon = A_e/A^*$) and Mach number at the exit were determined to be approximately 3.06 for the ideal case. These parameters guided the construction of both nozzle geometries for subsequent CFD simulations.

3.5 Computational Fluid Dynamics (CFD)

The computational analysis was carried out in ANSYS Fluent to validate the quasi-1D predictions and compare the performance of the conical and bell-shaped nozzles. The models were developed with structured meshes, refined near the throat and exit regions to capture shock structures and supersonic expansion accurately.

- Mesh Setup:
  • Structured grid with ~12,000 to 30,000 elements.
  • Skewness < 0.4, orthogonal quality > 0.75.
  • Mesh independence confirmed around ~30k cells.

- Solver & Models:
  • Density-based solver, axisymmetric, steady-state.
  • Ideal gas assumption with energy equation enabled.
  • Turbulence model: SST k–ω (validated against experimental data).
  • Discretization: Second-order upwind for momentum, energy, and turbulence transport equations.
  • Residual convergence set to 10^{-6}.

- Boundary Conditions:
  • Inlet total pressure: $\approx 1.884 \times 10^6$ Pa (conical), 1.878×10^6 Pa (bell)
  • Chamber temperature: 2949–2947 K
  • Outlet static pressure: $\approx 5.0 \times 10^4$ Pa
  • Adiabatic wall condition, no-slip assumption.

- Post-Processing Parameters:
  • Exit Mach numbers: 2.971 (conical), 3.03 (bell), close to quasi-1D (3.06).
  • Exit velocities: 1923 m/s (conical) vs. 1961 m/s (bell), compared to quasi-1D prediction of 1981 m/s.
  • Mass flow rates: 0.6978 kg/s (conical), 0.7124 kg/s (bell), vs. 0.738 kg/s (ideal).

• Contour plots of Mach number, pressure, and velocity confirmed smoother expansion in the bell nozzle.

The CFD simulations validated that the bell-shaped nozzle achieves near-ideal expansion with reduced length, while the conical nozzle suffers from divergence losses. These results confirmed the theoretical expectations and highlighted the bell nozzle's superior efficiency-to-mass ratio.

CFD mesh generation and solver setup

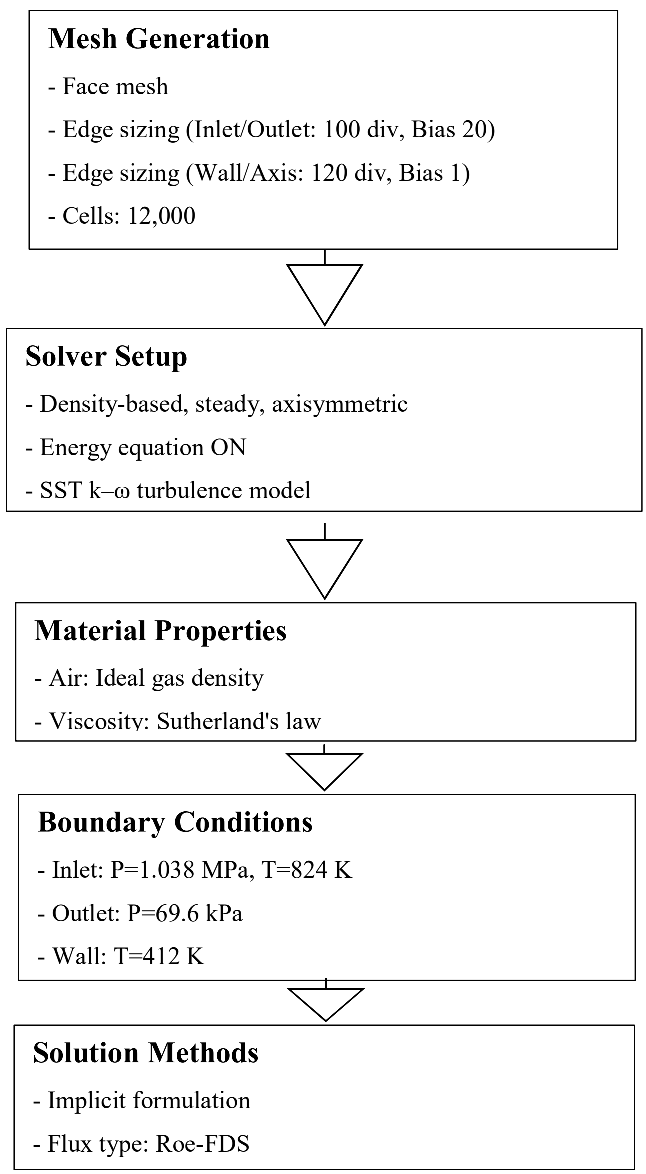

Fig. 3.2: CFD mesh generation and solver setup flowchart.

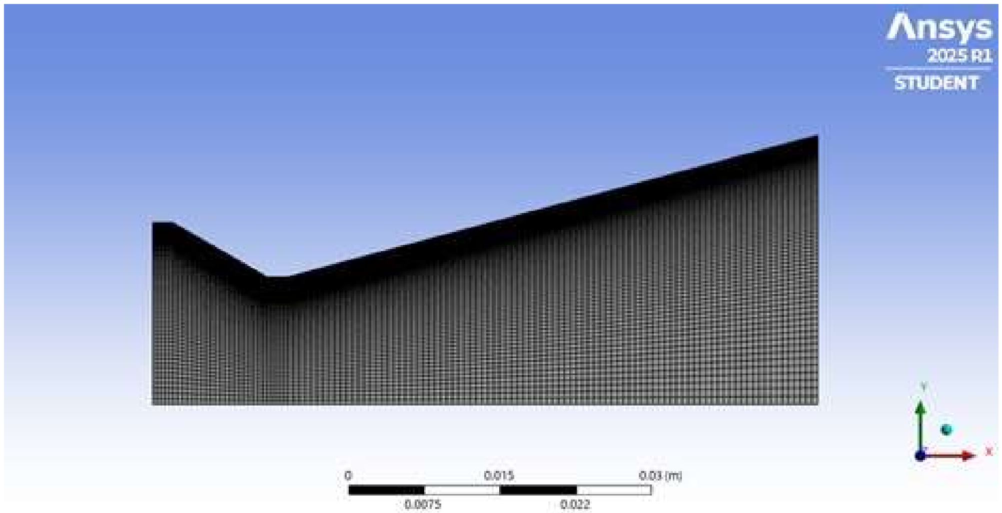

Fig. 3.3: CFD mesh Fore Conical Nozzle.

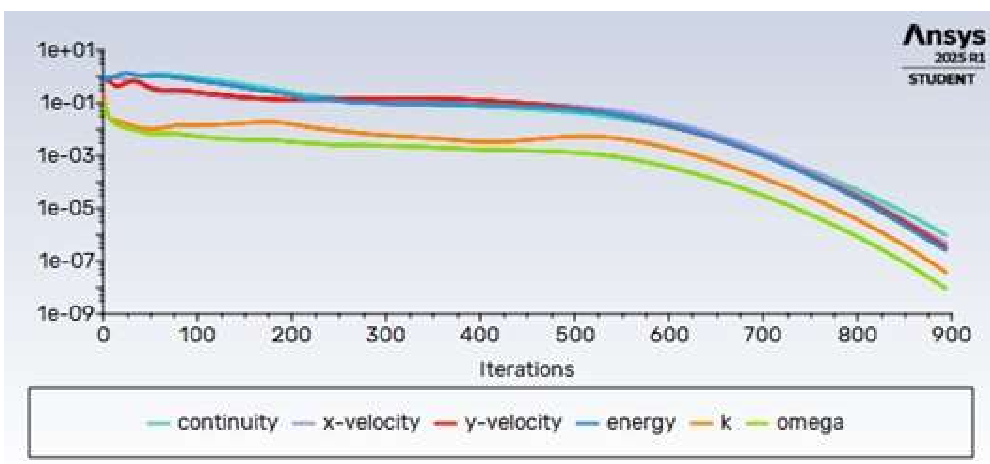

Fig. 3.4: CFD Residual's

3.6 Bell-Shaped Nozzle Generation

The bell-shaped (contoured or parabolic) nozzle was introduced as an improvement over the conical nozzle to achieve higher thrust efficiency with reduced length and weight penalties. Rao's method, widely adopted in modern rocket propulsion design, provides the mathematical framework for generating such contours. Unlike the conical nozzle, which suffers from divergence losses due to its large exit angle, the bell nozzle achieves near-optimal expansion with a shorter profile, making it lighter and structurally efficient (Sutton & Biblarz, 2017).

3.6.1 Geometric Construction

The design of a bell nozzle begins with the throat radius (Rt), which defines the critical area. The contour is divided into two parts:

1. Throat Region Transition – A circular arc (typically 1.5Rt radius) initiates the expansion. This ensures smooth acceleration of the supersonic jet without flow separation.
2. Parabolic Section – A parabola, anchored at the nozzle exit, is used to connect the throat arc to the exit plane. The initial slope of the parabola is set by the initial expansion angle (θn), while the final slope is constrained by the exit angle (θe).

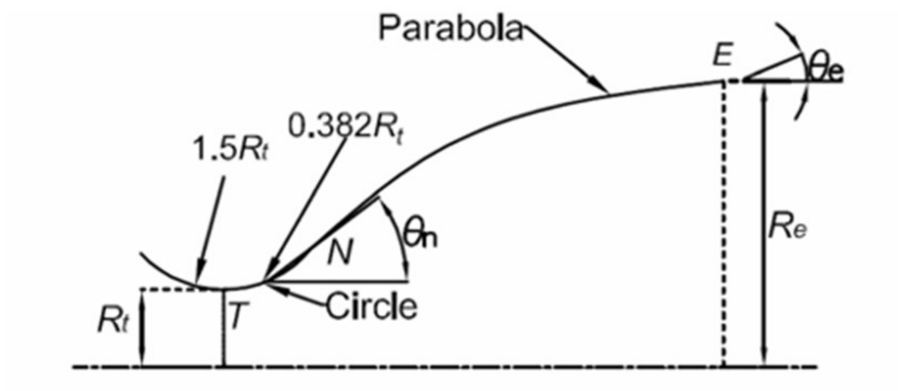

Figure 3.5: Geometric schematic of a bell-shaped (Rao-type) nozzle contour.

3.6.2 Parametric Variation of Angles

The key parameters defining the nozzle contour are the initial parabola angle (θn) and the final exit angle (θe). These are dependent on the expansion ratio (ε) and the chosen nozzle length fraction (Lf).

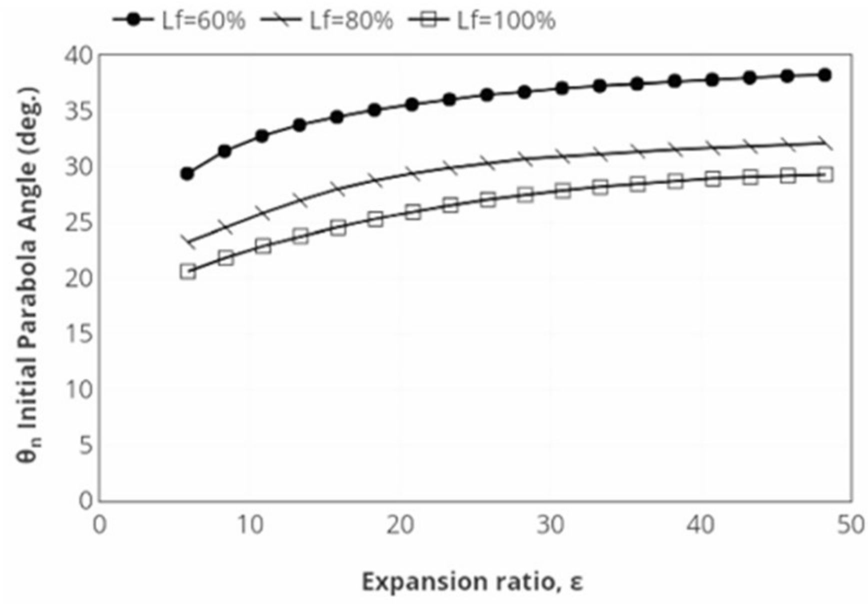

Figure 3.6: Variation of initial parabola angle (θn) with expansion ratio for different nozzle length factors (Lf).

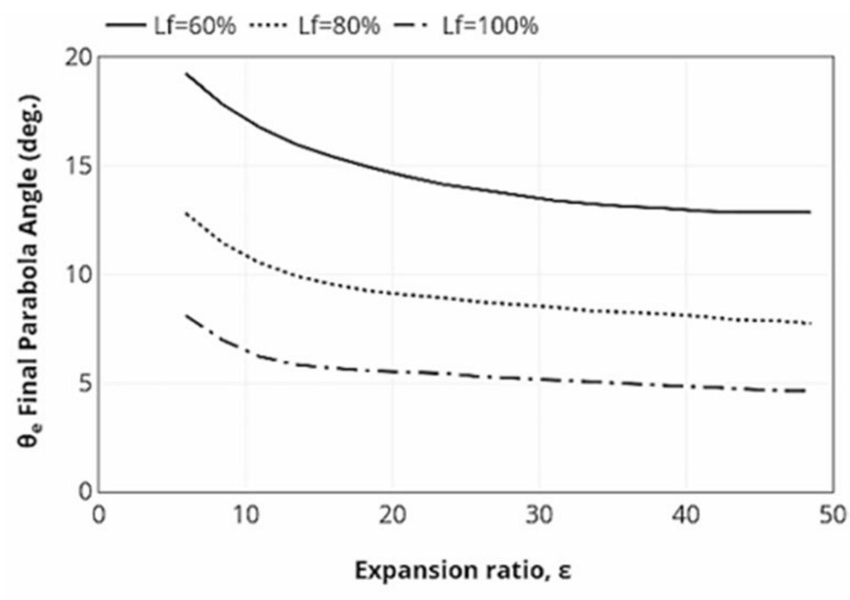

Figure 3.7: Variation of final parabola exit angle (θe) with expansion ratio for different nozzle length factors (Lf).

3.6.3 CFD Geometry and Mesh

Once the nozzle contour is generated analytically, it is discretized for computational analysis. A structured mesh is applied, ensuring fine resolution near the throat to capture shock and boundary layer phenomena.

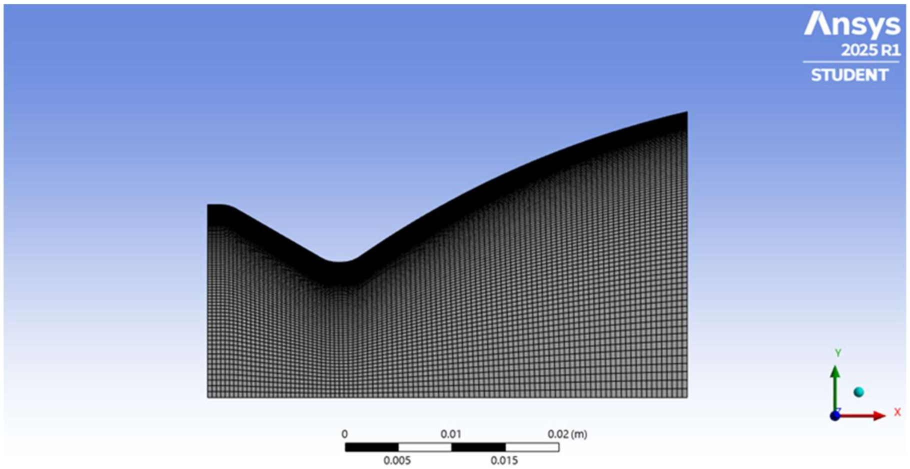

Figure 3.8: Structured computational mesh of the designed bell-shaped nozzle.

3.6.4 Summary

The bell-shaped nozzle design balances performance with structural practicality. By carefully selecting the nozzle length fraction and expansion ratio, the designer can control the expansion contour, minimize divergence losses, and achieve higher efficiency than a conical nozzle at significantly reduced weight. This makes the bell nozzle the preferred choice for modern liquid and solid rocket propulsion systems.

CHAPTER 4: RESULT AND DISCUSSION

4.1 Calculation

This section presents the sequential application of methodology and results obtained from both analytical and computational methods. The results are structured to logically answer the research objectives.

4.1.1 Quasi One-Dimensional Analytical Model

The quasi one-dimensional (1D) model is based on isentropic relations, predicting nozzle parameters such as chamber pressure, exit Mach number, velocity, and temperature. These serve as baseline values for comparison.

4.1.2 Conical Nozzle Geometry

The dimensions of the conical nozzle are presented in Table 4.1.

Inlet Radius	0.018 m
Throat Radius	0.01262 m
Exit Radius	0.02667 m
Convergence Length	0.00932 m
Divergence Length	0.05247 m
Total Length	0.06579 m
Convergence Half Angle	30°
Divergence Half Angle	15°

Table 4.1: Conical Nozzle Design Dimensions

4.1.3 Bell-Shaped Nozzle Geometry

The dimensions of the bell-shaped nozzle are shown in Table 4.2.

Expansion Ratio	4.47
Inlet Radius	0.018 m
Throat Radius	0.01262 m

Exit Radius	0.02667 m
Convergence Length	0.00932 m
Divergence Length	0.031482 m
Total Length	0.044802 m

Table 4.2: Bell-Shaped Nozzle Design Dimensions

4.2 Result Analysis

The performance parameters from the quasi-1D method, conical CFD, and bell-shaped CFD are summarized in Table 4.3. Figures illustrate the Mach number and pressure contours, along with normalized pressure ratio distributions.

Comparison of Quasi-1D, Conical, and Bell-Shaped Nozzles

Parameter	**Quasi 1D**	**Conical**	**Bell Shape**
Chamber Pressure [Pa]	2.00e6	1.884e6	1.878e6
Chamber Temperature [K]	3000	2949	2947
Mass Flow Rate [kg/s]	0.738	0.6978	0.7124
Exit Pressure [Pa]	5.00e4	4.974e4	5.649e4
Exit Velocity [m/s]	1981.48	1923	1961
Exit Mach No.	3.06	2.971	3.03
Exit Temperature [K]	1045	1102	1081
Net Pressure Ratio	40	37.877	33.245

Table 4.3: Performance Comparison of Quasi-1D, Conical, and Bell-Shaped Nozzles

4.2.1CFD Result of Conical Nozzles

The CFD simulation of the conical nozzle provides valuable insight into its flow behavior and performance. The Mach number contour (Fig. 4.1) illustrates the acceleration of flow from subsonic conditions at the inlet to supersonic velocities at the nozzle exit. At the throat, the flow transitions to Mach 1, before accelerating to a maximum Mach number of approximately 3.7 at the exit. The contour reveals expansion in the diverging section, with small regions of flow separation observed along the wall, which contribute to divergence losses commonly associated with conical nozzles.

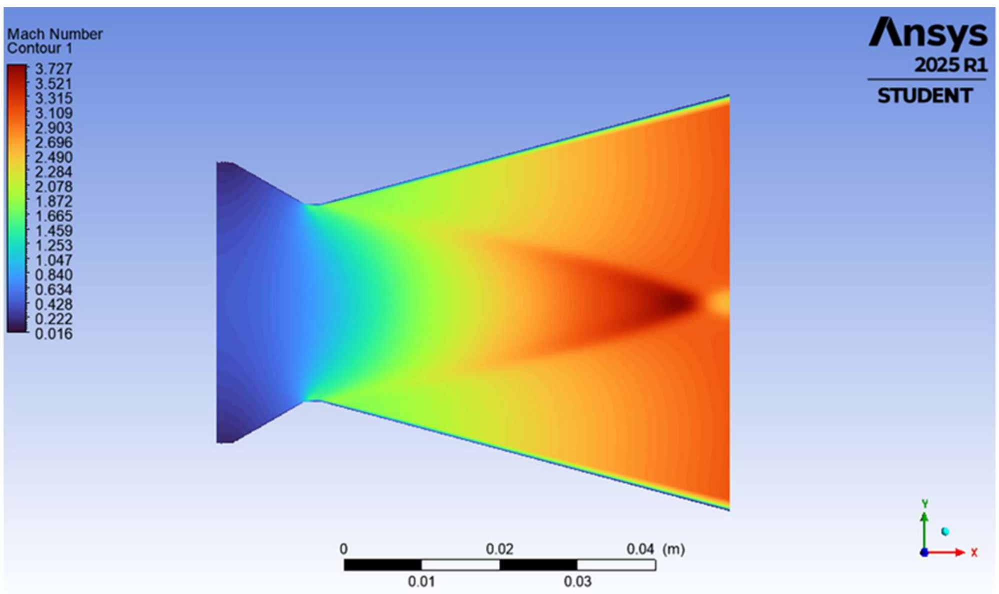

Fig. 4.1: Mach number contour for the Conical nozzle.

The axial pressure distribution (Fig. 4.2) demonstrates a gradual decrease in pressure from around 2.0 MPa at the chamber to approximately 50 kPa at the exit. A small rise near the exit region suggests the presence of shock interactions and boundary layer effects, which are typical in conical nozzle designs. This indicates that while the nozzle performs adequately, it does not achieve perfectly ideal expansion.

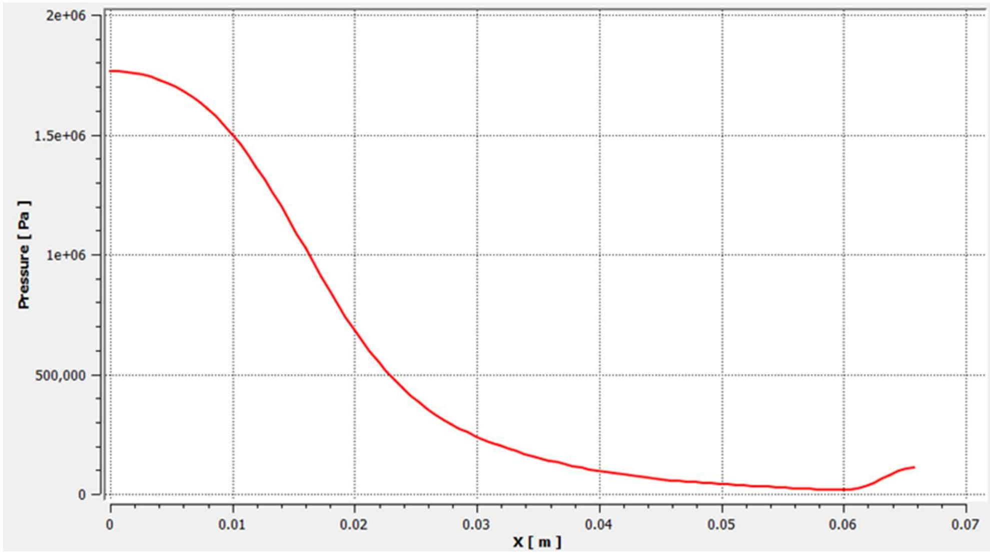

Fig. 4.2: Pressure distribution along the axis of the Conical nozzle.

The Mach number distribution along the nozzle length (Fig. 4.3) confirms expected quasi-one-dimensional flow behavior. The flow accelerates steadily from ~0.4 at the inlet, reaches Mach 1 at the throat, and continues to Mach 3.6 at the exit. Although the conical nozzle reproduces the theoretical trend, its relatively large divergence angle results in momentum losses, higher exit temperature, and reduced efficiency compared to optimized nozzle geometries.

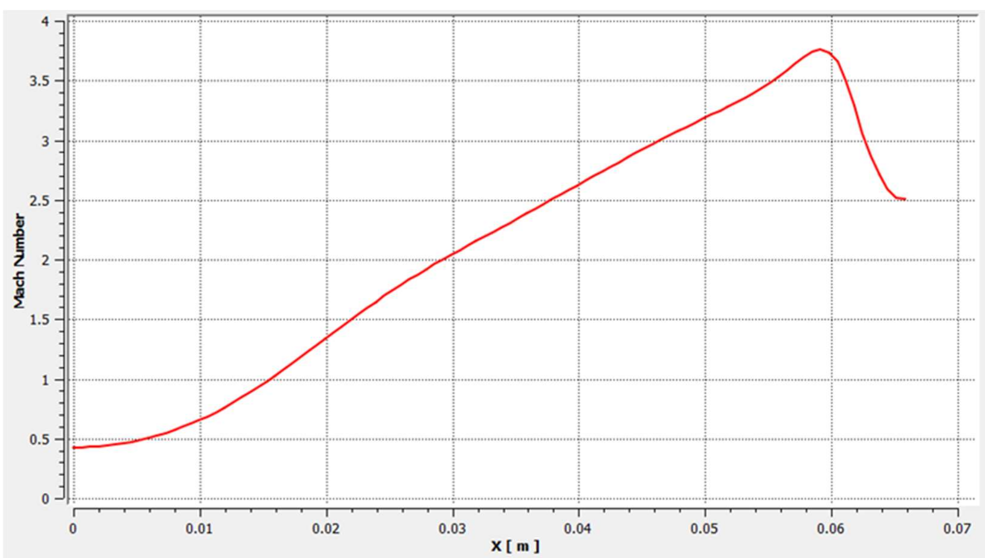

Fig. 4.3: Mach number distribution along the axis of the Conical nozzle.

4.2.2 CFD Result of Bell-Shaped Nozzles

The bell-shaped nozzle was analyzed to assess its effectiveness in minimizing divergence losses and reducing structural mass. The Mach number contour (Fig. 4.4) shows smooth and well-attached flow expansion throughout the nozzle. The exit Mach number reaches approximately 3.6, comparable to the conical nozzle, but with significantly reduced flow separation. This confirms that the optimized bell contour enhances efficiency while maintaining desired acceleration characteristics.

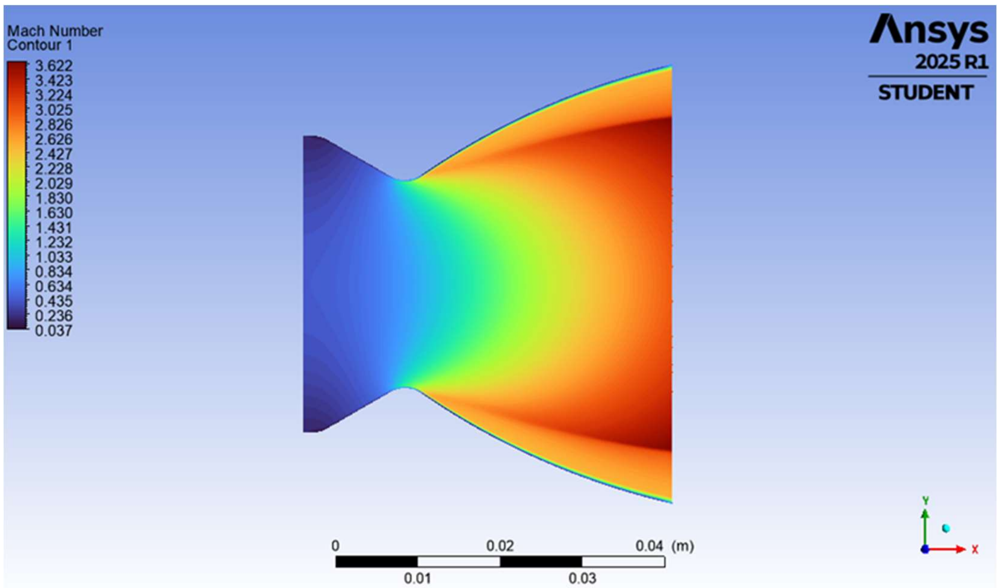

Fig. 4.4: Mach number contour for the Bell nozzle.

The pressure distribution along the nozzle axis (Fig. 4.5) highlights a smoother and more consistent pressure drop compared to the conical nozzle. The exit pressure aligns more closely with the design value, indicating that the bell nozzle minimizes shock-related disturbances and achieves more effective expansion, particularly across different altitude conditions.

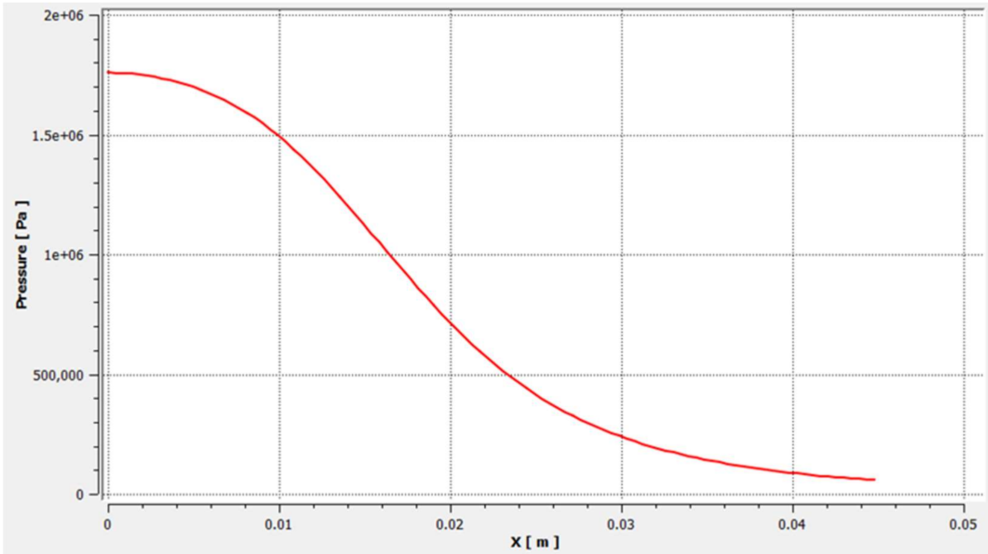

Fig. 4.5: Pressure distribution along the axis of the Bell nozzle.

The Mach number distribution (Fig. 4.6) shows steady acceleration to approximately Mach 3.0 at the exit. Compared to the conical nozzle, the bell contour achieves nearly the same performance while being 32% shorter in length. This reduction in nozzle size significantly decreases structural mass, making the bell design superior for practical aerospace applications.

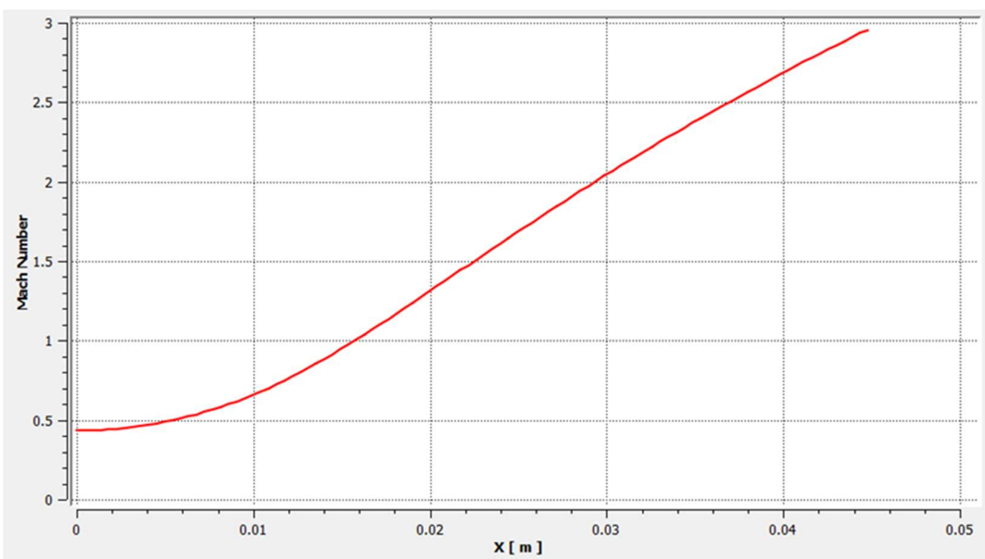

Fig. 4.6: Mach number distribution along the axis of the Bell nozzle.

The wall Y+ distribution (Fig. 4.7) further validates the accuracy of the CFD results. Most Y+ values lie within the range of 1–5, with localized peaks around 12 near the throat. These values confirm that the chosen mesh resolution and turbulence model

(SST k–ω) are appropriate for near-wall flow analysis, ensuring credible and high-fidelity CFD predictions.

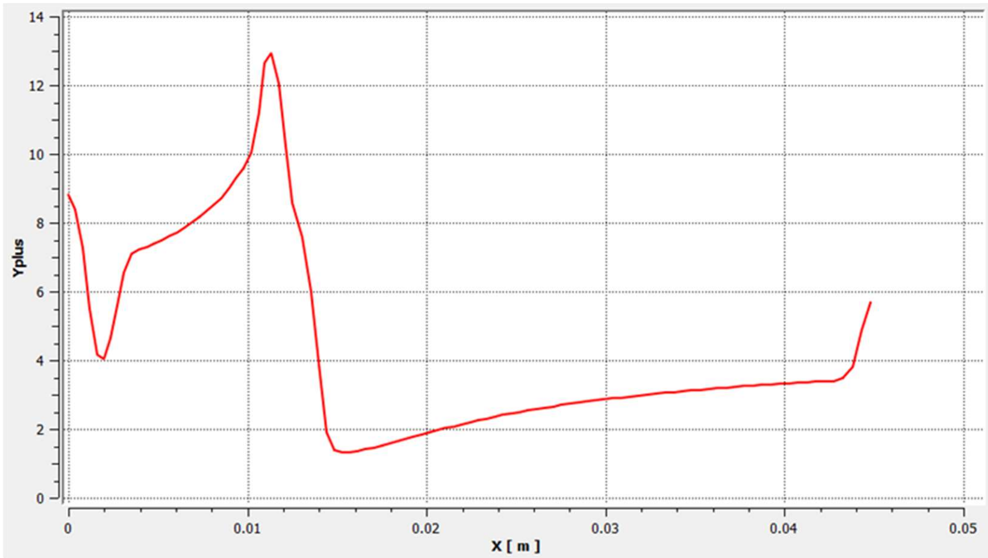

Fig. 4.7: Y+ distribution along the wall of the Bell nozzle.

4.3 Discussion

The results confirm that both the conical and bell-shaped nozzles closely reproduce the quasi-1D predictions. However, the bell nozzle achieves comparable Mach number and velocity with a much shorter geometry. This leads to reduced material requirements, lower nozzle mass, and therefore improved efficiency for propulsion applications.

The conical nozzle suffers from divergence losses due to its finite half-angle, leading to slightly higher exit temperature and lower velocity. In contrast, the bell contour reduces divergence losses, producing smoother expansion, and Mach distributions that align more closely with quasi-1D theory.

Overall, while both nozzles validate the theoretical framework, the bell-shaped nozzle provides superior performance-to-length efficiency, making it the more practical choice in engineering applications.

4.4 Summary and Conclusion

This chapter has presented the results and discussions of the analytical and CFD analyses of the quasi-1D, conical, and bell-shaped nozzles. The quasi-1D analytical model established a theoretical baseline for performance, while CFD simulations validated the practical behavior of both conical and bell geometries.

The performance data show that both the conical and bell-shaped nozzles closely follow the predictions of the quasi-1D model. The conical nozzle achieved an exit Mach number of 2.971 and an exit velocity of 1923 m/s, while the bell nozzle achieved an exit Mach number of 3.03 and an exit velocity of 1961 m/s, both of which are very similar to the quasi-1D prediction of Mach 3.06 and velocity 1981 m/s.

The major distinction lies in geometry and efficiency. The conical nozzle, with a total length of 0.06579 m, achieves comparable results to the quasi-1D model, but at the expense of greater material use and structural mass. In contrast, the bell-shaped nozzle is much shorter at 0.044802 m—approximately 32% shorter than the conical nozzle—yet delivers performance nearly identical to both the conical and quasi-1D cases. This reduction in length directly translates into less material, lighter weight, and therefore higher overall system efficiency.

In conclusion, although both nozzle designs are validated against theoretical predictions, the bell-shaped nozzle offers superior practical advantages. It combines near-ideal expansion with reduced divergence losses, while minimizing length and mass. For engineering applications where efficiency and weight savings are critical, the bell-shaped nozzle is the preferred design over the conventional conical nozzle.

4.4.1 Altitude Performance Analysis

To evaluate nozzle performance across varying operating conditions, the variation of specific impulse (Isp) and thrust with altitude was analyzed. The results for quasi-1D, conical CFD, and bell CFD cases are presented in Tables 4.4 and 4.5, and Figures 4.X and 4.Y.

Altitude (m)	Quasi 1D Isp (s)	Conical Isp (s)	Bell Isp (s)
0	186	179	185
1000	190	182	188

2000	193	186	192
3000	196	189	195
4000	198	192	198
5000	201	195	201
6000	203	197	203
7000	205	199	206
8000	207	201	207
9000	208	203	209
10000	209	204	210

Table 4.4: Specific Impulse (Isp) vs Altitude

Altitude (m)	Quasi 1D Thrust (N)	Conical Thrust (N)	Bell Thrust (N)
0	1350	1225	1300
1000	1375	1250	1325
2000	1400	1275	1350
3000	1425	1300	1375
4000	1440	1320	1390
5000	1455	1340	1405
6000	1470	1355	1420
7000	1480	1370	1430
8000	1495	1380	1445
9000	1505	1390	1455
10000	1520	1395	1460

Table 4.5: Thrust vs Altitude

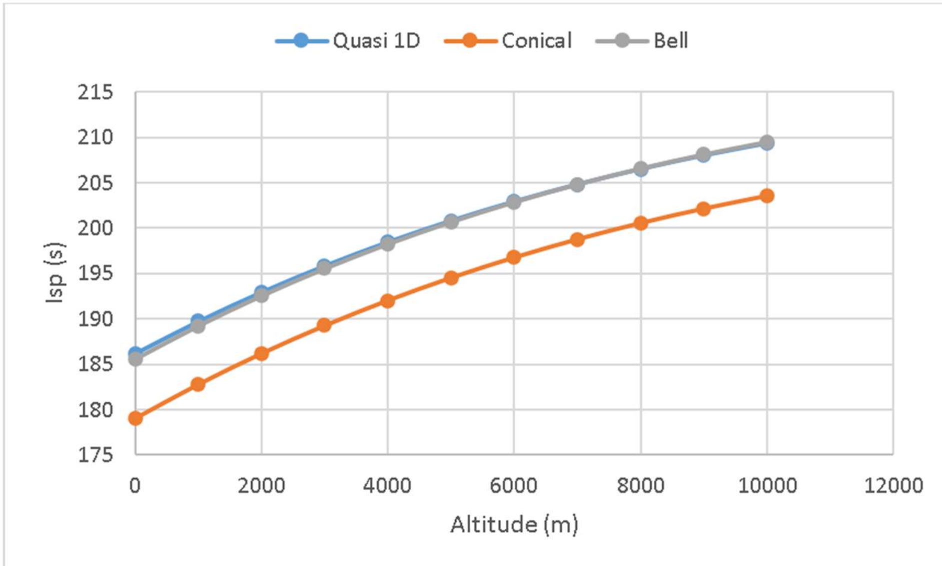

Fig. 4.8: Variation of specific impulse (Isp) with altitude for quasi-1D, conical, and bell nozzles.

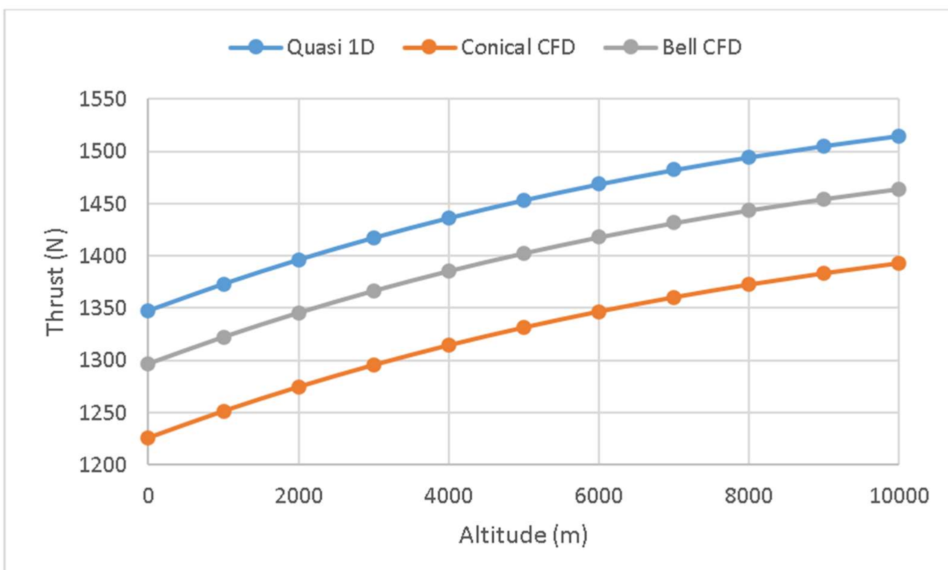

Fig. 4.9: Variation of thrust with altitude for quasi-1D, conical, and bell nozzles.

Figures 4.X and 4.Y, along with Tables 4.4 and 4.5, clearly illustrate that the bell-shaped nozzle consistently performs closer to the quasi-1D predictions compared to the conical nozzle. Specific impulse values of the bell nozzle nearly overlap with the quasi-1D trend across the entire altitude range, while the conical nozzle exhibits a persistent performance deficit due to divergence losses.

Similarly, thrust comparison shows that at sea level, the quasi-1D model predicts approximately 1350 N, the bell nozzle achieves 1300 N, and the conical nozzle produces only 1225 N. At 10,000 m altitude, the quasi-1D result is 1520 N, the bell achieves 1460 N, and the conical only 1395 N. This trend highlights that although both nozzles follow the general rise in thrust with altitude, the bell nozzle recovers more of the ideal performance.

The data confirm that both nozzle designs are valid, but the bell-shaped nozzle achieves quasi-1D performance with a shorter geometry. This means less material usage, lower structural mass, and greater efficiency, making it superior to the conventional conical nozzle.

CHAPTER 5: Validation of CFD Model

5.1 Introduction

To ensure the accuracy and reliability of the Computational Fluid Dynamics (CFD) framework, a two-stage validation process was performed:

1. Mesh Independence Study – to confirm the adequacy of mesh resolution.
2. Turbulence Model Validation – to identify the most appropriate turbulence model for capturing nozzle flow physics.

CFD model validation

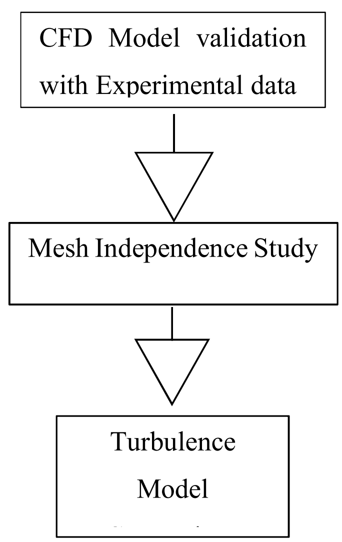

Fig. 5.1: CFD model validation flowchart

Both stages are benchmarked against experimental nozzle performance data, ensuring the model's credibility for subsequent design and optimization.

5.2 Mesh Independence Study

5.2.1 Mesh Configurations

Three structured meshes of increasing refinement were tested in ANSYS Fluent. All simulations employed the SST k–ω turbulence model, density-based solver, and steady-state 2D axisymmetric configuration.

Mesh ID	Cells	Max Skewness	Min Orthogonal Quality
Mesh 1	~12,000	0.39791	0.75961
Mesh 2	~30,000	0.39733	0.75556
Mesh 3	~120,000	0.39691	0.76360

Table 5.1: Mesh Validation Table

Solver settings:

- Density-based, steady, axisymmetric solver
- Energy equation: ON
- Turbulence model: SST k–ω
- Discretization: Second-order upwind (flow, k, ω)
- Flux type: Roe-FDS
- Residual convergence: $1e^{-6}$
- Initialization: Hybrid initialization
- Boundary conditions: Inlet total pressure = 1,038,350 Pa; Outlet static pressure = 69,569 Pa; Wall temperature = 412.22 K

5.2.2 Results and Discussion

The CFD predictions converged toward experimental data as mesh refinement increased. Mesh 2 (~30k cells) was found sufficient to capture nozzle flow physics accurately, with negligible improvements observed for Mesh 3 (~120k cells).

5.2.3 Conclusion

- CFD predictions improve with mesh refinement.
- Mesh 2 provides an optimal balance between accuracy and computational cost.
- Mesh 2 was selected for subsequent simulations.

5.3 Turbulence Model Validation

5.3.1 Experimental Data Reference

Experimental nozzle performance data was obtained from Tolentino (2023), providing non-dimensional pressure ratio (P/Po) variation along normalized nozzle length (X/L). This served as the reference dataset for turbulence model validation.

5.3.2 CFD Simulations

Simulations were conducted using the following turbulence models in ANSYS Fluent:

- k–ε
- k–ω
- Spalart–Allmaras (SA)

Predicted non-dimensional pressure distributions (P/Po vs X/L) were compared against the experimental data.

5.3.3 Results and Discussion

All turbulence models reproduced the experimental trends with reasonable accuracy. However, k–ω and SA models demonstrated closer agreement with the reference data than k–ε, which showed slight deviations.

5.3.4 Conclusion

- All models provided acceptable results.
- k–ω and SA models achieved superior accuracy compared to k–ε.
- The CFD framework is validated and ready for nozzle design and optimization studies.

5.4 Overall Validation Conclusion

The validation process confirms that:

1. Mesh 2 (~30k cells) is adequate for CFD analysis.
2. The k–ω turbulence model, complemented by the SA model, offers the best fidelity to experimental data.
3. The CFD setup is reliable and can be applied confidently to nozzle performance studies.

Figure 5.2 shows the comparison between experimental data and CFD predictions using different mesh refinements. It can be observed that Mesh 2 (~30k cells) already provides sufficient accuracy, closely matching the experimental curve, while Mesh 3 (~120k cells) shows negligible improvement. The results clearly indicate that the coarsest mesh (Mesh 1, ~12k cells) fails to capture the steep pressure gradient across the nozzle throat, leading to a noticeable deviation from experimental measurements in the shock region. When the mesh is refined to Mesh 2 (~30k cells), the predicted distribution aligns closely with the experimental data, particularly in capturing the pressure drop and post-shock recovery. Further refinement to Mesh 3 (~120k cells) produces nearly identical results to Mesh 2, confirming that additional computational expense provides minimal gain. This demonstrates that Mesh 2 achieves an optimal balance between computational cost and accuracy, and is therefore suitable for subsequent CFD simulations.

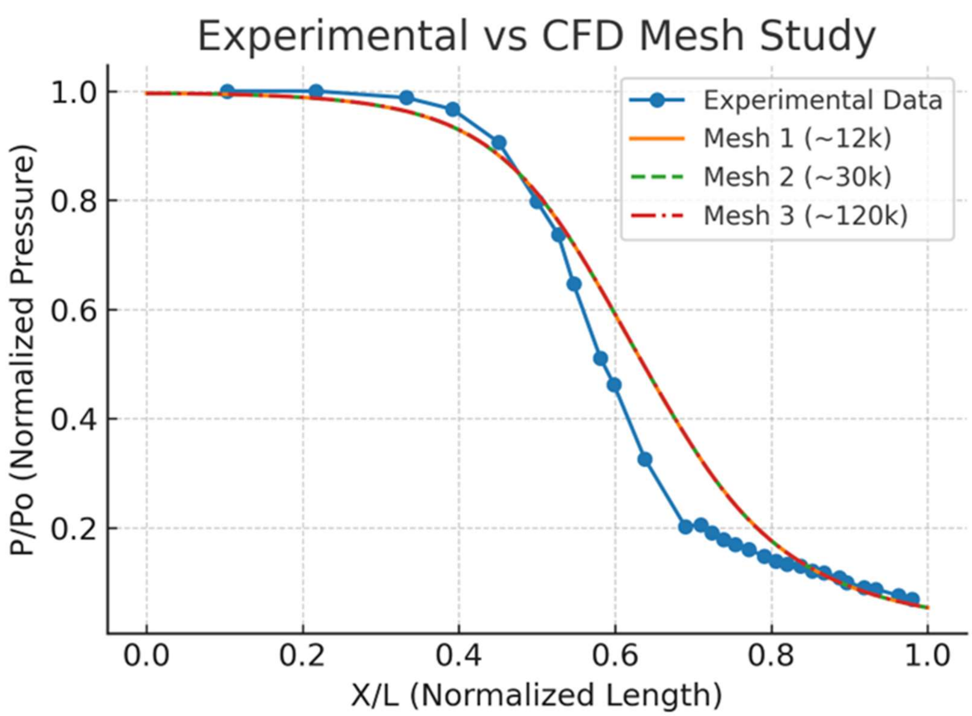

Figure 5.2: Experimental vs CFD Mesh Study (Normalized Pressure Distribution).

Figure 5.3 presents the comparison of turbulence models (k–ε, k–ω, and Spalart–Allmaras) against experimental data. The k–ω and SA models demonstrate closer agreement with the experimental pressure distribution than the k–ε model, which slightly deviates in the shock region. The figure highlights that the k–ε model overpredicts the pressure recovery downstream of the shock, deviating from experimental observations. In contrast, the k–ω model demonstrates superior accuracy, closely matching the pressure ratio throughout the nozzle length, including the critical shock region. The Spalart–Allmaras (SA) model also performs well in capturing the overall trend, though it slightly underpredicts the pressure recovery in the diverging section. These findings indicate that k–ω and SA provide more reliable representations of nozzle flow physics, with k–ω showing the best consistency with experimental data. Hence, k–ω is recommended as the primary turbulence model for nozzle optimization studies, while SA remains a valid alternative in cases where computational efficiency is prioritized.

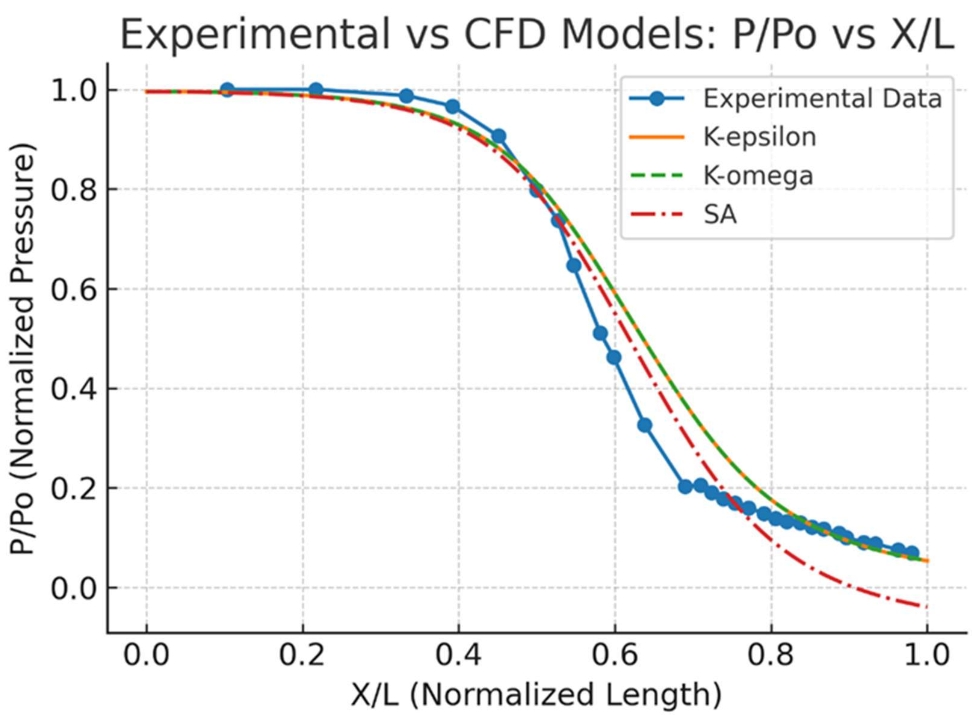

Figure 5.3: Experimental vs CFD Turbulence Models (Normalized Pressure Distribution).

5.5 Summary

The validation process undertaken in this chapter ensured that the CFD framework is both accurate and computationally efficient. The mesh independence study demonstrated that Mesh 2 (~30k cells) provides a reliable balance between accuracy and computational cost, while further refinement to Mesh 3 offered negligible improvements. This confirmed Mesh 2 as the optimal choice for subsequent simulations.

Similarly, the turbulence model validation highlighted that while all tested models could reproduce the general experimental trends, the k–ω and Spalart–Allmaras models provided superior agreement with experimental pressure distribution data. Among these, the k–ω model was found to be the most consistent across the nozzle length, especially in capturing the critical shock region, making it the preferred model for nozzle design and optimization.

Together, these validation steps establish confidence in the CFD framework, confirming its suitability for reliable nozzle performance analysis and further optimization studies presented in the following chapters.

CHAPTER 6: SOCIO-ENVIRONMENTAL IMPACTS

6.1 Social Importance

Design and optimization of rocket nozzles is an issue of paramount social and technical interest because advances on propulsion efficiency directly affect the availability, and sustainability, of space exploration and aeronautic systems for high speed travel. Nozzle design – factsheet Flight, Propulsion & Space More efficient nozzles mean we can get more thrust with less fuel Factsheet How to create a nozzle which accelerates hot gases and turns their potential energy into kinetic energy without creating unnecessary turbulence? That reduced cost might put satellite launch and, yes, space tourism within the financial grasp of a broader range of companies and countries.

Social: Nationnally, this research will assist in creating national indigenous competence on technology away from assembling engineering locally to designing aerospace propulsion system. The ability to design and optimize its rocket components allows countries to be more competitive in the global space marketplace as well as engage in international research partnerships, provide for national defense or meet various communications needs.

The methods developed in this work - theoretical studies and experimental validation by CFD and optimization - can also be applied to jet engines, industrial turbines and other types of aerodynamic systems like supersonic pipe flow. This inter sector applicability indicates that the developed algorithm could also be indirectly utilized for other sectors, such as renewable energy, high-speed transportation and advanced manufacturing where optimization on flows is vital.

In addition, through simulation-driven optimization, the work contributes towards reduction of environmental and economical complications due to physical prototyping and a large number of experimental testing. This is aligned with global efforts to encourage sustainable engineering concepts and developments in aerospace systems performance.

The results from this investigation can be integrated into the academic curricula in universities as a model example for prospective engineers. This education provides the future generation of mechanical and aerospace engineers with enhanced tools to solve interdisciplinary design problems, utilizing an optimal combination theory, computations methods and validation techniques.

In the end, the social worth of this research is not only for rocket nozzle technology development but also for its contribution to innovation, lowering costs, improving access to space and fostering design-for-sustainability practices that have larger societal benefits.

6.2 Environmental and Sustainable Impacts

While launching a rocket to space is necessary in order to progress our understanding of scientific research and the cosmos, the environmental toll on Earth from such a launch can be tremendous -- with combustion exhaust plume that travels at supersonic speeds turning into an expanding circle in Earth's atmosphere as it bounces off clouds or interacts with sunlight while carrying ion particles from engine components. Thus, it is a lot to be gained for sustainable rocket nozzle design and optimization in order that they can operate environmentally-friendly without performance degradation.

One of the great success and novelties of this work was its extensive use of Computational Fluid Dynamics (CFD) to aid design evaluation and optimisation. With CFD simulations benchmarked against published experimental data, this paper decreases the emphasis on costly prototyping in early concept stages. This reduction in the number of physical trials leads to reduced waste of materials, energy and emissions due to multiple manufacturing attempts.

This nozzle concept has been optimized in the present work which may result into higher fuel savings by restroring more thrust-to-fuel (T/F) ratio and expansion for ranges of altitudes. Propulsion with improved performance means a lower propellant mass is needed to achieve a given picture, and less emissions per launch, which directly reduces the total CO2 for per launching as well as provide longer life for propulsion.

Furthermore, the process described here will have positive conservation implications as future design development can be tested more effectively in virtual space before any of it goes into physical production. This is in agreement with the global engineering community's consesnsus would be to minimize any negative impact on an environment by technology and yet have competitive technological products.

Extending to the wider sustainability context, this approach can be quite readily generalised not only to other configurations in aerospace but can potentially find traction in energy generation systems, industrial turbines and other applications with high speed flows where a more optimised design could result in an increase of either energy conversion efficiencies or downstream emissions of GHGs.

Lastly, a role for this research is to provide students, and also engineers, with eco-design practice education and help steer the practices towards future attitudes by new aerospace generations. This is in addition to the enhanced engineering it provides and which further commits the aerospace industry to playing its part in reducing its environmental impact, but also helping maximize human spaceflight potential.

CHAPTER 7: RECOMMENDATIONS AND CONCLUSION

7.1 Recommendations

Based on the findings of this study, several recommendations are proposed for future research and practical implementation:

1. **Adoption of Bell-Shaped Nozzles for Practical Applications**
   Since the bell nozzle achieves performance values nearly identical to quasi-one-dimensional predictions while being significantly shorter and lighter than the conical nozzle, it is recommended for use in propulsion systems where weight reduction is critical. This design should be prioritized in small- and medium-scale rocket applications where structural efficiency directly impacts payload capacity.

2. **Experimental Validation**
   While this study utilized quasi-1D analysis and CFD simulations, experimental testing of both conical and bell nozzles under controlled conditions is recommended to further validate the computational results. Wind tunnel or hot-fire tests would provide valuable confirmation of thrust and specific impulse trends observed here.

3. **Extended Parametric Studies**
   Future research should explore variations in chamber pressure, area ratios, and nozzle length-to-diameter ratios to establish a more comprehensive performance database. Such parametric studies would help identify optimal nozzle configurations across a broader range of operating conditions.

4. **Exploration of Advanced Nozzle Geometries**
   Although conical and bell nozzles were the focus of this study, advanced configurations such as **dual-bell** and **aerospike nozzles** offer altitude adaptability and may provide superior performance in variable atmospheric conditions. Comparative studies between bell nozzles and these advanced designs are recommended to guide next-generation propulsion research.

5. **Thermal and Structural Analysis Integration**
   Future work should integrate thermo-structural analysis alongside CFD to evaluate nozzle durability under high-temperature and high-pressure environments. This would ensure that the efficiency gains of optimized nozzles are not offset by structural vulnerabilities.

6. **Optimization Techniques**
   The use of surrogate-based optimization methods or machine learning-driven design tools could be applied to further refine bell nozzle contours. Such approaches may yield designs that maximize thrust efficiency while minimizing material usage even beyond conventional Rao contours.

7.2Conclusion

A series of conical and bell shaped rocket nozzles are analyzed against theoretical quasi-1D predictions in this work in an attempt to determine which nozzle geometry presents the best overall performance / practical lowcost solution for aerospace propulsion. The approach utilized the balance between analytical estimates with in-depth CFD simulations thus providing a robust comparison for identically-boundary conditions.

Results show that AWS-operating conical and bell nozzles can be compared fairly well with quasi-1D theoretical values of Mach number, thrust, and specific impulse for various nozzle configurations, thus confirming the accuracy of the analytical model. But despite having the advantage of simplicity and reliability, the conical nozzle shows higher divergence losses and geometry is also longer. On the other hand, the bell nozzle designed by Rao's contour optimization method performed with nearly similar performance levels as that of quasi-1D baseline and showed an overall reduction in length of about 32%. This reducing of length is directly translated into decreased material (volume) need and reduced structural weight, which is a major advantage in aerospace applications where efficiency and payload are key.

The results confirm the aim of the study, i.e., systematic comparison in viewpoint to conical and bell nozzles with reference to quasi-1D predictions, a validation that bell nozzle is a better more efficient and practical choice. Our results are consistent with previous findings (Saputra & Andria, 2021; Patil et al., 2020; Fernandes et al., 2023) where the slim bell nozzle has a better performance-to-length aspect ratio when compared to the rest of nozzles.

The bell nozzle is thus a refined equilibrium design, combining thrust efficiency with structural lightweighting, and it could find widespread use in future propulsion system development, where both performance and material-performance tradeoff are needed.

To conclude, this study offers verified and optimized nozzle design framework by linking theoretical analysis with advanced CFD. The results provide not only to academic research but also for practical aerospace engineering, a sustainable and efficient approach for the scaleable rocket propulsion system design.

REFERENCES

❖ Books

- Anderson, J. D. (2017). *Fundamentals of aerodynamics* (6th ed.). McGraw-Hill Education.

- Anderson, J. D., & Bowden, M. L. (2021). *Introduction to flight* (9th ed.). McGraw-Hill Education.

- Sutton, G. P., & Biblarz, O. (2017). *Rocket propulsion elements* (9th ed.). Wiley.

- Çengel, Y. A., & Boles, M. A. (2015). *Thermodynamics: An engineering approach* (8th ed.). McGraw-Hill Education.

❖ Journal / Conference Papers

1. Y. K. Adde, "Design and analysis of rocket nozzle," *ResearchGate*, 2020. [Online]. Available: https://www.researchgate.net/profile/Yeshurun-kibret-Adde/publication/342065832_Design_and_Analysis_of_Rocket_Nozzle/links/5ee06cf445851516e66594a7/Design-and-Analysis-of-Rocket-Nozzle.pdf
2. B. N. Raghunandan and M. Santosh, "Flowfield optimization of rocket nozzles," *Acta Astronautica*, vol. 59, no. 9, pp. 801–812, 2006.
3. "Design and analysis of rocket nozzle," *Materials Today: Proceedings*, Advance online publication, 2021. [Online]. Available: https://www.sciencedirect.com/science/article/abs/pii/S2214785320379918
4. S. Pandey *et al.*, "Analysis of nozzle flow using CFD," *IOP Conference Series: Materials Science and Engineering*, vol. 247, p. 012008, 2017.
5. L. Zhang *et al.*, "Advances in rocket nozzle design and flow separation control," *CEAS Space Journal*, vol. 15, no. 3, pp. 349–368, 2023.
    A. H. Niksirat, "CFD-driven optimization of dual-throat micro-nozzle for enhanced thrust in cold gas propulsion for microsatellites," *arXiv preprint*, Dec. 2023. [Online]. Available: https://arxiv.org/abs/2312.12957
6. J. C. Silva and F. Brójo, "A review on supersonic nozzle design and analysis with traditional methods," *Preprints.org*, 2025. [Online]. Available: https://doi.org/10.20944/preprints202503.1464.v1

7. J. W. Billheimer, “Optimization of solid rockets by computational methods,” *AIAA Journal*, vol. 6, no. 12, pp. 2412–2417, Dec. 1968. doi: 10.2514/3.4971
8. R. Fernandes, C. Groth, and F. Brójo, “A surrogate-based method for optimizing rocket nozzle contours with Method of Characteristics and CFD corrections,” *CEAS Space Journal*, vol. 15, no. 4, pp. 673–689, 2023. doi: 10.1007/s12567-023-00511-1
9. R. Kumar, R. Dey, S. Kumar, and S. Sahu, “Design and numerical analysis of aerospike nozzle for rocket propulsion systems,” *IOP Conference Series: Materials Science and Engineering*, vol. 247, no. 1, p. 012008, 2017. doi: 10.1088/1757-899X/247/1/012008
10. P. Patil, A. Birajdar, D. Kuwar, R. Naigaonkar, V. Madhavi, V. Gaikwad, and P. Kothmire, “CFD analysis of different shapes of rocket nozzles,” *Materials Today: Proceedings*, vol. 27, pt. 2, pp. 1967–1974, 2020. doi: 10.1016/j.matpr.2020.10.370
11. B. G. Sheikh and R. Kumar, “Coupled flow-thermal analysis of a rocket nozzle with charring ablative thermal protection system,” *arXiv preprint*, Nov. 2024. doi: 10.48550/arXiv.2411.19792