Optimizing Rocket Propulsion Efficiency through CFD Simulation of Nozzle Designs

Veer Shukla [a], Reetu Jain [b]

[a] Student Researcher, V3-1401, Vista, The Address by Wadhwa Groups Lal Bahadur Shastri Road, Opposite R City Mall, Ghatkopar West, Mumbai, Maharashtra 400086

Email id: ***shuklaveer13@gmail.com***

[b] Mentor, On My Own Technology Pvt. Ltd. Mumbai, India - 400053

Email-id: ***reetu.jain@onmyowntechnology.com***

ORC-ID: ***0000-0002-7199-2823***

Abstract—*The Nozzle of the rocket is one of the most crucial parts for creating thrust. Many researchers are working on the different types of rocket nozzle to achieve the optimum thrust efficiency. In this research, CFD simulation was carried out. A 2D convergent-divergent nozzle was made in the FLUENT, further, a parametric setup was carried out. In this setup a total of 47 design points were created keeping the inlet and outlet length of the nozzle as variable. To obtain the correlation between the design point length and the functional parameters like temperature, pressure, and velocity a heatmap was created. In the result, it was found that when the nozzle inlet and outlet angle are near 45 degrees and the pressure at both ends maintains an equal ratio, the nozzle provides the optimum thrust.*

Keywords—CFD, 2D Nozzle design, Parametric design, data analysis.

1. Introduction

1.1. Background Research

The background research entailed delving into 17 different previous research papers, all based on rocket nozzles. They all assisted in the research process by providing the various faucets that may be important for optimizing rocket engines and aerospace propulsion efficiency. An aerospike nozzle is, however, presented for virtue in the initial report, showing a 2% increase over conventional bell nozzles. More learning is developed in these so-called optimization strategies for flow separation at different altitudes, which rely on the influence of factors like nozzle pressure ratio, thickness, and Mach number. Calculation tools like Computer Fluid Dynamics (CFD), which uses governing equations of mass, momentum, and energy in predicting fluid flow, helped determine the ideal nozzle divergent angle. Some other studies focused on the design of nozzles that work under non-design conditions. For instance, the following altitude-adapted supersonic nozzles were designed to work using methods integrated into numerical modeling: the prediction of specific impulse in dual bell nozzles and the effect of changes in the inlet and outlet diameters on thrust force and volumetric flow rate, overall demonstrating the effectiveness in using the numerical approach. Later efforts then focused on optimizing the nozzle for a rocket, performance evaluation of dual bell nozzle configurations with altitude variation, and performance improvement through multi-inlet combustion chambers. Pioneering efforts also moved towards the understanding of the flow dynamics toward the use of different pintle shapes for efficiency improvement in propulsion. This forms the basis of the research paper, which delves into the intricacies and possibilities of the rocket nozzle design and optimum performance.

1.2. Your research questions and objective

Using CFD, what effects do angles and convergent/divergent length have on rocket propulsion efficiency?

1.3. Significance to Industry

The aerospace industry constantly develops better propulsion systems for rockets to improve efficiency and performance at lower costs. Understanding how nozzle dimensions affect propulsion efficiency is crucial for designing designs that sustain peak efficiency at any altitude or environmental condition. Direct reduction in nozzle efficiency minimizes fuel usage, resulting in cost savings for longer-duration missions. Efficient nozzles also improve the thrust-to-weight ratio, allowing for larger payloads, which is beneficial for commercial satellite launch services and deep-space mission deployment. Research in Computational Fluid Dynamics can help achieve revolutionary breakthroughs in propulsion technology, giving organizations a competitive advantage. Understanding how nozzle dimensions affect performance can lead to designs that sustain peak efficiency at any altitude or environmental condition.

1.4. Scope of the Research

This research aims to understand how nozzle dimension variations, such as angles and lengths of convergent and divergent sections, affect rocket propulsion systems' efficiency. It will use Computational Fluid Dynamics (CFD) simulations to identify key nozzle features, such as angles of convergence and divergence, convergent length, and divergent length and ranges. The simulations will manipulate fluid dynamics in different nozzles, determining key performance indicators like thrust, specific impulse, Mach number, static pressure, and temperature distributions at the nozzle exit. The research will help identify the best nozzle dimensions for highest propulsion efficiency by comparing different configurations and finding a trade-off between design parameters. Validation of CFD results with experimental data or theoretical models will ensure accuracy and reliability. The research will provide practical insights and design recommendations for next-generation rockets, improving performance, cost-effectiveness, and adaptability to different operational conditions.

1.5. Literature Review

Sam Daniel Fenny et. al[1], aimed to improve the efficiency of the rocket nozzle by analyzing and developing aerospike nozzles and comparing them to the traditional bell nozzle. The methodology involves computer simulations modeling fluid dynamics to study the flow characteristics of the Aerospike Nozzle. The results showed that using the aerospike design results in a 2% increase in efficiency over the existing bell nozzle. **Ijas Muhammaed et al[2],** investigated flow separation in rocket nozzles under varying altitudes to understand the effect of ambient density and pressure in truncated ideal contour (TIC) nozzles. Researchers conducted a numerical analysis of the flow characteristics of a TIC at six different altitudes, considering cold and hot flow simulations. The study explores how the separation position shifts upstream with increasing altitude for a particular nozzle pressure ratio, revealing that separation position depends on the operation height for TIC rocket nozzles. The results capture the separation position at different altitudes, aligning with the experimental results. However, there is the possibility of inadequacy within the turbulence model to simulate flow phenomena accurately under the specific conditions observed in the experimental and numerical results. **Raman Ravi Shankar et al[3],** aimed to design a supersonic wind tunnel nozzle for Mach 2 testing using the Method of Characteristics and CFD simulations to achieve efficient supersonic flow, evaluating nozzle performance and drawing comparisons with turbulence models. The nozzle design aimed to achieve a Mach number range from 1.78 to 2 with excellent efficiency. The nozzle geometry was meticulously crafted in SolidWorks, and CFD simulation was conducted in Ansys Fluent R22 software. Mesh generation was structured, and thermodynamic properties of the gas (Nitrogen) used were considered for the simulations. Results showed excellent agreement between simulation and analytical data, showcasing exceptional accuracy and meticulousness in the study. **Khizar Ahmed Pathan et al[4],** aimed to see how characteristics like nozzle pressure ratio, nozzle thickness, and Mach number affected nozzle deformation and equivalent stress. With an emphasis on variables like nozzle thickness and factor of safety at different levels, the study investigates material optimization options for a range of flow and geometrical characteristics. The methodology involves CFD simulations across many scenarios with pressure profiles serving as the critical boundary conditions for static structure analysis. Additionally, Fear of Safety(FOS) is calculated for all simulations with findings shown in the Minitab software, this emphasizes the importance of CFD in studying fluid flow problems. The findings showed that nozzle thickness and FOS are directly proportional, meaning when nozzle thickness increases so does FOS. There is a research gap in this field because the specific topic of material optimization for nozzles has not received much attention in the literature to date.**Tushar Agarwal[5],** aims to optimise rocket engine nozzles at different divergence angles using CFD. The study specifically looks into simulating and solving fluid flow problems, investigating shock phenomena and velocity variations in the nozzle. The study uses a 2-dimensional asymmetric model and ANSYS FLUENT software for CFD analysis to consider the diverging angles of 5°, 10°, and 15° . The plotting and calculation of parameters like Mach Number, Static pressure, velocity, and pressure drop resulted in the ideal divergent angle of 15° divergence angle which eliminated shocks and satisfied thrust demands. However, despite advances in computational and experimental skills, there are still unresolved problems in the field, suggesting possible limits in the current work.

Sidali Haif et al[6], aimed to improve the performance of aerospace propulsion nozzles in non-design work conditions such as Sea-Level and High-Altitude modes where they operate outside their optimal design parameters. The methodology involved developing altitude-adapted axisymmetric supersonic nozzles for greater performance, utilizing fluent code for simulations of viscous flow, conducting inviscid simulations to compare outcomes with a simulation showing a good agreement, employing the method of characteristics and Prandtl-Meyer Expansion for design optimization, comparing the performance of the E-D nozzle with a plug nozzle under various design conditions. The transition process in the dual bell nozzle requires active control due to a gap in the transition pressure ratio at lower values. **B Sham et al[7],** addressed the design and structural analysis of rocket motor nozzles, optimization of exhaust nozzle contour for thrust, and numerical simulation of supersonic conical nozzle exhaust for thrust optimization. The CATIA V5 and ANSYS software were used for designing and analyzing the nozzles with focus on optimizing the thrust and doing a structural analysis. **Akella and Sai Raja[8],** aim to develop a numerical model for predicting the specific impulse in the dual bell nozzle of a solid rocket motor, crucial for missile and rocket system design. The methodology involved CFD simulations conducted on the bell nozzle at various altitudes and conditions, including sea level; 4, 8, 12,16,20 km above sea level and vacuum. The simulations used an inlet pressure of 7.3Mpa, a temperature of 3410 K, and a mass flow rate of 5.96 kg/s of combustion gases. A mesh with 2,26,500 cells was used, and the simulation involved 7,680 iterations. The results included a graphical representation of the relationship between altitude and specific impulse, thrust force, outlet velocity, and maximum pressure, providing a visual understanding of the performance variations at different altitudes. Some gaps in the research and limitations with the paper include a lack of validation with experimental data, assumptions and simplifications made in the numerical model that may impact the accuracy of the results, and limited discussion on the practical applications of the study. **Kiral and Hizarci aim[9],** to investigate the effect of varying the inlet and exit diameters on thrust force and the volumetric flow rate of convergent conical nozzles commonly used in applications like air-blowing guns and rockets. The study does theoretical analysis using Euler's equations, numerical simulations with ANSYS Fluent, and experimental measurements to compare results. The governing equations of continuity, momentum, and energy are discretized using the finite volume method and solved with a pressure-based coupled solver in ANSYS Fluent for numerical simulations. The breakthroughs include the discharge coefficient for Type 3 being closer to 1, indicating better agreement between numerical and experimental results. Numerical results are generally closer to experimental data than analytical results. Then, the study ranks the results in order of accuracy for theoretical and numerical approaches, showing the effectiveness of numerical simulations in predicting experimental outcomes. Some gaps include negligence of viscous losses in the theoretical analysis which could impact the accuracy of the results, and the complexity of convergent-conical nozzle flows due to compressibility effects, shock waves, turbulence, and boundary layers, which are not fully captured in the theoretical and numerical approaches. **Jayaprakash et al[10],** aimed to design a rocket C-D nozzle for various purposes like manned missions or satellite launches, focusing on the nozzles for achieving optimal performance. The design process involved utilizing knowledge of compressible flow to determine temperature, pressure, and velocity at different sections of the nozzle. The shape, insulation, and cooling arrangements were based on these values, with thrust calculations performed based on the design. Theoretical calculations and Computational Fluid Dynamics (CFD) simulations were conducted to validate the design. The methodology included comparing results obtained from theoretical calculations and CFD simulations. The pressure drop at the exit was crucial for maximizing thrust, with pressure decreasing from inlet to exit due to shock formation at the throat. The exit static pressure was 0.927 bar, the exit temperature was 1468 K, and the results from CFD and theoretical approaches were found to be almost identical. One potential limitation of the study could be the lack of detailed discussion on the specific challenges faced during the design process or any unexpected findings that could have influenced the results. This information could have provided a deeper insight into the design and analysis of the rocket C-D nozzle.

Chada et al.[11], conducted a study to investigate the configuration and operation of a dual bell nozzle across different altitudes through the application of Computational Fluid Dynamics (CFD). The research entailed the design and assessment of dual bell nozzles at altitudes ranging from 2 to 15, with specified inlet conditions including a mass flow rate of 5 kg/s, inlet temperature of 1200 K, and NRP value of 12. The utilization of SolidWorks 2020 facilitated the simulation process by adjusting the fluid's inlet pressure and accounting for ambient pressure variations corresponding to altitude changes. The flow analysis was executed on a three-dimensional model, emphasizing the enhancement of thrust generation in the dual bell nozzle design at varying altitudes while aiming to minimize the occurrence of shock waves. The anticipated outcomes are anticipated to provide valuable insights into the nozzle's performance, effectiveness, and thrust attributes under diverse circumstances. However, some notable areas for further research include the absence of experimental validation to corroborate the CFD simulations, limited discourse on the practical application or real-world experimentation of the dual bell nozzle design, and a lack of comparisons with other nozzle configurations or alternative methodologies for performance assessment.The research done by **Bhaskar and Sahu[12],** focuses on understanding the operational efficiency of convergent-divergent nozzles with a multi-inlet combustion chamber in a rocket engine. The main object is to gain insight into flow dynamics, pressure, temperature, and velocity distribution in the system. The analysis uses basic equations about continuity, momentum, and energy. The second-order upwind scheme is used for discretizing the equations relating to momentum and energy.

The computer model is compared to existing research to ensure its validity. 3 different configurations are used: a convergent-divergent nozzle with a combustion chamber with 4 inlets, 6 inlets, and an extended combustion chamber length. The configuration with 6 inlets exhibits a 26.3% rise in the exit velocity and the modified combustion chamber shows a 38.6% increase. However, a more thorough analysis of the effect of the metrics on the system's performance should have been done. **Chada et al aim[13],** to understand the flow dynamics in nozzles and the usage of pintle shapes to improve the propulsion efficiency. The research is centered on the development and optimization of A pintle nozzle for a rocket. The method employs CFD simulations to understand the flow inside the nozzle. Parametric analysis of algorithms for shape optimization was used to boost the nozzle's effectiveness. Results encompass information on the impact of nozzle configurations on the flow metrics like rate, velocity, and pressure, affecting propulsion efficacy. However, the research lacks specifics like the exact CFD software or algorithms used. A constraint may be present in putting the optimized nozzle designs in real-world rocket systems indicating a research gap for future research. **Joshi et al[14] ,** work to address the challenge of making nozzles efficient for rockets that operate in atmospheres other than Earth's, with gases like hydrogen, helium, methane, and carbon dioxide present. The method involves the designing and comparing of various nozzles using CFD analysis in ANSYS fluent and Catia V5 considering factors such as Mach velocity, Temperature, and pressure are considered in the analysis. A series of CFD simulations are conducted to analyze the efficiency of nozzles under different temperatures. The results show that the velocity components at the inlet start at Mach 1.61x10^-1, then increase to 8.23x10^-1 and finally reach supersonic flow with a value of 2.81. A constraint is that the research paper lacks details on the performance metrics used to evaluate the efficiency of designed nozzles. More information on these criteria will improve the paper's clarity and usefulness. **Khalid et al[15]** aimed to reduce the problems in flow characteristics like pressure, Mach number, and velocity in a converging-diverging nozzle by using CFD with the k-ε turbulence model. The method employed uses the Finite Volume Method solver within CFD to simulate flow within the nozzle. Nozzle configuration is done using Designmodeler and flow examination is done using ANSYS Fluent. The presence of a shockwave in the nozzle divergent section is explored using CFD. The outcomes of the experiment are compared to theoretically computed results, showing a minimal difference. This praises the precision of the simulation. A constraint could be the assumption and usage of ideal conditions in simulations.

1.3. Motivation and Novelties

Rocket propulsion efficiency is paramount in advancing aerospace technology, driving the need for innovative approaches to nozzle design. Traditional nozzles, while effective, often fall short in optimizing thrust efficiency under varying operational conditions. This research, through Computational Fluid Dynamics (CFD) simulation, aims to address these limitations by exploring a wide array of nozzle configurations. By conducting detailed simulations and parametric studies on a 2D convergent-divergent (CD) nozzle, this study seeks to uncover optimal geometrical parameters that maximize thrust while maintaining stable operational characteristics. The ultimate goal is to contribute to the development of more efficient propulsion systems, reducing fuel consumption and improving payload capacity, thereby benefiting both commercial and scientific aerospace endeavors.

The novelty of this research lies in its comprehensive parametric analysis of rocket nozzle design using CFD. Unlike previous studies that often focus on specific nozzle types or limited design variations, this research investigates 47 distinct design points by varying the inlet and outlet lengths of a 2D CD nozzle. The creation of heatmaps to correlate design parameters with functional outcomes such as temperature, pressure, and velocity is particularly innovative. This approach not only identifies optimal geometrical configurations but also provides a detailed understanding of how these configurations impact overall propulsion efficiency. The findings, notably the discovery that nozzles with inlet and outlet angles near 45 degrees and balanced pressure ratios yield optimal thrust, offer valuable insights that can inform future nozzle design and optimization strategies in the aerospace industry.

1.4. Project Execution Timeline

To achieve the aims and objectives of the research, my work was divided into 5 phases.

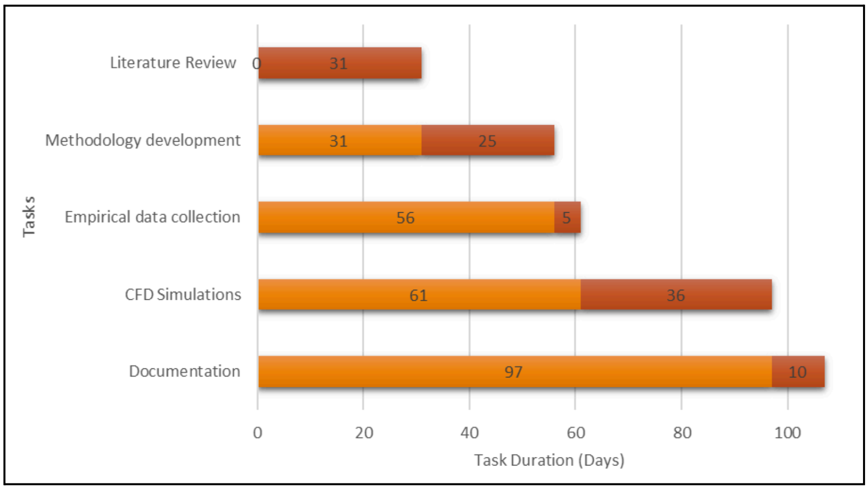

Fig. 4: Gantt chart for the research project

My research was structured into five phases with specific timelines to ensure systematic progress and thorough analysis. The literature review began on 15 March 2024, and concluded on 15 April 2024, laying a solid foundation of existing knowledge and helping me understand what knowledge I can build on in this project. Methodology development commenced immediately after, from 15 April to 10 May 2024, wherein I debated and discussed the approach we want to take and established the research framework. During both these periods I also learnt about CFD and how to carry out such simulations.

Empirical data collection was carried out from 10 May to 15 May 2024, to verify the theoretical calculations of airship volume. Subsequently, CFD simulations were conducted from 15 May to 20 June 2024 optimising the airship design. Most simulations were run overnight because they took extremely long. Finally, the documentation phase spanned from 20 June to 30 June 2024, wherein I covered the research findings comprehensively.

Task	Start Date	End Date
Literature Review	15 April 2024	6 May 2024
Methodology development	3 April 2024	10 May 2024
Empirical data collection	10 May 2024	15 May 2024
CFD Simulations	15 May 2024	20 June 2024
Documentation	20 June 2024	30 June 2024

Table 1: Timeline and Duration of Project

2. The Case Study

This section provides an in-depth examination of the case study, detailing the core problem, the assumptions made, and the mathematical modeling employed. By analyzing the optimization of rocket propulsion efficiency through CFD simulation of nozzle designs, the study aims to elucidate the relationships between nozzle geometry and performance, thereby offering insights into enhancing propulsion systems in aerospace engineering.

2.1. Problem Statement

Rocket propulsion efficiency is a critical factor in the performance and cost-effectiveness of aerospace missions. Traditional nozzle designs, while effective, often fail to maximize thrust efficiency under varying operational conditions. This research addresses the core problem of optimizing rocket nozzle design to achieve superior propulsion efficiency. By employing CFD simulations to analyze a range of 2D convergent-divergent nozzle configurations, the study seeks to determine the optimal geometrical parameters that enhance thrust while maintaining stability in temperature, pressure, and velocity distributions. The ultimate goal is to develop nozzle designs that significantly improve propulsion efficiency, reduce fuel consumption, and increase payload capacities for both commercial and scientific missions.

- **Performance and Cost-Effectiveness**: Achieving high propulsion efficiency is crucial for enhancing the performance and cost-effectiveness of aerospace missions.
- **Limitations of Traditional Nozzle Designs**: Conventional nozzles often fall short in maximizing thrust efficiency under varying conditions.
- **Optimization Objective**: The research aims to optimize rocket nozzle design to achieve superior propulsion efficiency through CFD simulations.
- **Geometrical Parameter Analysis**: The study analyzes a range of 2D convergent-divergent nozzle configurations to identify the optimal geometrical parameters.
- **Thrust Enhancement**: The goal is to enhance thrust while maintaining stability in temperature, pressure, and velocity distributions.
- **Overall Impact**: Developing optimized nozzle designs that improve propulsion efficiency will lead to reduced fuel consumption and increased payload capacities, benefiting both commercial and scientific aerospace missions.

2.2. Assumptions

To facilitate a structured and focused analysis, several assumptions were made:

1. The fluid flow through the nozzle is compressible and follows the principles of gas dynamics.
2. The specific heat ratio (Gamma) is assumed to be 1.4, based on standard atmospheric air properties.
3. The gas constant per unit weight (R) is 287 J/Kg-K, which is typical for air.
4. The specific heat at constant pressure (Cp) is 1005 J/Kg-K, and at constant volume (Cv) is 718 J/Kg-K.
5. The chamber pressure is maintained at 3000K for all simulations.
6. The mass flow rate through the nozzle is constant at 10 Kg/s.
7. The simulations assume steady-state conditions to simplify the computational model.
8. Boundary conditions are defined based on typical operating environments of rocket nozzles, with inlet and outlet pressures being variable but within realistic ranges.

2.3. Problem Formulation

By following this structured approach, the study aims to identify the key design parameters that maximize rocket propulsion efficiency and provide practical recommendations for future nozzle designs in aerospace applications.

The problem is formulated mathematically and methodologically through the following steps:

1. **Geometric Design and Setup**:
    - **Initial Design**: A base 2D convergent-divergent nozzle design is created using ANSYS Fluent. The design incorporates standard dimensions and geometries derived from existing literature and theoretical models.
    - **Design Variations**: The nozzle's convergent and divergent lengths are systematically varied to generate 47 different design points. This extensive parametric study allows for a comprehensive analysis of how geometric changes impact nozzle performance.
2. **CFD Simulation**:
    - **Solver Selection**: Using CFD, the fluid dynamics within each nozzle design are simulated. A density-based solver is chosen to accurately capture the compressible flow characteristics of the high-speed gases within the nozzle.
    - **Turbulence Model**: The k-epsilon turbulence model is employed to ensure precise prediction of boundary layer behavior and overall flow dynamics, critical for accurate thrust calculation.
3. **Meshing and Boundary Conditions**:
    - **Mesh Generation**: Optimal meshing is performed to balance computational efficiency and accuracy. The mesh is refined in areas with high gradients, such as near the nozzle throat, to capture detailed flow features.
    - **Boundary Condition Definition**: Inlet and outlet boundary conditions are specified based on realistic operating environments. The inlet conditions include specified pressure and temperature, while the outlet conditions allow for varying pressure to simulate different operational scenarios.
4. **Data Collection and Analysis**:
    - **Output Parameters**: Key performance indicators, including thrust, specific impulse, Mach number, static pressure, and temperature distributions at the nozzle exit, are collected for each design configuration.
    - **Dataset Creation**: A comprehensive dataset is created from the simulation results, with outlier readings filtered out to ensure data integrity. This dataset forms the basis for subsequent analysis.
5. **Optimization Criteria**:
    - **Performance Metrics**: The optimization focuses on maximizing thrust and specific impulse while ensuring stability in pressure and temperature distributions. Designs are evaluated based on their ability to meet these criteria under different operational conditions.
    - **Trade-Off Analysis**: A trade-off analysis is conducted to balance the competing objectives of thrust enhancement and stability. This involves identifying designs that provide the best overall performance, even if some individual metrics are not maximized.
6. **Iterative Refinement**:
    - **Design Iteration**: Based on initial findings, further iterations of design and simulation are performed to refine the nozzle geometries. This iterative process ensures that the final designs are optimized for peak performance.
    - **Sensitivity Analysis**: A sensitivity analysis is conducted to determine how small changes in design parameters affect overall performance. This helps in identifying the most critical design variables.

3. Methodology

3.1. Analytical Calculation:-

To achieve the optimum efficiency of the rocket nozzle by varying the dimension parameters, analytical calculations were performed at the start to get a base 2D CD nozzle. At the beginning of the calculation, some constants were assumed like Gamma (Specific heat ratio)=1.4, The gas constant per unit weight(R)= 287 J/Kg-K, the specific heat at constant pressure (C_p)= 1005 J/Kg-K, Specific heat at constant volume (C_v) = 718 J/Kg-K. Po - chamber pressure, Pe - expanding-to pressure. The literature also showed that optimum thrust can be achieved when the expansion ratio is between 0.7 and 1 atm. To keep this objective, the mass flow rate was kept at 10Kg/s and the chamber pressure was considered 3000K.

3.2. CFD Analysis:-

Based on the above values the conversion and diversion dimensions of the Nozzle were calculated and a base CD Nozzle design was made in the ANSYS geometric section. The same 2D Nozzle design is shown in Figure 1 below.

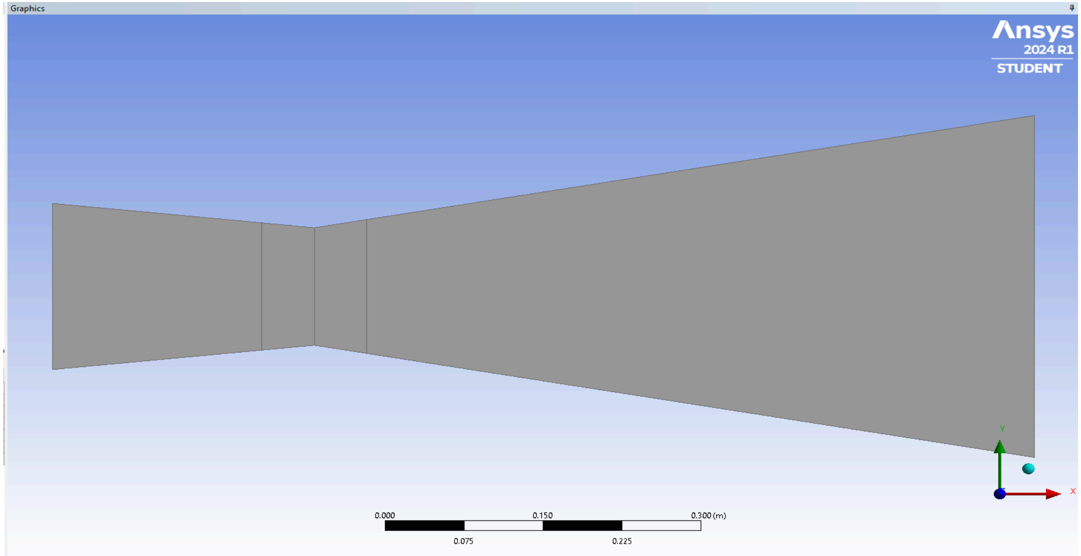

Fig 2: 2D CD- Nozzle geometry

The 2D CD-Nozzle shown in Figure 1 was used as a base nozzle to achieve the optimum expansion ratio. A parametric setup was carried out in the second stage shown in Figure 2. The 47 types of design parameters were set, keeping divergent and conversion length as variables.

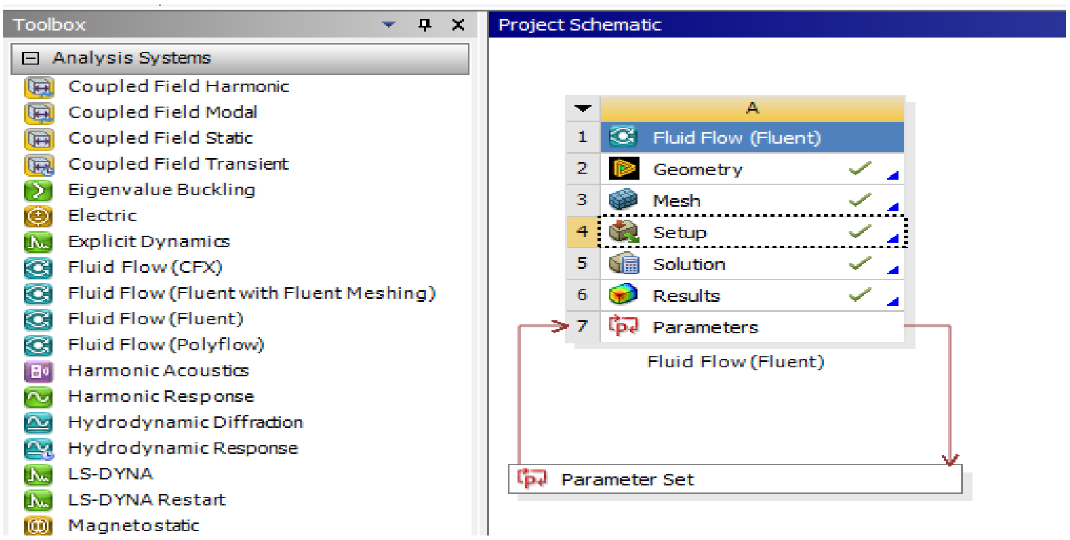

Fig 3: Parametric Setup in ANSYS Workbench

In the next step, meshing was performed while keeping the optimal orthogonality and skewness ratio. This will not only help to reduce CPU processing power but also reduce the number of reiterations. Different naming sections are allotted to the 2D geometry to give the boundary condition. Figure 3 shows the meshing structure of the geometry with inlet and outlet sections represented with an arrow vector.

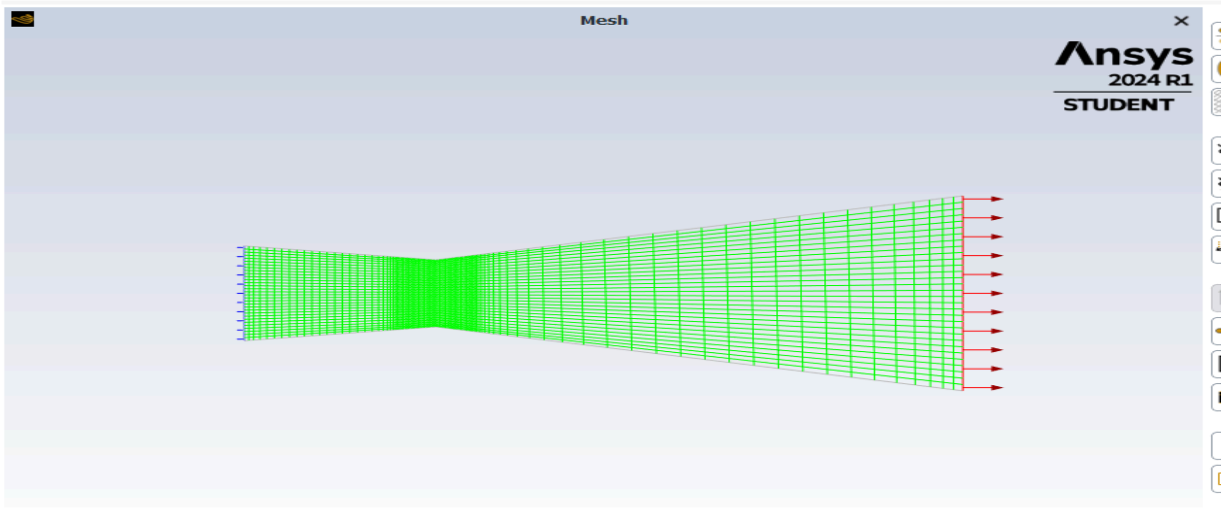

Fig 4: 2D mesh with inlet and outlet section.

The mesh structure is then imported to the setup section of the Fluent module. Since the fluid is compressible, a density-based solver has been used with time transient analysis. To capture the boundary level separation with optimum accuracy k-epsilon viscus model was used. Velocity and combustion chamber have been given as input to the CFD model. The calculation method has been selected as a couple and input from the inlet is taken as a reference value. The number of iterations is set to be 1000 (till it converges). A total of 47 iterations were run for all 47 design points. The different results received from the CFD are plotted in the next sections of the research paper.

3.3. Data Analysis:-

A data sheet was prepared once the CFD analysis of all 47 geometry was done. This includes the design point in terms of length as input and pressure at inlet, outlet, temperature at inlet, outlet, velocity at inlet, and outlet of nozzle are considered as output parameters. This data is further exported into a CSV file and outlier readings are separated. In the next set data preparation has been done by assigning column names. Using the phytone algorithm a data analysis has been performed on the above data and the correlation between the input and output parameters is plotted. This plot has been shown in the next section of the research paper.

4. Results

Once the CFD solution was converged for all 47 design points various types of output values were received concerning the design point. Figure 4 shows the velocity at the nozzle outlet. From the figure, we can see that as the design point of the nozzle changes i.e. the inlet length varies with repost to the outlet nozzle length, there is substantial change in the velocity from 400 to 600m/sec.

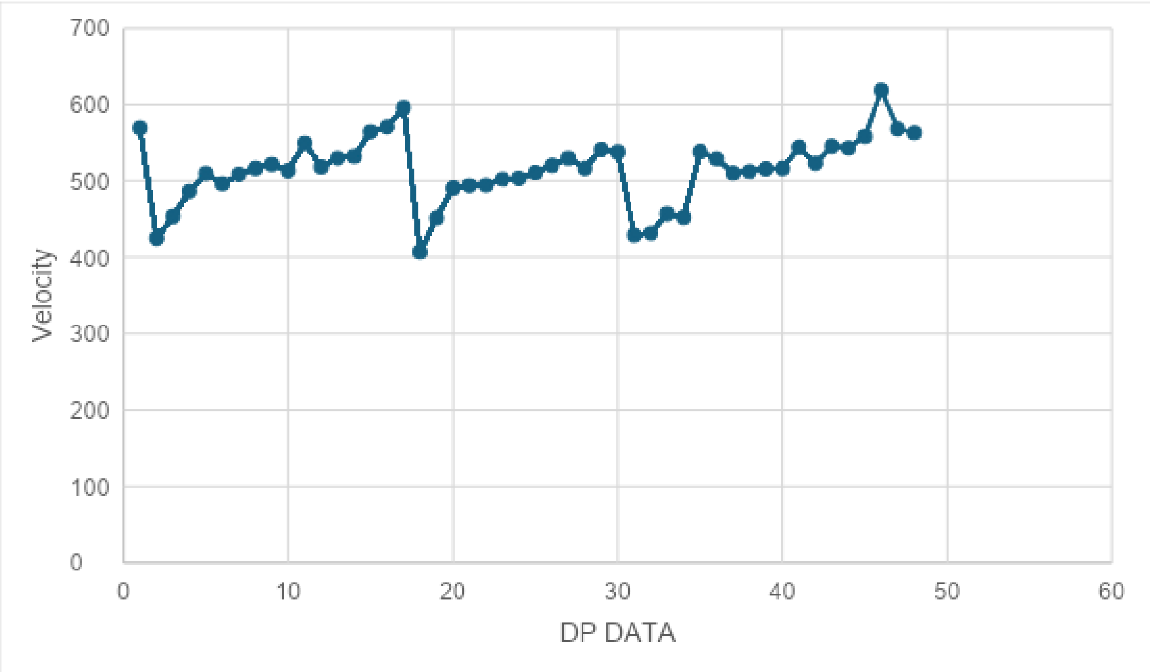

Fig 5: Design points VS Velocity at nozzle outlet

When the pressure graph was plotted from the CFD simulation, it was evident that the overall high fluctuation of nozzle pressure for all the design points can be observed in Figure 5.

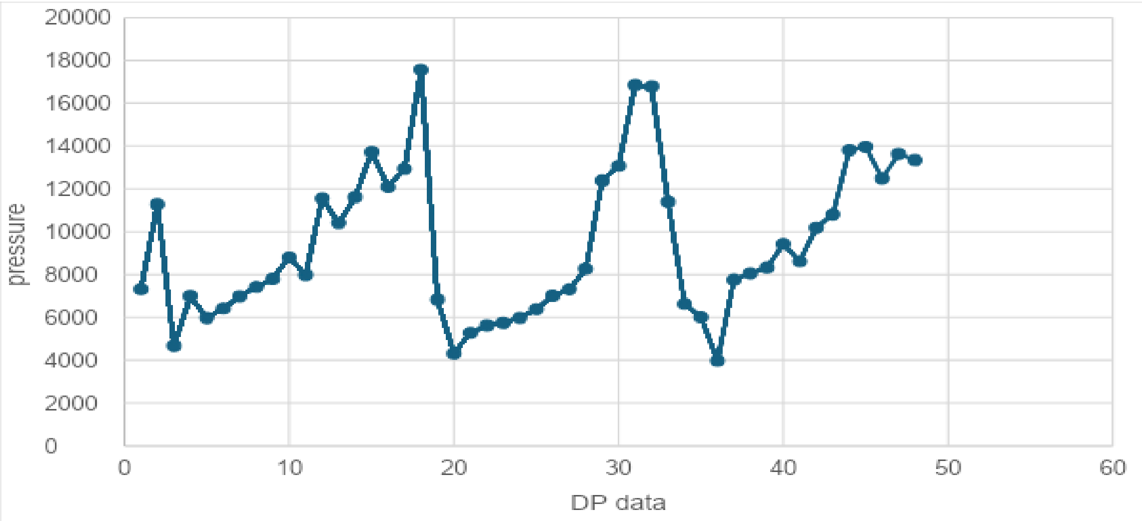

Fig 6: Design point VS maximum pressure into the nozzle

In terms of temperature, all the design points show stable changes concerning the inlet and outlet temperatures. Figure 6 shows the temperature profile of all 47 design points.

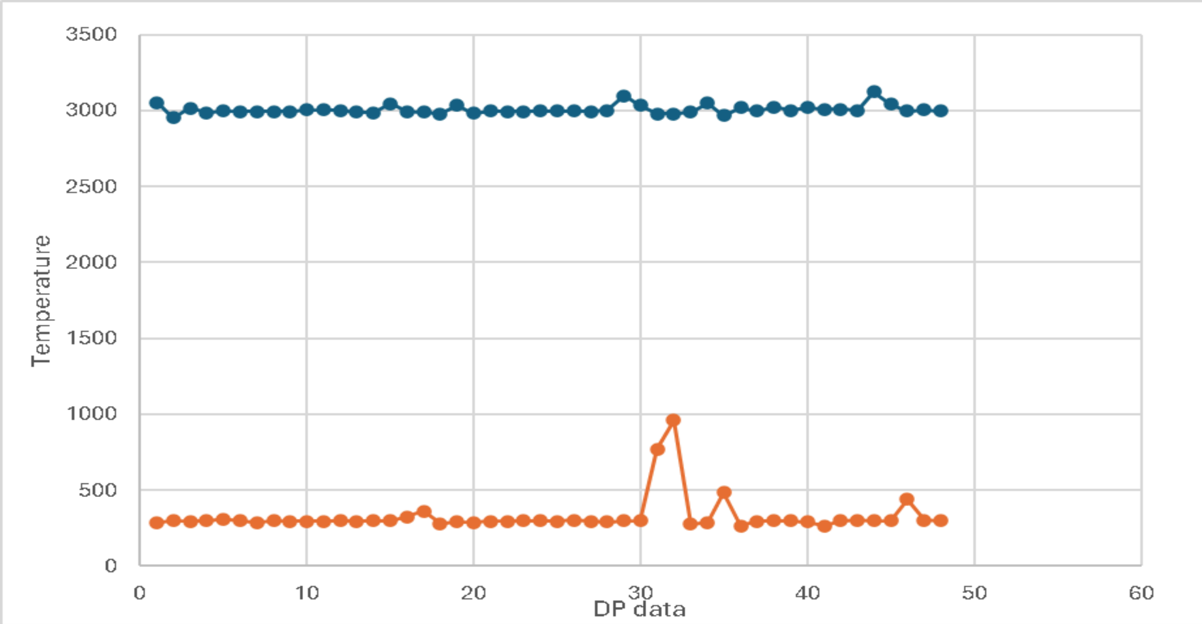

Fig 7: Design point VS maximum temperature of the nozzle

While observing the thrust developed by each nozzle design, there were substantial concerning the design points. Figure 7 shows the thrust developed by each of the design points.

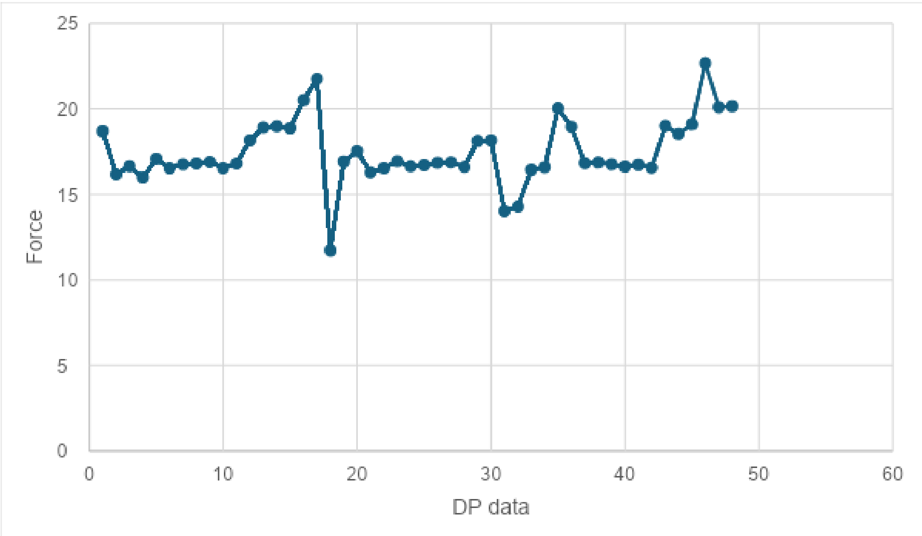

Fig 8: Design point VS thrust developed

When this data was collected and further used to the data analysis. A correlation between all input and output parameters was carried out. Figure 8 shows the correlation between design length p1+p2 and the velocity, temperature, and pressure. From the data analysis, it was evident that variation length gives an inverse relationship with force, pressure, and temperature, whereas as a direct relationship with velocity.

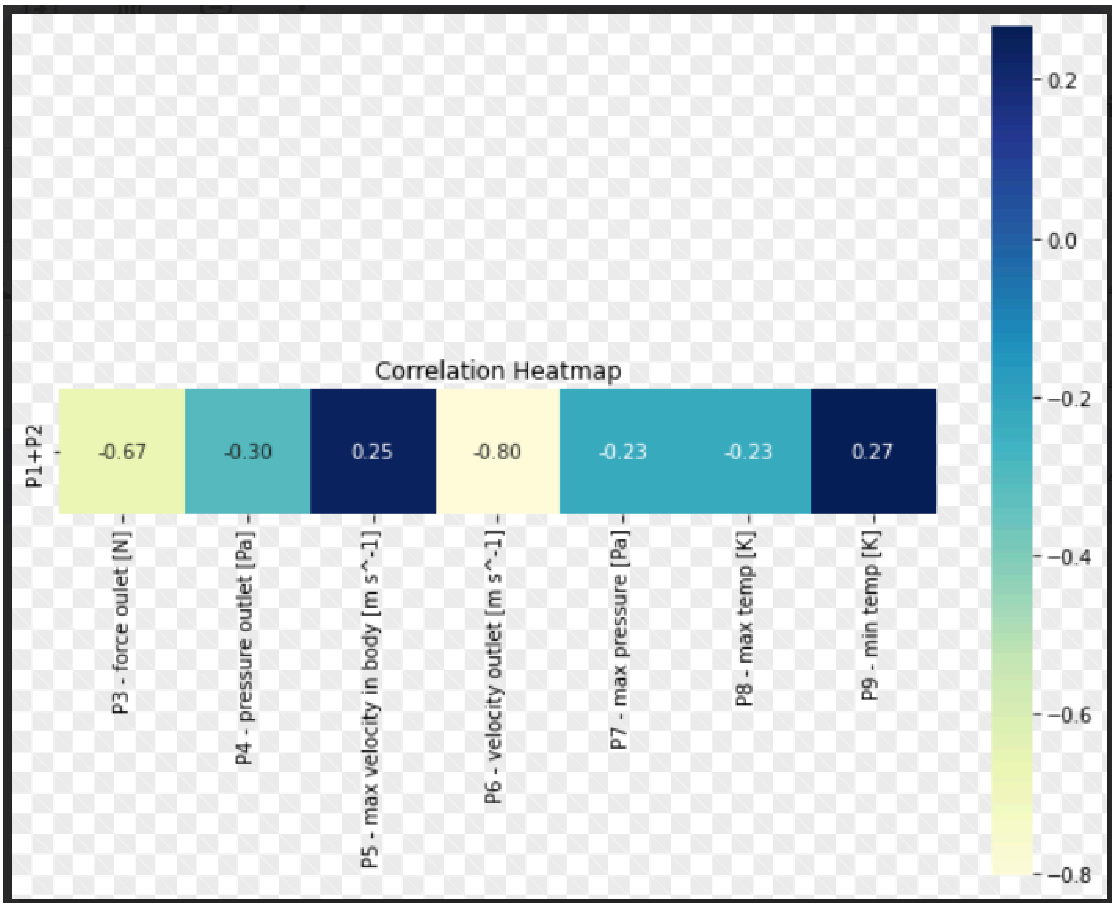

Fig 9: Correlation heatmap from data analysis.

Based on the simulation and data analysis, it was found that small changes in the input or output length of the CD nozzle can strongly vary the other parameters such as pressure, temperature, and velocity. The other concurrent results are mentioned in the next section of the research.

5. Conclusion

To optimize the rocket nozzle in this research a CFD simulation is carried out. to determine the optimal result further data analysis is done using the data collected from the CFD simulation. From this research following observation can be carried out.:-

1. The velocity variation from all the design points varies between 400 to 600m/s, which shows the keeping flow rate is the same but varying length can change the velocity in a limited range.

2. In terms of pressure there is high fluctuation across all the design points.

3. It was found that for design point 47 when the inlet and outlet pressure are all most same, maximum thrust can be obtained.

4. Data analysis gives the exact relationship between nozzle length and other functional parameters like temperature, pressure, and velocity.

Further research can be carried out on nozzle material and 3D simulation which may give a better idea in terms of the real-time effect on the nozzle.

6. Acknowledgements

I would like to express my gratitude to my mentors for their help in carrying out the project.

Dr. Reetu Jain (On My Own Technology Pvt. Ltd.) provided me with the resources to take on the project. The On My Own Technology lab in Lokhandwala, Mumbai is where the project was conducted. I am grateful to her for allowing me to use their desktops at all hours to run the computationally intensive CFD simulations. I am also grateful to her for providing the space within the building to conduct the experimental helium balloon testing.

Mr. Hitendra Vaishnav (On My Own Technology Pvt. Ltd.) taught me how to progress with a research paper, including finding the right sources for literature review and actually writing a research paper. Further, he helped me understand the payload considerations and calculate the possible volume of the airship. He pushed me to think creatively and holistically about all the payload that would be required. He also introduced me to ballonets and understanding their impact on the airship's design.

7. Ethical and Safety Standards

During the course of my project, I adhered to proper ethical standards. While using digital resources I took care to cite them appropriately in the References section to avoid plagiarism and misconstrue any work as my own. Additionally, the CFD simulations were conducted on desktop PCs overnight with minimal power use to ensure that there was no need to turn on the air conditioner for the whole night which could cause immense pollution. Lastly, the results were presented authentically with no misrepresentation.

The primary safety concern was during the empirical testing with helium balloons. I took care to ensure that we tethered the balloon so that there is no possible safety concern with the balloon going too far away and potentially bursting. Furthermore, the masses attached to the balloon were tightly sealed in zip lock bags to prevent them falling from a height and injuring anyone.

8. References

[1] Sampathkumar, Deepak, and Nishant B. Mayekar. "Development and Numerical validation of an Aerospike nozzle Contour Design." MATEC Web of Conferences. Vol. 393. EDP Sciences, 2024.

[2] Muhammed, Ijas, et al. "Computational study of flow separation in truncated ideal contour nozzles under high-altitude conditions." International Journal of Fluid Engineering 1.1 (2024).

[3] Raman, Ravi Shankar, et al. "Design and CFD Simulation of Supersonic Nozzle by Komega turbulence model for Supersonic Wind Tunnel." E3S Web of Conferences. Vol. 507. EDP Sciences, 2024.

[4] Pathan, Khizar Ahmed, et al. "Optimization of Nozzle Design for Weight Reduction using Variable Wall Thickness." Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 112.2 (2023): 86-101.

[5] Agarwal, Tushar. "Computational Fluid Analysis and Optimization of Rocket Engine Nozzles at Various Divergent Angles." (2023).

[6] Haif, Sidali, Hakim Kbab, and Amina Benkhedda. "Altitude-compensating axisymmetric supersonic nozzle design and flow analysis." INCAS Bulletin 15.2 (2023): 33-47.

[7] Sham, B., Ass Prof S. Nagarajan, and Ass Prof S. Preethi. "DESIGN AND STRUCTURAL ANALYSIS OF ROCKET MOTOR NOZZLE."

[8] Chada, J. S. R., and S. R. D. Akella. "Prediction of Solid Rocket Motor Performance in a Pintle-Dual Bell Nozzle Using Computational Fluid Dynamics." J Aeronaut Aerospace Eng 12 (2023): 298.

[9] Hızarcı, Berkan, and Zeki Kıral. "Theoretical, Numerical and Experimental Investigation of the Inlet and Exit Diameter Effect of Convergent-Conical Nozzles on Thrust Force and Volumetric Flow Rate." Dokuz Eylül Üniversitesi Mühendislik Fakültesi Fen ve Mühendislik Dergisi 25.75: 525-538.

[10] Jayaprakash, P., and Durlab Das. "Design and analysis of a rocket CD nozzle."

[11] Chada, Jithendra Sai Raja, et al. "Conceptual Design and Study of Flow through a Dual Bell Nozzle at Different Altitudes using Computational Fluid Dynamics." i-Manager's Journal on Mechanical Engineering 11.3 (2021): 39.

[12] Bhaskar, Arun, and Mithilesh K. Sahu. "Numerical investigation on performance of convergent–divergent nozzle with multi‐inlet combustion chamber of a rocket engine." Heat Transfer 51.1 (2022): 5-21.

[13] Chada, Jithendra Sai Raja, Priya Chandini Ragu, and A. Phani Bhaskar. "Study on Conceptual Design and Shape Optimization of Pintle Nozzle of a Rocket." i-Manager's Journal on Mechanical Engineering 10.4 (2020): 14.

[14] Joshi, Prapti, Tarun Gandhi, and Sabiha Parveen. "Critical designing and flow analysis of various nozzles using CFD analysis." International Journal of Engineering, Research & Technology 9.02 (2020): 421-424.

[15] Khalid, Muhammad Waqas, and Muhammad Ahsan. "Computational fluid dynamics analysis of compressible flow through a converging-diverging nozzle using the K-ε turbulence model." Engineering, Technology & Applied Science Research 10.1 (2020): 5180-5185.