Computational Fluid Dynamics (CFD) Analysis Study On Rocket Nozzles For Design And Performance Optimization - A Review

Kabir S M
Dept of Aerospace Engineering
Madras Institute of Technology Campus,Anna University, Chennai, Tamil Nadu, India

Abstract*- The nozzle is an important component used extensively in aerospace research and industries. The main component used in rocket engines is the convergent-divergent (CD) nozzle. Researchers all around the world are focused on designing nozzles that meet the current day trends across the aerospace sector. Also, nozzles are an extensive research area in aerospace domains. In engineering and technology, nozzles play an important role. The design and performance optimization of rocket nozzles are crucial nowadays. These optimizations are done with the help of computational fluid dynamics (CFD) analysis. Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that makes use of numerical methods, formulas, and algorithms to solve and analyze fluid flow. Computational fluid dynamics analysis usage in engineering and technology is extensive. Recently, researchers have focussed on the optimization of the design and performance of rocket engine nozzles utilizing computational fluid dynamics techniques. This review paper briefly describes the computational fluid dynamics (CFD) analysis study on various rocket engine nozzles for the design and performance optimization.*

Keywords- Rocket engine nozzle, Computational fluid dynamics (CFD), Optimization, Aerospace research and Convergent-Divergent (CD) nozzle.

I. INTRODUCTION

A nozzle is used to convert the chemical-thermal energy generated in the combustion chamber into kinetic energy. The nozzle converts the low-velocity, high-pressure, high-temperature gas in the combustion chamber into high-velocity gas of lower pressure and temperature. Swedish engineer of French descent who, in trying to develop a more efficient steam engine, designed a turbine that was turned by jets of steam. The critical component – the one in which the heat energy of the hot high-pressure steam from the boiler was converted into kinetic energy – was the nozzle from which the jet blew onto the wheel. De Laval found that the most efficient conversion occurred when the nozzle first narrowed, increasing the speed of the jet to the speed of sound, and then expanded again. Above the speed of sound (but not below it) this expansion caused a further increase in the speed of the jet and led to a very efficient conversion of heat energy to motion. The theory of air resistance was first proposed by Sir Isaac Newton in 1726. According to him, an aerodynamic force depends on the density and velocity of the fluid, and the shape and the size of the displacing object. Newton"s theory was soon followed by other theoretical solutions to fluid motion problems. All these were restricted to flow under idealized conditions, i.e. air was assumed to posses' constant density and to move in response to pressure and inertia. Nowadays steam turbines are the preferred power source of electric power stations and large ships, although they usually have a different design to make the best use of the fast steam jet, de Laval"s turbine had to run at an impractically high speed. But for rockets, the de Laval nozzle was just what was needed.

A nozzle is an isentropic device where there is zero change in entropy and the fluid flowing in the nozzle has an adiabatic and steady flow i.e. it does not impart any heat to the surrounding walls. As the fluid flows from the inlet to the exit its velocity at the end increases with a decrease in parameters like pressure and density. The nozzle is used to accelerate the flow of gases and produce a jet of stream with high velocity in the expense of enthalpy drop. The hot pressurized gases, passing through the nozzle are converted to a supersonic speed due to the conversion of heat energy into kinetic energy. The primary function of exhaust nozzles is the efficient conversion of the potential energy of the exhausting gas to kinetic energy by increasing the exhaust velocity, which is accomplished by efficiently expanding the exhausting gases to the ambient pressure. The pressure ratio across the nozzle controls the expansion process and the maximum uninstalled thrust for a given engine is obtained when the exit pressure (pe) equals the ambient pressure ($p0$). A rocket nozzle can be visualized as a 3d pipe with variable cross-sectional area. For huge values of thrust, the active vitality of the fluid must be high which requires high speed fumes. For designing a perfect nozzle, the area ratio must be kept in mind. The area ratio is optimized to provide the maximum performance during a critical time of the engine operating period.

The nozzle works on Bernoulli’s principle and Newton’s third law of motion. Bernoulli’s principle states that as the speed of a moving fluid increases, the pressure within the fluid decreases which explains the reason behind the increase in velocity and drop in pressure value of the fluid at the outlet. The third law of motion in turn explains that the amount of thrust produced in the engine depends on the mass flow rate exiting through the nozzle and the exit pressure. This is the main reason for the research and advancement in the rocket nozzle these days aiming towards the improvement of the rocket design that allows an increase in the mass flow rate thereby increasing the thrust.

A convergent-divergent nozzle is the most important component of the rocket engine and it is a combustion chamber extension. The main role of nozzle is to expand and accelerate the gas. The rocket nozzle is designed to control the direction and it is tube-shaped through which hot gases flow. Nozzles are frequently used to control the rate of flow, flow direction, and the pressure of the stream that emerges from them. The size and shape of the nozzle depend on the mission of the rocket and its application. Nozzles are used even in the turbojet and ramjet engines. During engine startup and shutdown, the rocket nozzle will operate in an over-expanded mode where the ambient/operating pressure is greater than the internal pressure acting on the nozzle wall. In the nozzle, the flow first converges down to the throat or minimum area of the nozzle and then it is expanded through the divergent section to the exit.The nozzle (especially the CD nozzle) shows its strong use in fields like rocket propulsion, jet engines to propel aircraft, aerospace research, electricity production, supersonic flow measurement, wind tunnels, gas expansion, cooling, etc.

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that makes use of numerical methods, formulas, and algorithms to solve and analyze fluid flow. CFD has allowed us to study and predict the behavior of fluids (either liquids or gases) in various physical conditions without the actual need for conducting physical experiments. It allows us to perform numerical simulation, offers flexibility in geometry design, and gives us a platform to carry out various operations of turbulence model, heat transfer, multi-phase flow, boundary condition, etc. The use is not just limited to aerospace engineering and automobile engineering, rather it is widely used in biomedical engineering, electronics, chemical processes, marine engineering, nuclear engineering, etc. Computational Fluid Dynamics (CFD) is an engineering tool that assists experimentation. Its scope is not limited to fluid dynamics; CFD could be applied to any process that involves transport phenomena with it. To solve an engineering problem, we can make use of various methods like the analytical method, and experimental methods using prototypes. The analytical method is complicated. The experimental methods are very costly. If any errors in the design were detected during the prototype testing, another prototype is to be made clarifying all the errors and again testing. This is a time-consuming as well as a cost-consuming process. The introduction of Computational Fluid Dynamics has overcome this difficulty as well as revolutionized the field of engineering. In CFD a problem is simulated in software and the transport equations associated with the problem are mathematically solved with computer assistance. Thus, we would be able to predict the results of a problem before experimentation.

This review paper briefly presents the computational fluid dynamics (CFD) analysis study on various rocket nozzles for design and performance optimization.

II. LITERATURE REVIEW

[1] Vinayaka Nagarajaiah, Sara Solomon Raj, Ayampalayam Nanjappan Swaminathen, Paluru Kiran Kumar, Chintala Indira Priyadarsini, Kodipaka Mamatha. Optimization of Process Parameters for Flow Nozzle with Different Geometry using Computational Fluid Dynamics Method. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 118, Issue 1 (2024) 142-154.

This research focuses on a comparative study of two kinds of nozzles, namely bell and conical nozzles, at various Mach numbers to conclude which results in the better optimal flow of hot gases to enhance nozzle performance and optimize thrust delivery. Computational fluid dynamics (CFD) is used for the modeling of flow parameters for these nozzles. The CFD problem formulation aimed to ensure the optimal thrust pressure integral throughout the supersonic nozzle by consideration of base pressure. This research work takes consideration of boundary layer impact thus focusing on irrotational flow. Results were concluded based on the implementation of flow mach numbers (Mach 1 and Mach 2) for a comparative study of two nozzles. The performance of these nozzles was better in the case of Mach 2. Based on the nozzle's ability to handle higher temperatures and supersonic flows, the bell-shaped nozzle outperformed the conical-shaped nozzle.

[2] Jayaprakash P, Dhinarakaran, Durlab Das. Design and analysis of a rocket C-D nozzle. 2022 International Journal of Health Sciences, 6(S5), 3545–3559 https://doi.org/10.53730/ijhs.v6nS5.9404

This research work focuses on optimizing the design of rocket C-D nozzle for various applications by taking the considerations of previous rocket nozzle reference paper design. The values of pressure, temperature, velocity, and Mach numbers are needed at every section of the nozzle to design the shape, insulation, and cooling arrangements for the rocket C-D nozzle. The calculations and the results obtained from using ANSYS Fluent CFD software and the theoretical calculations are compared based on their values to optimize the nozzle design and thrust delivery. Considering factors like shock waves, boundary layer effects, velocity components, and more, the values of CFD and theoretical methods are slightly different. Finally, it is clear that the nozzle designed was more efficient and produced more thrust than existing nozzles.

[3] Nikhil D. Raut. A CFD Analysis & Study on effect over Mach of Rocket CD nozzle by varying its geometric parameters. International Journal of Mechanical and Production Engineering, ISSN: 2320-2092, Volume- 4, Issue-3, Mar.-2016.

This paper focuses on computational fluid dynamics analysis of rocket nozzle to study the effects of Mach number by varying the CD nozzle geometric parameters such as providing curvature at the throat section, varying inlet, outlet diameter, throat diameter, and changing the length of the divergent section by ANSYS Fluent CFD software. By providing curvature at the throat of 0.5 times the throat diameter, about 2.77 Mach speed can be achieved. Divergent length is reduced by 0.9 and can provide a speed of nearly 2.92 Mach. Reducing throat diameter by 0.75, 3.12 Mach speed at exit can be obtained. Increasing the exit diameter by 1.4 times the actual exit diameter by 3.19 Mach speed can be achieved.

[4] Raghu Ande, Venkata N. Kumar Yerraboina. Numerical Investigation on Effect of Divergent Angle in Convergent-Divergent Rocket Engine Nozzle. Chemical Engineering Transactions. VOL. 66, 2018. DOI: 10.3303/CET1866132 A publication of The Italian Association of Chemical Engineering.

This research study numerically investigates different divergent angles to find the effect of divergent angles on Mach number and static pressure. The different divergent angles used for CFD analysis are 9°, 12°, 15° and 18°. The research results were evaluated and compared with the help of different graphs and contours to figure out the optimum divergent angle with the maximum Mach number. At the divergent section of the nozzle, the velocity distribution was found to be increasing with an increase in divergent angle. At the exit section of the nozzle, the static pressure decreased with an increase in divergent angle. At a divergent angle of 15°, the optimum value of Mach number is obtained.

[5] Sreenath K R, Mubarak A K. Design and analysis of contour bell nozzle and comparison with dual bell nozzle. IJRE | Vol. 03 No. 06 | June 2016

This research presents the computational fluid dynamics study on bell nozzles for better performance and thrust delivery. This paper focuses on the development of CD bell nozzles with negligible shock waves. The flow inside the nozzle is purely cold flow (303K). The analysis was carried out with the help of ANSYS Fluent CFD software. The nozzle walls were set as adiabatic and assumed to be hydraulically smooth. The flow behavior of bell nozzles was studied. The dual bell nozzle has better overall performance than other nozzles. High fuel efficiency and thrust efficiency can be obtained by using the dual bell nozzle.

[6] Karna S. Patel. Flow Analysis and Optimization of Supersonic Rocket Engine Nozzle at Various Divergent Angle using Computational Fluid Dynamics (CFD). IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 11, Issue 6 Ver. IV (Nov- Dec. 2014), PP 01-10

This research study aims to investigate the best-suited divergent angle for a CD nozzle for optimum performance. The oblique shock was visualized and it was found that at 15 degrees of divergent angle, the oblique shock was eliminated from the nozzle. From the research study, it was found that the intensity of velocity was found to have an increasing trend with increment in divergent angle thereby obtaining an optimum divergent angle which would eliminate dynamic instabilities due to shock waves and satisfy the thrust performance and fluid flow inside the nozzle. It was inferred that the oblique shock was eliminated from the nozzle when the divergent angle increased to 15 degrees, thus this could be considered as a good design. The static pressure decreases with increasing divergent angles. The efficiency of the supersonic rocket engine nozzle increases as we increase the divergent angle of the nozzle up to a certain level.

[7] Zhihong Wang, Chunguang Wang and Weiping Tian. Study on Separation Characteristics of Nozzles with Large

Expansion Ratio of Solid Rocket Motors. Aerospace 2023, 10, 4. https:// doi.org/10.3390/aerospace10010004

This research study focuses on the numerical simulation analysis of the flow characteristics of a nozzle with a large expansion ratio and its influence on the force on the nozzle, ground cold flow test and a fluid-structure coupling. The separation flow field under cold flow conditions was simulated using the CFD commercial calculation software Fluent to verify the correctness of the numerical calculations. The fluid-structure coupling analysis was carried out on a large expansion ratio (maximum expansion ratio = 48) full-scale nozzle, and the structural deformation characteristics of the nozzle under the separation conditions were studied. The research study shows that flow separation occurs in the nozzle with a large expansion ratio in underground conditions. After the reach of the separation point, the pressure pulsation increases, and the turbulent pulsation effect is enhanced. Under the combined influence of the ground conditions and low total pressure, the large lateral load caused by the asymmetry of the separation flow field will cause the deformation of the nozzle structure to increase by 5.5 times. This work describes the important reference for the design optimization and experiment of rocket nozzles with a large expansion ratio.

[8] Bogdan-Alexandru Belega, Trung Duc Nguyen. Analysis of Flow in Convergent-Divergent Rocket Engine Nozzle using Computational Fluid Dynamics. INTERNATIONAL CONFERENCE of SCIENTIFIC PAPER AFASES 2015 Brasov, 28-30 May 2015

This research paper presents the flow through the CD nozzle and the study was carried out by using a finite volume rewarding code and ANSYS Fluent. The computational results were in good acceptance with the experimental results taken from the literature. From the analysis, it was observed that the nozzle created based on exit parameters is in accord with the scope.

[9] Tejas Christopher, Prajwal Ashok Kumar, Siddalingappa P K, N K S Rajan. Flow Analysis of Hybrid Rocket Nozzle Exhaust and Its Effects on Launch Pad and Guide Stand. International Journal of Recent Technology and Engineering (IJRTE) ISSN: 2277-3878, Volume-8 Issue-3, September 2019.

This paper presents the CFD analysis work on hybrid rocket engine nozzle exhaust. The exhaust speeds vary depending upon the expansion proportion of the nozzle. The flow features of the three different jets obtained for three different geometries are like each other in terms of the formation of the shock structure. The temperature contours help in determining the type of material that can be used to withstand this high temperature.

[10] Yashraj Asthana. CFD Analysis of Different Types of Advanced Rocket Nozzles in Ansys. International Journal of Advancements in Technology. Research Article.

This research paper describes the testing of different types of nozzles in a rocket by calculating their average velocity at the outlet, kinetic energy, pressure, velocity, and temperature contours. The analysis was carried out on ANSYS Fluent CFD software. Based on the results from the CFD, a nozzle that is reliable to use is selected. It is inferred that the greater value for exit velocity is in the case of a double-bell nozzle. Thus, we conclude that the double-bell nozzle shows better performance optimization than other nozzles that were tested. Double bell nozzle stands as best among the nozzles we analyzed so far in the research literature.

[11] Csaba Jéger, Árpád Veress. Novell Application of CFD for Rocket Engine Nozzle Optimization. Periodica Polytechnica Transportation Engineering. 47(2), pp. 131-135, 2019. https://doi.org/10.3311/PPtr.11490

This work describes the numerical computational study of the rocket nozzle for design optimization. The nozzle design optimization of the nozzles that were optimized are conical and bell-shaped nozzles. The Results are in better agreement with existing nozzle flow fields. The optimization loop built upon the model was then tested on the diffusor of a one-parameter conical nozzle and a two-parameter bell nozzle to yield the largest obtainable thrust for the prescribed conditions.

[12] Prapti Joshi, Tarun Gandhi, Sabiha Parveen. Critical Designing and Flow Analysis of Various Nozzles using CFD Analysis. International Journal of Engineering Research & Technology (IJERT). ISSN: 2278-0181. Vol. 9 Issue 02, February-2020.

This study includes the design of several nozzles and comparing them with the existing ones. The research incorporates various parameters such as Mach velocity, temperature, and pressure for the critical design optimization analysis. The CFD simulations are carried out to understand the nozzle efficiency at different temperature conditions in ANSYS Fluent CFD software. From the research study, it is inferred that the contours of the bell nozzle offer far better pressure, velocity, and temperature curves than the contours of the conical nozzle.

[13] N. Shalom, R. Rabin, A. J. Abinesh, Abi Sam EA, Aadith. B. Roshan. Convergent Divergent Nozzle Design

and Analysis to Increase Working Effectiveness. International Journal for Research in Applied Science & Engineering Technology (IJRASET) ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538 Volume 11 Issue VIII Aug 2023.

This work aims to develop and examine a CD nozzle to lower the nozzle costs. Utilizing the ANSYS Fluent program, the work is done, and the calculations and results are validated for the case of high heat absorbent material. This work analyzed the CD nozzle's parameters such as stress, strain, deformation, and heat flux. After various simulations, it is concluded that the titanium is producing effective outcomes in terms of results. The heat flow of the titanium material is high, so titanium material is preferred.

[14] Mohamed Tarmizi Ahmad, Anudiipnath Jagannathan, Amzari Zhahir, Muhammad Hanafi Azami, Razali Abidin, Mohd Nor Hafizi Noordin, Norzaima Nordin. CFD Analysis of De Laval Nozzle of a Hybrid Rocket Engine. E3S Web of Conferences 477, 00027 (2024)

This study focuses on CFD methods to compare factors with those derived from classical theory such as pressure, velocity, temperature, and area ratios. The results of the computational modeling of CFD simulation are comparable to those found in theoretical calculations. The nozzle is examined under various conditions at 1000,000 Pa of pressure and 2200K of temperature.

[15] Muhammad Waqas Khalid, 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. Vol. 10, No. 1, 2020, 5180-5185

This research article aims to analyze the variation of flow parameters like pressure, Mach number, velocity, temperature, and density using a finite volume method solver with a standard k-ε turbulence model in computational fluid dynamics. The investigation of shock waves and boundary layer are studied inside the CD nozzle. The CFD model results are compared with the theoretically calculated results. The difference is negligible. Simulations were carried out to study and analyze the behavior of exhaust flow inside the nozzle. There are no wall frictions as the shape of the shock is straight.

[16] Munipally Prathibha, M. Satyanarayana Gupta, Simhachalam Naidu. CFD Analysis on a Different Advanced Rocket Nozzles. International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249-8958 (Online), Volume-4 Issue-6, August 2015.

This research work describes the various nozzle concepts and comparative study of the nozzles with the conventional nozzles. The CFD flow simulations were carried out in ANSYS Fluent software. After comparing the CFD simulations and various parameters, the CD nozzle' performance is best among all other nozzles.

[17] Rishab Kumar Agrawal, Sivaram Munagala. Parametric Output of Penetration Length in De Laval Nozzle using Computational Fluid Dynamics. International Journal of Recent Technology and Engineering (IJRTE) ISSN: 2277-3878 (Online), Volume-8 Issue-6, March 2020.

This research work focuses on bell nozzle CFD simulations with constant divergent lengths. The CFD model is validated and verified with NPR (Nozzle Pressure Ration) computationally and experimentally. This work confirms that the penetration length influences the effects of mass-weighted average Mach number. The maximum Mach number attained in the modeled nozzle with minimal losses is at an angle of 35°. If the penetration length is of the order of 75% of the total nozzle length, the flow expands optimally to give the maximum thrust output at a defined pressure ratio across the nozzle. The thrust obtained in the nozzle is directly proportional to the exit velocity of the flow. Thus, the exit Mach number for 35-degree divergent half angles is concluded to generate the maximum thrust.

[18] Amar Gandge, Basawaraj and Chennabasappa Hampali. Design and numerical analysis on supersonic rocket nozzle. JOURNAL OF MINES, METALS & FUELS. ICAMMME 2021. Volume 69, Issue 12A, December 2021

This research work focuses on the various flow parameters such as pressure, temperature, velocity, and Mach number of the mass flow CFD simulations in the nozzle by changing the divergent angle. CFD simulations were studied for various divergent angles such as 5°, 10° 11°, 12°, 15°, and 20°. By considering all the results at different divergent angles, the 11° angle is more efficient and found all the flow parameters such as velocity, temperature, pressure, and Mach number of the mass flow are at the required condition.

[19] Balaji Krushna.P, P. SrinivasaRao, B. Balakrishna. Analysis of Dual Bell Rocket Nozzle using Computational Fluid Dynamics. IJRET: International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321-7308. Volume: 02 Issue: 11 | Nov-2013

This research paper investigates the CFD simulation results of Dual Bell Nozzle for design optimization and to benchmark with various nozzles. The result shows the variation in the Mach number, pressure, temperature distribution, and turbulence intensity. The dual bell nozzle has 90% overall better performance than the conventional bell-shaped nozzle. The dual bell design is suitable for single-stage to-orbit (SSTO) flight. Other advantages are that the dual bell nozzle makes better use of the base area, and has higher thrust efficiency and thus a higher average specific impulse. The Mach number at the exit is found to be 1.2 Mach (Supersonic). The velocity value of the Nozzle increases from inlet to exit the velocity at the exit is found to be 3.29 e+02.

[20] Shamsher Ali Ansari and Pramod Chaudhary. CFD Analysis of CD Nozzle with Sharp Throat and CD Nozzle with Curved Throat. Online Journal of Robotics & Automation Technology. DOI: 10.33552/OJRAT.2024.02.000542. ISSN: 2832-790X Volume 2 - Issue 4, 2024

This research paper investigates a comparative simulation study of CFD on a CD nozzle with a sharp throat and a CD nozzle with a curved throat. Design and simulation of nozzles was carried out using ANSYS Fluent CFD software. In the nozzle with a sharp throat region, there is a sudden variation of the properties. The properties show exponential variation near the throat region. While in the nozzle with the curved throat, the variation in the properties is gradual and shows almost linear variation.

[21] Dipen R. Dangi, Parth B. Thaker, Atal B. Harichandan. Flow Analysis of Rocket Nozzle using Method of Characteristics (MoC). ICRTESM – 17. ISBN 978-93-86171-21-4

The research work investigates the CFD analysis of the bell-type nozzle with the help of the Shear Stress Transport k-ω (SSTKW) turbulence model using ANSYS Fluent CFD software and also by implementing the technique of method of characteristics with different Mach numbers. In this research article, CFD simulations are used to develop the best geometry of the rocket nozzle with consideration of various thermodynamic properties like pressure, velocity, density, temperature, and Mach number. The performance of the rocket nozzle increases, as the side loads are reduced on the rocket nozzle wall. In the case of Mach number 3.0 maximum Mach number and velocity achieved by simulation are 2.89 and 600.64 m/s respectively. However, the Authors have observed that, by using the Modified Method of Characteristic (MMOC), the Mach number is achieved beyond 2.89 in the case of designing of rocket nozzle at Mach number 3.0.

[22] Tiago Fernandes, Alain Souza, Frederico Afonso. A shape design optimization methodology based on the method of characteristics for rocket nozzles. CEAS Space Journal (2023) 15:867–879 https://doi.org/10.1007/s12567-023-00511-1.

The present work aims to develop a low-fidelity and fast method to conduct nozzle shape optimization. This method consists of using the free-form deformation (FFD) parameterization technique to control the nozzle shape utilizing an optimization algorithm to maximize the coefficient of thrust determined by a two-dimensional method of characteristics (MoC). To verify the reliability of the proposed method, a similar optimization process is carried out, recurring to high-fidelity simulations, namely using a Euler solver, in the open-source framework SU2. This latter optimization process is established as a surrogate-based optimization (SBO) not only to mitigate the SU2 framework limitations in performing shape optimization on nozzles but also as a way to reduce computational power. A good agreement between the results from both methods is achieved, displaying solely a small offset concerning the optimal contour width and the coefficient of thrust. Hence, this proves the usefulness of the developed shape optimization strategy based on the MoC for the preliminary design of nozzles. An improvement of 3.08% is achieved for the coefficient of thrust in vacuum CTvac by the developed optimization process using multiple design variables and a [9 × 2] FFD grid. The present work proved that the MoC is a strong and reliable tool for preliminary design optimization since it reduces the computational cost regarding CFD-based optimization for a small loss in accuracy. The method developed throughout this work is ideal for a multi-fidelity approach.

[23] Shridevi S Keralamatti, and Nagappa Pradhani. Fluid Structure Interaction Based Investigation on Convergent-Divergent Nozzle and Study of Coating Material. Journal of Aeronautics & Aerospace Engineering. J Aeronaut Aerospace Eng, Vol. 8 Iss. 1 No: 216.

This research study investigates and analyzes aerodynamic flow in a rocket CD nozzle and performs a Fluid-Structure Interaction simulation for the thermo-mechanical interaction between solid and fluid structures. The nozzle is made with Rhenium and Iridium. The FSI interaction analysis was carried out with the help of StarCCM+ software by creating various layers. It is inferred that, from the cases studied, when pressure values decrease in the CD nozzle, the velocity increases. Iridium coating reduces the thermal loading on material structures. Also, the iridium coating is capable of preventing damage. Thus, Iridium coating can be used for high-temperature applications.

[24] Anup Singh, Nikita Shukla. Rocket Engine Nozzle Optimization. International Journal of Innovative Research in Science, Engineering and Technology. Vol. 9, Issue 2, February 2020.

This research work focuses on numerical and analytical analysis with geometrical optimization of the parabolic nozzle. This study includes a computational fluid dynamics-based comparative study of conical and parabolic nozzle contour using ANSYS Fluent CFD software confirming that the performance of parabolic nozzle contour is better than conical nozzle contour. The difference in their respective Mach Numbers is 18.62%. This study also investigates the geometric optimization of parabolic nozzles using the Taguchi Methodology for quality improvement and making products and processes incentive to manufacturing and environmental variations. Optimization resulted in maximization of thrust and Mach number considering the area ratio, angle ratio, and length ratio as constraints. The exit Mach number is increased by 12.97% in the Thrust Optimized Parabolic Nozzle.

III. CONCLUSION

Various computational fluid dynamics (CFD) analysis studies on rocket nozzles are discussed in this review literature. The differences between various rocket nozzles, their parameters, and their flow properties are studied. From the literature review, it is clear that the geometric parameters of the nozzle influence nozzle performance and efficiency. The comparative study between various nozzles, their contours, and graphs are reviewed. Thus, the design & performance optimization of a nozzle and the flow properties of a nozzle are influenced by its geometric parameters. The nozzle design optimization for aerospace purposes is reviewed. This review literature helps aerospace researchers and industry experts to design rocket nozzles with better performance and efficiency.

REFERENCES

[1] Vinayaka Nagarajaiah, Sara Solomon Raj, Ayampalayam Nanjappan Swaminathen, Paluru Kiran Kumar, Chintala Indira Priyadarsini, Kodipaka Mamatha. Optimization of Process Parameters for Flow Nozzle with Different Geometry using Computational Fluid Dynamics Method. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences 118, Issue 1 (2024) 142-154.

[2] Jayaprakash P, Dhinarakaran, Durlab Das. Design and analysis of a rocket C-D nozzle. 2022 International Journal of Health Sciences, 6(S5), 3545–3559 https://doi.org/10.53730/ijhs.v6nS5.9404

[3] Nikhil D. Raut. A CFD Analysis & Study on effect over Mach of Rocket CD nozzle by varying its geometric parameters. International Journal of Mechanical and Production Engineering, ISSN: 2320-2092, Volume- 4, Issue-3, Mar.-2016.

[4] Raghu Ande, Venkata N. Kumar Yerraboina. Numerical Investigation on Effect of Divergent Angle in Convergent-Divergent Rocket Engine Nozzle. Chemical Engineering Transactions. VOL. 66, 2018. DOI: 10.3303/CET1866132 A publication of The Italian Association of Chemical Engineering.

[5] Sreenath K R, Mubarak A K. Design and analysis of contour bell nozzle and comparison with dual bell nozzle. IJRE | Vol. 03 No. 06 | June 2016

[6] Karna S. Patel. Flow Analysis and Optimization of Supersonic Rocket Engine Nozzle at Various Divergent Angle using Computational Fluid Dynamics (CFD). IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684, p-ISSN: 2320-334X, Volume 11, Issue 6 Ver. IV (Nov- Dec. 2014), PP 01-10

[7] Zhihong Wang, Chunguang Wang and Weiping Tian. Study on Separation Characteristics of Nozzles with Large Expansion Ratio of Solid Rocket Motors. Aerospace 2023, 10, 4. https://doi.org/10.3390/aerospace10010004

[8] Bogdan-Alexandru Belega, Trung Duc Nguyen. Analysis of Flow in Convergent-Divergent Rocket Engine Nozzle using Computational Fluid Dynamics. INTERNATIONAL CONFERENCE of SCIENTIFIC PAPER AFASES 2015 Brasov, 28-30 May 2015

[9] Tejas Christopher, Prajwal Ashok Kumar, Siddalingappa P K, N K S Rajan. Flow Analysis of Hybrid Rocket Nozzle Exhaust and Its Effects on Launch Pad and Guide Stand. International Journal of Recent Technology and Engineering (IJRTE) ISSN: 2277-3878, Volume-8 Issue-3, September 2019.

[10] Yashraj Asthana. CFD Analysis of Different Types of Advanced Rocket Nozzles in Ansys. International Journal of Advancements in Technology. Research Article.

[11] Csaba Jéger, Árpád Veress. Novell Application of CFD for Rocket Engine Nozzle Optimization. Periodica Polytechnica Transportation Engineering. 47(2), pp. 131-135, 2019. https://doi.org/10.3311/PPtr.11490

[12] Prapti Joshi, Tarun Gandhi, Sabiha Parveen. Critical Designing and Flow Analysis of Various Nozzles using CFD Analysis. International Journal of Engineering Research & Technology (IJERT). ISSN: 2278-0181. Vol. 9 Issue 02, February-2020.

[13] N. Shalom, R. Rabin, A. J. Abinesh, Abi Sam EA, Aadith. B. Roshan. Convergent Divergent Nozzle Design and Analysis to Increase Working Effectiveness. International Journal for Research in Applied Science & Engineering Technology (IJRASET) ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538 Volume 11 Issue VIII Aug 2023.

[14] Mohamed Tarmizi Ahmad, Anudiipnath Jagannathan, Amzari Zhahir, Muhammad Hanafi Azami, Razali Abidin, Mohd Nor Hafizi Noordin, Norzaima Nordin. CFD Analysis of De Laval Nozzle of a Hybrid Rocket Engine. E3S Web of Conferences 477, 00027 (2024)

[15] Muhammad Waqas Khalid, 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. Vol. 10, No. 1, 2020, 5180-5185

[16] Munipally Prathibha, M. Satyanarayana Gupta, Simhachalam Naidu. CFD Analysis on a Different Advanced Rocket Nozzles. International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249-8958 (Online), Volume-4 Issue-6, August 2015.

[17] Rishab Kumar Agrawal, Sivaram Munagala. Parametric Output of Penetration Length in De Laval Nozzle using Computational Fluid Dynamics. International Journal of Recent Technology and Engineering (IJRTE) ISSN: 2277-3878 (Online), Volume-8 Issue-6, March 2020.

[18] Amar Gandge, Basawaraj and Chennabasappa Hampali. Design and numerical analysis on supersonic rocket nozzle. JOURNAL OF MINES, METALS & FUELS. ICAMMME 2021. Volume 69, Issue 12A, December 2021

[19] Balaji Krushna.P, P. SrinivasaRao, B. Balakrishna. Analysis of Dual Bell Rocket Nozzle using Computational Fluid Dynamics. IJRET: International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321-7308. Volume: 02 Issue: 11 | Nov-2013

[20] Shamsher Ali Ansari and Pramod Chaudhary. CFD Analysis of CD Nozzle with Sharp Throat and CD Nozzle with Curved Throat. Online Journal of Robotics & Automation Technology. DOI: 10.33552/OJRAT.2024.02.000542. ISSN: 2832-790X Volume 2 - Issue 4, 2024

[21] Dipen R. Dangi, Parth B. Thaker, Atal B. Harichandan. Flow Analysis of Rocket Nozzle using Method of Characteristics (MoC). ICRTESM – 17. ISBN 978-93-86171-21-4

[22] Tiago Fernandes, Alain Souza, Frederico Afonso. A shape design optimization methodology based on the method of characteristics for rocket nozzles. CEAS Space Journal (2023) 15:867–879 https://doi.org/10.1007/s12567-023-00511-1.

[23] Shridevi S Keralamatti, and Nagappa Pradhani. Fluid Structure Interaction Based Investigation on Convergent-Divergent Nozzle and Study of Coating Material. Journal of Aeronautics & Aerospace Engineering. J Aeronaut Aerospace Eng, Vol. 8 Iss. 1 No: 216.

[24] Anup Singh, Nikita Shukla. Rocket Engine Nozzle Optimization. International Journal of Innovative Research in Science, Engineering and Technology. Vol. 9, Issue 2, February 2020.