



e-ISSN: 2319-8753 | p-ISSN: 2347-6710

IJIRSET

International Journal of Innovative Research in
SCIENCE | ENGINEERING | TECHNOLOGY

INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 12, Issue 9, September 2023

ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA

Impact Factor: 8.423

9940 572 462

6381 907 438

ijirset@gmail.com

www.ijirset.com

Study on Fluid Thermal Structural Using Rectangular Nozzles

Priyashree B^{1*}, Sonali Gupta²

M.Tech Research Scholars, Department of Aeronautical Engineering, Nitte Meenakshi Institute of Technology, Bengaluru, Karnataka, India¹

Assistant Professor, Department of Aeronautical Engineering, Nitte Meenakshi Institute of Technology, Bengaluru, Karnataka, India²

ABSTRACT: The study underscores the importance of Fluid-Thermal-Structural Interaction (FTSI) for assessing designs and validating new ones through multi-physics simulations on rectangular nozzles. It confirms the feasibility of an improved design and suggests FTSI's applicability across product design cycles. The paper focuses on numerical analysis of supersonic flow within a converging-diverging nozzle, aiming to identify optimal nozzle designs for high exit velocity. ANSYS 16 software is used for simulations, comparing results to theoretical data. The study gives detailed idea on area ratio effects in case of suddenly expanded ducts and also the base pressure control using computational fluid dynamics (CFD), emphasizing the pivotal role of area ratio in determining base pressure values. The paper investigates micro-jet utilization for base pressure control in planar ducts through numerical simulations with turbulence models, validating results against commercial CFD software. It discusses the use of CFD for analyzing flow characteristics in a two-dimensional convergent-divergent nozzle, validating the simulation by comparing it to the standard nozzle equation. The study examines microjet control in a convergent-divergent nozzle, optimizing Mach number, nozzle pressure ratio and length-to-diameter ratio for maximum pressure increase. Finite element (FE) analysis and the K-ε turbulence model are employed, validated with ANSYS. The paper analyzes the variation of area ratio, nozzle pressure ratio, velocity, pressure, and Mach number in CD Nozzles, emphasizing their impact on resultant forces. Additionally, it discusses theoretical aspects and performance parameters of propellants used in solid rocket motors, including material selection.

I. INTRODUCTION

Nozzles play a vital role in various industries, including aerospace and power plants. They are instrumental in controlling the velocity of rockets and other fluid systems. Among these nozzles, the convergent-divergent nozzle is specially designed to transform high-pressure as well as high-temperature gas to high-velocity gas with lower pressure and temperature, achieving supersonic speeds. In this study, we employ ANSYS 16 software to conduct a comprehensive analysis, with a primary focus on achieving supersonic speeds and investigating fluid behavior within the nozzle. A crucial parameter in our analysis is the Mach number, which increases from the inlet to outlet. The modeling, meshing, and pre-processing stages are carried out using ANSYS 16, and we utilize the Navier-Stokes equation to calculate both the Mach number and velocity. This research aims to deepen our understanding of nozzle performance in these critical applications. [1]

Rectangular supersonic nozzles have garnered attention due to their advantages in terms of reduced noise production and simplified design and manufacturing processes compared to circular nozzles. Experimental and numerical investigations have been performed to explore noise generation and flow characteristics within rectangular nozzles, with a specific focus on phenomena like shock formation, turbulent mixing, and screech tones. Rectangular nozzles, featuring sharp throats and conical sections, exhibit a distinctive double-diamond shock pattern that has repercussions



on structural integrity, wave drag, and side loads. While finite element analysis has been applied during the design phase, the combined impact of multi-physics modeling has not been mentioned comprehensively. In supersonic flow, the impact of Shock wave Boundary Layer Interaction on nozzle structure is of paramount importance. Therefore, this study proposes the adoption of Fluid-Thermal-Structural Interaction modeling to gain observation on the interconnected effects of flow physics and structural integrity within rectangular nozzles. [2]

This paper examines the phenomenon of sudden expansion flows in high-speed applications, with a particular focus on the significant influence of microjets. Such flows have the potential to create sub-atmospheric pressure conditions, resulting in increased drag and impacting the performance of aerospace vehicles.[3]

The introduction brings attention to the issue of unanticipated flow expansion, both in subsonic and supersonic conditions, and underscores the significance of jet efficiency across diverse applications. It notes that prior research has primarily concentrated on managing base pressure in cases of sudden expansion within ducts, yet there exists a noticeable gap in the literature concerning active base pressure control within simulations. The current study is prepared to fill this gap by investigating the control of base pressure through the application of micro-jets in conjunction with CFD techniques. Additionally, the study showcases the utility of turbulence models and the comparison of results obtained with commercial CFD software. [4]

The introduction delves into the nozzle's function, which involves transforming high-temperature, high-pressure gases into lower-temperature, lower-pressure, and higher-velocity gases. It emphasizes the nozzle's crucial role in managing sound levels and enhancing efficiency within jet engines. The introduction underscores the benefits of employing computational fluid dynamics (CFD) as a means to study nozzle flow characteristics, drawing comparisons with alternative approaches like experimental and analytical methods. Furthermore, the introduction references prior research endeavors that have explored aspects such as divergent angles and Mach numbers in nozzles. It also alludes to the utilization of analytical solutions and CFD simulations in the examination of flow rates and fluid properties. [5]

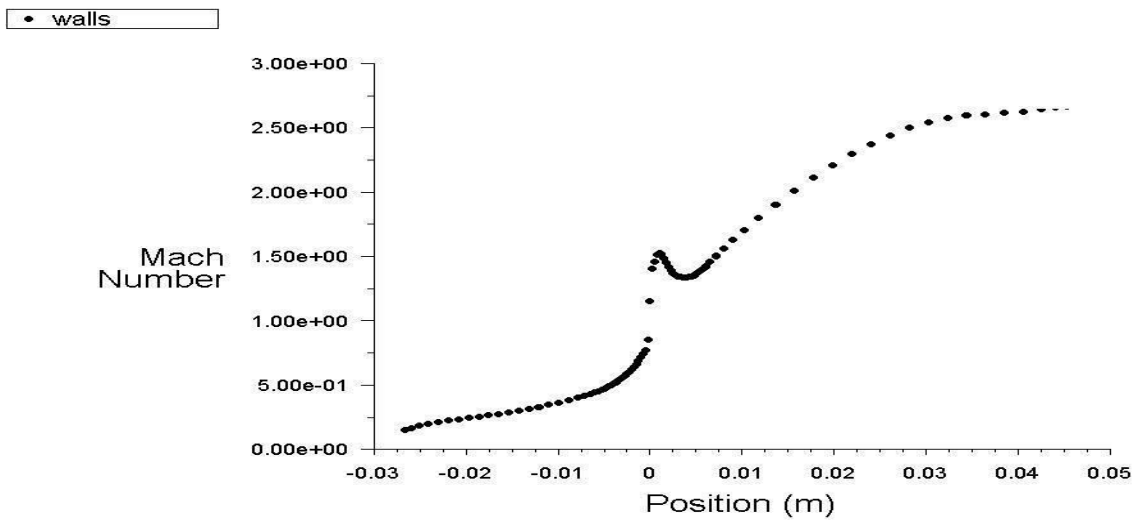
The paper addresses a common engineering challenge known as sudden expansion at the base, which often leads to elevated base drag. This study centers on mitigating base pressure and drag by employing microjets as an active control method. The research investigates the efficacy of microjets in regulating flow conditions when sudden expansion occurs. While previous studies have examined fluid behavior in abruptly expanded ducts, no prior work has explored the use of active control methods through experiments and numerical simulations. The main intention of this investigation is to assess the impact of micro jet control within a convergent-divergent nozzle subjected to sudden expansion. This assessment is performed using CFD, and the obtained output is validated through experimental measurements. Additionally, the study delves into the complexities associated with compressible flow regions and discusses the selection of appropriate turbulence models to provide closure for Reynolds-averaged Navier-Stokes equations. [6]

The introduction offers an overview of the Convergent-Divergent (CD) nozzle, a device employed for managing the speed, direction, and flow characteristics of fluids. This nozzle is structured into three distinct sections: the converging segment, throat section, and diverging portion, each fulfilling a specific role in flow control. The converging section facilitates the transformation of high-pressure gas by converting thermal energy into kinetic energy. Meanwhile, the throat section serves as the conduit connecting the nozzle's inlet and outlet. Lastly, the diverging section of the CD nozzle enables gas expansion, leading to the acceleration of gas flow to supersonic speeds. This process converts heat energy into kinetic energy, resulting in high-speed exhaust. CD nozzles find a wide range of applications, including powering nozzles in automobiles, jet engines, high-speed missiles, rocket propulsion systems, steam turbines, and supersonic jet engines.[7]



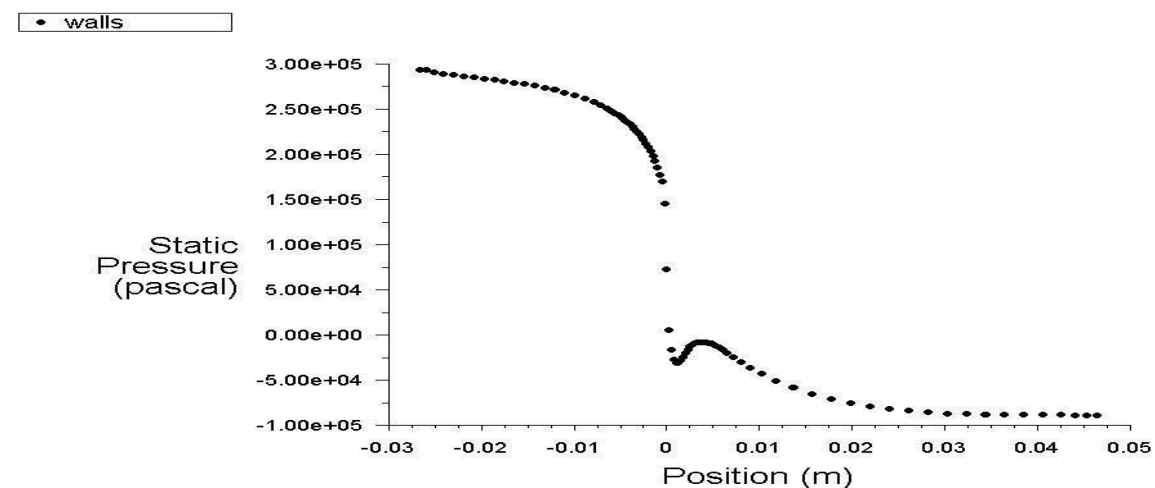
II. RESULTS & DISCUSSION

The analysis of the convergent-divergent nozzle using CFD simulation showed a continuous increase in velocity from the inlet to the outlet, with no presence of shocks. The Mach number at the outlet was found to be 2.50e00, indicating supersonic flow. The paper aimed to acquire the finest nozzle that provides high exit velocity in nozzle cross-section. However, specific results regarding the comparison of different nozzle designs or cross-sections are not mentioned in the provided sources. The authors performed a proper analysis of the convergent-divergent nozzle, focusing on the behavior of the fluid during its movement inside the nozzle. The CFD analysis using ANSYS 16 software allowed them to determine the Mach number and velocity distribution inside the nozzle.[1]



Mach Number ANSYS Fluent Release 16.0 (2d, dbns imp, lam) Mar 28, 2020

Fig 1: Mach number vs position [1]



Static Pressure ANSYS Fluent Release 16.0 (2d, dbns imp, lam) Mar 28, 2020

Fig 2: static pressure vs pressure [1]

The paper evaluates the impact of Fluid-Thermal-Structural Interaction (FTSI) in the existing rectangular nozzle geometry and the advanced design. The structural deformation and boundary layer thickness are calculated for the existing design to understand the influence of internal shock formation on boundary layer flow. The design is evaluated at off-design conditions. [2]

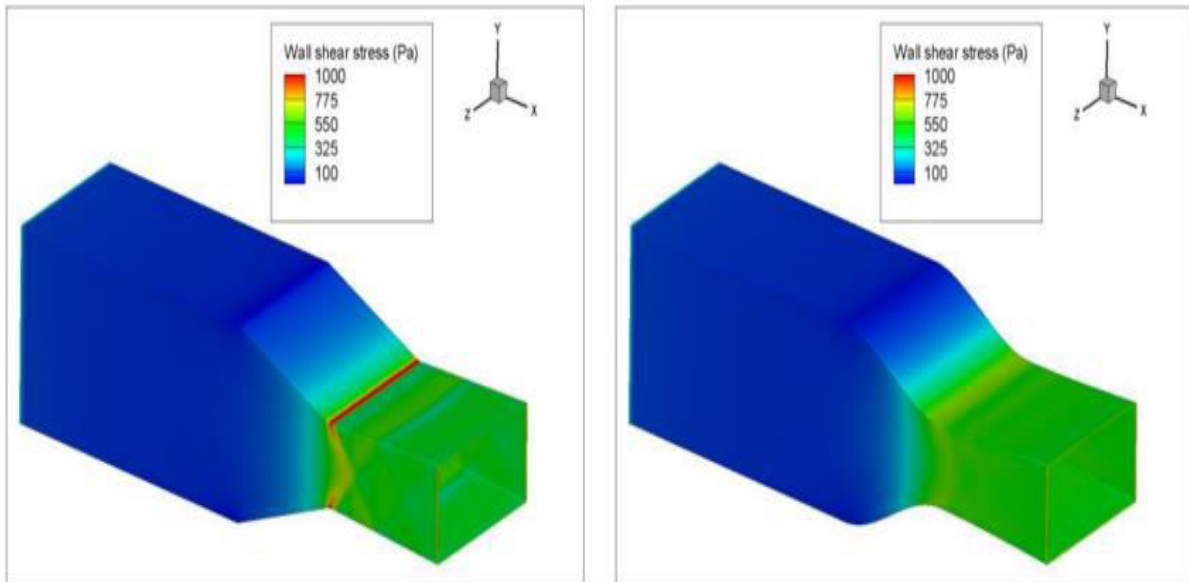


Fig 3: Wall shear stress on nozzle walls existing design and improved design. [2]

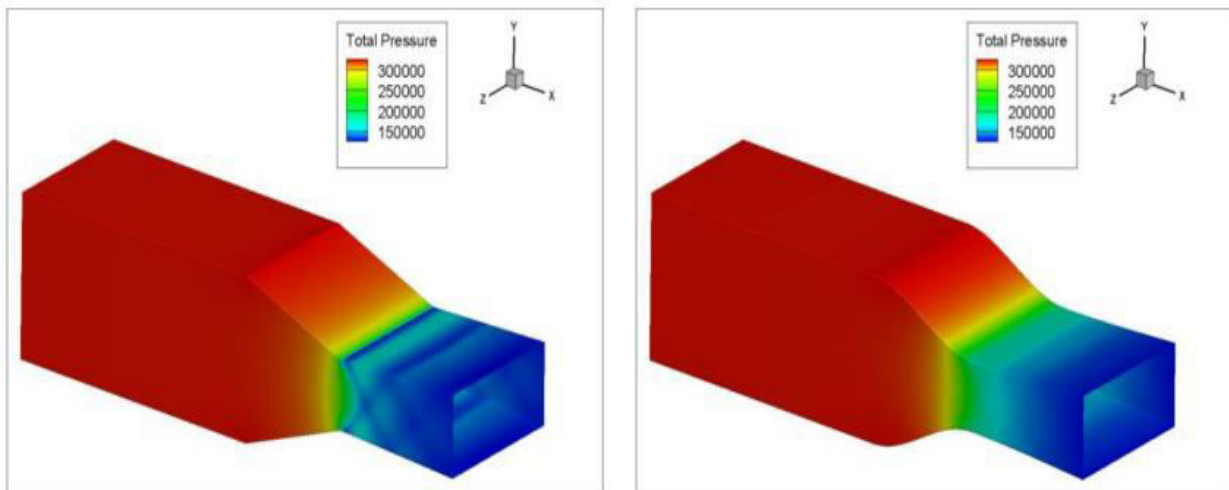


Fig 4: total pressure on nozzle wall in existing design and improved design. [2]

The paper showcases the effects of area ratio in a suddenly expanded duct and base pressure control with micro jets using CFD simulations. The results showed that the area ratio is important to determining the base pressure values. The present results were validated by comparing them with experimental data from Khan et al. The comparison between the present results and the experimental work showed good agreement, indicating the accuracy of the finite element model used in the study.[3]

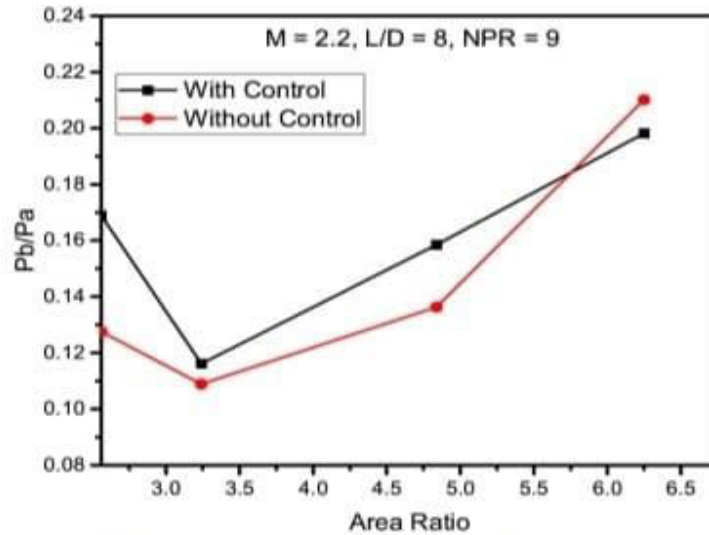


Fig 5: Base pressure distribution [3]

The paper validates the finite element model by comparing the results obtained by Khan et al with the present finite element outputs. The potency of micro-jets in controlling base pressure in suddenly expanded two-dimensional planar ducts was studied. The study explains on two micro-jets placed at 90° intervals along a pitch circle distance of 1.5 times the nozzle exit diameter. The simulations were conducted using CFD analysis with the ANSYS software. The turbulence models, specifically the K-ε standard wall function turbulence model, were used in the simulations. The calibrated turbulence model from the code was verified the with commercial CFD software.[4]

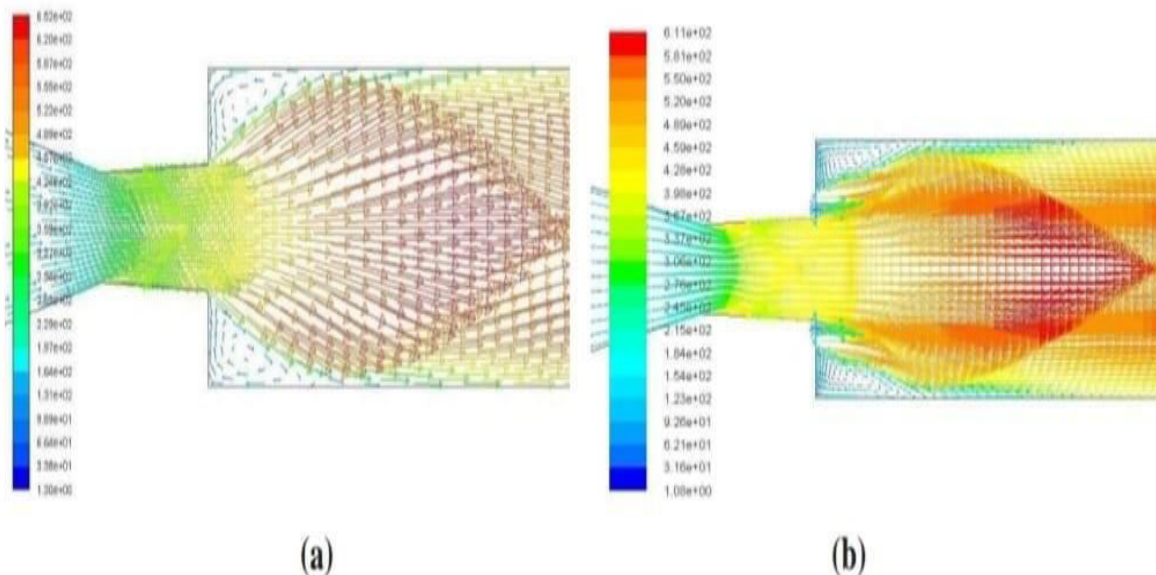
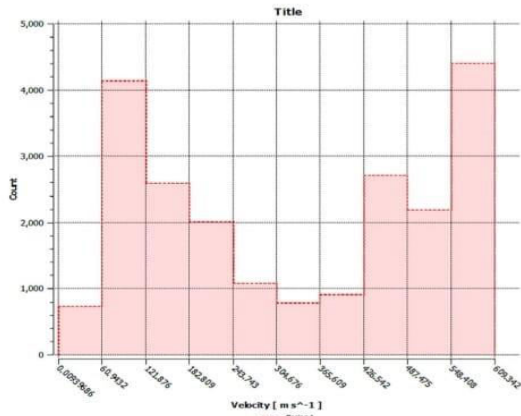


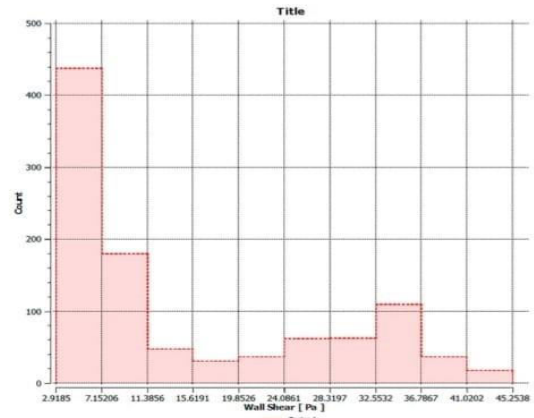
Fig 6: Velocity vector for NPR 3 (a) without micro-jet control (b) micro jet control. [4]



The paper presents contour plots showing the pressure distribution at the nozzle outlet, with a value of -87094.9 Pa. The CFD simulation analysis provided finest results for temperature, static pressure, velocity, and static temperature in the flow. The output obtained from the CFD simulation were compared with the manually calculated standard nozzle equation, indicating the validation of the simulation findings.[5]

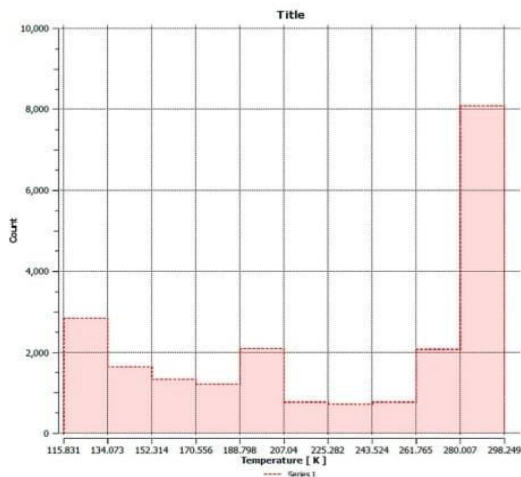


Flow count vs. velocity distribution.

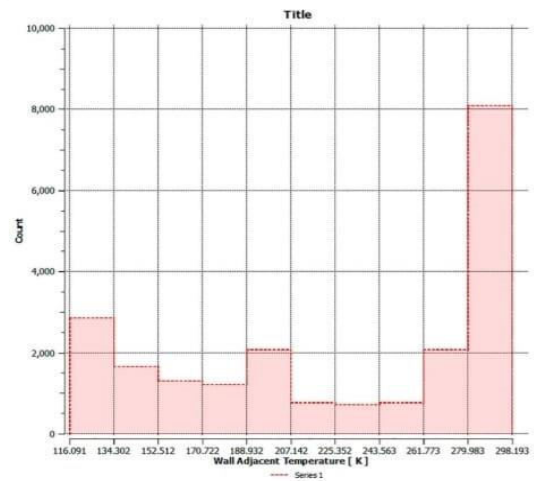


Flow count vs. wall shear.

Fig 7: flow count vs velocity distribution and flow count Vs wall shear [5]



Flow count vs. adjacent wall temperature.



Flow count vs. temperature.

pressures is -87094.9 [pa] in the nozzle outlet (Figure

Fig 8: Flow count vs. adjacent wall temperature and flow count vs temperature [5]

The main intention of the study was to enhance the flow field at various nozzle pressure ratios and investigate the impact of microjet control. The study used a finite element method and CFD techniques to analyze the flow field in CD nozzle with sudden expansion in it. The outputs showed that the presence of microjets in the base region of the nozzle did not majorly affect the flow field. The pressure, Mach number contours, and plots were obtained to observe the behavior of the flow field. The study identified the Mach number, NPR, and length-to-diameter ratio that leads to increase in pressure. The study demonstrated the effectiveness of microjet control in controlling the base pressure and optimizing the flow field in the CD nozzle.[6]



The paper is the study on the variation of area ratio, nozzle pressure ratio, velocity, pressure, and Mach number with and without micro-jets inserted at the base region. The results show that there is impact of resultant forces on area ratio, nozzle pressure ratio, and Mach number. The study also highlights the relationship between Mach number and NPR. It is observed that Mach number increases as the NPR value increases for base pressure control with jets, while for base pressure without control jets, the Mach number decreases with an increase in NPR. The impact of Mach number on pressure ratio shows an inverse relation, where an increase in pressure ratio leads to a decrease in Mach number. The paper also mentions the minimum loss of pressure for micro-jets inserted at the base region compared to without micro-jets, and the decline in pressure ratio after a certain point as the Mach number increases.[7]

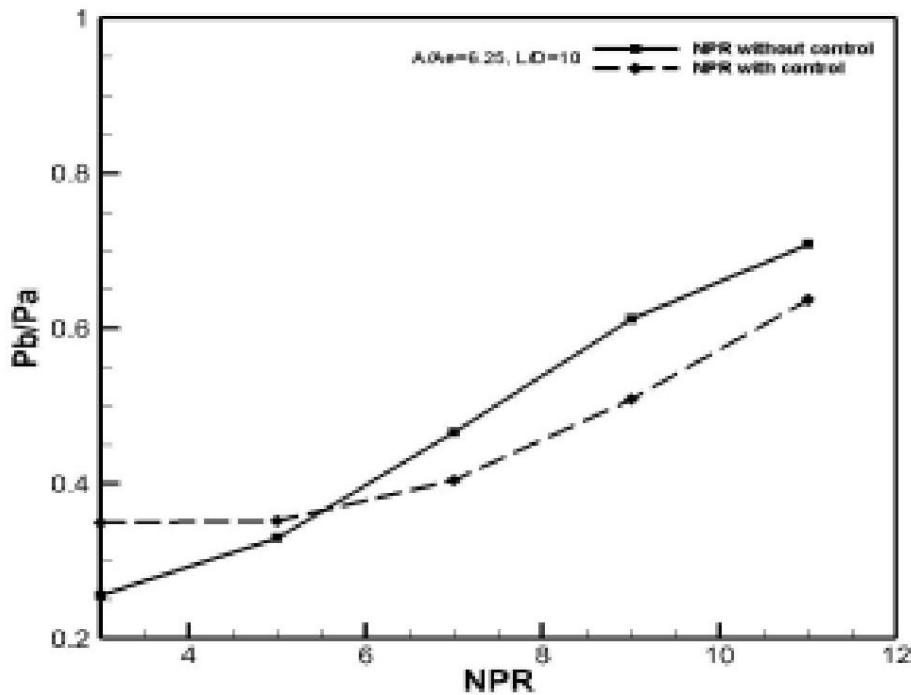


Fig 9: NPR Vs Base pressure [7]

III. CONCLUSION

After grasping the main ideas from the mentioned papers, the key points that can be summarized is as follows:

The study revealed that Mach number increases with higher nozzle pressure ratios (NPR) for base pressure control with jets, but decreases without control jets as NPR rises. This demonstrates an inverse relationship between Mach number and pressure ratio, highlighting the efficiency of micro-jets in minimizing pressure loss at the base. The research emphasized the substantial impact of resultant forces on area ratio, nozzle pressure ratio, and Mach number. While specific numerical data were not provided, the overall findings underscored the importance of these factors for optimizing thrust in rocket engines. The study demonstrated the efficiency of micro-jet control in reducing base suction and regulating base pressure in a convergent-divergent (CD) nozzle during sudden expansion. The micro-jets achieved these objectives without adversely affecting the wall pressure field. Factors influencing base pressure in the nozzle were identified as Mach number, flow relief, level of expansion, and length-to-diameter ratio (L/D). The level of expansion played a significant role in determining base pressure values. The study observed that the total wall pressure and pressure values were highest when the nozzle pressure ratio (NPR) was 11. This suggests that optimizing Mach number, NPR, and L/D can maximize pressure increase within the nozzle. In the nozzle, increasing the divergent angle led to higher fluid flow speed but lower static pressure. The study also indicated that turbulence temperatures increased

with velocity, implying dynamic numerical solutions of turbulence. The research noted that the rectangular nozzle exhibited a greater pressure drop compared to other shapes, with a substantial drop in temperature as well. The paper presented a numerical study using computational fluid dynamics analysis with ANSYS software to assess the efficiency of micro-jets in controlling base pressure in suddenly expanded to two-dimensional planar ducts. The study concluded that area ratio significantly influences base pressure values, with higher area ratios yielding higher base pressures. Microjets were most effective for lower area ratios, while their impact diminished for higher area ratios. The paper primarily focused on analyzing fluid behavior within a convergent-divergent nozzle using CFD analysis with ANSYS 16 software. It confirmed the presence of supersonic flow with a Mach number of 2.50e00 at the outlet. Specific conclusions or comparisons of different nozzle designs were not provided in the paper, as its main objective was to investigate fluid behavior and determine Mach number and velocity distribution within the nozzle.

REFERENCES

- [1] Rajat Mishra, Devendra Lohia, CFD ANALYSIS OF CONVERGENT AND DIVERGENT NOZZLE, International Research Journal of Engineering and Technology (IRJET) Volume: 07, Issue: 05, May 2020.
- [2] Kalyani Bhide *, Kiran Siddappaji and Shaaban Abdallah, Influence of Fluid–Thermal–Structural Interaction on Boundary Layer Flow in Rectangular Supersonic Nozzles, Department of Mechanical Engineering, University of Cincinnati, Cincinnati, OH 45221, USA; s2kn@mail.uc.edu (K.S.); shaaban.abdallah@uc.edu (S.A.) * Correspondence: bhidekr@mail.uc.edu ; Tel.: +1-513-556-332.
- [3] Sher Afghan Khan^{1*}, Abdul Aabid¹, Fharukh Ahmed Mehaboobali Ghasi^{2,3}, Abdulrahman Abdullah Al-Robaian⁴, Ali Sulaiman Alsagri⁴, Analysis of Area Ratio In a CD Nozzle with Suddenly Expanded Duct using CFD Method, PENERBIT AKADEMI BARU, Issue 5 (2019)
- [4] SHER AFGHAN KHAN¹, ABDUL AABID² AND MAUGHAL AHMED ALI BAIG³, CFD ANALYSIS OF CD NOZZLE AND EFFECT OF NOZZLE PRESSURE RATIO ON PRESSURE AND VELOCITY FOR SUDDENLY EXPANDED FLOWS, international journal of mechanical and production engineering research and development(IJMPERD), Volume 8, Issue 3, June 2018.
- [5] R. Ramesh Kumar and Yuvarajan Devarajan, CFD simulation analysis of two-dimensional convergent-divergent nozzle, International journal of ambient energy, volume 41.
- [6] Abdul Aabid¹, Zakir Ilahi Chaudhary², Sher Afghan Khan¹, Modelling and Analysis of Convergent Divergent Nozzle with Sudden Expansion Duct using Finite Element Method, Journal of advanced research in fluid mechanics and thermal science, Issue 1(2019).
- [7] Shreya Mane¹, Prasannalakshmi Thota², Yash Deshmukh³, Rishab Gupta⁴, Effect of Area Ratio and Nozzle Pressure Ratio for CD Nozzle, International Journal of All Research Education and Scientific Methods (IJARESM), Volume 10, Issue 5, May-2022.



INNO SPACE
SJIF Scientific Journal Impact Factor
Impact Factor: 8.423



ISSN INTERNATIONAL
STANDARD
SERIAL
NUMBER
INDIA



INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

9940 572 462 6381 907 438 ijirset@gmail.com



www.ijirset.com

Scan to save the contact details