J. Electrical Systems 20-4s (2024): 766-772

[1] Sampreetananda Bharali
[2] Shubham
[3] Sarthak Joshi
[4] M. Zunaid

Optimization of Bypass Flow Introduction Angle in Dual-Throat Nozzle

JES
Journal of Electrical Systems

Abstract: - This research paper investigates the optimal angle for introducing a bypass jet in rocket engines for thrust vectoring. Numerical simulations using a finite-volume method were performed to test a range of bypass jet angles and their effect on flow performance. Results indicate that the optimal bypass jet angle is between 40 and 50 degrees, with larger angles leading to decreased thrust. These findings demonstrate the potential for bypass jet introduction to improving thrust vectoring in rocket engines, especially in the supersonic regime. Such insights may inform the development of more efficient and effective rocket engines for various applications, including satellite launches, space exploration, and defense. Overall, this study represents a valuable contribution to the field of rocket propulsion and may drive advances in engine design and performance.

Keywords: Nozzle; Thrust vectoring; CFD; Computational Analysis; BDTN; Bypass nozzle.

I. Introduction

The design of efficient and high-performing rocket propulsion systems has been a crucial aspect of aerospace engineering for several decades. The development of rocket engines requires careful consideration of various factors, including thrust performance, fuel consumption, and operational safety. One of the critical factors that affect the thrust performance of rocket engines is the direction and intensity of the exhaust jet.

Thrust vectoring is a technique of changing the direction of the exhaust jet to control the rocket's flight path. This technique is particularly useful in situations where precise control of a rocket's trajectory is required, such as when a spacecraft is tethered, maneuvering, or landing. It can help recover or even control the stall regions of a flight along with providing STOL capabilities [1]. Thrust vectoring can be achieved by a variety of methods, including mechanical vanes, fluid jets, and nozzle formation. One promising technology that has received increasing attention in recent years is bypass jet launching. Although there are other methods to use fluidic end effectors to control an aircraft as highlighted by Wilde et al[2], this paper focuses on thrust vectoring.

Introducing a bypass jet involves introducing a secondary gas jet into the exhaust stream to change the direction and intensity of the exhaust jet. This technique can be used to achieve thrust vectoring by pointing the secondary beam in a specific direction. The introduction of bypass jets also improves exhaust gas mixing, leading to better combustion and higher thrust performance. However, the effectiveness of the bypass jet introduction for thrust vectoring depends on several factors, including the angle of introduction and the flow velocity of the secondary jet.

Therefore, determining the optimal entry angle of the bypass jet is critical to maximizing the effectiveness of thrust vectoring. Several studies have investigated the effect of entry angle on the thrust vectoring capability of a rocket engine. However, these studies have produced mixed results and there is still no consensus on the optimum angle of the introduction of the shunt beam thrust vector.

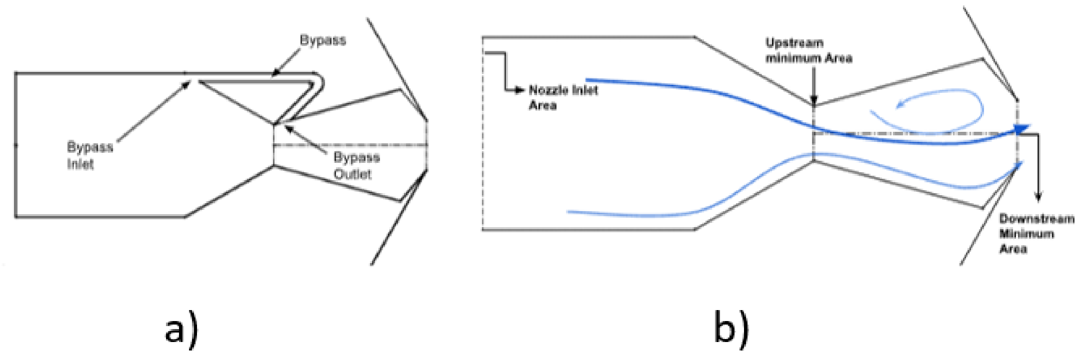


Fig. 1. a) Bypass dual throat nozzle b) Expected fluid flow lines [1]

[1] Student, Mechanical Engineering Department, Delhi Technological University, New Delhi, India. sampreetbharali@gmail.coms
[2] Student, Mechanical Engineering Department, Delhi Technological University, New Delhi, India. shmalik97@gmail.com
[3] Student, Mechanical Engineering Department, Delhi Technological University, New Delhi, India. sarthakjoshi2001@gmail.com
[4] Assistant Professor, Mechanical Engineering Department, Delhi Technological University, New Delhi, India. mzunaid3k@gmail.com

1.1 Literature Review

In the research paper titled "Experimental and numerical investigations of a bypass dual throat nozzle" by Gu et al.[3], the authors found that bypass flow can increase the thrust vectoring efficiency, and the thrust vectoring effect was enhanced with a decrease in the bypass slot height.

Gal-Or's book[4], "Vectored Propulsion Supermaneuverability and Robot Aircraft," discusses the concept of vector propulsion and its applications in supermaneuverability and robot aircraft. The book covers various aspects of vector propulsion and its implementation in different aircraft.

Deere et al[5], a paper titled, "Summary of Fluidic Thrust Vectoring Research Conducted at NASA Langley Research Center",discusses the different fluidic thrust vectoring concepts, their advantages, and their applications in aircraft.

Karen A. Deere et al[6], provides valuable insights into the design and performance of thrust vectoring nozzles for supersonic aircraft. The proposed dual throat nozzle is shown to be a promising alternative to traditional mechanical thrust vectoring systems and the results of the CFD simulations provide valuable guidance for future research in this area.

P.J. Yagle et al[7], in a paper titled "Demonstration of Fluidic Throat Skewing for Thrust Vectoring in Structurally Fixed Nozzles" provides a review of the existing literature on thrust vectoring and fluidic thrust vectoring, highlighting the benefits and limitations of these approaches. They then describe the design and performance of their proposed fluidic throat skewing system and present the results of experiments that were conducted to evaluate its performance. The authors find that their fluidic throat skewing system can effectively vector the thrust in a fixed nozzle and that it performs similarly to a mechanically actuated nozzle in terms of thrust vectoring angle and efficiency.

Shin et al[8] provides valuable insights into the performance and potential of the dual throat nozzle for use in thrust vectoring applications. The study's findings suggest that the nozzle has several advantages over other designs, and could be a valuable addition to the range of available thrust vectoring nozzles. Further research and development in this area may lead to improvements in the performance and control of jet engines.

In the research paper titled "Experimental study of an axisymmetric dual throat fluidic thrust vectoring nozzle for supersonic aircraft application" by Flamm et al[9]. The authors found that the nozzle exhibited excellent thrust vectoring capability and reduced drag in the off-design conditions.

Shi et al.[10], in the paper, "Investigation on flow characteristics of SVC nozzles," presents numerical simulations to study the effects of various design parameters on the flow characteristics of SVC nozzles.

Anderson et al.'s paper[11], investigates the feasibility of hybrid fluidic/mechanical thrust vectoring for fixed-exit exhaust nozzles. The authors conducted experimental tests and found that hybrid fluidic/mechanical thrust vectoring is feasible and can provide better performance compared to conventional thrust vectoring methods.

Mason and Crowther's paper[12], "Fluidic Thrust Vectoring for Low Observable Air Vehicles," The authors present a fluidic thrust vectoring concept that can be used to reduce the infrared signature of the aircraft. They also discuss the potential of fluidic thrust vectoring for reducing the radar cross-section of the aircraft.

In their paper, "Visualization and analysis on the thrust vectoring control in three-dimensional dual-throat nozzles", Wu et. al.[13], provides a comprehensive analysis of the use of TVC in three-dimensional dual-throat nozzles, using a combination of numerical simulations and experimental data. The study contributes to the understanding of the flow physics and thrust performance of TVC nozzles, and provides valuable insights for the design and optimization of such systems.

Afridi et.al.[14], in their paper "Multi-objective nozzle design optimization for maximum thrust vectoring performance" investigates the effect of various parameters such as nozzle bypass angle, convergence angle, and bypass width on the performance of the bypass dual throat nozzle. The results show that bypass width and bypass angle significantly affected thrust vectoring.

The paper titled "Computational study of axisymmetric divergent bypass dual throat nozzle" by Wang et. Al[15], provides a comprehensive analysis of the flow characteristics and performance of an axisymmetric divergent bypass dual throat nozzle, using numerical simulations. The study contributes to the understanding of the design and optimization of such nozzles and provides valuable insights for the development of advanced aircraft propulsion systems

These recent studies demonstrate the potential of bypass jet introduction for improving the thrust performance and control of rocket engines. However, there is still a need for further research to determine the optimal angle of introduction for bypass jet thrust vectoring. In this paper, we aim to build on these recent studies by investigating the optimal angle of bypass jet introduction for thrust vectoring using CFD simulations.

In this paper, we aim to investigate the optimal angle of bypass jet introduction for thrust vectoring using computational fluid dynamics (CFD) simulations. The study will involve varying the angle of introduction and flow rate of the secondary jet and evaluating the effect on the rocket engine's thrust vectoring capability and overall thrust performance. The results of this study will provide valuable insights into the effectiveness of thrust vectoring for introducing bypass jets and informing the design of future rocket propulsion systems. Additionally, the study will demonstrate the potential of CFD simulations to assess rocket engine performance and develop optimized designs.

1.2. Structure

The nozzle used to analyze BDTN is shown in Fig. 2. The cavity divergence angle is 15.69 degrees and the cavity convergence angle is 50 degrees. The nozzle throat radius is 9mm and the cavity is 26mm. The bypass incident angle is 45 degrees. Table 1 lists other geometric variables such as expansion rate (A2/A1), shunt width (We), and cavity bottom (Rz).

Table 1. Geometric Values of BDTN

Geometric Variable	Values
R_2/R_1	1.22
A_2/A_1	1.49
W_e	3.7mm
R_2	11mm
θ	30deg, 40 deg, 45 deg, 50deg, 60 deg, 70deg, 80deg, 90deg

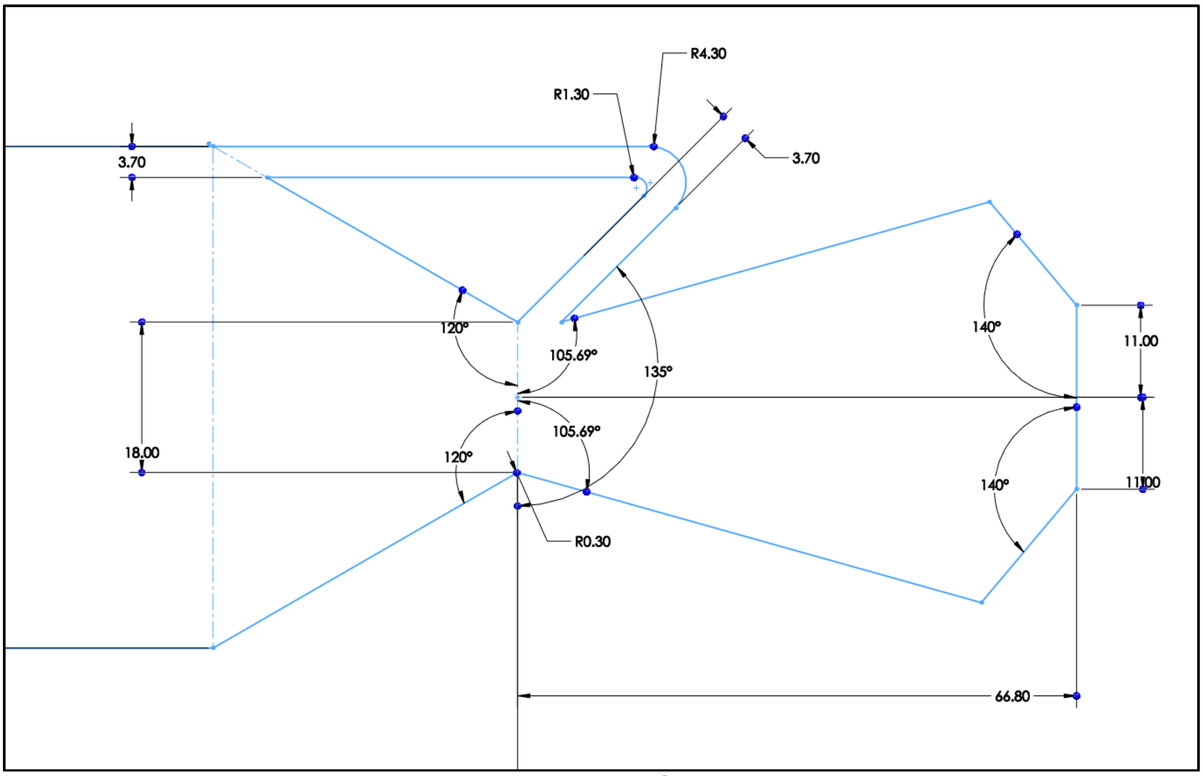


Fig. 2. BDTN geometric dimensions;

II. Computational Fluid Dynamics

2.1 CFD Model Calibration

The Dual Throat Bypass Nozzle was designed in the Design Modeller environment of Ansys Workbench. The analysis would be on a 2D nozzle model. Next, the meshing operations were carried out with the general mesh size set as 2mm. An edge-sizing operation was performed along the central axis of the nozzle with a mesh size of 1mm, hard enforced. The total number of mesh nodes was 386974.

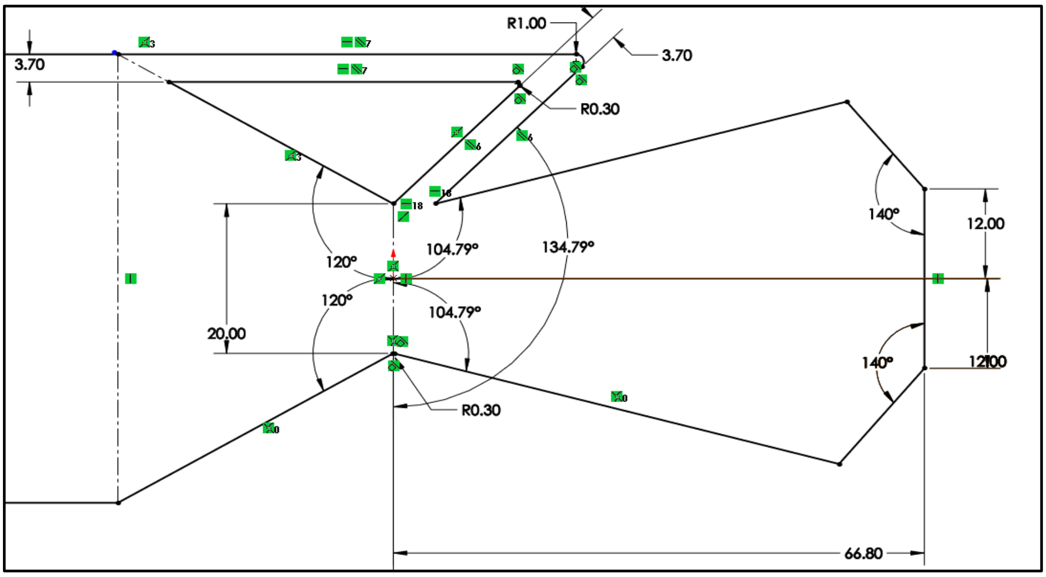


Fig.3. Model for CFD method calibration

For the analysis setup in Fluent, the following configurations were used:
Launcher configurations: Double precision, parallel solver with 4 processes and ACT enabled

- Energy equations: On
- Solver method: K-ε model (2-equation model)
- Conditions: Realizable, Standard wall model
- Air density model: Ideal gas
- Viscosity model: Sutherland
- Inlet type: Pressure Inlet
- Initial pressure at inlet: 405300 Pa
- Gauge pressure at inlet: 101325 Pa
- Output type: Pressure Output
- Reference values: Standard, Computed from the inlet
- Solution methods:
  - Turbulent Dissipation Rate: Second Order Upwind
  - Gradient: Least Squares Cell-based
  - Flow: Second Order Upwind
  - Turbulent KE: Second Order Upwind
- Residuals convergence criterion: 10-3 for all values
- Initialization: Standard, relative cell initialization from the inlet

The CFD setup uses mesh topology that is refined near boundary areas and uses edge sizing along the central axis for more accurate values along the axis, especially near throat regions. Dimensions of the calibration model are shown in fig. 3. The comparison of the data points extracted from the analysis to the experimental data points recovered from previous literature (Hamedi-Estakhrsar et al[16], Gu et al[1]) is shown in Fig.4. As it shows to correctly simulate the flow details, a standard realizable wall function with the k-e turbulent model will be used.

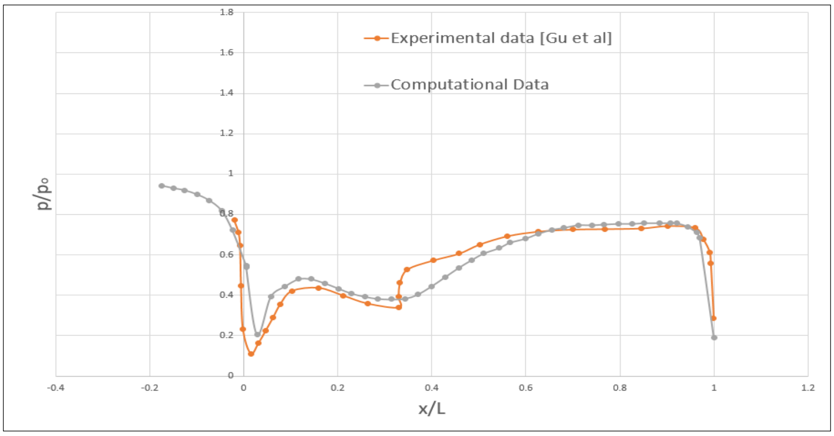


Fig. 4. Comparison of experimental data points;

2.2. Grid Independance Test

The meshing procedure followed a standard mesh size of 2mm. An edge-sizing operation was cascaded with the X-axis as the principal axis. The element size for edge sizing is 1mm, hardbound behavior. Both the general mesh element size and edge element sizing have been considered as the input parameters for the grid independence test. The maximum mach speed at nozzle outlet has been considered as the output parameter to monitor the effect of mesh element density:

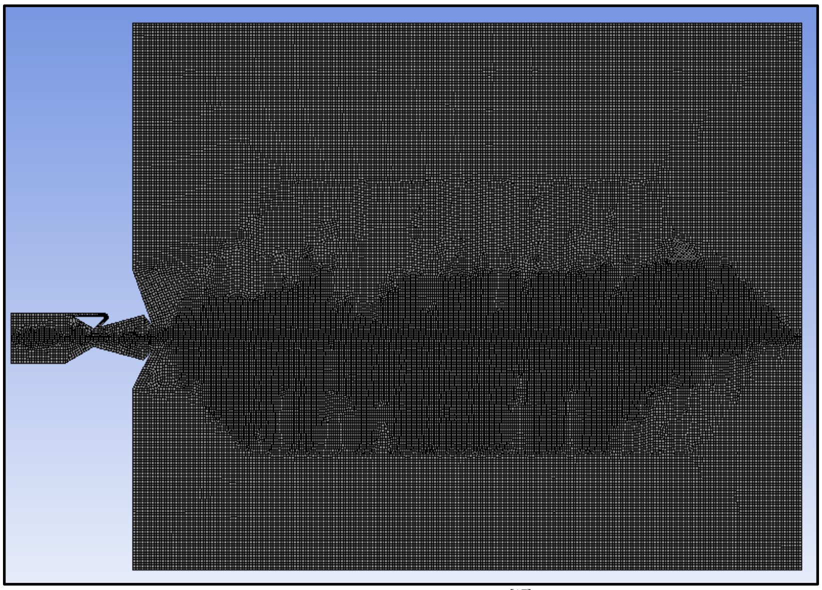
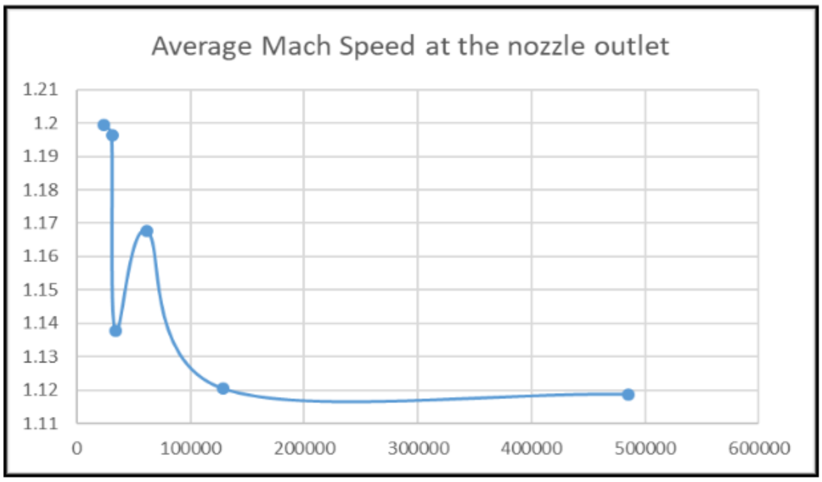


Fig. 5. Mesh visualization[17]



Along x-axis: Number of mesh elements
Along y-axis: Maximum Mach Speed at the nozzle outlet

Fig. 6. Graphical representation of the grid independence study

The maximum mach number of the jet stream leaving the nozzle at the outlet has been calculated for multiple meshing configurations, with the number of mesh elements varying from 25000 to 500000. The variation of values have been plotted. From the above graphs, it is clear that the values stabilize considerably when number of elements exceed 100000. So a mesh with total of 128584 elements has been selected to get accurate and consistent results while not exceeding computational requirements of the system.

III. Results and Discussion

The nozzle bypass introduction angle has been tested for 30°, 40°, 45°, 50°, 60°, 70°, 80°, and 90° with the horizontal. The Nozzle Pressure Ratio (NPR) is kept at 4 and the area ratio is 1.49. The Mach number and pressure contours are shown in figures 8 and 9. From it, we can clearly see that the maximum mass flow rate is achieved when the bypass introduction is done at an angle between 40° and 50° with the horizontal. Further fine-tuned analysis shows that the optimal angle for flow introduction is at an angle of 45° with the horizontal.

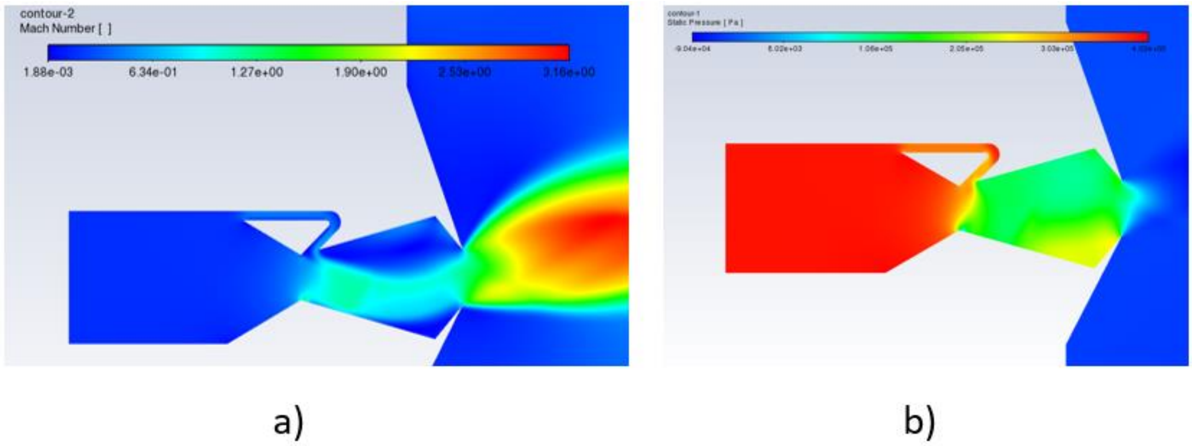


Fig. 7. Contours of (a) Mach number at bypass jet introduction angle (θ) = 45° (b) Pressure at bypass jet introduction angle (θ) = 45°

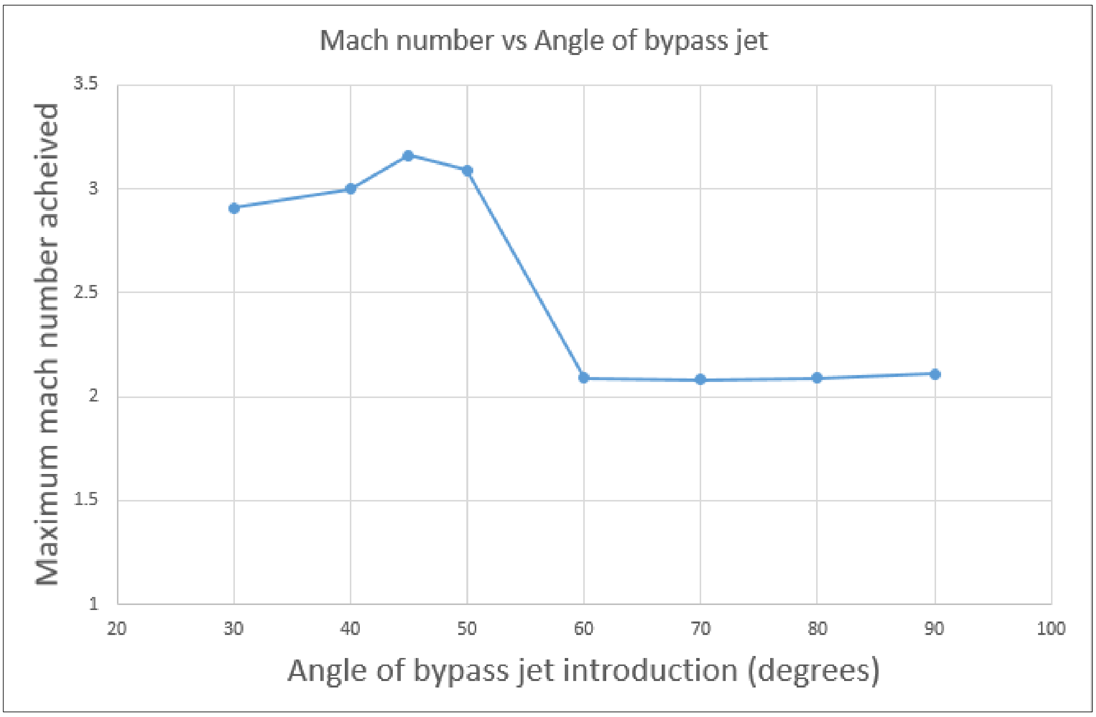


Fig. 8. Plot of maximum Mach number achieved vs the angle of bypass flow introduction

The initial angle from 30° shows the increment of jet velocity with the increase in the angle of introduction. The velocity peaks between 40° and 50°. Post 50°, the velocity reduces and stabilizes at around 60°. The Mach number values stay between 2 and 2.1 till the angle of introduction reaches 80°. Beyond that, the Mach number rises slightly till the angle is 90°. Fig. 8 shows the variation in the maximum Mach number with respect to the angle of flow introduction. The highest flow velocity was achieved by an angle of 45^0 where the maximum recorded mach number of the fluid was 3.16. The pressure contours and velocity contours for the same are shown in fig.7.

IV. CONCLUSION

The paper successfully explored and calibrated the computational method for a flow analysis of a dual throat bypass nozzle for thrust vectoring and compared it to the experimental values achieved by previous research. The experiment yielded a high degree of similarity to the experimental setup and results were validated by other publications.

The experiment to find the most effective bypass angle was conducted by creating multiple geometries and meshing then using the same approach as the calibration model. Each geometry was analyzed at NPR 4 an area ratio of 1.49. The only variable was the bypass flow introduction angle.

The comparison of various angles of bypass stream introduction as per performance parameters shows that the introduction of the flow at 45° results in minimum loss of jet velocity for the given yaw angle divergence at a nozzle pressure ratio of 4. The maximum output velocity of the fluid at this angle is 45°, and the average nozzle pressure at the outlet for the same is 103083.1 Pa.

V. ACKNOWLEDGEMENTS

The authors would like to thank the Department of Mechanical Engineering of Delhi Technological University for the support and resources provided to carry out this research.

REFERENCES

[1] J.W. Shi, Z.X. Wang, L. Zhou, X.L. Sun Investigation on flow characteristics of SVC nozzles J. Appl. Fluid Mech., 11 (2) (2018), pp. 331-342

[2] P.I.A. Wilde, W.J. Crowther, A. Buonanno, A. Savvaris Aircraft Control Using Fluidic Maneuver Effectors AIAA Paper 2008-6406 (2008)

[3] Gu, Rui, Jinglei Xu, and Shuai Guo. "Experimental and numerical investigations of a bypass dual throat nozzle." *Journal of Engineering for Gas Turbines and Power* 136.8 (2014).

[4] Gal-Or, Book.,1990, Vectored Propulsion Supermaneuverability and Robot Aircraft, Springer-Verlag, New York.

[5] Deere, K. A.,2003, "Summary of Fluidic Thrust Vectoring Research Conducted at NASA LangleyResearchCenter," AIAAPaper No. 2003-3800. 10.2514/6.2003-3800

[6] Deere, Karen, et al. "Computational study of an axisymmetric dual throat fluidic thrust vectoring nozzle for a supersonic aircraft application." *43rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*. 2007.

[7] P.J. Yagle, D.N. Miller, K.B. Ginn, J.W. Hamstra Demonstration of fluidic throat skewing for thrust vectoring in structurally fixed nozzles J. Eng. Gas Turbines Power, 123 (3) (2001), pp. 502-507

[8] Shin, Choon Sik, et al. "A computational study of thrust vectoring control using dual throat nozzle." *Journal of Thermal Science* 19 (2010): 486-490.

[9] Flamm, Jeffrey, et al. "Experimental study of an axisymmetric dual throat fluidic thrust vectoring nozzle for supersonic aircraft application." *43rd AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit*. 2007.

[10] Shi, Jingwei, et al. "Investigation on flow characteristics of SVC nozzles." *Journal of Applied Fluid Mechanics* 11.2 (2018): 331-342.

[11] Anderson, Christopher, et al. "Investigation of hybrid fluidic/mechanical thrust vectoring for fixed-exit exhaust nozzles." *33rd Joint Propulsion Conference and Exhibit*. 1997.

[12] M.S. Mason, W.J. Crowther Fluidic Thrust Vectoring for Low Observable Air Vehicles AIAA Paper 2004-2210 (2004)

[13] Wu, Kexin, Tae Ho Kim, and Heuy Dong Kim. "Visualization and analysis on the thrust vectoring control in three-dimensional dual-throat nozzles." Journal of Visualization 24 (2021): 891-915.

[14] Afridi, Saadia, and Tariq Amin Khan. "Multi-objective nozzle design optimization for maximum thrust vectoring performance." Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering 237.3 (2023): 587-599.

[15] Wang, Y., Xu, J., Huang, S., Lin, Y., & Jiang, J. (2019). Computational study of axisymmetric divergent bypass dual throat nozzle. Aerospace Science and Technology, 86, 177-190.

[16] Hamedi-Estakhrsar MH, Mahdavy-Moghaddam H. Experimental evaluation and numerical simulation of performance of the bypass dual throat nozzle. Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering. 2021;235(7):768-781. doi:10.1177/0954410020959886

[17] Aniket. (2020, November 19). You will need to manually split the faces [Comment on the online forum post *How can I make a uniform meshing in the bypass area of this dual throat nozzle?*]. Ansys forum. https://forum.ansys.com/forums/topic/how-can-i-make-a-uniform-meshing-in-the-bypass-area-of-this-dual-throat-nozzle/