Design and Optimization of Divergent Angle of a CD Rocket Engine Nozzle Using Computational Fluid Dynamics

[1] Anushka Mendjoge, [2] Sudhanshu Shinde, [3] Yash Kulkarni,

[4] Prof. (Dr.) Sameer Bhosale

[1,2,3,4] Department of Mechanical Engineering, PES Modern College of Engineering

Abstract

The present study involves the Computational Fluid Dynamics (CFD) analysis of a rocket engine nozzle to investigate the behaviour of supersonic flow at different divergent angles. A two-dimensional axi-symmetric model is employed, and the governing equations are solved using the finite-volume method in ANSYS FLUENT® software. The inlet boundary conditions are set based on available experimental data. The analysis focuses on studying the variations in key parameters such as the Mach number and static pressure. Additionally, the phenomenon of oblique shock is visualized, and the movement of the shock wave with respect to the divergent angle is observed. The results indicate that at a divergent angle of 8°, the shock wave is completely eliminated from the nozzle. Furthermore, it is observed that the Mach number shows an increasing trend as the divergent angle increases, suggesting the existence of an optimal divergent angle that minimizes shock-related instabilities and meets the thrust requirements for the rocket engine. The findings from this analysis provide valuable insights for the design and optimization of rocket engine nozzles, with potential applications in enhancing overall engine performance and stability.

I. Introduction

This research paper investigates the design and optimization of the divergent angle of a converging-diverging (CD) rocket engine nozzle using Computational Fluid Dynamics (CFD). The divergent angle plays a critical role in the nozzle's performance, affecting the expansion and acceleration of exhaust gases. By utilizing CFD simulations, this study aims to analyze the flow behaviour within the CD nozzle and determine the optimal divergent angle that maximizes thrust efficiency. The findings from this research contribute to the field of rocket engine design by providing valuable insights into nozzle optimization using CFD techniques.

Computational Fluid Dynamics (CFD) is an invaluable engineering tool that facilitates experimentation across a broad range of processes involving transport phenomena. Its applications extend beyond fluid dynamics, offering a versatile approach to problem-solving. Traditional methods, such as analytical techniques or experimental testing with prototypes, present challenges with complexity, cost, and time. However, the introduction of CFD has revolutionized the engineering field by addressing these difficulties.

In CFD, problems are simulated using specialized software, allowing for mathematical solutions to the associated transport equations with the aid of computers. This enables us to predict outcomes before physical experimentation, saving both time and resources. This research focuses on determining the optimal divergent angle for a Converging-Diverging Rocket engine nozzle to achieve maximum outlet velocity and thrust requirements. Shockwaves are generated inside the nozzle that lead to flow instabilities resulting in reduced exit Mach number and engine thrust. The divergent angle of the nozzle can be changed to mitigate these issues.

The analysis includes Divergent angles of 5˚,6˚,7˚,8˚,9˚,10˚,11˚,12˚. As physical experimentation of this would be very costly and time consuming, CFD is a tool that helps us to perform simulation on the nozzle and help address this issue.

II. Literature Review

1. CFD Analysis of Compressible Flows in A Convergent-Divergent Nozzle
The study conducted by S. A. Khan, O. M. Ibrahim, and A. Aabid [1] demonstrated the flow behaviour in a convergent-divergent nozzle using computational fluid dynamics (CFD) techniques. The objective of this research was to gain insights into the performance characteristics and optimize the design of the nozzle for efficient and effective operation.

2. Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics
Biju Kuttan P. and M. Sajesh [2] present a comprehensive study on determining the optimum divergent angle for a rocket engine nozzle. The paper addressed the challenges of flow instabilities caused by shock formation within the nozzle and proposed varying the divergent angle as a potential solution. The authors conducted CFD simulations on nozzles with divergent angles of 4°, 7°, 10°, 13°, and 15° to investigate the effects on flow behaviour and nozzle performance. The results indicated that increasing the divergent angle leads to an increase in the
Mach number at both the exit and throat sections of the nozzle. The Mach number ranged from 2.20 for a 4° angle to the highest value of 4.82 for a 15° angle at the exit. Similarly, at the throat, the Mach number increased from 8.26 at 4° to 1.25 at 15°. The static pressure decreased with the divergent angle, indicating a favourable relationship between the angle and pressure distribution.

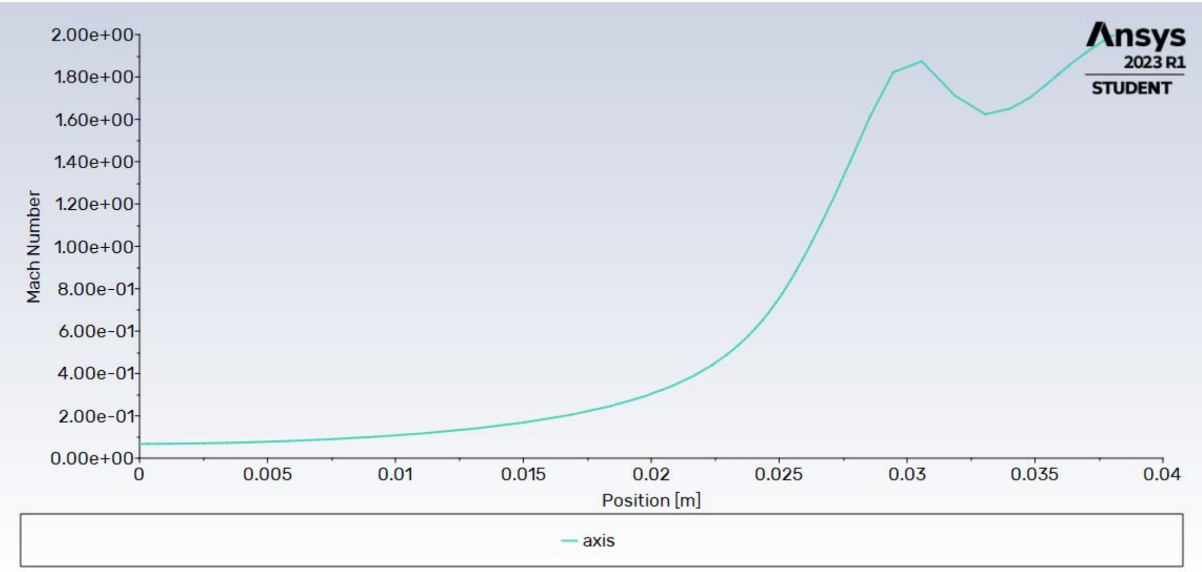
3. Numerical study of convergent-divergent nozzle at different throat diameters and divergence angles
The study conducted by Meena et al. [4] focuses on the numerical investigation of a convergent divergent nozzle with varying throat diameters and divergence angles. The purpose of the research is to understand the flow behaviour and performance characteristics of the nozzle under different geometric configurations. The authors employ computational fluid dynamics (CFD) simulations to analyse the nozzle's behaviour and assess its suitability for specific applications.
The study begins by emphasizing the importance of convergent-divergent nozzles in various engineering fields, such as aerospace propulsion systems and gas turbines. It highlights the significance of optimizing nozzle geometry to achieve desired flow characteristics, including maximum thrust and efficiency. The review also discusses previous studies that have explored the influence of throat diameter and divergence angle on nozzle performance.

III. Objectives

1. Understanding the flow behaviour and performance characteristics of the CD rocket engine nozzle, including Mach number, static pressure, and shock formation, under various divergent angles.
2. Investigating the effects of divergent angle variations on the nozzle performance.
3. Identifying the optimal divergent angle that balances the trade-off between maximizing thrust efficiency and minimizing flow instabilities caused by shock formation.
4. Developing a reliable and accurate CFD simulation approach to model the flow behaviour and performance of the CD rocket engine nozzle.
5. Validating the CFD simulation results by comparing them with theoretical models to ensure the accuracy and reliability of the simulation approach.

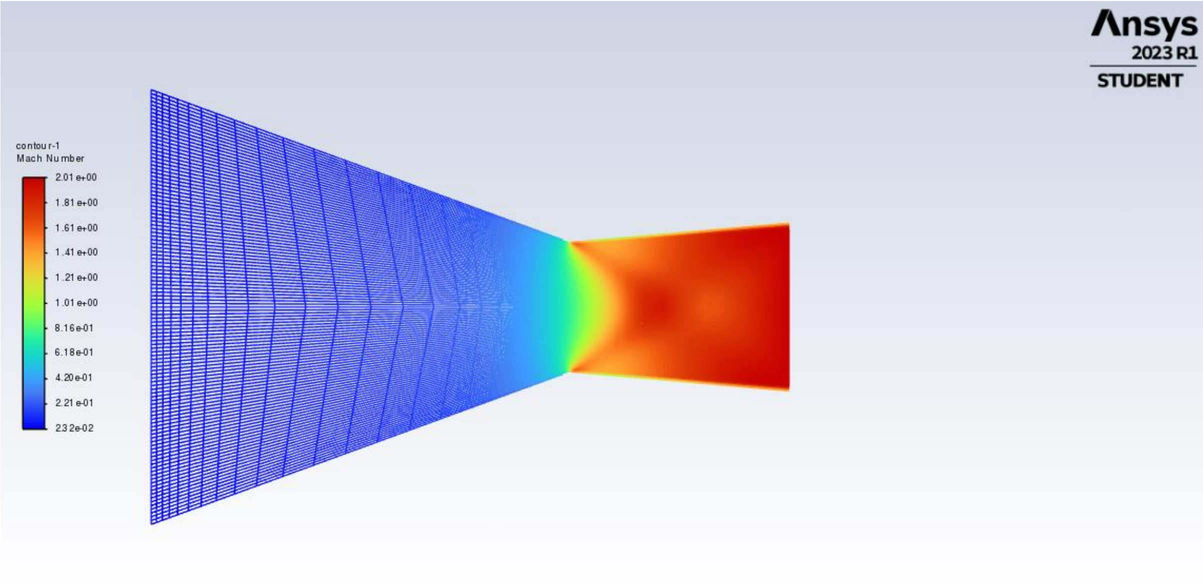
IV. Results and Discussion

Mach vs position for shock



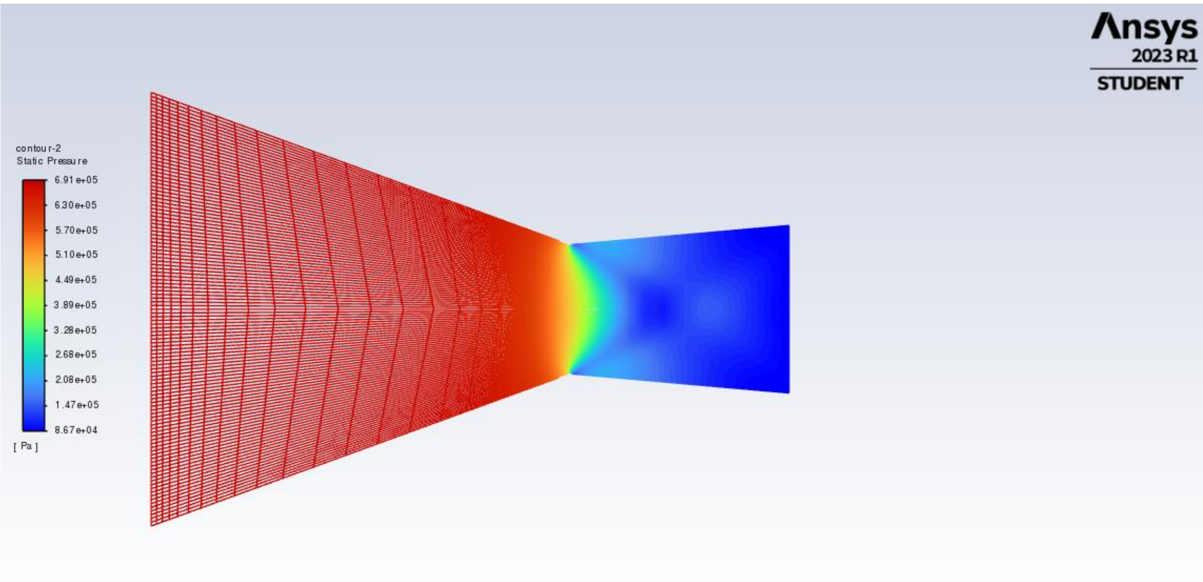
This is the Mach VS Position for shock graph for 5˚ Divergent angle. It clearly shows that a shockwave is generated at the ending of the Divergent side. The position of the shock is to be seen at 0.0328.

Mach contour



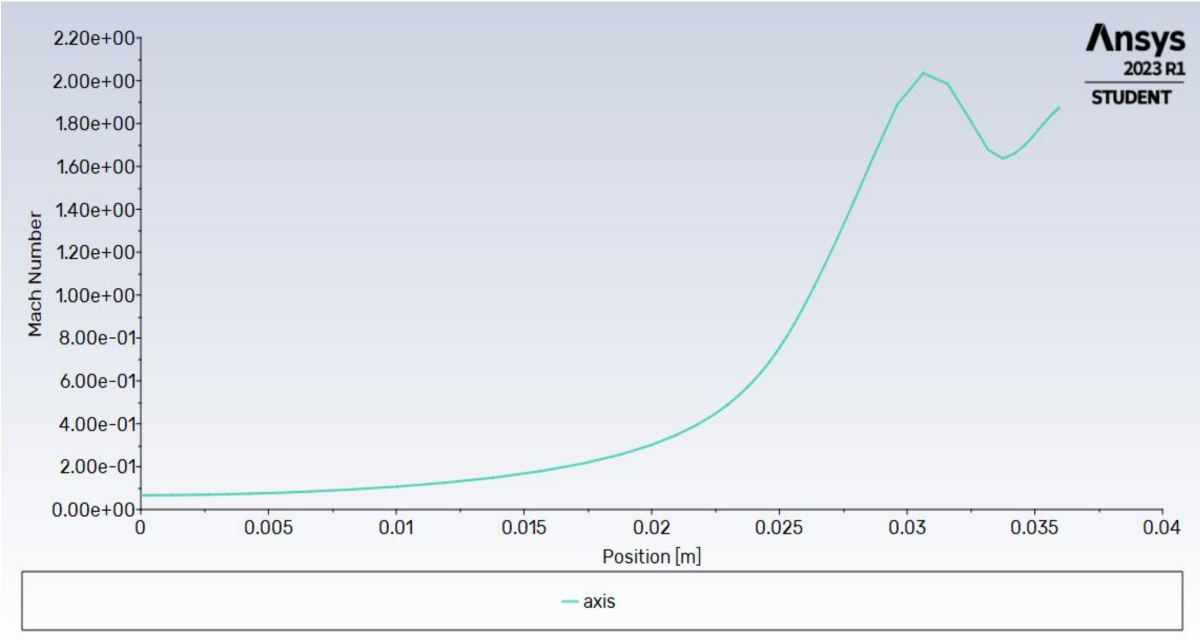
The above figure shows the Mach contour formed for 5˚ Divergent angle.

Static Pressure contour



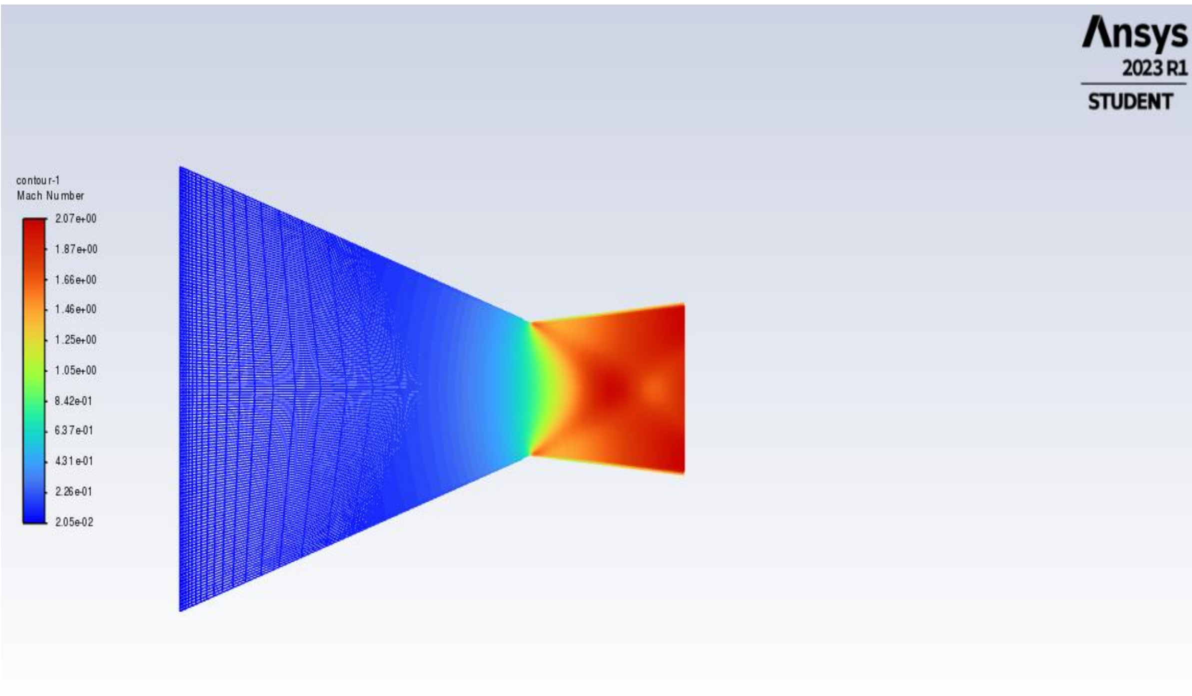
The above figure shows Static Pressure contour for 5˚ Divergent angle.

Mach vs position for shock



This is the Mach VS Position for shock graph for 6˚ Divergent angle. It clearly shows that a shockwave is generated at the ending of the Divergent side. The position of the shock is to be seen at 0.0329.

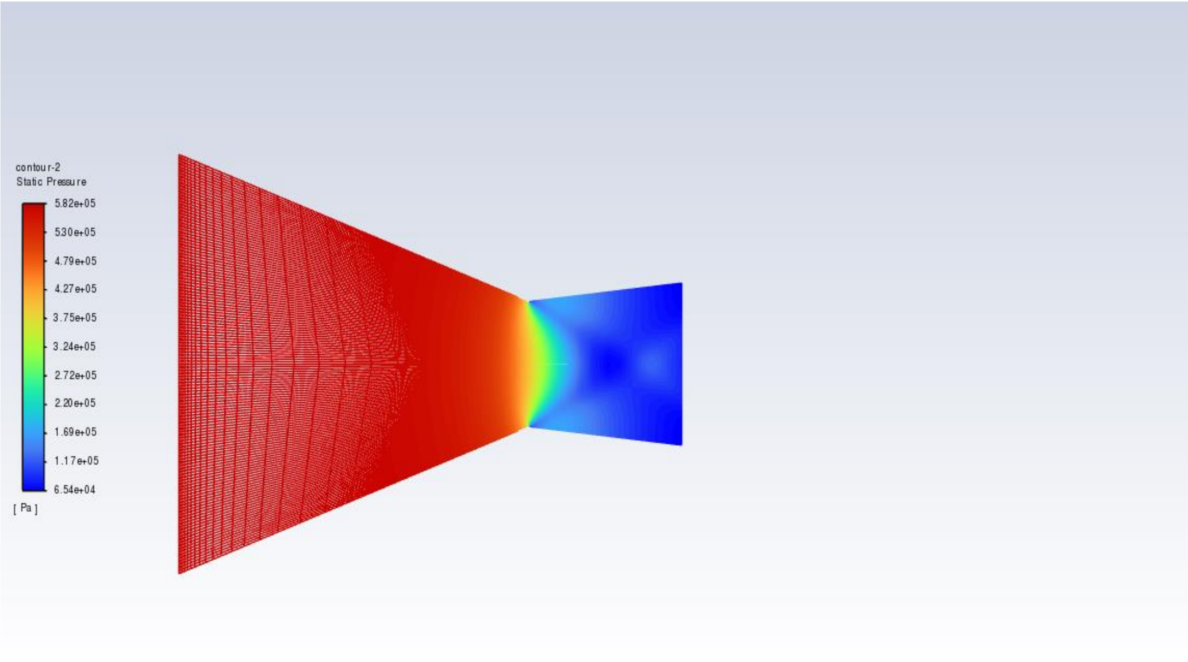
Mach Contour



The above figure shows the Mach contour formed for 6˚ Divergent angle.

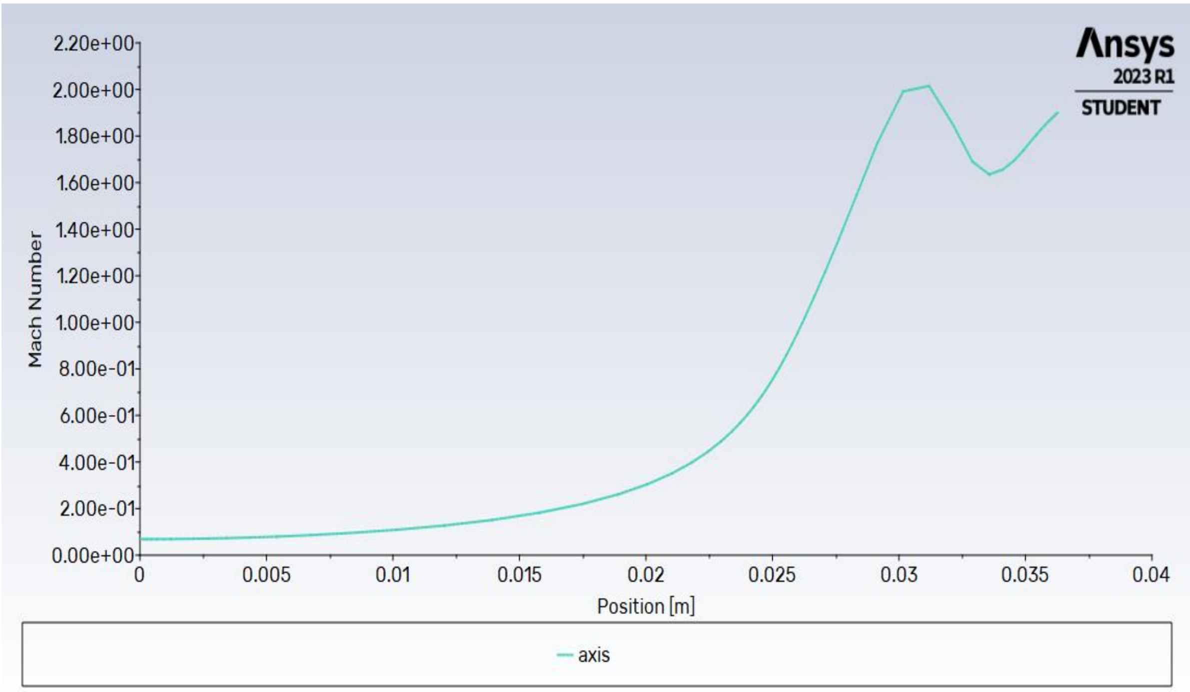
International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 07 | July - 2023 SJIF Rating: 8.176 ISSN: 2582-3930

Static Pressure contour



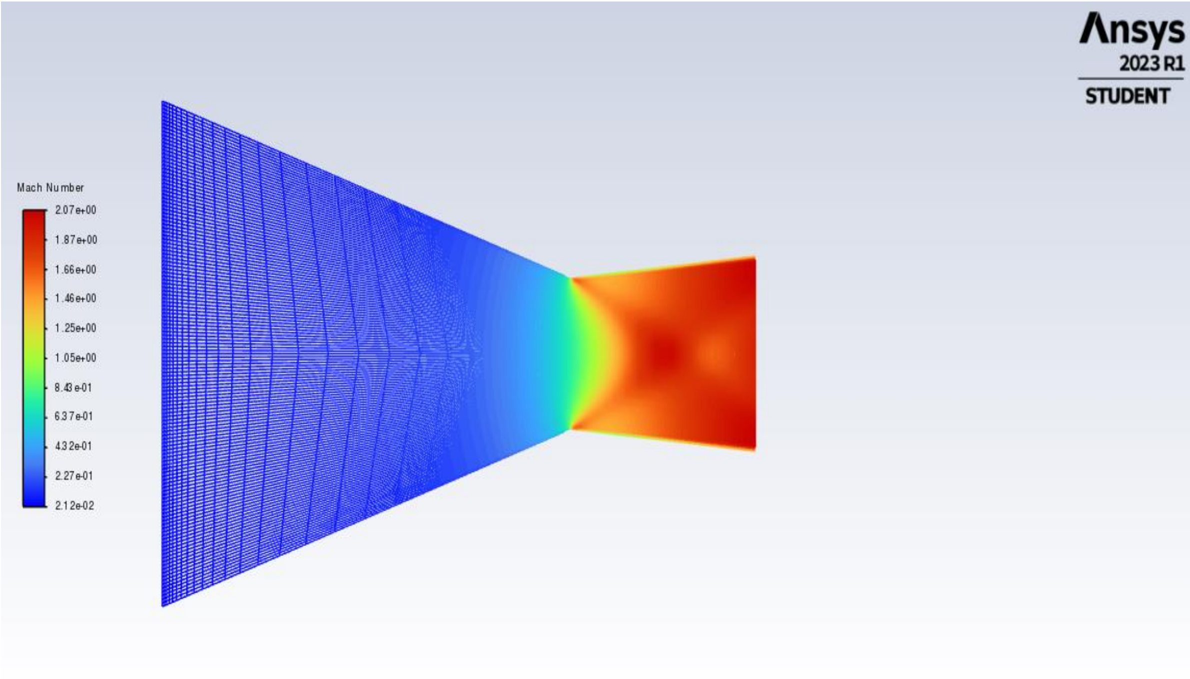
The above figure shows Static Pressure contour for 6˚ Divergent angle.

Mach vs position for shock



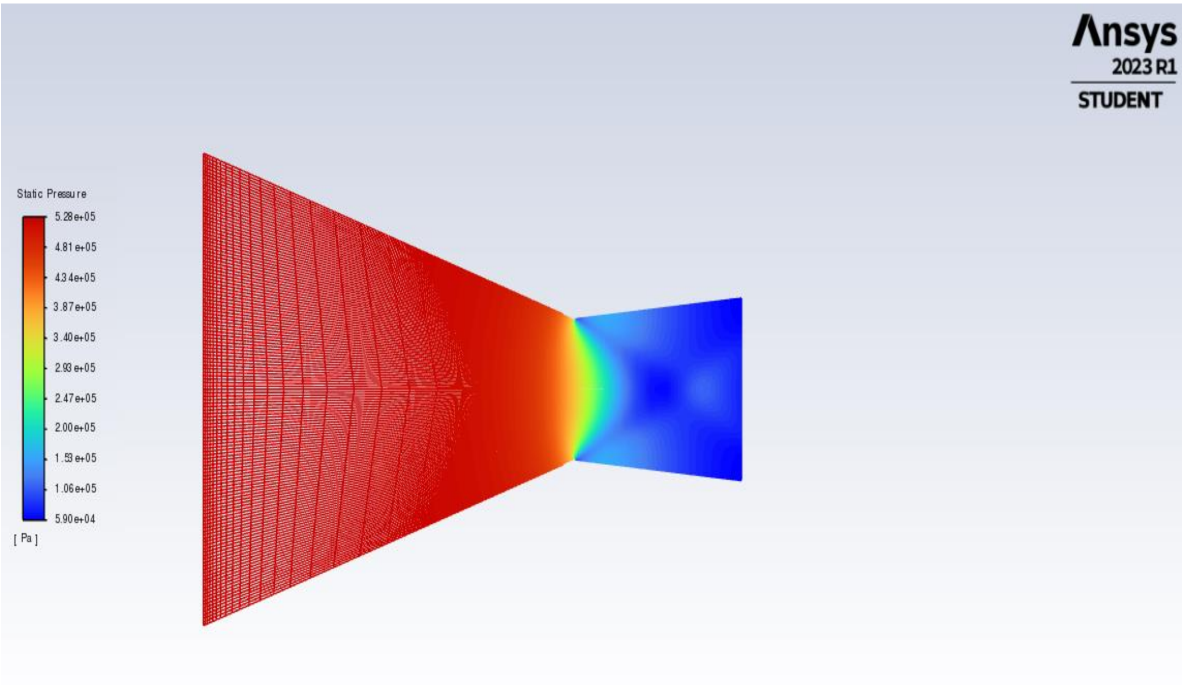
This is the Mach VS Position for shock graph for 7˚ Divergent angle. It clearly shows that a shockwave is generated at the ending of the Divergent side. The position of the shock is to be seen at 0.0334.

Mach Contour



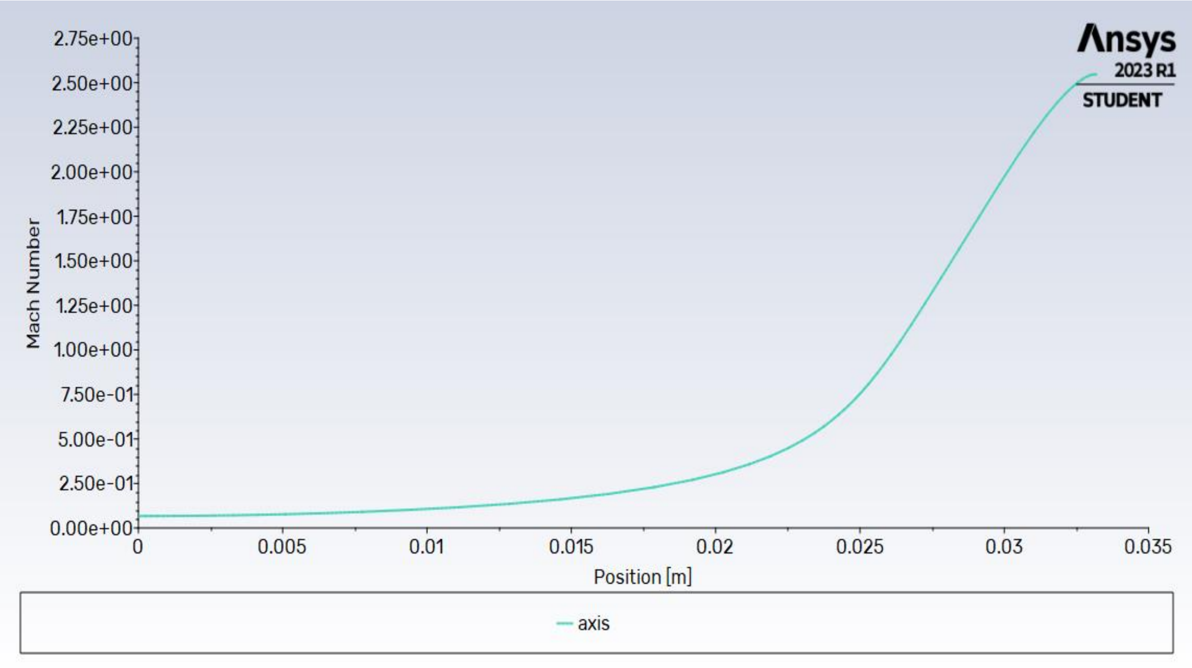
The above figure shows the Mach contour formed for 7˚ Divergent angle.

Static Pressure contour



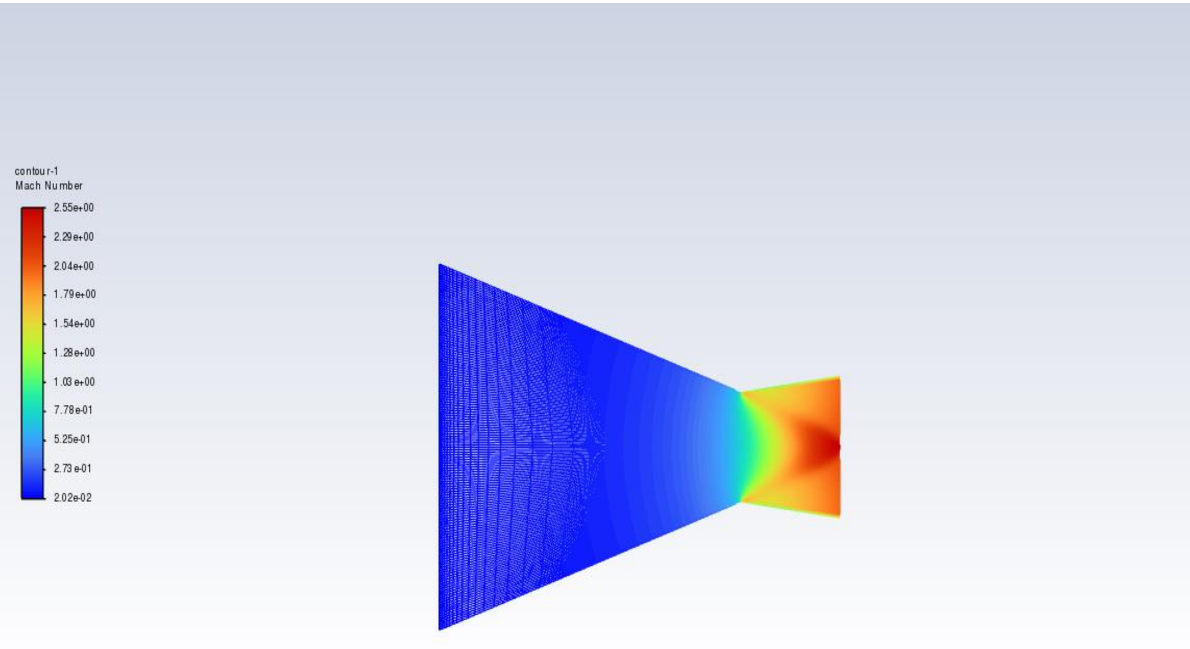
The above figure shows Static Pressure contour for 7˚ Divergent angle.

Mach vs position for shock



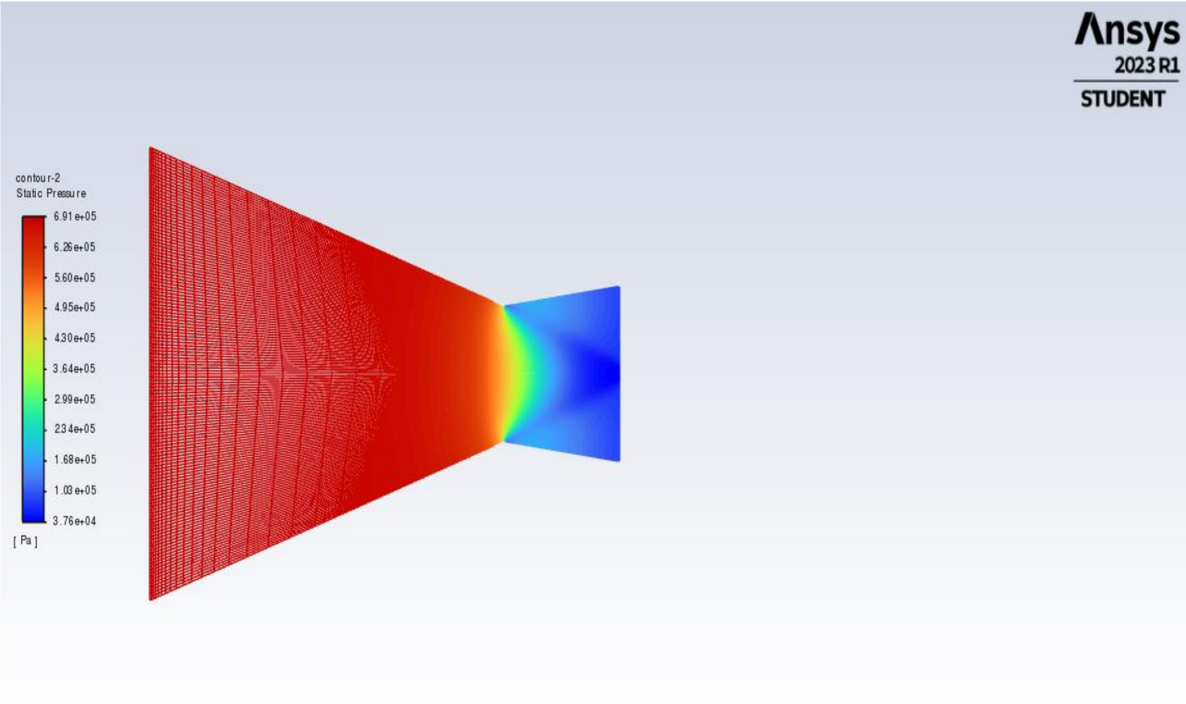
This is the Mach VS Position for shock graph for 8˚ Divergent angle. It clearly shows that a shockwave is generated just after the exit from the nozzle. This is the optimal condition considering which the nozzle will be designed and manufactured as it shows shockwaves just after the exit condition, the Mach number increases and Static Pressure decreases.

Mach contour



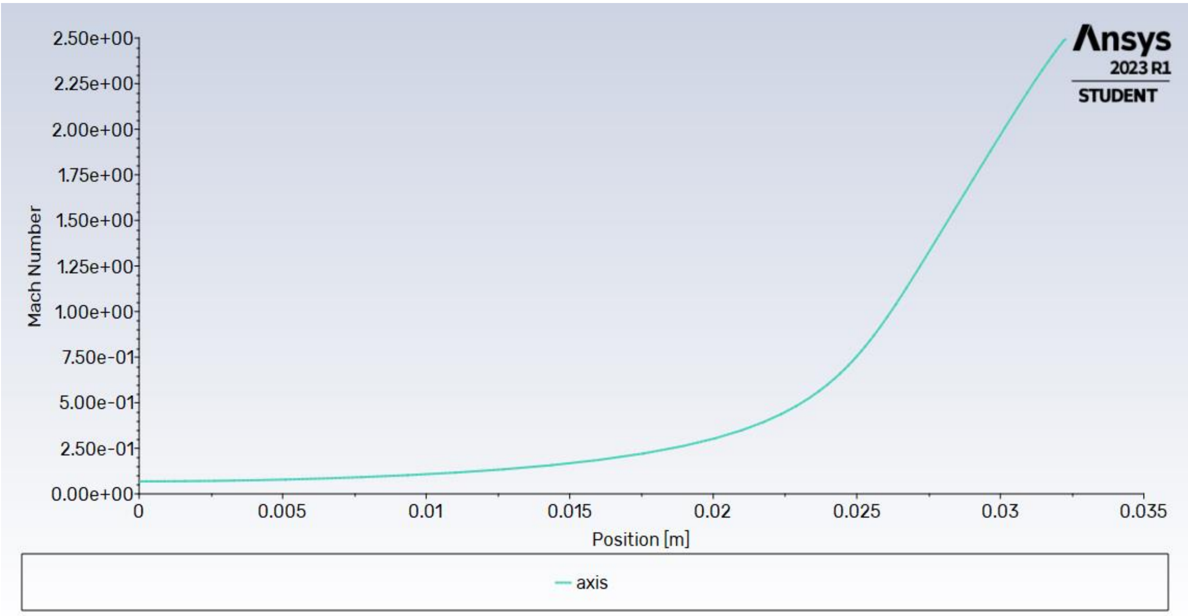
The above figure shows the Mach contour formed for 8˚ Divergent angle.

Static Pressure contour



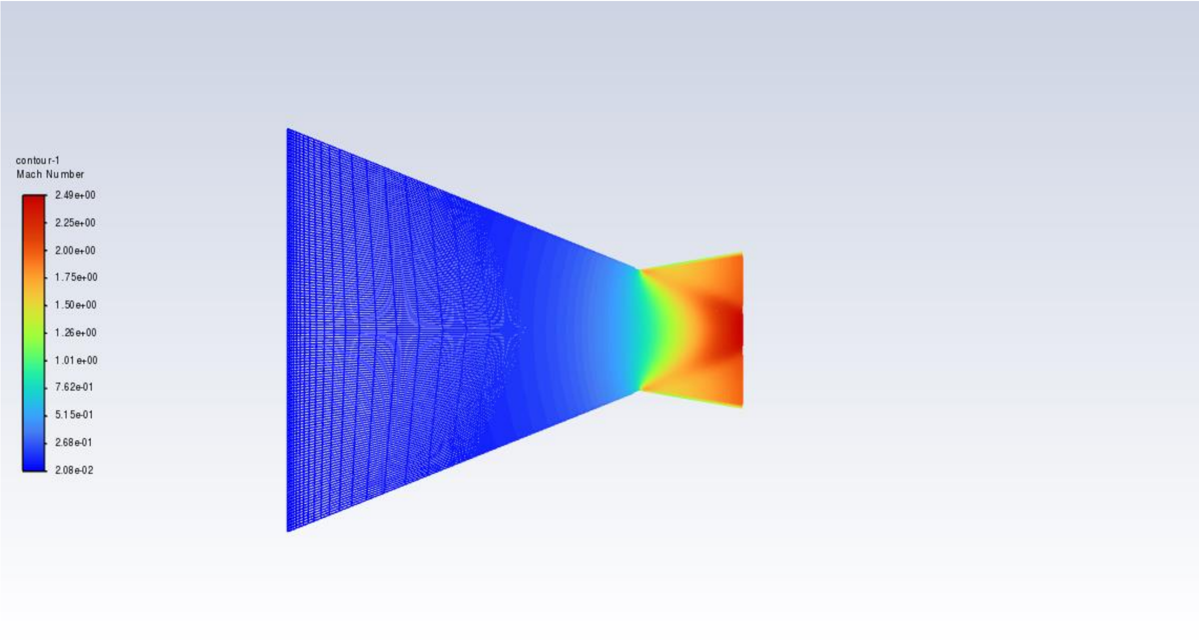
The above figure shows Static Pressure contour for 8˚ Divergent angle.

Mach vs position for shock



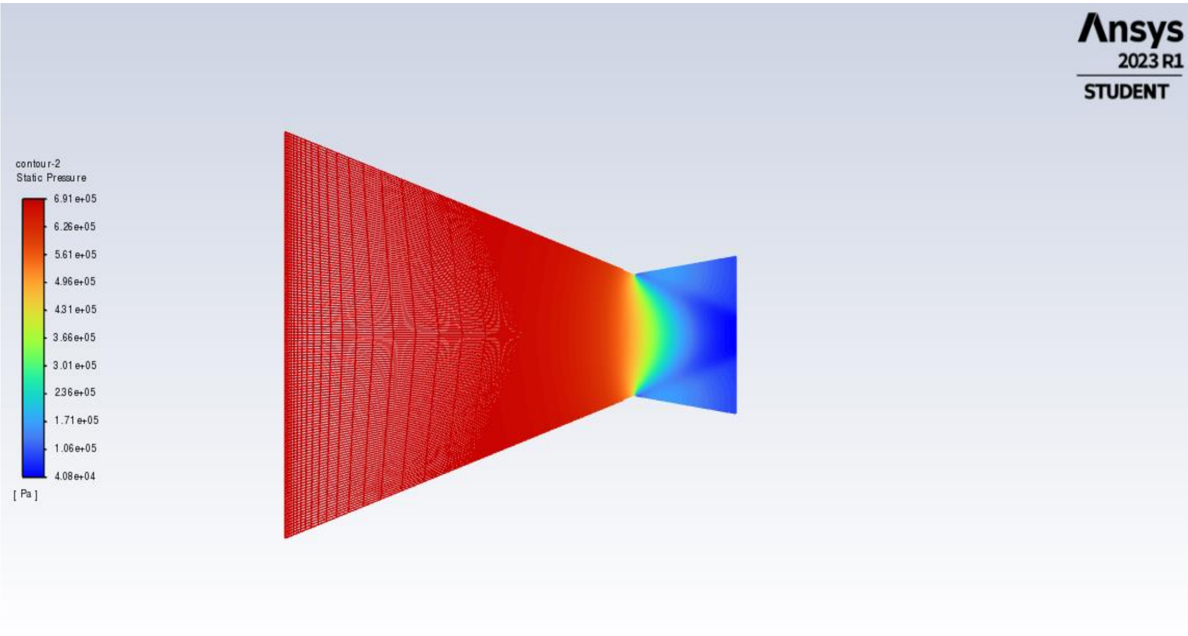
This is the Mach VS Position for shock graph for 9˚ Divergent angle. It clearly shows that no shockwave is generated.

Mach contour



The above figure shows the Mach contour formed for 9˚ Divergent angle.

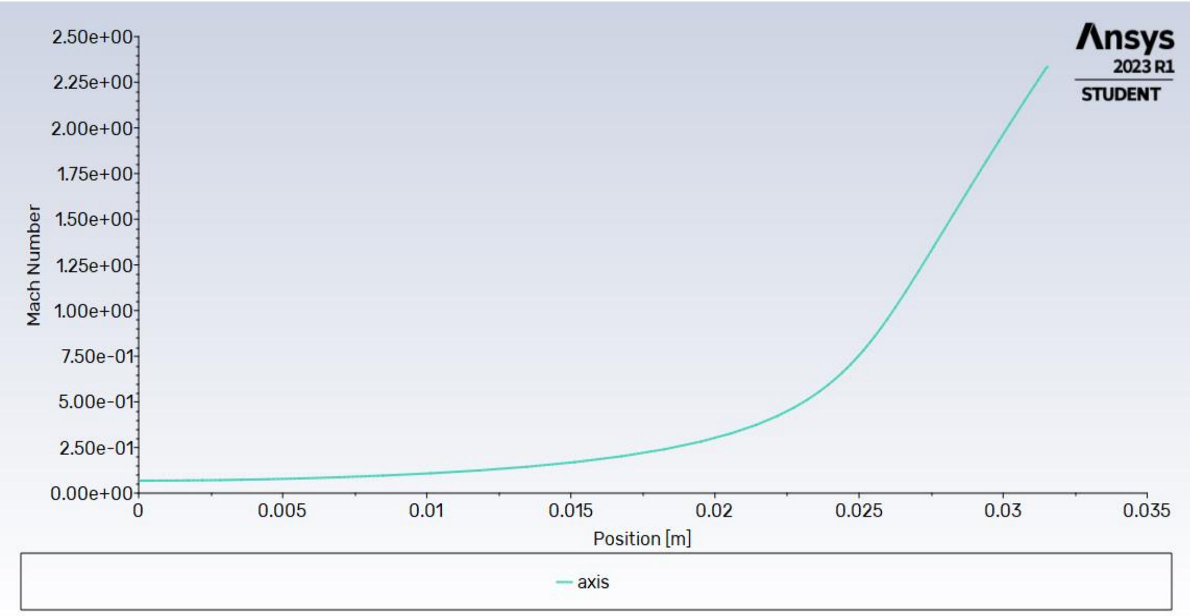
Static Pressure contour



The above figure shows Static Pressure contour for 9˚ Divergent angle.

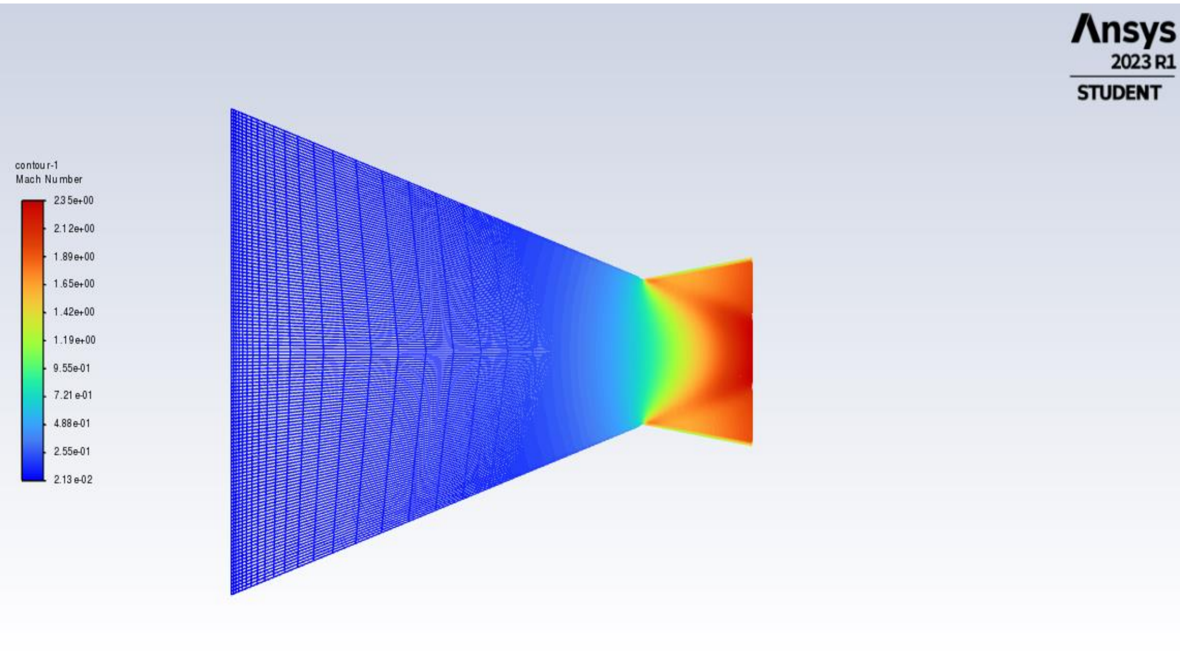
International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 07 Issue: 07 | July - 2023 SJIF Rating: 8.176 ISSN: 2582-3930

Mach vs position for shock



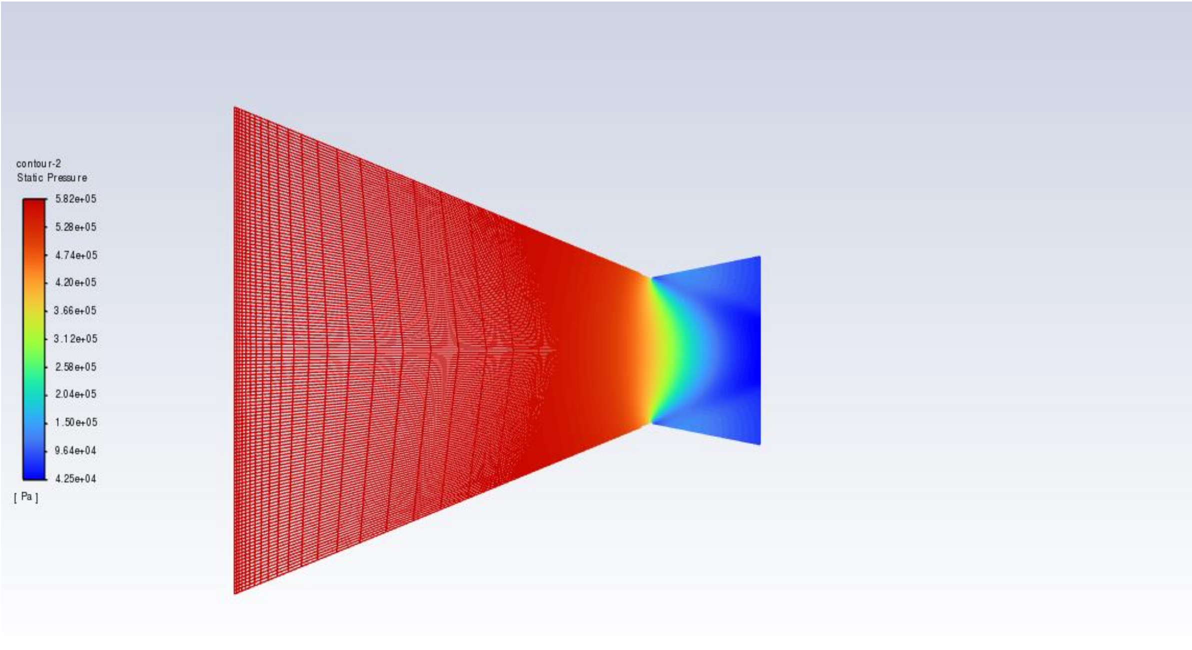
This is the Mach VS Position for shock graph for 10˚ Divergent angle. It clearly shows that no shockwave is generated.

Mach contour



The above figure shows the Mach contour formed for 10˚ Divergent angle.

Static Pressure contour



The above figure shows Static Pressure contour for 10˚ Divergent angle.

Sr. No.	Angle	Shock Position (m)	Exit Mach Number	Exit Static Pressure (Pa)
1	5	0.0328	2.01e+00	8.67e+04
2	6	0.0329	2.07e+00	6.54e+04
3	7	0.0334	2.07e+00	5.90e+04
4	8	Just after exit	2.53e+00	3.76e+04
5	9	No shock	2.49e+00	4.08e+04
6	10	No shock	2.35e+00	4.25e+04
7	11	No shock	2.27e+00	5.80e+04
8	12	No shock	2.21e+00	6.30e+04

The graphs and contours for 11˚ and 12˚ Divergent angles has not been added as they show the same behaviour as the graph and contour plot of 10˚ Divergent angle. There are no shockwaves generated with Divergent angles 11˚ and 12˚ too.

V. Conclusions

1. Since the Mach number reaches it maximum value and the Static Pressure reaches its minimum value at the divergent angle of 8˚; while the shock too, is eliminated, it is selected as the optimum angle for the considered nozzle geometry.
2. The decrease in Mach number and the increase in Static Pressure post the divergent angle of 8˚ is attributed to the nozzle being under-expanded for the given geometry for higher Mach numbers.
3. Post the divergent angle of 8˚, the shocks have found to be eliminated.
4. The CFD simulations have found to be consistent with the theoretical calculations.

VI. Acknowledgement

The authors would like to acknowledge Prof. (Dr.) S. Y. Bhosale, Head of Department, Mechanical Engineering, PES Modern College of Engineering, who has always been a source of inspiration.