STUDY ON EFFECT OF CONVERGING DIVERGING ANGLE ON THE PERFORMANCE CHARACTERISTICS OF A ROCKET NOZZLE

ANANTHESH D SHENOY[1], SATISH HIREMATH[2]

[1]*Graduate Mechanical Engineer, Ernakulam, Kerala, India*
[2]*Assistant Professor, ACS College of Engineering, Bengaluru, Karnataka, India*

Abstract - *Nozzles are flow controlling devices used to either accelerate or decelerate the flow of the gases produced from combustion. Rocket nozzles are convergent-divergent nozzles or otherwise known as the de Laval nozzle. They converts the high temperature, high pressure, and low velocity gas into high velocity and low pressure gas at the exit and hence achieve supersonic speeds. This paper aims to study the variation in flow parameters by modifying the nozzle divergence angle. CFD Analysis is carried out for diverging angles of 5^0, 10^0, 11^0, 12^0 and 15^0. Variation in Mach number and static pressure at the nozzle outlet is studied and the optimum divergence angle for maximum Mach number is found out.*

Key Words: CFD, divergence angle, nozzle

1. INTRODUCTION

A nozzle is a mechanical device which is used to regulate the flow characteristics of fluids as they exit the medium, chamber or pipe through which it is flowing. Nozzles can be converging or diverging. In a converging nozzle, the fluid pressure is increased and the velocity is decreased at the exit and for a divergent nozzle the exit velocity is high and pressure reduced. Nozzles usually control the rate of flow, direction and pressure of the fluid passing through them. Other examples of nozzles are propelling nozzles, spraying nozzles, magnetic nozzles, atomizer nozzles, air-aspirating nozzles, swirl nozzles etc.

The convergent-divergent or CD nozzle is the most common type of nozzle used in rocket engines, supersonic jet engines and in some steam turbines. It is also known as the de-Laval nozzle. Its working depends on the properties of the gas passing through it at subsonic, sonic and supersonic speeds.

The gas passing through a CD nozzle is isentropic and adiabatic in nature. From Figure 1 we can see that first the flow is subsonic. This happens at the converging nozzle where the gas is compressed to higher pressure at the throat the area is the smallest and the Mach number is 1. Here the flows becomes sonic in nature. This phenomenon is called "Choked Flow". Next as the gas passes through the divergent section, the nozzle cross-sectional area increases. The gas undergoes expansion and reaches supersonic velocity.

An important condition for supersonic flow to occur is that the ambient pressure should not be greater than twice or thrice the pressure of the in the supersonic gas at the nozzle exit region.

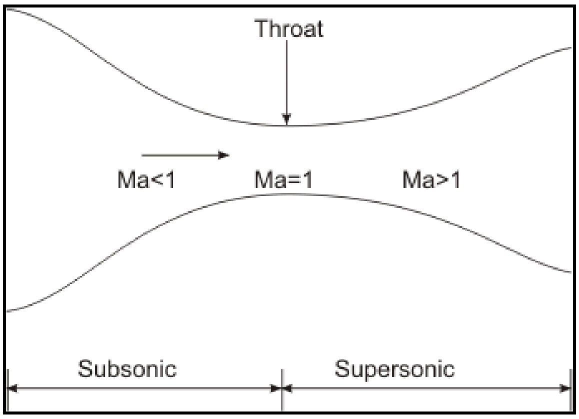

Fig -1: Converging-diverging nozzle

CFD is a computational tool which is used to solve fluid mechanics problems by modelling a physical phenomenon mathematically and solving the corresponding set of governing equations. The mathematical equations are based on the conservation of mass, moment and energy and the most common mathematical formulation is that of the Navier-Stokes equation. Before the invention of CFD, specific codes were required to solve each and every problem or a class of problems. With the invention of CFD solvers there came a generalised set of procedures to solve fluid flow related problems, CFD is not just limited to fluid flow but can be applied in areas of transport phenomena as well. This technique has a lot of advantages over analytical and experimental techniques. The analytical methods are very complex and time consuming to solve and the experimental techniques would incur a lot of expenses. CFD helps us in predicting the result of a particular problem without even performing the experiment. This has helped in reducing the design time period and hence the cost involved. Due to its ability to deliver very precise and cost-effective results, CFD has helped revolutionize the engineering domain. It is now a major tool used in R&D's of most of the industries. CFD is applied to a lot of industries. These include aerospace, automotive, petroleum, power, metrology, defence, chemical etc.

2. BASIC EQUATIONS

The basic equations corresponding to nozzle design are as follows:

$$\dot{m} = \frac{F}{Ve}$$

$$\dot{m} = \frac{Apt}{\sqrt{Tt}} \cdot \sqrt{\left(\frac{\gamma}{R}\right) M} \cdot \left(1 + \frac{\gamma-1}{2} Ma^2\right)^{\frac{(\gamma+1)}{2(\gamma-1)}}$$

$$Ve = \sqrt{\left(\frac{TR}{M}\right)\left(\frac{2\gamma}{\gamma-1}\right) \cdot \left(1 - \left(\frac{Pe}{P}\right)^{\frac{\gamma-1}{\gamma}}\right)}$$

Where

$\dot{m}$: Mass flow rate

A: Cross-sectional area

T: Absolute temperature of the inlet gas

T_t: Total temperature

γ: C_p/C_v

C_p: Specific heat of the gas at constant pressure

C_v: Specific heat of the gas at constant volume

M_g: Molecular weight of the gas

M: Mach number

F: Exerted force

V_e: Velocity of gas at nozzle exit

Ap_t: Cross-sectional area at the throat

R: Universal gas constant

P: Absolute pressure of the gas at inlet

P_e: Absolute pressure of the gas at exit

3. MODELLING AND SIMULATION

The various steps involved in the simulation process are: modeling, meshing, pre-processing and solution.

i) Modeling: The nozzle was modelled using CATIA V5 software and saved in the .igs format so that it could be imported into ANSYS workbench later. The standard dimensions for the nozzle were obtained by referring journal from Research Gate. The dimensions are tabulated in Table 1. For each simulation the divergent angle is modified in CATIA V5 and subsequently the model is exported to ANSYS.

Table -1: Nozzle Dimensions

Parameter	Dimension
Inlet diameter	1000 mm
Throat diameter	304 mm
Outlet diameter	861 mm
Converging length	640 mm
Convergent angle	30^0
Divergence angle	5^0, 10^0, 11^0, 12^0, and 15^0

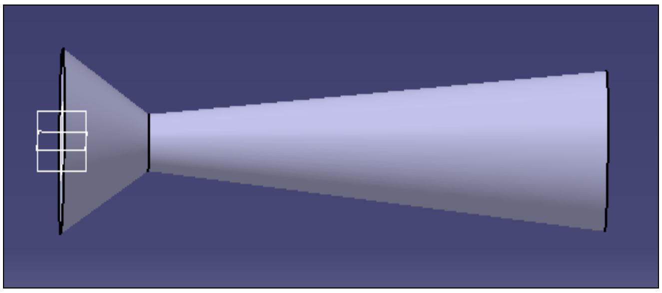

Fig -2: 3D modelled Nozzle

ii) Meshing: After the modelling is done, the nozzle file is imported into ANSYS workbench. First the model is sliced and then the 2D surface is created from it. Meshing is done with an element size of 50mm.

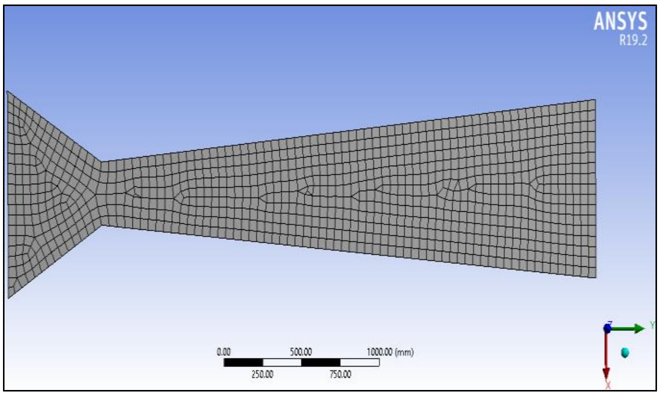

Fig -3: Nozzle after meshing

iii) Pre-Processing: This is done in ANSYS Fluent. The various parameters are tabulated in Table 2.

Table -2: CFD input parameters

Parameter	Dimension
Viscous model	k-epsilon (2 eqn)
Material	air
Inlet mass-flow	826 kg/s
Inlet temperature	3400K
Inlet pressure	45 bar
Space discretization	Coupled
Number of iterations	500

iv) Solution: The simulation is run for 500 iterations and the contours for the static pressure and Mach number are plotted.

CONTOURS FOR DIVERGENT ANGLE OF 5^0

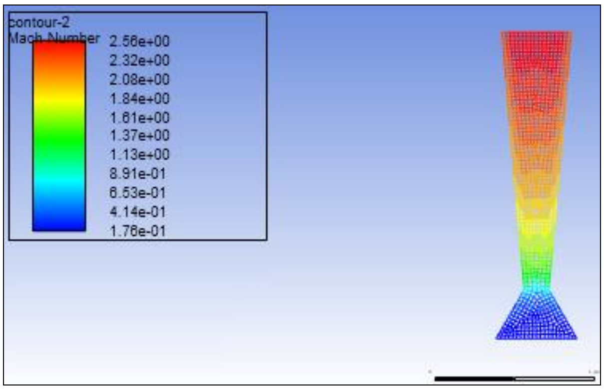

Fig -4: Mach number contour for 5^0 divergent angle

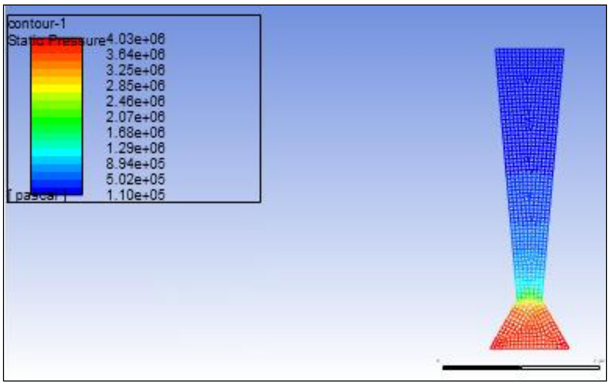

Fig -5: Static pressure contour for 5^0 divergent angle

CONTOURS FOR DIVERGENT ANGLE OF 10^0

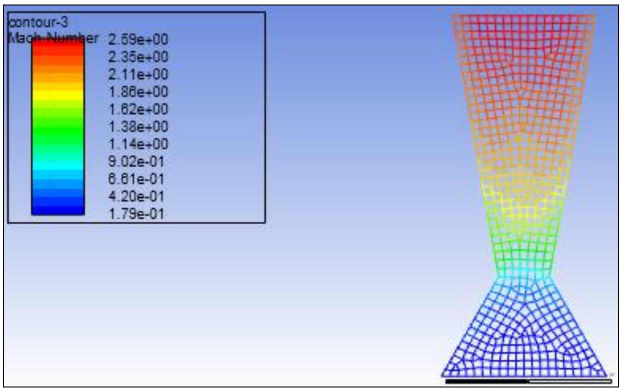

Fig -6: Mach number contour for 10^0 divergent angle

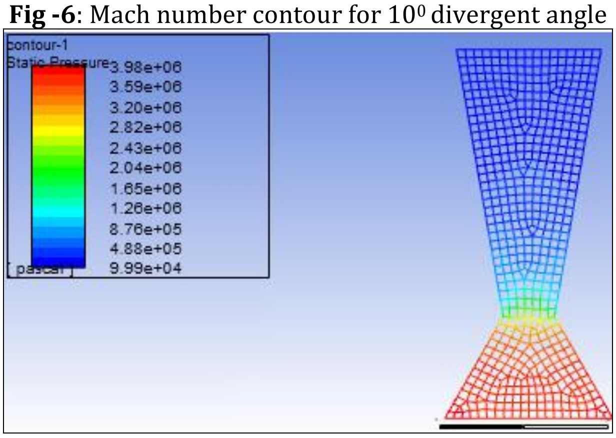

Fig -7: Static pressure contour for 10^0 divergent angle

CONTOURS FOR DIVERGENT ANGLE OF 11^0

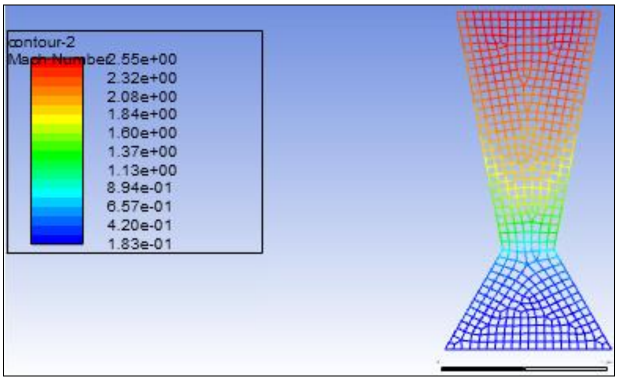

Fig -8: Mach number contour for 11^0 divergent angle

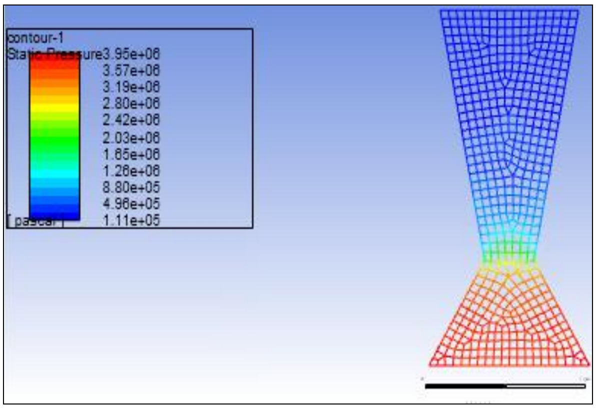

Fig -9: Static pressure contour for 5^0 divergent angle

CONTOURS FOR DIVERGENT ANGLE OF 12^0

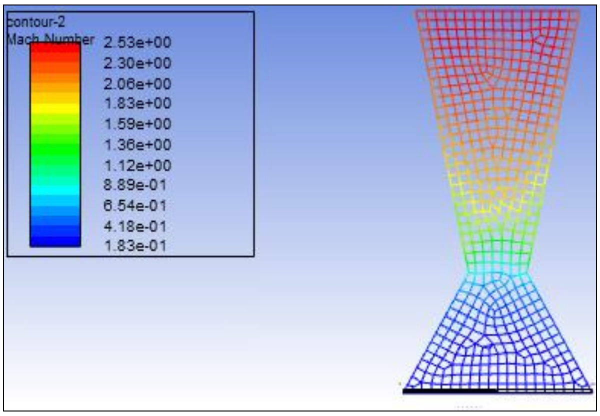

Fig -10: Mach number contour for 12^0 divergent angle

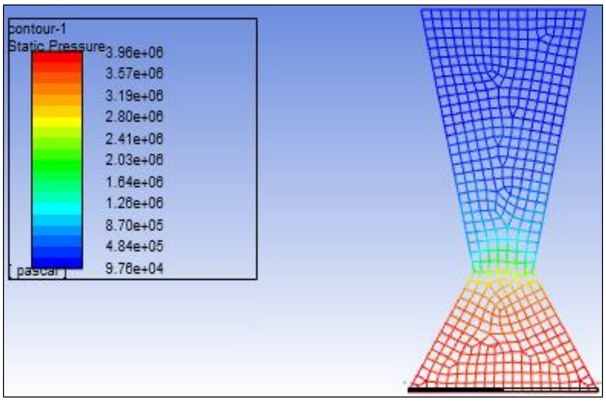

Fig -11: Static pressure contour for 12^0 divergent angle

CONTOURS FOR DIVERGENT ANGLE OF 15^0

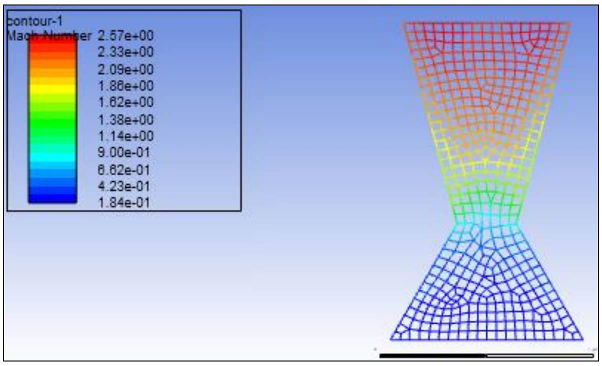

Fig -12: Mach number contour for 15^0 divergent angle

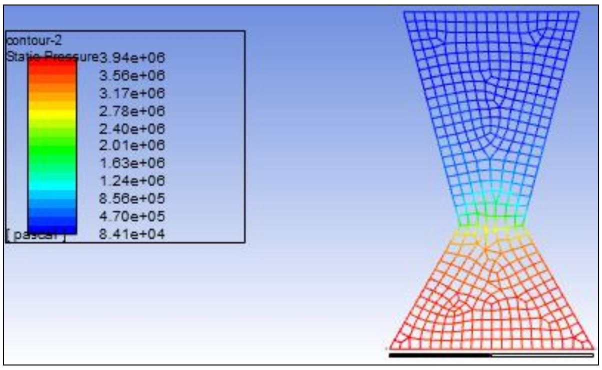

Fig -13: Static pressure contour for 15^0 divergent angle

4. RESULTS

The values of Mach number and static pressure at the exit region obtained from the CFD simulation for various divergent angles of the nozzle are presented in Table 3.

Table -3: Mach number and static pressure values at nozzle exit

Convergence angle (β)	Divergence angle (α)	Mach number	Static pressure (Pa)
30^0	5^0	2.56e+00	1.10e+05
30^0	10^0	2.59e+00	9.99e+04
30^0	11^0	2.55e+00	1.11e+05
30^0	12^0	2.53e+00	9.76e+04
30^0	15^0	2.57e+00	8.41e+05

5. CONCLUSION

CFD study employing the k-epsilon viscous model successfully simulated the behaviour of compressible gas via a converging-diverging nozzle. The change in Mach number and static pressure throughout the nozzle sections is studied for various divergence angle values. When going from the converging to the diverging section, the static pressure is observed to decrease. It reaches its lowest point near the nozzle exit, when the Mach value is the highest. The optimal Mach value is found at a divergence angle of 10^0.

ACKNOWLEDGEMENT

I would like to extend my heartfelt gratitude to Mr. Satish Hiremath, Assistant Professor, ACS College of Engineering, for his invaluable technical advice, encouragement and unwavering support during the study endeavour. It is through his support that I was able to overcome the obstacles experienced during the research phase.

REFERENCES

[1] 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

[2] Vishwajeet Yadav, Prof. Pawan Kumar Tiwari, “A Review on CFD Analysis of Effects of Convergence and Divergence Angles on the Performance of a Nozzle”

[3] B.P. Madhu , G. Mahendramani , K. Bhaskar, “Numerical analysis on flow properties in convergent – Divergent nozzle for different divergence angle” October 2020

[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.

[5] Mohammad Hadi Hamedi-Estakhrsar, Mohamed Jahromi, Hossein Mahdavy-Moghaddam, “Investigation of effects of convergence and divergence half-angles on the performance of a nozzle for different operating conditions, Journal of the Brazilian Society of Mechanical Sciences and Engineering, July 2018, DOI: 10.1007/s40430-018-1271-9