Effect of Convergent Angle on different Flow Parameters of a Convergent-Divergent Nozzle

Nawaf Mehmood Malik[1*], *Mansoor Ali* Zaheer[2] and *Muhammad Ali* Farooq[2]

[1] Department of Mechanical Engineering, School of Mechanical and Manufacturing Engineering (SMME), National University of Sciences and Technology, Islamabad 44000, Pakistan
[2] Department of Mechanical Engineering, College of Engineering and Technology, University of Sargodha, Sargodha 40100, Punjab, Pakistan

Abstract. Nozzle is defined as a device which is being used to convert the pressure energy into the kinetic energy. It basically controls the mass flow rate, pressure and velocity of the fluid stream. Consisting of three sections convergent, throat and divergent section, this device basically converts subsonic flow into supersonic flow. An extensive study have been conducted by the authors in the past about the divergent angle effect on flow parameters of a C-D (Convergent-Divergent) nozzle. In this paper, detailed study have been done to find the most optimum value of convergent angle of a C-D nozzle for the best flow parameters. Convergent angles under consideration are 15º, 20º, 25º, 30º and 35º.

1 Introduction

In engineering terminology C-D (Convergent-Divergent) nozzle is a mechanical device which is being used to convert the fluid thermal energy into kinetic energy. Fluid which possess high pressure, low velocity and high temperature enters the C-D nozzle and it converts it into low pressure, high velocity and low temperature fluid as compared to when it enters the nozzle. So basically we used to control the flow parameters (pressure, velocity) of the fluid through C-D Nozzle. If we talk with reference to the fluid speed so basically a convergent-divergent nozzle converts sub-sonic flow into a supersonic flow. Mostly C-D nozzle is being used in steam turbines, rocket engines and also in jet engines [1]. In a C-D Nozzle there are two sections named as convergent and divergent section. In a convergent section subsonic flow is converted to sonic and further super-sonic flow in a divergent section [2]. If C-D nozzles are classified based on the arrangement or configuration it has 3 types named as Bell, Conical and Contour Nozzles. Selection of Nozzle type totally depends upon the application for which we are going to use the Nozzle [3]. Physical fluid motion in real life is in 3D (Three Dimensions), but we usually convert it into 2D or 1D motion for the sake of simplicity. There are mainly 3 governing equations used in fluid flow known as equation of continuity, momentum equation and the energy equation [4].

Prapti Joshi conducted a study on the design and flow analysis of bell-shaped and conical nozzle. It was observed that the pressure and velocity contours of pressure and velocity

* Corresponding author: nawaf998428@gmail.com

curves of bell shaped nozzle were much better than conical nozzles but shock formation was observed at various divergent angles and other critical conditions [5]. Later on this same year B.P. Madhu conducted a study contour and conical shaped nozzle for different divergent angles (i.e., 9°, 11° and 13°) while keeping the convergent angle and throat diameter constant. Contour nozzle gave higher value for Mach number as compared to conical nozzle at the same geometry but higher intensity shock was being observed at lower divergent angles for contour nozzle as compared to conical nozzle [6]. Shock waves are not found when operating under the nozzle pressure ratio (NPR) of 1.52 [7]. Sher Afghan Khan conducted a study on the area ratio of a C-D Nozzle and found that area ratio have a significant effect on the base pressure and no effect of area ratio was found on any other flow parameter. When area ratio value was increased the base pressure also increased [8].

Most extensive studies are conducted on the divergent angle of the C-D nozzle. Effect on the divergent angle variation have been done and its effect on various flow parameters were studied in detail. Studies conducted have shown that the discharge coefficient increases with the increase in the value of the divergent angle until it reaches an optimum value [9]. Keeping in consideration Mach number, velocity and static pressure the best value of divergent angle found was 15° [10-12]. In this research our focus would be to find the optimum value of convergent angle for minimum value of static pressure and maximum value of velocity and Mach number at exit of the C-D nozzle without the formation of shockwave at any shock wave inside the nozzle.

2 Methodology

We will use the CFD method to analyse the impact of convergent angle on different flow parameters of a convergent-divergent nozzle. ANSYS Fluent Software have been used to design and carry out the CFD simulations in order to find the most optimized value of convergent angle maximum velocity and Mach number values and minimum value of static pressure at the exit of the C-D Nozzle. Different values used for convergent section are shown in the Table 1. A figure of C-D Nozzle with the respective dimensions as in the Table 1 have been attach shown below:

Table 1. Dimensions of C-D Nozzle

Geometric Parameters	Dimensions
Convergent Section Length	100 mm
Divergent Section Length	150 mm
Throat Diameter	30 mm
Outlet Diameter	110.38 mm
Divergent Angle	15°
Convergent Angle	15°, 20°, 25°, 30° and 35°

Table 2. Boundary Conditions of CD Nozzle

Boundary Conditions	**Values**
Inlet Pressure	300×10^{3} Pa
Outlet Pressure	0 Pa
Wall condition	No slip

ANSYS Fluid Flow (Fluent) software is being used to solve the Navier-Stokes equation.

$$\frac{\partial v}{\partial t} + (v \cdot \nabla) v = -\frac{1}{\rho}\nabla p + \nu \nabla^{2} v \qquad (1)$$

Here v is the velocity, t represents time, p represents the pressure field, ρ is fluid density and v represents the kinematic viscosity.

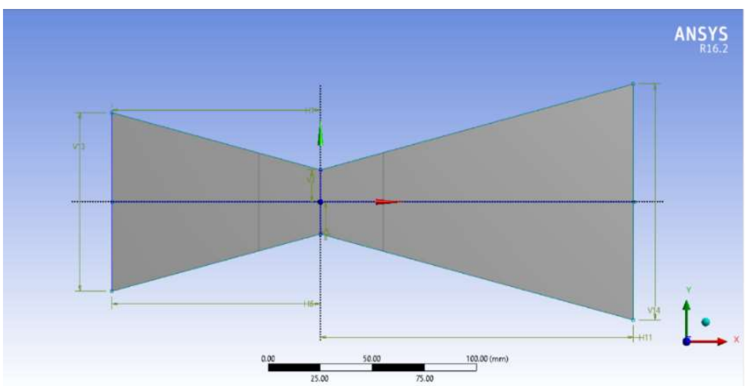

Fig. 1. 2D model of a Convergent-Divergent Nozzle in ANSYS Fluid Fluent.

Structured type of meshing have been done to simulate the model.

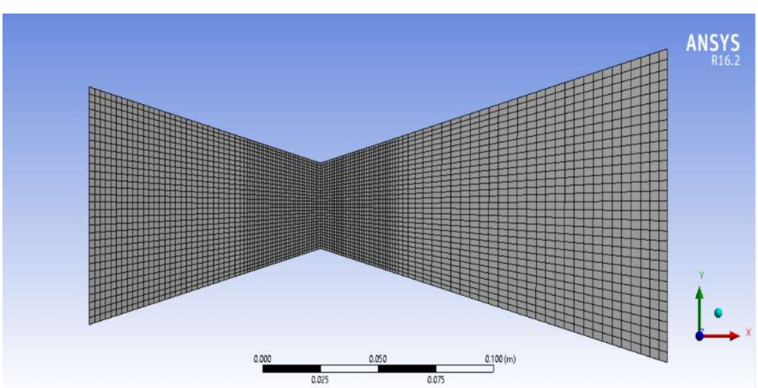

Fig. 2. Structured Mesh of Convergent-Divergent Nozzle

Working fluid used is ideal gas with a specific heat of 1006.43 J/kg K, thermal conductivity 0.0242 W/Mk, and while running calculations number of iterations were kept 500.

3 Results and Discussions

After the preparation and meshing of a 2-D model of the C-D nozzle in ANSYS Fluent, CFD analysis have been carried out while keeping in consideration the above mentioned conditions. Basically a comparative study have carried out in order to find the most optimum convergent angle at which best values of flow parameters can be attained without the formation of shock wave of any type.

Results have been shown in the form of contours of Velocity and Static pressure of C-D nozzle for the convergent angles of 15°, 20°, 25°, 30° and 35° as shown in the figures given below:

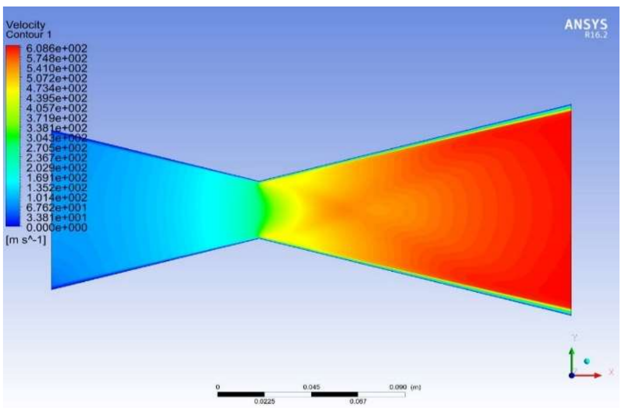

Fig. 3. Velocity Contour of C-D Nozzle (15° convergent angle)

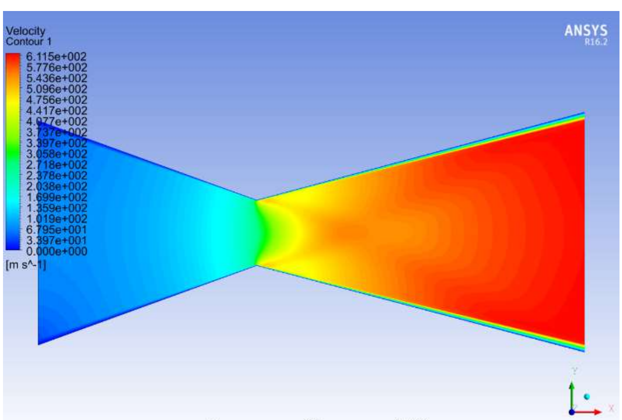

Fig. 4. Velocity Contour of C-D Nozzle (20° convergent angle)

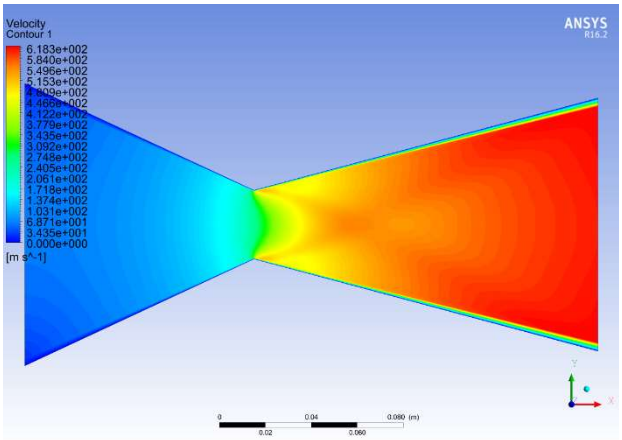

Fig. 5. Velocity Contour of C-D Nozzle (25 ^{o} convergent angle)

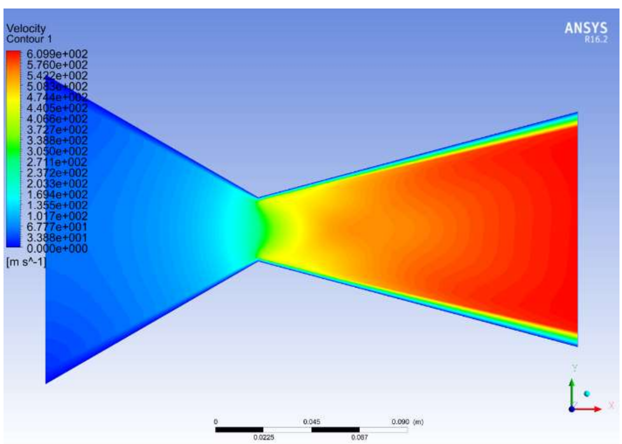

Fig. 6. Velocity Contour of C-D Nozzle (30 ^{o} convergent angle)

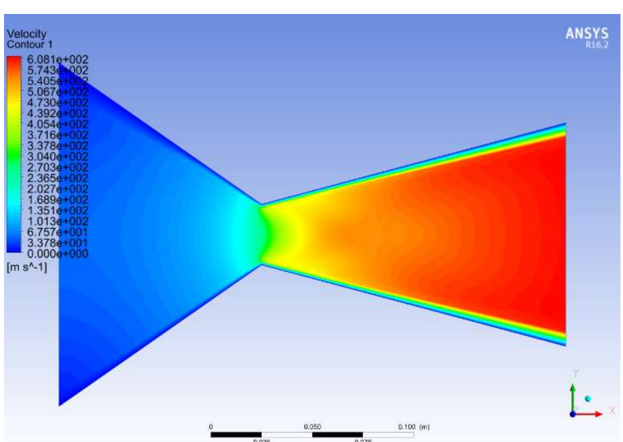

Fig. 7. Velocity Contour of C-D Nozzle (35 ^{o} convergent angle)

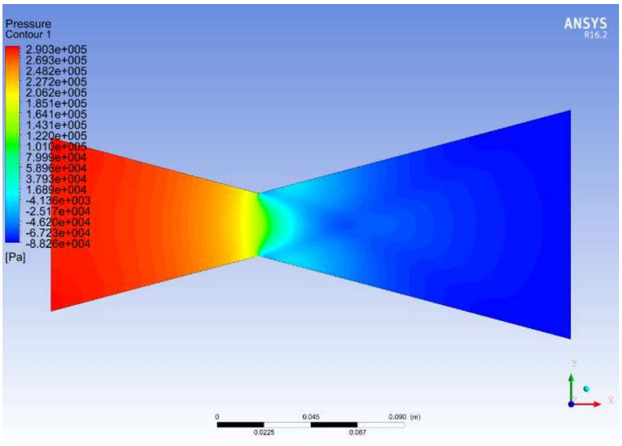

Fig. 8. Pressure contour of C-D Nozzle (15° convergent angle)

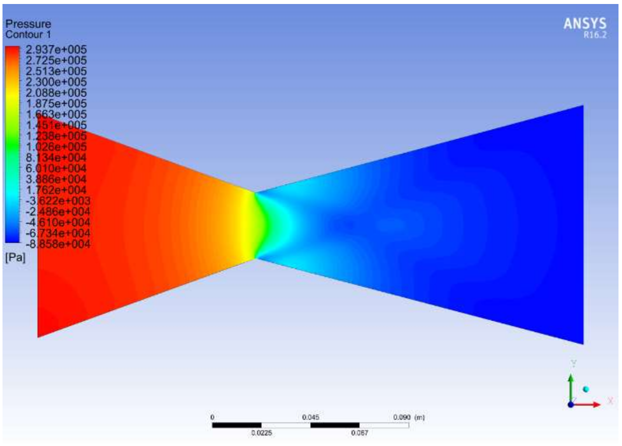

Fig. 9. Pressure Contour of C-D Nozzle (20° convergent angle)

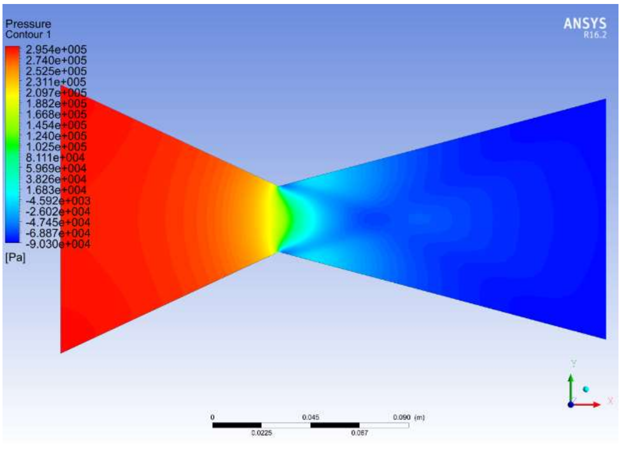

Fig. 10. Pressure Contour of C-D Nozzle (25° convergent angle)

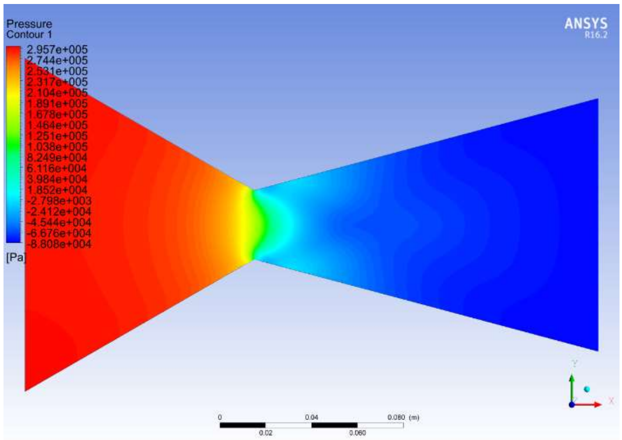

Fig. 11. Pressure Contour of C-D Nozzle (30 ° convergent angle)

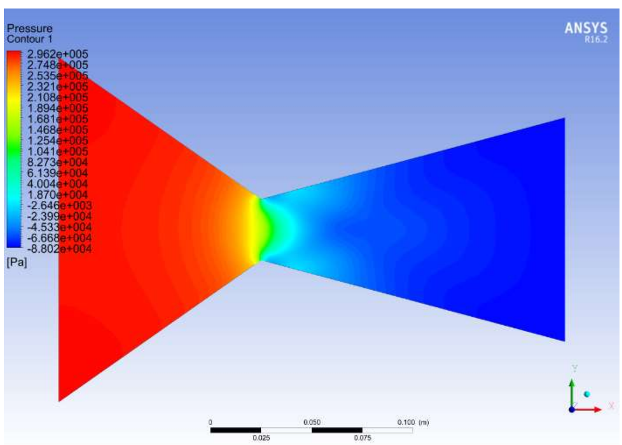

Fig. 12. Pressure Contour of C-D Nozzle (35 ° convergent angle)

For better understanding a table have been made to show exit velocity and pressure at the exit of C-D Nozzle having different indication angles under consideration.

Table 3. Velocity, Mach number and Pressure at exit of C-D Nozzle at different convergent angles.

Convergent Angle	Velocity at exit	Pressure at exit	Mach Number
15°	608.6 m/s	-8.826×10^{4} Pa	2.88
20°	611.5 m/s	-8.858×10^{4} Pa	2.90
25°	618.3 m/s	-9.030×10^{4} Pa	2.97
30°	609.9 m/s	-8.808×10^{4} Pa	2.87
35°	608.1 m/s	-8.802×10^{4} Pa	2.87

4 Conclusion

After conducting the CFD analysis of Convergent-Divergent angles at different convergent angles it have been found that the velocity magnitude at the exit of C-D nozzle kept increasing with the increment of convergent angle up to 25 ^{o} and then start to decrease. In pressure case, it was found to decreasing up to 25 ^{o} and then start to increase. Mach number trend at C-D nozzle exit is analogous to velocity i.e., kept on increasing up to 25 ^{o} and then start to decrease. Hence it can be concluded that best flow parameters can be attained at the convergent of 25 ^{o} without the observation of shock waves of any kind.

References

1. K.P.S.S Narayana, and K.S. Reddy, Simulation of convergent divergent rocket nozzle using CFD analysis. IOSR Journal of Mechanical and Civil Engineering, 2016. **13**(4): p. 58-65.
2. L. Meena, M.S. Niranjan, Aman, Gautam, Gagandeep, G. Kumar, M. Zunaid, Numerical study of convergent-divergent nozzle at different throat diameters and divergence angles. Materials Today: Proceedings, 2021. **46**: p. 10676-10680.
3. O.J. Shariatzadeh, A. Abrishamkar, and A.J. Jafari, Computational Modeling of a Typical Supersonic Converging-Diverging Nozzle and Validation by Real Measured Data. Journal of Clean Energy Technologies, 2015. **3**(3): p. 220-225.
4. S.A. Khan, O.M. Ibrahim, and A. Aabid, CFD analysis of compressible flows in a convergent-divergent nozzle. Materials Today: Proceedings, 2021. **46**: p. 2835-2842.
5. P.Joshi, T. Gandhi, and S. Parveen, Critical designing and flow analysis of various nozzles using CFD analysis. International Journal of Engineering, Research & Technology, 2020. **9**(02): p. 421-424.
6. B.P. Madhu, G. Mahendramani, and K. Bhaskar, Numerical analysis on flow properties in convergent–Divergent nozzle for different divergence angle. Materials Today: Proceedings, 2021. **45**: p. 207-215.
7. P.Padmanathan, and S. Vaidyanathan, Computational Analysis of Shockwave in Convergent Divergent Nozzle. International Journal of Engineering Research and Applications (IJERA), ISSN, 2012: p. 2248-9622.
8. S.A. Khan, A. Aabid, F.A.M Ghasi, A.A. Al-Robaian, A.S. Alsagri, Analysis of area ratio in a CD nozzle with suddenly expanded duct using CFD method. CFD Letters, 2019. **11**(5): p. 61-71.
9. H. Estakhrsar, M. Jahromi, Numerical simulation of turbulent compressible flows in a CD nozzle with different divergence angles. Journal of Heat and Mass Transfer Research, 2014. **1**(2): p. 93-100.
10. K.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, 2014. **11**(6): p. 01-10.
11. P.Roy, A. Mondal, and B. Barai, CFD analysis of rocket engine nozzle. International Journal of Advanced Engineering Research and Science (IJAERS), 2016. **3**(1): p. 39-46.
12. R. Ande, and V.N.K. Yerraboina, Numerical investigation on effect of divergent angle in convergent-divergent rocket engine nozzle. Chemical Engineering Transactions, 2018. **66**: p. 787-792.