International Journal of Science and Research (IJSR)
ISSN: 2319-7064
SJIF (2022): 7.942

Flow Dynamics and Thrust Generation in De Laval Nozzles: A Study Using ANSYS Fluent

Kamal Gautam

Graduate Student, MIME Department, The University of Toledo, Ohio, USA
Email: *Kamalgautam643[at]gmail.com*

Abstract: ***De Laval nozzle is a converging - diverging (CD) nozzle that can convert chemical energy with high pressure into kinetic energy with high velocity and low pressure. Manipulating variables such as area ratio and backpressure govern the flowing nature at the nozzle outlet to have either a supersonic flow or the existence of shock waves. This work reported a comprehensive flow simulation in a typical supersonic converging - diverging nozzle. This study is concerned with the study of the flow nature in the CD when the backpressure ratio was 10 kPa and 15 kPa for the given design of the nozzle. Mach number was supersonic for the pressure of 10 kPa. However, there was normal shock at the outlet with the Mach number reducing to 1 in case of 15 kPa backpressure. The pressure and density have been lowered, and velocity has been drastically increased at the outlet, producing greater thrust in the case of jet engines. The models were designed and analyzed using the ANSYS Fluent program. The results obtained from the ANSYS Fluent program were compared with the theoretical value calculated using isentropic flow equations for a nozzle.***

Keywords: De Laval Nozzle, Mach Number, Shock Waves, Theoretical Validation, ANSYS FLUENT®

1. Introduction

The nozzle is a device designed to control working fluid properties like pressure, density, temperature, and velocity. Based on their cross - sectional area, nozzles are of two major types: convergent and divergent. The convergent nozzle has a reducing cross - section up to the throat, at which the fluid gains maximum velocity by converting pressure and heat into kinetic energy. The highest speed a fluid can accelerate is sonic speed, which occurs at the exit. However, various applications in propulsion systems such as rocket engines and after - burners of jet engines, steam turbines, gas turbines, and supersonic wind tunnels require supersonic speed to achieve higher thrust or better efficiency of the engines. Thus, the diverging part is attached at the exit of a convergent nozzle to increase the velocity of fluid beyond subsonic speeds (Ma >1). This nozzle type is called Converging Diverging (CD) or De - Laval Nozzle. [1]

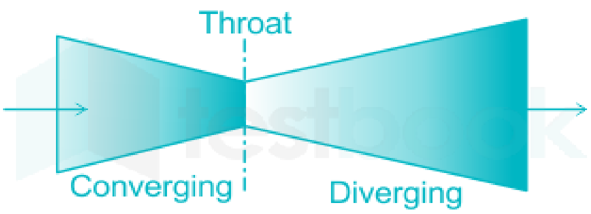

Figure 1: Schematic of a converging-diverging nozzle

A CD nozzle is a variable area passage with a minimum area occurring at a specific location called the throat. Downstream of the throat, the area increases, creating the diverging section of the CD nozzle, as shown in Figure 1. [2] The performance of this nozzle in giving suitable supersonic flow strongly depends upon flow properties and characteristics as it passes through the throat section (interconnection region between convergent and divergent section). [3] Compressible fluid has the characteristic of compressibility, which is the magnitude of specific volume change when the pressure changes. Geometry and flow parameters play a significant role in the supersonic fluid nozzle, like cross - section areas of throat and exit, stagnation pressure at the inlet side, and back pressure at exit one. [4] Ideally, a fluid process in the nozzle is isentropic, where the friction is minimal and usually negligible, and the analysis uses the ideal gas and is entropically expansion equations. [5]

In this paper, CFD analysis of a compressible flow in a CD nozzle is done using ANSYS Fluent to obtain results for various parameters like pressure, temperature, velocity, and Mach number across the CD nozzle for a given design and operating parameters along with the study of flow pattern in the nozzle. The results will be analyzed using a pressure plot, a Mach number plot, and different contour plots for pressure, temperature, density, velocity, and Mach number.

2. Analytical Relations

2.1 Isentropic Flow Through Nozzles

During fluid flow through many devices such as nozzles, diffusers, and turbine blade passages, flow quantities vary primarily in the flow direction only, and the flow can be approximated as one - dimensional isentropic flow with reasonable accuracy. We begin our investigation by seeking relationships among the pressure, temperature, density, velocity, flow area, and Mach number for one - dimensional isentropic flow. We assume the flow is isentropic and the gas has constant specific heats (ideal gas). The equations are based on the reference [6] [7].

The continuity equation is given by,

$$\rho_1 A_1 V_1 = \rho_2 A_2 V_2 \tag{1}$$

Then, the ideal gas equation is given by

$$P = \rho R T \tag{2}$$

For an ideal gas speed of sound is given by,

$$c = \sqrt{kRT} \tag{3}$$

Mach number is defined as the ratio of the speed of flow to the speed of sound,

$$Ma = \frac{v}{c} \tag{4}$$

International Journal of Science and Research (IJSR)
ISSN: 2319-7064
SJIF (2022): 7.942

Now, during a stagnation process, the kinetic energy of the fluid is converted to enthalpy, increasing fluid temperature and pressure. The properties of the fluid at the stagnation state are called stagnation properties. T_0 is stagnation temperature, which represents the temperature an ideal gas attains when brought to rest adiabatically. P_0 is stagnation pressure, which fluid attains when brought to rest isentropically. Similarly, ρ_0 is stagnation density. Their relation with the static temperature, pressure, and density are as follows:

$$T_0 = T + \frac{v^2}{2C_p} \tag{5}$$

$$\frac{P_0}{P} = \left(\frac{T_0}{T}\right)^{\frac{k}{k-1}} \tag{6}$$

$$\frac{\rho_0}{\rho} = \left(\frac{T_0}{T}\right)^{\frac{k}{k-1}} \tag{7}$$

However, in the nozzle case, we express the above relations regarding the specific heat ratio k and the Mach number M. Using continuity and the isentropic relations, we can get the following equations relating the area to the Mach number called the area - Mach number relation.

$$\frac{A}{A^*} = \frac{1}{M}\left[\frac{2}{k+1}\left(1 + \frac{k-1}{2}M^2\right)\right]^{\frac{k+1}{2\,(k-1)}} \tag{8}$$

Where A^* is throat area

It shows that $M = f\left(\frac{A}{A^*}\right)$ i. e., the Mach number at any location in the duct is a function of the ratio of the local duct area to the sonic throat area.

Noting that $C_p = \frac{kR}{k-1}, c^2 = kRT, and\ M = V/c$

The pressure variation is given by,

$$\frac{P_0}{P} = \left(1 + \left(\frac{k-1}{2}\right)M^2\right)^{\frac{k}{k-1}} \tag{9}$$

The temperature variation is given by,

$$\frac{T_0}{T} = \left(1 + \left(\frac{k-1}{2}\right)M^2\right) \tag{10}$$

The density variation is given by,

$$\frac{\rho_0}{\rho} = \left(1 + \left(\frac{k-1}{2}\right)M^2\right)^{\frac{k}{k-1}} \tag{11}$$

The mass flow rate is given by,

$$\dot{m} = PAM\sqrt{\frac{k}{RT}} \tag{12}$$

2.2 Numerical Model

2.2.1 CFD Simulation Principle

Unlike incompressible flow, density is no longer a constant. However, it becomes an additional variable in the equation set of Navier - Stokes equations. However, in this simulation, we limit to ideal gases. When we apply the ideal gas law, we introduce yet another unknown, namely, temperature T. Hence, energy equations need to be solved along with the compressible forms of the equations of conservation of mass and conservation of momentum.

While solving compressible flow problems with CFD, the pressure inlet needs to be specified with both stagnation pressure and static pressure, along with stagnation temperature.

The CFD analysis was done in the following steps:

- Modeling
- Meshing
- Preprocessing
- Solver/Processing
- Post Processing

2.2.2 Modeling

To design and model the nozzle, various guidelines have to be followed. The basic profile design of nozzles is created using linear, parabolic, and circular design equations. At the same time, the method of characteristics is one of the standard methods in designing a divergent section of a supersonic nozzle.

In the present study, axisymmetric two - dimensional modeling was done using the ANSYS DesignModeler. First, lines were drawn, and a 2D surface was created using the surface from edges command. Then, face split is used to create five faces in the geometry, as shown in Figure 2. The CD nozzle dimensions used for this study are as follows:

Table 1: Design parameters

Parameter	Dimensions (mm)
The total length of the Nozzle	200
Inlet diameter	100
Outlet diameter	140
Throat diameter	57.92
Converging angle	40 °
Diverging angle	20 °
Converging length	66.53
Diverging length	133.47

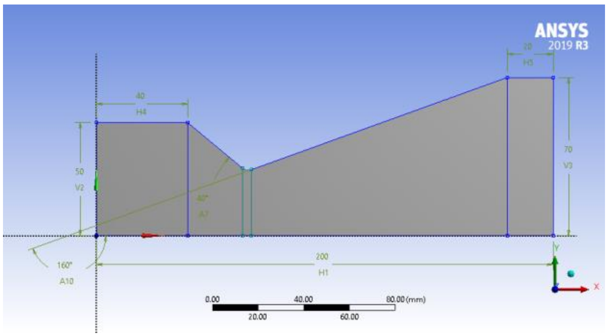

Figure 2: Schematic diagram of 2D axisymmetric CD nozzle used in this analysis

2.2.3 Meshing

The model created using the above dimensions was meshed in the mesh mode of ANSYS Workbench 2019. It was done to create a suitable numerical domain over the geometric feature in which the solver solves the required equations. The software automatically generated unstructured tetrahedral, pyramid, and wedge structured mesh. However, proper refinement and the desired mesh were obtained using the following controls.

Edge Sizing has been used with varying numbers of divisions, from 20 in the throat section, which has a small length, to 200 in the diverging section of the nose. Then, **Face mesh** was used to create the structured mesh with all quadrilateral cells, as shown in Figure 3. Therefore, the number of nodes was 42, 521, and the number of elements was 42000. The physical preference for mesh generation was CFD, and the solver preference was Fluent. At the same time, other elements were set to the default by the software.

International Journal of Science and Research (IJSR)
ISSN: 2319-7064
SJIF (2022): 7.942

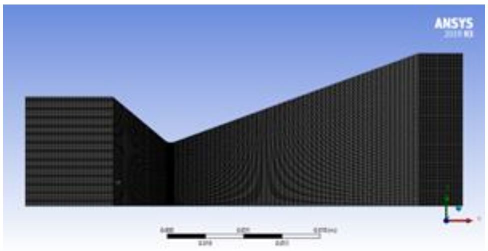

Figure 3: Meshing in the CD nozzle

Mesh quality was determined as skewness, as shown in Figure 4. The maximum skewness was 0.444, and on average, it was 0.1057. According to the definition of skewness, a value of 0.25 – 0.5 indicates good cell quality, and less than 0.25 is excellent quality. Thus, the quadrilateral cell mesh generated in these 2D CD nozzles is fine and should not be a problem for the convergence of the solution.

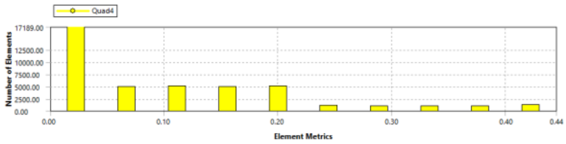

Figure 4: Skewness plot in the CD nozzle mesh

2.2.4 Preprocessing

Preprocessing of the Nozzle was done in ANSYS FLUENT.2 - D and double - precision settings were used while reading the mesh. The computational economy is achieved by exploiting the symmetry of the nozzle, using the axisymmetric formulation for the entire computational domain. The density - based steady solver with absolute velocity is adopted for the numerical simulations instead of the pressure - based solver as fluid is compressible with higher speed. Energy equations are turned on for the simulation of compressible flow to solve the vast effect of temperature variation due to the nature of the nozzle. The material chosen was the air with ideal gas for density variations.

These simulations analyzed two conditions: supersonic flow and typical shock Mach number study. The detailed boundary condition used is as follows:

a) **Pressure Inlet:** The pressure inlet boundary condition is applied to the nozzle inlet boundary (A), as shown in Figure 6. Static pressure of 100000 Pa, stagnation pressure of 101325 Pa, and constant temperature of 300 K are used for all the cases.
b) **Nozzle walls:** The nozzle wall (C) is specified by the wall boundary condition with the adiabatic and no - slip conditions used.
c) **Axis:** To make a computationally economical numerical investigation, the computational domain is axisymmetric. Thus, the axis boundary condition (D) is specified to the axis, as shown in Figure 6.
d) **Pressure Outlet:** The pressure outlet boundary condition is specified for the domain boundary layer B. The outlet pressure is 10 kPa for supersonic flow and 15 kPa for normal shock.

These cases have been analyzed assuming inviscid flow as the effect of viscosity on the flow is not appropriate for high - Reynolds number applications of compressible flow where inertial forces tend to dominate viscous forces.

The detail of the preprocessing is highlighted in the table below:

Table 2: Preprocessing details used in this simulation

General	**Solver - Type:** Density - Based **2D Space:** Axisymmetric **Time:** Steady State
Models	**Energy:** ON **Viscous:** Inviscid
Material	**Fluid:** Air **Density:** Ideal Gas **Cp:** 1006.43 J/kg - K **k** =1.4 **Molecular Weight:** 28.966 kg/mole
Boundary Condition	**Inlet:** Pressure Inlet **Outlet:** Pressure Outlet **Wall:** No Slip **Axis:** Axisymmetric **Values as mentioned above**
Reference Values	**Computer From:** Inlet **Reference Zone:** Solid Surface body

2.2.5 Solver/Processing

The implicit scheme and the convergence criteria of 1e - 06 are chosen for the convergence of the solutions. In solver, the solution is initialized, and calculation proceeds with the desired number of iterations along with a selection of required data quantities that are desired to calculate.

The solver details for various conditions are as follows:

Table 3: Solver details used in this analysis

Solution Control	Courant number = 5
Residuals	Continuity, X - velocity, Y - velocity, Energy, K, and Epsilon= 1e - 06
Solution Initialization	Compute from = Standard Initialization from Inlet
Run Calculations	Number of iterations = 1500 (supersonic) and 3000 (normal shock)

2.2.6 Post Processing

The post - processing is done in ANSYS CFD - POST for contour plots option for various parameters like pressure, velocity, Mach number, and temperature. At the same time, plots for those variables were made in both CFD posts. MS Excel is used for better normalization and comparisons of various parameters.

3. Results and Discussion

3.1 Convergence History

Figure 6 shows the convergence of the solution, which is relatively unstable initially but later slowly converges after a rapid drop in error at around 650 iterations. The termination of the solution was at 1500 with a residual error of around 1e - 05.

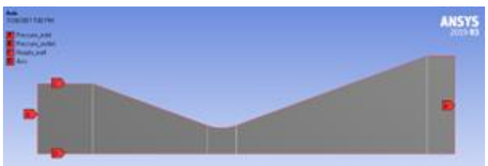

Figure 5: Boundary Conditions

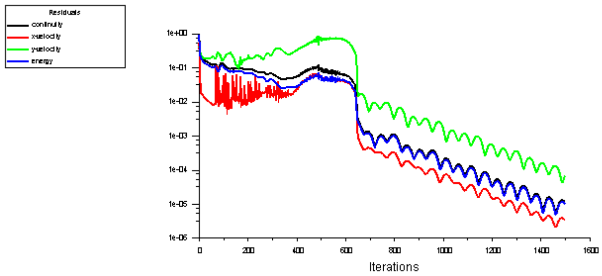

Figure 6: Schematic for the convergence of the solution

3.2 Mach Number Variation

Figure 7 shows the variation of Mach number at the axis of the CD diverging nozzle for the given back pressure and pressure ratio ($P_b/P_{0,inlet}$) of 10 kPa and 0.1 respectively. It shows no normal or oblique shocks in the nozzle and is fully supersonic flow. The Mach number approaches unity when A/A* is unity at the throat. However, the diverging section flow further accelerates to the supersonic region. Thus, without a diverging section, the maximum Mach number we can get is 1 for the converging nozzle only.

Now, when the backpressure ratio is increased from 10 kPa to 15 kPa, i. e., the pressure ratio ($P_b/P_{0,inlet}$) of 0.15, normal shock occurs at the exit of the nozzle, as illustrated in Figure 8. As the normal shock occurs, the supersonic Mach number instantly reduces to subsonic values, which will affect other parameter values to reduce aftershock.

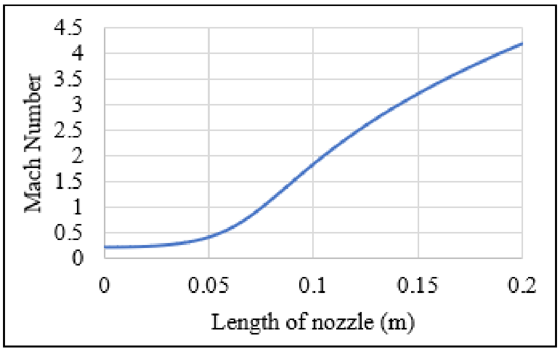

Figure 7: Mach number variation across the length of CD nozzle for supersonic flow

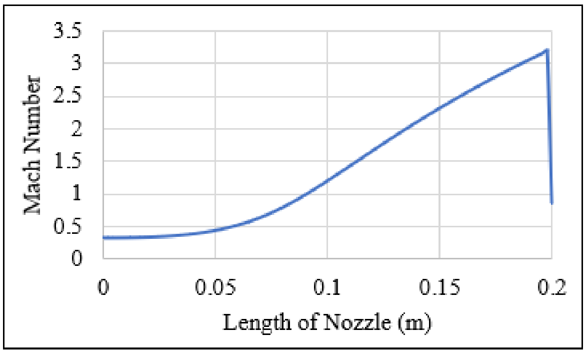

Figure 8: Mach number variation in case of normal shock

Now, parameters like pressure, velocity, temperature, and density are only analyzed further for the supersonic flow in the CD nozzle. Due to the device's computation limitation, other parameters like pressure, temperature, velocity, and density could not be visualized.

3.3 Pressure Variation

Pressure is maximum at the inlet section, with a slight reduction in pressure going towards the throat section. As fluid leaves the throat to a divergent section, there is a drastic reduction in the pressure, and it ultimately attains the minimum value at the outlet of the nozzle, as shown in Figure 10. This is due to the acceleration of the fluid either sub - sonically or supersonically or the expansion of fluid in diverging sections of the CD nozzle. The maximum value of pressure is 98536.91 Pa, and the minimum value is 507.14 Pa in this case for the given back pressure and pressure ratio ($P_b/P_{0,inlet}$) of 10 kPa and 0.1 respectively.

3.4 Velocity Variation

The velocity is minimum at the inlet and increases until the nozzle exits, as shown in Figure 10.

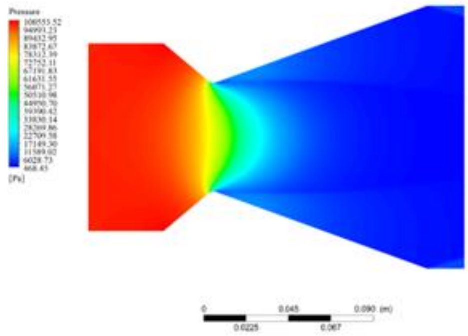

Figure 9: Pressure contour for an outlet pressure of 10 kPa

The velocity magnitude is Mach 1 at the throat section of the Nozzle having A/A* of 1. This is called the choked flow condition when the pressure ratio ($P_b/P_{0,inlet}$) of 10 kPa is given as a boundary condition. Due to the non - uniform convergence and divergence section, the velocity gradient was not uniform at the outlet compared to the inlet. We can see from the figure that velocity is maximum towards the center and relatively low from the axis. Steep edges and higher velocity away from the throat also cause the flow separation at the divergence section, creating a beam flow for the radius equal to the throat. There is a higher velocity change after the throat before attaining a value close to the maximum, and it slowly rises after that value. The velocity rise is also due to the reduction of the area, which incorporates higher velocity for conserving mass flow rate. Maximum velocity is 684.88 m/s at the outlet, and a minimum of 20.90 m/s at the edge of the convergence section can be seen. However, the minimum velocity at the centerline of the inlet is 69.18 m/s.

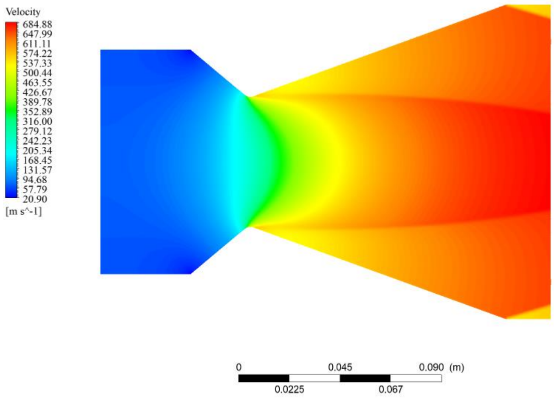

Figure 10: Velocity contour for the outlet pressure of 10kPa

3.5 Temperature Variation

Temperature variation is similar to pressure variation, with a maximum at the inlet and a minimum at the outlet. The same is true with velocity contours; separation layers of temperature can also be seen away from the throat radius downstream of flow. The maximum temperature value is 297.62 K at the inlet, and the minimum value is 66.22 K around the axis region of the outlet. The decrease in the temperature of working fluid away from the throat section is due to the isentropic expansion of fluid at the diverging section, which is the same as the pressure drop illustrated in Figure 11.

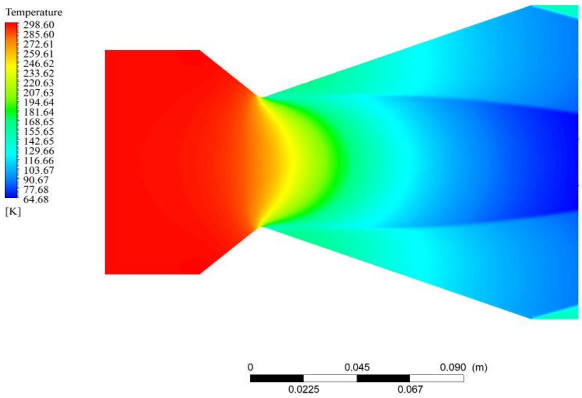

Figure 11: Temperature contours across the nozzle

3.6 Density Variation

The flow in the CD nozzle is compressible, so the density changes from the inlet to the outlet. The flow density is maximum at the inlet and minimum at the outlet. The maximum value of density is 1.15 kg/m3, and the minimum value is 0.02 kg/m3. Its variation is also the same as the pressure variation, with a sharp decrease in the density just after the throat section, as shown in Figure 13.

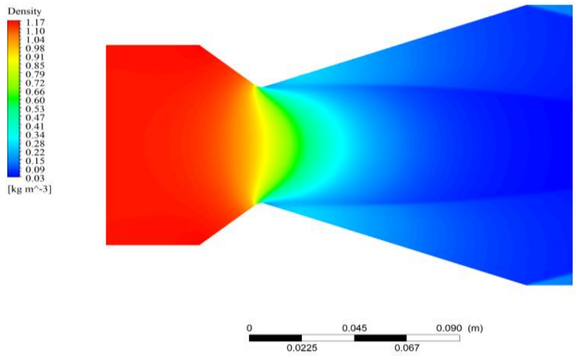

Figure 12: Density contours plot

3.7 Validation with theoretical calculations

The exact value for a few important locations of various parameters for the given model has been computed below with the help of Excel using the isentropic and nozzle equations presented above:

Table 4: Theoretical calculation of pressure, temperature, and velocity

X (m)	A/A*	M	P Pa	T K	V m/s
0.000	2.98	0.20	96973.28	297.37	72.59
0.052	1.90	0.43	87921.60	289.16	147.52
0.064	1.06	0.66	74649.82	275.95	219.77
0.067	1.00	0.72	70589.08	271.58	238.93
0.074	1.17	0.95	56160.44	254.40	302.64
0.082	1.40	1.19	41636.97	233.56	365.34
0.112	2.44	2.41	6700.437	138.58	569.45
0.142	3.76	3.04	2555.711	105.22	625.53
0.172	5.36	3.70	994.6921	80.35	664.27
0.200	5.84	4.21	497.8042	65.93	685.73

Now, a comparison of theoretical and CFD values of Mach number, pressure, temperature, and velocity along the axis are illustrated in Figures 13, 14, 15, and 16 below:

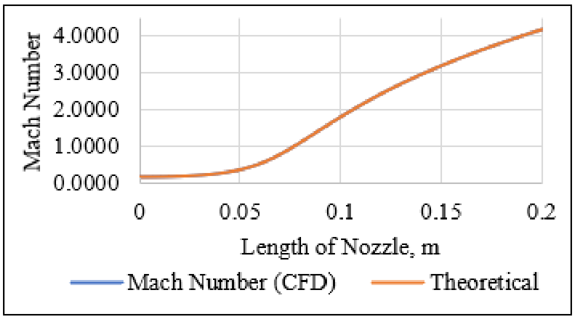

Figure 13: Mach number - theoretical vs CFD along the axis

Paper ID: SR24707080829 DOI: https://dx.doi.org/10.21275/SR24707080829

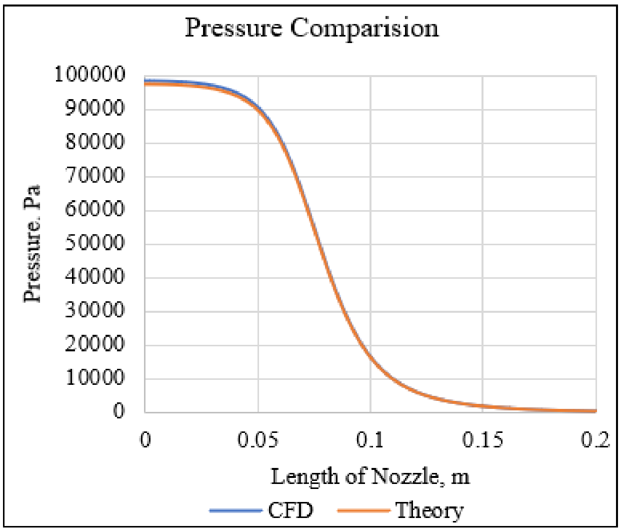

Figure 14: Comparison of CFD pressure with theoretical along the axis

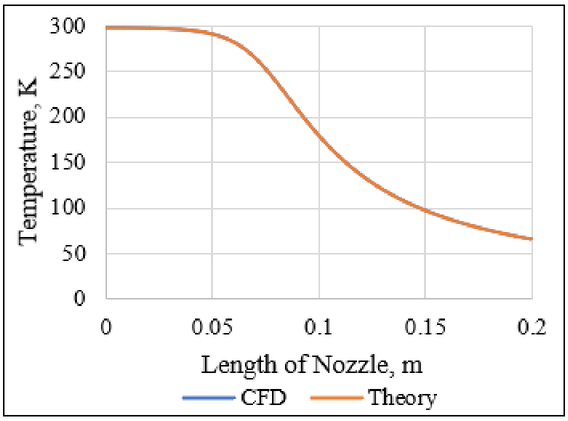

Figure 15: Temperature comparison between theoretical and CFD along the axis

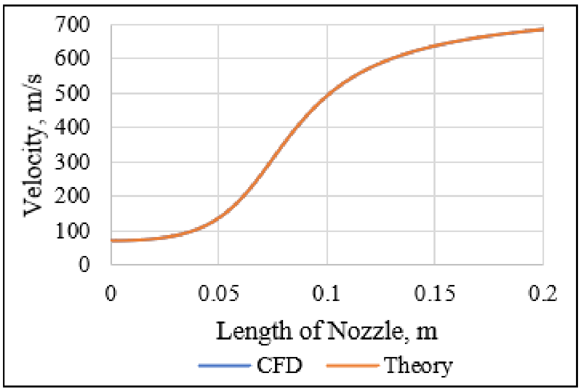

Figure 16: Velocity Comparison along the axis

From the equation of nozzles 2 - 12, it is clear that pressure, temperature, and velocities directly depend on the Mach number of the flowing fluid. The values calculated from CFD and theoretical equation are almost the same as seen from the above figures, with little difference in Mach number, which resulted in deviations from other parameters. Otherwise, the nature of the graphs is similar to that discussed in the contour plot. Thus, CFD results are accurate for given conditions compared to theoretical calculations using isentropic nozzle equations with an error of 1.36 %.

4. Conclusions

This study discussed the basic concepts connected with the converging - diverging nozzle. It provided an analysis of the supersonic flow condition. In the geometry mentioned above, the maximum value of the Mach number is in a place where pressure is minimum. At the throat region, pressure values are near the back pressure. Besides this, various conclusions drawn from this analysis are as follows:

- When the backpressure was raised above 15, 000 Pa, Mach number curves were not smooth due to shockwave formation after the throat but before the outlet boundary, as expected from theoretical understandings.
- There was no significant difference in the value of nozzle parameters by the theoretical method, which uses formulae and CFD for the inviscid flow.
- At the outlet pressure, the temperature and density of fluid decrease while the flow velocity significantly increases, which produces greater thrust in jet propulsion engines.
- CFD calculations would have produced different outcomes with schemes and initial values such as grid, application, turbulence model, method, parameter magnitude, etc.
- Nozzle geometry and outlet pressure will have a significant impact on nozzle performance. A lower diverging angle and no elongated throat would have given the best critical pressure ratio and a low - pressure drop. Thus, a precise study of the nozzle geometry and operating conditions is required to achieve maximum efficiency with the CD nozzle.

References

[1] T. Stevens and H. Hobart, Steam turbine engineering, by T. Stevens and H. M. Hobart. New York: Macmillan, 1906.

[2] K. Pandey and S. Yadav, "CFD Analysis of a Rocket Nozzle with Four Inlets at Mach 2.1", International Journal of Chemical Engineering and Applications, pp.319 - 325, 2010. Available: 10.7763/ijcea.2010. v1.55.

[3] S. Sher Afghan Khan et al., "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, vol.8, no.3, pp.1146 - 1158, 2018.

[4] J. Singh, L. Zerpa, B. Partington and J. Gamboa, "Effect of nozzle geometry on critical - subcritical flow transitions, " Heliyon, vol.5, no.2, 2019. Available: 10.1016/j. heliyon.2019. e01273

[5] F. Fharukh Ahmed. G. M et al., "Numerical Analysis of Convergent - Divergent Nozzle Using Finite Element Method, " International Journal of Mechanical and Production Engineering Research and Development, vol.8, no.6, pp.373 - 382, 2018. Available: 10.24247/ijmperddec201842.

[6] B. Das, R. Sardar, S. Sarkar, and N. Manna, "Compressible Flow Through Convergent–Divergent Nozzle, " Lecture Notes in Mechanical Engineering, pp.345 - 353, 2021. Available: 10.1007/978 - 981 - 33 - 4165 - 4_32

[7] Y. Cengel and J. Cimbala, Fluid mechanics. New York, NY: McGraw - Hill Education, 2018, pp.663 - 681, 922 - 927.

Author Profile

Er. Kamal Gautam is a Mechanical Engineer and a research scholar pursuing his MSc at the University of Toledo. He has a Bachelor of Engineering in Mechanical Engineering from Kathmandu University, where this study was conducted. His research interests include biomechanics, renewable energy, and computational thermal fluid science. He can be reached at kamalgautam643[at]gmail.com.