Proceedings of 12th IOE Graduate Conference
Peer Reviewed
ISSN: 2350-8914 (Online), 2350-8906 (Print)
Year: 2022 Month: October Volume: 12

Numerical and Experimental Investigation of Exhaust Flow from Convergent-Divergent Nozzle

Santosh Ranabhat [a], Kamal Darlami [b], Sudip Bhattarai [c], Nischal Shrestha [d]

[a] *Department of Automobile and Mechanical Engineering, Thapathali Campus, IOE, Tribhuvan University, Nepal*
[b, c, d] *Department of Mechanical and Aerospace Engineering, Pulchowk Campus, IOE, Tribhuvan University, Nepal*
[a] santoshranabhat125@gmail.com , [b] darlami.kd@pcampus.edu.np,
[c] sudip@ioe.edu.np, [d] 076bas024.nischal@pcampus.edu.np

Abstract
Convergent-Divergent (CD) nozzle is a simple cross section varying tube and used extensively in almost all high speed propulsion systems. Exhaust flow from the nozzle have significant characteristics for the prediction of nozzle performance and hence proper understanding and investigation technique of nozzle flow and exhaust plume is worthwhile. The flow characteristic of fluid through CD nozzle vary significantly according to the pressure difference between inlet and exhaust ambient pressure. Critical pressure ratio producing sonic flow at the throat and higher pressure difference is responsible for the formation of normal shock wave within the diverging section and it travels from throat to outlet region as the pressure gradient increases due to the effect of back pressure. Further increase in pressure difference causes under-expanded, over-expanded or perfectly expanded flow according to the ambient pressure. Theoretically, there is no any formation of oblique shock for perfectly expanded condition and parallel and streamline exhaust exit flow is most efficient for optimum thrust development. A specific planar nozzle geometry with exit Mach 2 is designed and relevant thermodynamic parameters are calculated at various sections. Computational Fluid Dynamics is used with well posed boundary conditions, numerical simulations are run for different operating nozzle pressure ratios. A setup of flow visualization with Optical schlieren Imaging System is established and monochrome schlieren image of exhaust flow from the designed nozzle is captured for the validation of simulation results.

Keywords
Compressible Flow, Isentropic Relation, Convergent-Divergent Nozzle, Normal and Oblique Shock Waves, Schlieren Imaging

1. Introduction

Convergent-Divergent (CD) nozzle sometimes also called as *de laval nozzle*, is geometrically a simple cross section varying tube with minimum cross section, called throat, between inlet and outlet. Propulsion system is one of the most used application of CD nozzle where propulsion engine expands and accelerates combusted gas up to supersonic state. Most of the gas turbine and rocket engines use nozzle to improve thrust by accelerating hot exhaust gases. This action can be described by Newton's 3rd law of motion and the amount of thrust developed depends on various flow variables. The flow variables are pressure and velocity at the exit and mass flow rate from the nozzle [1].

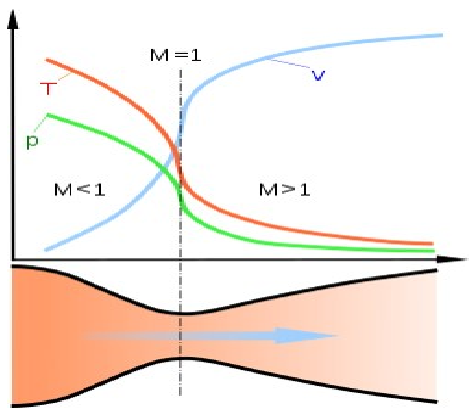

Figure 1: Magnitude of pressure, temperature and velocity across different locations of CD nozzle [2]

By accelerating the compressible fluid to supersonic speeds on its diverging section, this type of nozzle provides the axial thrust by converting the thermal energy of the fluid flowing. Figure 1 illustrates the area velocity relationship, which explains the flow velocity on the diverging portion increases beyond the local speed of sound [2].

The area velocity relationship is :

$$\frac{dA}{dV} = \frac{A}{V}\left(M^2 - 1\right) \tag{1}$$

Mach number in a duct is established by the ratio of the local duct area to the sonic throat area [3]. The area ratio of nozzle throat to exit (expansion ratio) with Mach number relationship is important for determination of exit area of nozzle. The relationship is as shown below.

$$\frac{A}{A_t} = \frac{1}{M}.\left(\frac{1+\frac{\gamma-1}{2}.M^2}{\frac{\gamma+1}{2}}\right)^{\frac{\gamma+1}{2(\gamma-1)}} \tag{2}$$

Equation 2 illustrates that for a specific area ratio, there exists two different values of the Mach number: a subsonic value and a supersonic value.

Figure 2 gives an example of a graphical method for determining the area ratio for a nozzle to be designed for Mach 2.

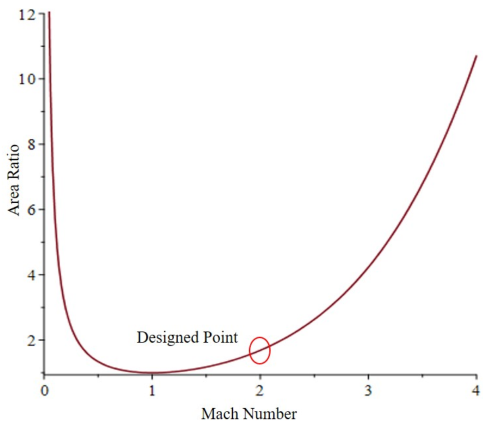

Figure 2: Area Ratio to Mach Number Relationship for $\gamma = 1.4$

Isentropic Relationship If the local Mach number is known, it is possible to calculate the ratio of the local isentropic stagnation property to the corresponding static property at any point in a flow for an ideal gas.

$$\frac{P_0}{P} = \left[1+\frac{\gamma-1}{2}M^2\right]^{\frac{\gamma}{\gamma-1}} \tag{3}$$

$$\frac{T_0}{T} = 1+\frac{\gamma-1}{2}M^2 \tag{4}$$

$$\frac{\rho_0}{\rho} = \left[1+\frac{\gamma-1}{2}M^2\right]^{\frac{1}{\gamma-1}} \tag{5}$$

Thermodynamic state of a system when the flow reaches sonic condition ($M = 1$) is used as the reference condition and termed as critical condition.

$$\frac{P_0}{P*} = \left[\frac{\gamma+1}{2}\right]^{\frac{\gamma}{\gamma-1}} \tag{6}$$

$$\frac{T_0}{T*} = \frac{\gamma+1}{2} \tag{7}$$

$$\frac{\rho_0}{\rho*} = \left[\frac{\gamma+1}{2}\right]^{\frac{1}{\gamma-1}} \tag{8}$$

The nozzle is said to be choked when throat conditions meet with the critical conditions.

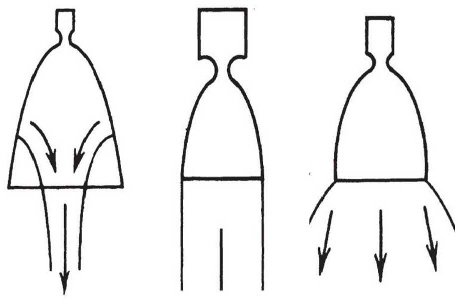

Figure 3: Effect of ambient pressure at exhaust on the performance of nozzle

Ambient Air Pressure and Nozzle Exit Pressure The exit pressure (P_e) must match the ambient air pressure (P_a) in order to get the exit exhaust to flow parallel. For the condition when ambient surrounding pressure is greater than the exhaust fluid pressure at exit, the exhaust flow is squeezed and the gas will not flow parallel to the nozzle. Higher gas pressure relative to the surrounding pressure at the exit causes the exhaust to expand as exit flow pushes against the surrounding air, deviating flow from intended parallel path. Hence nozzle area ratio is optimum when exhaust exit pressure is equal to ambient air pressure. Figure 3 depicts this with three different nozzles: the most effective nozzle being in the center.

2. Analytical Design of Nozzle

The design of nozzle is based on fundamental mathematical understanding of compressible flow and gas dynamics. This section determines the geometry of a specific nozzle as well its as various thermodynamic parameters.

2.1 Selection of Design Parameters

In this study, design parameters are taken in such a way that it can be manufactured easily and has experimental feasibility. For flow of gases inside and at exhaust of the nozzle, design parameters taken are $\gamma = 1.4$, $R = 287 \quad J/(Kg.K)$ and Mach numbers from $M = 2$ The ambient pressure on the nozzle exit is P_a, the ambient atmospheric pressure of 101325 Pascal. It is no longer valid to assume that the fluid from nozzle is an ideal gas at shock wave speeds higher than Mach 3 and the ratio of specific heat decreases from its ideal value of 1.4 for air [4].

2.2 Design of Geometry and diverging Wall Contour

The Mach area relation from Equation 2, the nozzle area ratio corresponding to the exit Mach number is determined for supersonic condition. Because the flow velocity in the converging section of the nozzle is relatively low, minimum energy losses can be achieved with well-rounded smoothed convergent nozzle surface. For the convergent, subsonic section, there is no one contour that is superior to another. There are guidelines based on real-world experience that are supported by subsonic flow theory. The performance of supersonic CD nozzle is mostly dictated by the diverging section because the flow is involved with extremely high velocities on that section [5].

The convergent section is developed using a smoothly varying cubic spline and is the same for all nozzles. The diverging contour coordinates are generated using general MOC theory which results in a uniform 1-D exit profile. In a supersonic flow, theory of MOC describes *lines* that are directed in particular directions and serve as conduits for the propagation of disturbances and pressure waves. It is a numerical method useful for solving two-dimensional compressible flow issues. This method allows for the calculation of velocity and its direction at various locations within a flow field [1]. The expansion of supersonic flow through Mach waves forms the cornerstone of the Prandtl and Busemann based MOC design technique [6]. The diverging contour profile generated from MOC based MATLAB is shown in Figure 4.

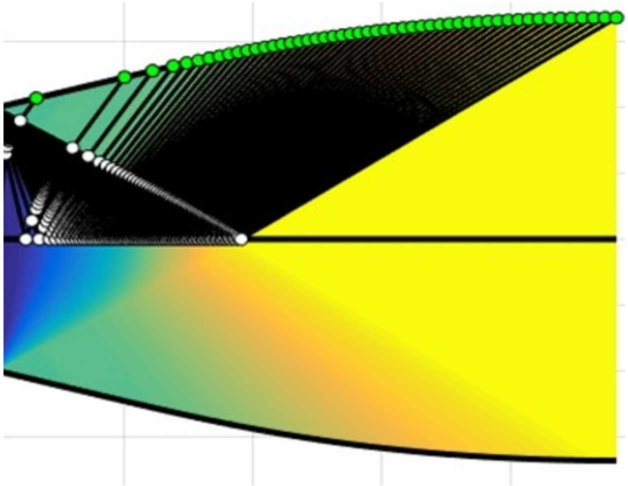

Figure 4: Diverging Contour of the nozzle developed using MOC from MATLAB

2.3 Thermodynamic Parameters and Performance

Inlet Chamber conditions For infinitely large inlet chamber, the flow velocity can be considered to be zero. In this case, the stagnation condition is equivalent to the inlet chamber condition ($P_c = P_0, \; T_c = T_0 \; and \; \rho_c = \rho_o$). Equation 3 can be used to calculate the pressure required inside the chamber, which is the pressure at the inlet of the nozzle. The equation can be rearranged as follows:

$$P_C = P_a \left[1 + \frac{\gamma - 1}{2} M^2 \right]^{\frac{\gamma}{\gamma - 1}} \tag{9}$$

As for optimum operation of nozzle,

$$P_e = P_a = 101325 \; Pascal$$

From above equation, chamber pressure (P_c) required for Mach 2 exit condition is calculated as 792362 Pascal. At this operating pressure, the exhaust from the nozzle flows with Mach 2 which is parallel to the axis of nozzle without any formation of normal or oblique shock waves and gives the optimum thrust performance.

Considering room temperature (T_c) of 300°K, the density at inlet region can be calculated using ideal gas equation.

$$\rho_c = \frac{P_c}{R \cdot T_c} = 9.21\ Kg/m^3$$

Throat conditions With a choked condition at throat, supersonic flow is achieved in the CD nozzle. At this condition, the flow at the throat reaches the local sonic condition and the properties is regarded as critical properties. ($P_t = P^*,\ \ T_t = T^*\ \ and \rho_t = \rho^*$). From Equation 6 to 8, the critical condition equations can be rearranged as

$$P_t = \left(1 + \frac{\gamma - 1}{2}\right)^{\frac{-\gamma}{1-\gamma}} \tag{10}$$

$$T_t = \frac{2 \cdot T_c}{\gamma + 1} \tag{11}$$

$$\rho_t = \rho_C \left[\frac{\gamma + 1}{2}\right]^{\frac{-1}{\gamma - 1}} \tag{12}$$

For Mach 2 exit flow requirement, the values obtained are as follows
$P_t = 418590\ \ Pascal$,

$T_t = 250\ \ K$ and

$\rho_t = 5.84\ \ Kg/m^3$
Also, local velocity of sound at throat can be calculated by
$a_t = \sqrt{K \cdot R \cdot T_t} = 293.43\ m/s$

Exit conditions The nozzle is designed to have an exit pressure that matches to the standard atmospheric pressure at designed operating condition ($P_e = P_a$). Isentropic relations can be used to calculate additional fundamental parameters such as temperature and density at exit condition. Equation 4 and 5 can be arranged as

$$T_e = \frac{2 \cdot T_c}{\gamma - 1} M^2 \tag{13}$$

$$\rho_e = \rho_c \left[\frac{\gamma + 11}{2} M^2\right]^{\frac{-1}{\gamma - 1}} \tag{14}$$

For Mach 2 exit flow requirement, following values are calculated.
$T_e = 166.67°K$ and $\rho_e = 2.12\ Kg/m^3$

Also, the local velocity of sound at exit can be calculated by
$a_e = \sqrt{K \cdot R \cdot T_e} = 239.58\ m/s$

Velocity at exit is determined using Mach number relationship.
$V_e = M \cdot a_e = 479.17\ m/s$

3. Numerical Simulation

ANSYS is a commercially available software package that is widely used for structural, thermal, fluid, electrical, magnetic, and other system analysis. A specific problem is defined in the software along with its boundary and other known conditions. The software then numerically computes the system by discretizing it in terms of algebraic equations and solves the problems. ANSYS Fluent is a CFD software program that can numerically solve compressible flows as well. It is written in the C programming language [7]. The student version of ANSYS 2021 R2 is used to simulate the flow through the nozzle.

In CAD software, a two-dimensional cross section surface of the nozzle along with exhaust flow domain is created and imported into fluent in *Initial Graphics Exchange Specification (iges)* file format.

3.1 Mesh Method and Control

Convergence of any CFD solution is expected only if the element shape, number, and distribution are properly taken into account. Figure 5 shows the discretized domain. The mesh is generated using three processes on the fluid domain starting from face splitting to edge sizing and final step of face meshing.

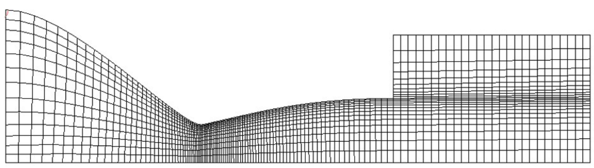

Figure 5: Domain Discretization

3.2 Problem Definition and Setup

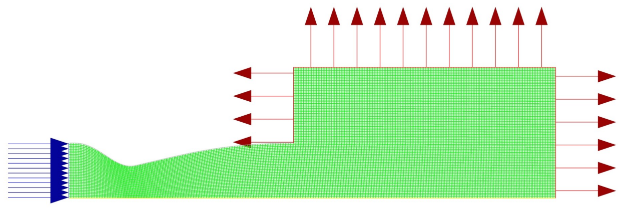

Figure 6: Boundary Conditions

Major CFD parameters for solvers are listed in the table below.

Table 1: Solver Setup Definitions

S.N.	Category	Parameters	Definitions
1	General	Solver Type	Density Based
		Space	Two Dimensional
		Domain Type	Axi-symmetric
		Time Marching	Steady
		Gravity	OFF
2	Model	Energy Equation	ON
		Viscous Model	$k - \varepsilon$ with standard wall function
3	Material	Density	Ideal Gas
		Cp	1006.43 J/Kg.K
		γ	1.4
		Mean Molecular Mass	28.966 g/mol
4	Boundary	Inlet Pressure	110000-1200000 Pascal
		Outlet Pressure	101325 Pascal
		Wall	Fixed
		Center-line	Symmetry

The fundamental equations of fluid flow is governed by the following laws: mass conservation, momentum conservation, and energy conservation. Total six scalar unknowns which are density (ρ), Pressure (p), three velocity component (V_i, V_j and V_k) and temperature (T) are determined by solving conservation of mass, three components of conservation of momentum, conservation of energy along with equation of state (total six scalar equations) simultaneously.

Density based solver with implicit formulation and steady state two-dimensional analysis is chosen in Fluent environment. Due to the fact that the density-based method was initially created for high-speed compressible flows, it has an accuracy advantage over the pressure-based solver for high speed supersonic conditions. Absolute velocity formulation without consideration of gravitation effect is used. The double precision calculation strategy is chosen over the single precision calculation strategy. It stores floating point numbers in the computer's memory and decrease round-off errors possibly improving accuracy and convergence, but the downside is the demand of more hardware memory.

3.3 Solution Strategy

The convergence criteria is the most challenging to comprehend during the initial simulations. The solution almost always diverged, resulting erratic simulation solutions. The simulation is carefully investigated, and it is found that the courant number swings at random throughout each iteration. This issue is solved by incorporating the *solution steering* feature available for density-based implicit solvers. The solution steering system assists in navigating the flow solution converging a initial guess to a final required solution with minimal involvement from the user side. The solver is allowed to take the solution so that the convergence criteria is satisfied as quickly as possible. Critical solver parameters are adjusted on the background as the solver proceeds with on each iteration to get a converged solution [8].

3.4 Mesh Independence Test

Controlling the number of divisions, on each edge on the domain, yields a series of systematically refined meshes. The simulation is run for all refined meshes and the Coefficient of Drag (C_d) calculated by *Fluent Function Generator* is listed and presented in Graph 7.

The nature of the error on (C_d) obtained is decreasing, with a highest rate at the beginning up to 5000 elements and a mediocre rate from 5000 to 10000 elements. This significant variation on the error of C_d with increase in elements number suggests further mesh refinement. Negligible variation on the error of C_d for mesh having greater than 20000 elements indicates no further requirement of mesh refinement.

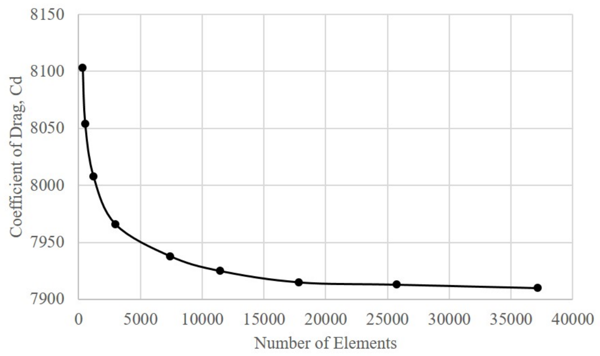

Figure 7: Mesh Independence Test

4. Experimental Setup

For experiment, the CD nozzle has a circular converging section which is blended over a rectangular throat, resulting in a planar nozzle at the output as shown in Figure 8.

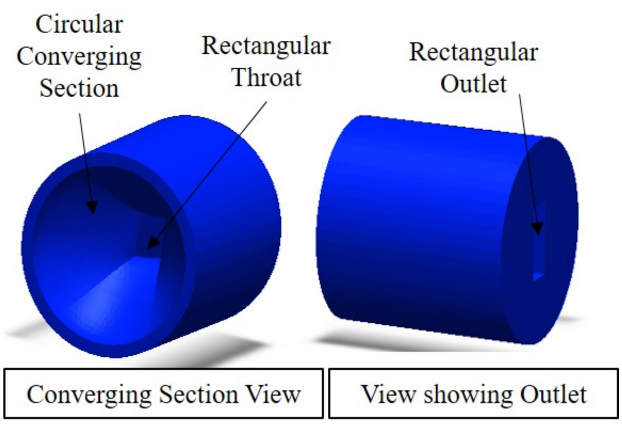

Figure 8: Geometry of nozzle used in the experiment

The designed nozzle is manufactured using a 3D printer, which is shown in Figure 9.

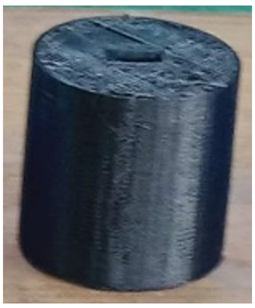

Figure 9: 3D Printed Nozzle

The inlet chamber is made from locally available one inch Galvanized Iron (GI) Pipe. The inlet side of nozzle is designed with around 1° draft value to get conical interference fit into inlet chamber ahead of the nozzle.

An external compressor with the proper reservoir setup is employed for providing air pressure at the inlet chamber of the nozzle. From the reservoir to the nozzle inlet chamber, pipe connections are installed with the least amount of head loss in mind. By adjusting the pressure on the reservoir, the required NPR for an experiment can be obtained.

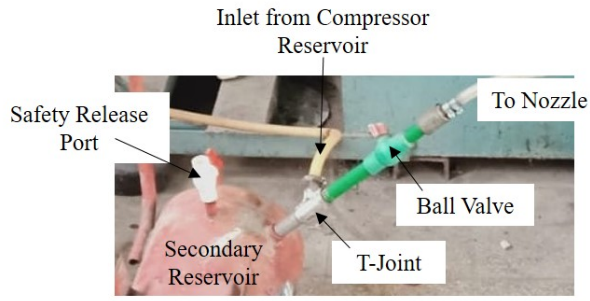

Figure 10: Connection with secondary Reservoir

The reservoir has a 15 cm exit orifice and is directly connected a ball valve using solid pipe. From ball valve is the connection to the nozzle inlet chamber with flexible hose pipe. The set up is shown by Figure 10.

4.1 Flow Visualization with Schlieren Method

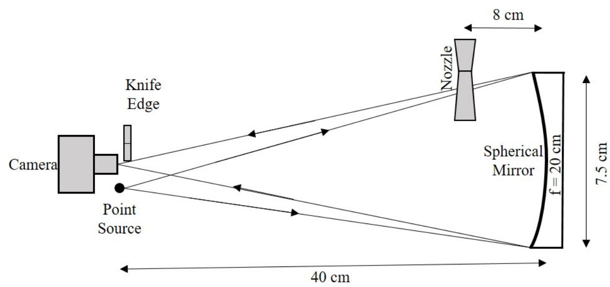

Figure 11: Schematic Schlieren Setup

The schlieren technique for visualizing flows depends on the fluid's changing density as it moves. A special optical system with a spherical mirror and a light source is employed, and light passing through the fluid produces an image that is used for flow visualization. Changes on either temperature or pressure causes the density of air flowing in front of the spherical mirror's to change. The air's resulting in-homogeneity causes the light rays it receives from the point source to reflect. This is the reason the schlieren is a density sensitive flow visualization technique and works on the principle of separating deflected light rays from unaffected light rays, typically by blocking off deflected light rays from reaching a screen on which the image of the flow is displayed. If such a deflection happens at a place in the test section and the deviated beam is then intercepted by a suitable filter device, the corresponding point on the picture plane appears darker. The suitable filter which acts as a cut-off device and it might be as simple as sharp edge of a knife [9].

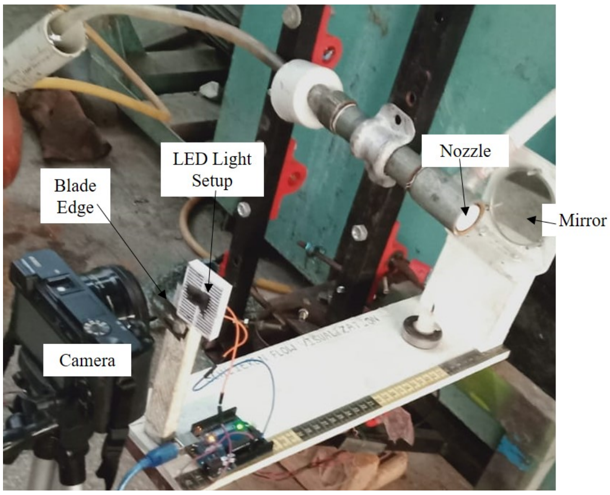

Figure 12: Schlieren Setup

A concave mirror having 20 cm focal length is used, and a red coloured Light Emitting Diode (LED) is used as a point light source at a distance twice the focal length, which is 40 cm from the center center of the mirror. This distance is actually the mirror's radius of curvature. Furthermore, a sharp blade of a paper cutter as a knife edge is placed very close to the LED to cut nearly half of the light reflected from the mirror. The camera is kept focused a few centimeters away from the knife edge to capture and record the view of the flow. The sides of the LED were taped to make it a point source, which focused the light that reaches the mirror and gets reflected to the camera. Figure 11 illustrates the schematic description of the setup.

Figure 12 portrays the actual schlieren configuration. The nozzle is kept about 8 mm in front of central axis of the mirror, and the LED is powered by an external Arduino.

5. Result and Discussion

5.1 Simulation Results

The result of simulation for nozzle designed for Mach 2 is discussed below.

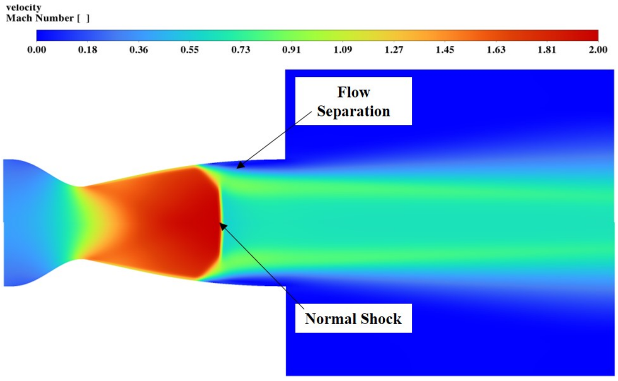

Figure 13: Velocity Contour for NPR 1.9

For a pressure ratio of 1.9 in Figure 13, abrupt velocity variation is observed across a thin line of discontinuity within the divergent section of the nozzle which is referred as a normal shock wave.

As the NPR rises, the normal shock shifts to the nozzle exit, where it crosses and the shock wave bends to produce the oblique shock phenomenon. Figure 14 apparently shows the phenomenon for NPR 4 condition. Interesting characteristics of the *Mach Disc* and *Mach Diamonds* are produced by the interaction of shock waves.

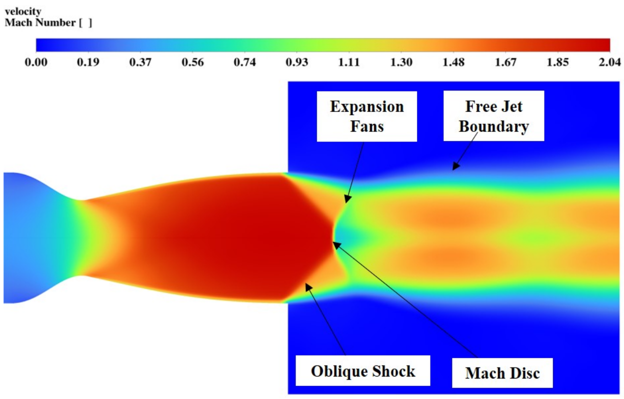

Figure 14: Velocity Contour for NPR 4

The oblique shock waves rotate to the flow direction from the nozzle exit as NPR is further increased. The flow from the nozzle is completely parallel to the flow direction and devoid of both normal and oblique shock waves at 9 NPR, as shown in Figure 15. This is necessary for the nozzle to operate optimally as it is intended to. The NPR value is substantially higher than the expected value of 7.82 which is calculated analytically using Equation 3. This indicates internal losses from the viscous and other nonlinear effects which is not considered in the analytical equation.

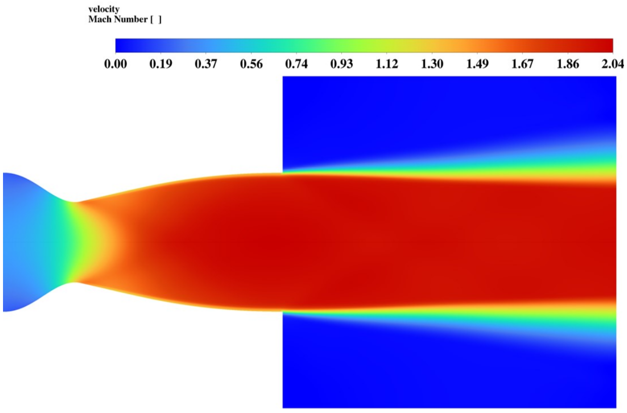

Figure 15: Velocity Contour for NPR 9

The expanded exhaust flow from the nozzle exhaust to the atmosphere is shown in Figure 16 for NPR 15. The pressure of the flow at the nozzle exit is expected to be greater than the atmospheric pressure of the surrounding and hence the flow has spread outward. The phenomenon of weak expansion fans and oblique shock waves and the yield of undesirable thrust components at the exhaust deteriorates nozzle's performance.

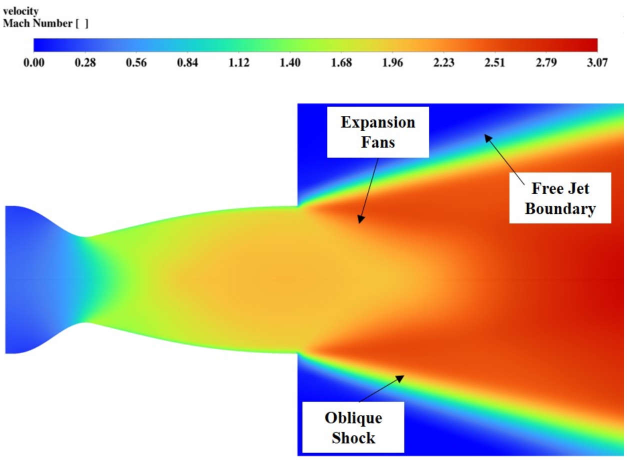

Figure 16: Velocity Contour for NPR 15

Based on the simulation results, the nozzle is expected to operate at NPR 9 with optimum performance and maximum thrust.

5.2 Schlieren flow visualization

The monochrome schlieren image captured by the schlieren setup for the exhaust flow from Mach 2.25 designed nozzle is primarily used for qualitative flow visualization. The variation of the first derivative index of refraction is directly related to the contrast seen in the image. The shock waves were depicted as a black lines.

Figure 17 depicts a comparison of *no flow* and *over-expanded flow conditions* with shock wave interactions for a Mach 2.25 nozzle. The video recorded on the camera clearly shows the movement of shock waves from free stream to nozzle exit as chamber pressure lowers with time during the experiment.

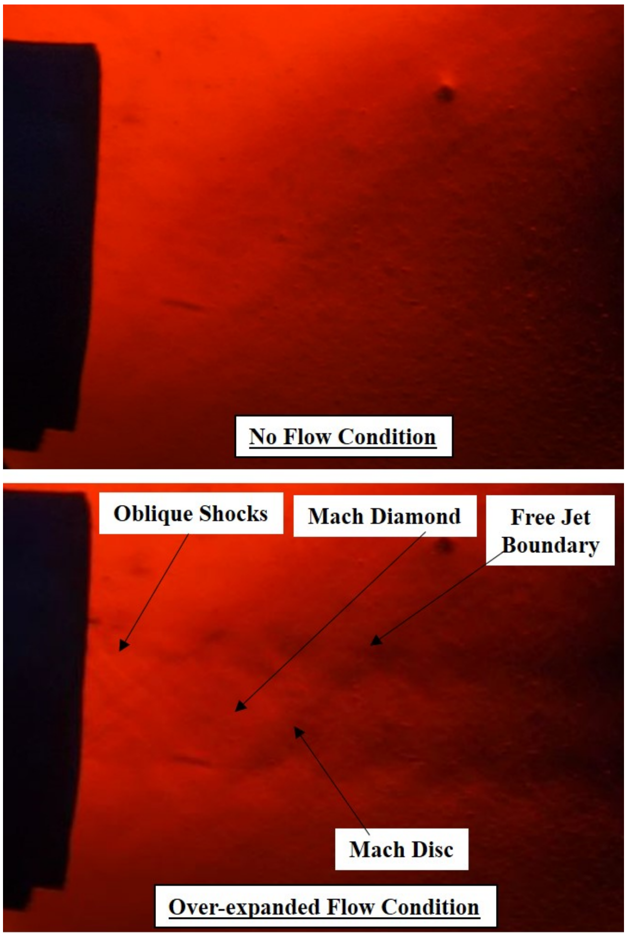

Figure 17: Comparison of No flow and Over-expanded flow visualization from Schlieren imaging

Though, the interaction between the shock waves was bluntly visualized, it illustrated a substantial impression on the supersonic flow phenomenon. A less-than-clear image of the transition phase was obtained by synchronizing the positions of the light source, the mirror and knife edge. The amount of reflected light that was cut-off at focus seems to have a significant influence on the image quality. This cut was made with a sharp blade. The position of camera behind the cut-off point also had an impact on the image quality. Premium grade spherical mirror is recommended for distinct and sharp image quality.

6. Conclusion and Recommendation

6.1 Conclusion

The NPR shows a considerable impact on the performance of the CD nozzle in simulation as well as experimental results. The exhaust flow from the nozzle is parallel at designed NPR, with no any formation of shock waves along the flow direction (perfectly expanded condition). Off-design operation generates oblique shock waves and expansion fans at the exhaust zone, reducing nozzle thrust and performance (over-expanded and under-expanded conditions). The result of numerical simulation exhibits the underlying phenomenon and interactions of shock waves with expansion fans, resulting in intriguing supersonic flow characteristics.

The Monochrome Schlieren image validates the simulation result for the over-expanded flow condition.

6.2 Recommendation

The primary importance of any optical schlieren setup is the use of high quality wider spherical mirrors. An effort to bring high-quality, large-size spherical mirrors add the value on high quality flow visualization.

Acknowledgments

The authors are grateful to University Grants Commission (UGC), Nepal for providing the financial support to complete this research.

References

[1] Md Akhtar Khan, Sanjay Kumar Sardiwal, MV Sai Sharath, and D Harika Chowdary. Design of a supersonic nozzle using method of characteristics. *International Journal of Engineering and Technology*, 2:19–24, 2013.

[2] George P Sutton and Oscar Biblarz. *Rocket propulsion elements*. John Wiley & Sons, 2016.

[3] John Anderson. *Fundamentals of Aerodynamics (SI units)*. McGraw Hill, 2011.

[4] McKenna L Roberts. *Comparison of an Axisymmetric and a Planar Shock Tunnel Nozzle*. PhD thesis, 2021.

[5] Yunus Emre Özkan. Design of a supersonic nozzle using method of chacteristics. 2021.

[6] Frederick Shope. Contour design techniques for super/hypersonic wind tunnel nozzles. In *24th AIAA Applied Aerodynamics Conference*, page 3665, 2006.

[7] John E Matsson. *An introduction to ANSYS fluent 2021*. SDC Publications, 2021.

[8] Ansys. Ansys fluent user's guide, release 18.1, 2017.

[9] Harald Kleine, Hans Grönig, and Kazuyoshi Takayama. Simultaneous shadow, schlieren and interferometric visualization of compressible flows. *Optics and Lasers in Engineering*, 44(3-4):170–189, 2006.