Experimental and numerical analysis of convergent nozzle

Srinivas G1, Bhupal Rakham*2

1Assistant Professor, Dept of Aero & Auto Engineering, Manipal Institute of Technology, Manipal University, Manipal, Udupi, Karnataka, India-576104.
2Assistant Professor, Dept of Aeronautical Engineering, MLR Institute of Technology-JNTU, Dundigal, Rangareddy District, Telangana State, India-500043.

Abstract. In this paper the main focus was given to convergent nozzle where both the experimental and numerical calculations were carried out with the support of standardized literature. In the recent years the field of air breathing and non-air breathing engine developments significantly increase its performance. To enhance the performance of both the type of engines the nozzle is the one of the component which will play a vital role, especially selecting the type of nozzle depends upon the vehicle speed requirement and aerodynamic behavior at most important in the field of propulsion. The convergent nozzle flow experimental analysis done using scaled apparatus and the similar setup was arranged artificially in the ANSYS software for doing the flow analysis across the convergent nozzle. The consistent calculation analysis are done based on the public literature survey to validate the experimental and numerical simulation results of convergent nozzle. Using these two experimental and numerical simulation approaches the best fit results will bring up to meet the design requirements. However the comparison also made to meet the reliability of the work on design criteria of convergent nozzle which can entrench in the field of propulsion applications.

1. Introduction
Today’s all the modern aircraft engines are using highly sophisticated components to achieve better fuel efficiency and thrust. These components are highly reliable and giving better service to the aerospace industry. However there are some technical problems which even today facing by all air breathing engines in each and every component like engine intakes, compressor, combustion chamber, turbine and most importantly the nozzle. Nozzle has a vital role in every air breathing and non-air breathing engines, it makes the complete expanded hot gas coming from the turbine to expel with the high acceleration [1]. The more massive expulsion of hot gas gives more forward motion to the aircraft or space craft as per the well-known theory Newton’s third law of motion [3]. In general discussion the word propulsion meaning gives the net force that results from imbalanced pressure. To understand more clearly can take the sealed container, gas or air under pressure in a sealed container applies equal forces or pressure on all sides of the container [2]. One more observation that all forces are equally balanced and there is no possibility of container to move. The purpose of this paper is to give straightforward information about the convergent nozzle basic engine operational theory and discussion on experimental results. Apart from this information this paper also describes pertinent information about problems that may phase during the experiment on
nozzle especially those cannot be solved through the simulations well and therefore cause confusion. However the consistent numerical simulation also done for better reliability of convergent nozzle [4].

In common the purpose of the nozzle in terms of the aeronautical applications it converts the chemical thermal energy formed in the combustion chamber into the required kinetic energy [6]. The nozzle makes the high velocity gas of lower temperature and pressure from high pressure, high temperature and low velocity of hot gas. According to the literature survey it is found that the nozzle generally gives exhaust velocity range from 2-4.5 km/sec [5]. If we closely observe the properties of the real fluid in the nozzle the inlet Mach number is less than one and near the throat section the flow is sonic. If we further attach the divergent section we may found that supersonic velocities depends upon the boundary conditions set in the simulation or experiment.

2. Literature on theoretical analysis on convergent nozzle

The convergent-divergent nozzle, the speed of the gas can increase from end to end, the mean speed of the molecules in the molecular structure of the gas will fall from end to end and at some point the speed of flow will equal the mean speed of the molecules [9]. We have seen that this is a special point where the duct must change from convergence to divergence. Then the flow in the convergence depends on expansion in the direction of flow and in the divergence on expansion across the flow.

The speed at the junction, that is, in the throat, is \( \sqrt{RT} \) where \( T \) the absolute temperature at the throat is of course. This immediately raises the issue of the use of the Mach number in connection with nozzles [10]. In the early part of the 20th century there was great optimism that the use of non-dimensional groups of physical properties would lead to significant improvements in the ways in which we store experimentally gathered data. One of the important groups is the Mach number denoted \( M \) and named in honor of Ernst Mach [7, 8]. It should be noted that in order to draw graph shown below to start with values for the stagnation conditions. At one time this would have meant that the calculations for drawing the graphs would have been very time consuming. That problem has disappeared with the emergence of mathematics packages on computers. We can draw as many graphs as we like almost instantly like figure 1.

![Figure 1. Representation of stagnation values of convergent nozzle.](image)

3. Experimental study on convergent nozzle

For a convergent nozzle where the area of cross-section decreases along the length of the nozzle, velocity increases according to the continuity equation as well as the Bernoulli’s equation. It is essential to demonstrate the working of the nozzle and study the velocity and pressure variations as the flow passes through the nozzle.
A convergent nozzle with surface/static pressure taps along the length of the nozzle is provided to measure the static pressure variation as the flow passes through the nozzle shown in figure 2-3. An orifice is provided in the upstream direction to measure the volume flow rate. Using the flow rate measurement and the geometry of the nozzle at various locations, it is observed that average velocity is varied along the nozzle. Comparisons can be made theoretically by obtaining the static variation using Bernoulli’s and continuity equations and practically using the measurements. Hence the working of the convergent nozzle can be demonstrated.

Figure 2. Experimental setup for convergent nozzle flow analysis.

Figure 3. Representation of convergent nozzle experimental setup manometer arrangement.

Figure 4. Geometrical details of convergent nozzle inlet and outlet diameters.
The set up consists of a blower unit coupled to D.C. motor and is connected to a convergent nozzle through a settling chamber with a hose. The discharge can be controlled by controlling the speed of the D.C. motor. The pressure tap rings (9 nos) are made in the nozzle surface and are connected to the multi-bank manometer. The orifice plate is fitted in the pipeline of the blower outlet, to measure the discharge of flow and is connected to differential manometer. The control panel consists of the mains-on indicator, control switch, D.C. motor speed controller, differential manometer and multi-bank manometer. The whole instrument is mounted on a self-contained sturdy table and is isolated from the blower unit so that vibration should not transfer to the study table.

Formulas used:

\[ Q = \text{Discharge} = C_d A \sqrt{\frac{2gh}{\rho_w}} \]
\[ g : \text{Acceleration due to gravity} = 9.8 \text{m/s}^2 \]
\[ \rho_w : \text{Density of water} = 1000 \text{kg/m}^3 \]
\[ \rho_a : \text{Density of air} = 1.22 \text{ kg/m}^3 \]
\[ h : \text{Differential manometer reading in meters of H}_2\text{O column} \]
\[ d : \text{Diameter of orifice} = 25 \text{mm} \]
Where \( C_d \) – Co-efficient of orifice = 0.64

\[
\text{Area of orifice} = \frac{\pi d^2}{4} = \frac{\pi (0.025)^2}{4} \text{ m}^2
\]

\[ V - \text{Velocity of nozzle at particular location} = \frac{Q}{A_n} \]
Where \( n \) – the nozzle number
\( A_n \) – Cross technical area at point n of the nozzle.

After successful iteration of the nozzle requirement in design point of view, numerically and experimentally the appropriate tabulation was formed. After conducting the experiment the observations has tabulated as shown in the table 1 and 2. From the figure 5 it is clear that the port number between 7 and 8 all the pressure lines meeting at one point because the area contacting in the nozzle has more influence. According to the theoretical understanding the exit velocity flow speed and discharge flow speed vary proportionally, same was observed in the experimental graph showed in figure 6 and figure 7. Similar performance also given from the figure 8 and figure 9 that velocity is directly proportional to the pressure head also discharge is proportional to the pressure head because of the area contraction at desired design point is quite good enough to achieve better velocity.

![Figure 5. Representation of pressure head in axial direction.](image-url)
Figure 6. Representation of exit velocity vs flow speed

Figure 7. Representation Discharge vs flow speed

Figure 8. Representation of velocity vs pressure head
Figure 9. Representation of discharge vs pressure head.

Table 1. Experimental tabulation static pressure values across the convergent nozzle.

| Speed | Static pressure head |
|-------|----------------------|
| %     | 1 2 3 4 5 6 7 8 9 H_ref |
| 30    | 16.6 16.6 16.6 16.6 16.8 16.7 16.6 16.9 17.2 17.2 |
| 35    | 16.6 16.7 16.7 16.7 16.8 16.7 16.7 16.9 17.3 17.3 |
| 40    | 16.5 16.5 16.5 16.7 16.7 16.7 16.9 17.5 17.5 |
| 45    | 16.4 16.4 16.4 16.5 16.7 16.7 16.6 16.9 17.7 17.7 |
| 50    | 16.4 16.4 16.4 16.4 16.7 16.6 16.6 17.9 17.9 |
| 55    | 16.3 16.3 16.3 16.4 16.6 16.5 16.6 17.1 19.1 18.1 |
| 60    | 16.3 16.3 16.3 16.3 16.5 16.5 16.6 17.1 18.1 18.1 |
| 65    | 16.1 16.1 16.1 16.1 16.3 16.4 16.5 17.1 18.7 18.7 |
| 70    | 16 16 16 16 16.2 16.3 16.6 17.2 18.9 18.9 |
| 75    | 15.9 15.9 15.9 15.9 16.2 16.2 16.6 17.2 19.3 19.3 |
| 80    | 15.7 15.7 15.7 15.8 16 16 16.5 17.3 19.8 19.8 |
| 85    | 15.5 15.5 15.5 15.5 15.7 15.8 16.4 17.4 20.5 20.5 |
| 90    | 15.3 15.3 15.3 15.3 15.6 15.7 16.4 17.4 21 21 |
| 95    | 15 15 15 15 15.3 15.4 16.3 17.5 22 22 |
| 100   | 15.1 14.7 14.7 14.8 15 15.1 16.2 17.6 22.8 22.8 |

Table 2. Experimental convergent nozzle tabulation values of manometer readings.

| Speed | Manometer head | Velocity | Discharge |
|-------|----------------|----------|-----------|
| %     | L  R  H (cm)   | m/sec    | m³/sec    |
| 30    | -0.6 0.4 1     | 9.80     | 1.39E-04  |
| 35    | -0.6 0.5 1.1   | 10.59    | 1.46E-04  |
| 40    | -0.8 0.1 0.9   | 12.66    | 1.32E-04  |
| 45    | -1 1 2        | 14.43    | 1.97E-04  |
| 50    | -1.4 1.3 2.7   | 15.50    | 2.29E-04  |
| 55    | -1.8 1.7 3.5   | 16.98    | 2.60E-04  |
| 60    | -2.2 2.1 4.3   | 16.98    | 2.89E-04  |
| 65    | -2.7 2.6 5.3   | 20.41    | 3.20E-04  |
| 70    | -3.2 3.1 6.3   | 21.55    | 3.49E-04  |
| 75    | -4 3.9 7.9     | 23.34    | 3.91E-04  |
4. Numerical simulation of convergent nozzle

To solve the numerical simulation on nozzle suitable turbulent model needed. The final solution of final step in the navier stokes equation is flow velocity. Equation (1) represents the navier stokes equation where the term ‘w’ indicates specific thermodynamic work, ‘v’ kinetic viscosity, ‘u’ velocity profile which helpful in many single phase fluid flows. Once the flow velocity is known other quantities such as temperature or pressure found easily. The Cauchy momentum equation derived originally from navier stoke momentum equation. Equation (2) shows conservative form of cauchy momentum equation, the left side of the equation designates acceleration and right side of the equation combination of body forces.

\[
\frac{\partial \mathbf{u}}{\partial t} + (u \nabla) \mathbf{u} - v \nabla^2 \mathbf{u} = -\nabla w + g. \tag{1}
\]

\[
\frac{\partial}{\partial t} (\rho \mathbf{u}) + \nabla (\rho \mathbf{u} \mathbf{u}) + \rho \mathbf{f} = \mathbf{g} \tag{2}
\]

The same convergent nozzle used for the experimental work, the numerical simulation has been conducted by creating the similar artificial environment in the ANSYS software. The basic geometric details are shown in figure 4. Once the geometry is created by using proportionate nozzle profile the boundary conditions are given before the meshing. The boundary conditions for the convergent nozzle is shown in table 3. The exit diameter of the nozzle obviously smaller comparatively the inlet diameter, therefore the velocity inlet was given to entry of the nozzle and pressure outlet was given to exit of the nozzle to find the accurate convergence solution.

| Description                          | Setup Details                                  |
|--------------------------------------|-----------------------------------------------|
| General values                       | Fluid type: Density type                       |
| Fluid Description                    | Viscous: Inviscid, Energy: on                 |
| Materials                            | Fluid: Air, Density: Real gas                 |
| Start-up cell zone condition         | Operating condition : 40 pa (Equivalent to experimental) |
| Boundary conditions                  | Inlet: velocity, Outlet: pressure, Temperature: 240 K |
| Solution control                     | Courant number: 6                             |
The surface of the nozzle taken as walls so the interference results predicted easily. The meshing plays a vital role in the simulation where the accuracy of the flow parameters like static pressure can found at each location. The mesh elements maintained almost very high density to have better flow characteristics across the nozzle. The mesh density representation over the walls shown in figure 10-11.

![Figure 10. Representation of convergent nozzle grid along XZ full symmetry in ANSYS.](image)

![Figure 11. Representation of nozzle wall mesh density high level contrast.](image)
**Figure 12.** Representation of ANSYS fluent solution convergence plot at convergence criteria $10^{-5}$.

**Figure 13.** Representation of ANSYS solution momentum and mass along the axis of convergent nozzle.

**Figure 14.** Representation of convergent nozzle mach contour.
**Figure 15.** Representation of convergent nozzle Total temperature.

**Figure 16.** Representation of convergent nozzle turbulent intensity.

**Figure 17.** Representation of convergent nozzle velocity contour.
The final most important step in the simulation giving boundary conditions. Based on subsonic standardized theory the flow boundary conditions given to convergent nozzle solver. The convergence criteria has been maintained $10^{-5}$ value so that accuracy can be improved. The convergent nozzle for the given set of mesh 2 million cells convergence solution obtained within the 500 iterations. The solution convergence plot as shown in figure 12. For the same boundary conditions the mass and momentum curves are further captured through ANSYS shown in figure 13. The flow aerodynamic behavior in convergent nozzle further explained through the Mach number, total temperature, turbulence model and velocity contour are shown from figure 14-17. The maximum total temperature found to be $3.02 \times 10^2$ K, the velocity at the exit of nozzle found to be $2.85 \times 10^3$ m/sec are optimum values comparatively with the experimental and theoretical results. From experimental, theoretical and numerical approached it is found that the port six values are very much optimum to the design point of view. The same representation of three approaches the desired values of convergent nozzle static pressure vs port number shown in figure 18. The mesh independence check also performed for

**Figure 18.** Representation of convergent nozzle static pressure vs nozzle port.

**Figure 19.** Representation of mesh independence check of convergent nozzle average temperature at exit.
optimum temperature values that has been shown in figure 19. The average temperature at outlet maintained to be 28 K which is best suitable for subsonic flow conditions in air breathing propulsion.

5. Conclusions
According to the theory studied on the nozzle and experiment conducted to the convergent nozzle it is understood that the area rule concept perfectly agreed with the numerical simulation results. After successful completion of numerical simulation, convergent nozzle experiment and theoretical calculation the desired mach value achieved at the exit of the convergent nozzle section found to be 1.8x10² and also the total temperature also found to be 3.02x10² K. Similarly the turbulent intensity was found that 5.50x10³ which is optimum velocity value for all subsonic air breathing propulsion. Because as the area decreases it increases the velocity especially at throat the flow is sonic and the flow further accelerates to supersonic if we attach additional divergent section. However the given boundary conditions are meeting by conducting the experiment on the convergent nozzle. It is also further understood that there is a scope to improve the same experimental setup to understand the convergent and divergent section nozzle three dimensional flow analysis for better aero mechanical features.

6. References
[1]. Pandey, K.M.; Singh, A.P. 2010 CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software, International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, ISSN: 2010-0221.
[2]. Natta, Pardhasaradhi.; Kumar, V.Ranjith.; Rao, Dr. Y.V. Hanumantha; 2012 Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (Cfd), International Journal of Engineering Research and Applications (IJERA), ISSN: 2248-9622, Vol. 2, Issue 5, pp.1226-1235.
[3]. P. Padmanathan, Dr. S. Vaidyanathan, 2012 Computational Analysis of Shockwave in Convergent Divergent Nozzle, International Journal of Engineering Research and Applications (IJERA), ISSN: 2248-9622, Vol. 2, Issue 2, pp.1597-1605.
[4]. Biju kuttan. P, M. Sajesh., 2013, Optimization of Divergent angle of a rocket engine nozzle using Computational Fluid Dynamics The IJES, vol.2, Issue2, Pages 196-207.
[5]. C.Satheesh, A. Arulmurugu., 2015 Design and analysis of C-D Nozzle increase the efficiency using CFD IJMTER –PP 490 - 494.
[6]. Mayur Chakravarti 2011 Design analysis of flow in convergent divergent rocket nozzle – University of Petroleum and energy studies, Department of Aerospace Engineering, Dehradun, India.
[7]. Vipul Sharma, Ravi Shankar, Gaurav Sharma, 2011, CFD Analysis of Rocket Nozzle – University of Petroleum and energy studies, Department of Aerospace Engineering, Dehradun, India.
[8]. Yasuhiro Tani, 2005, Fundamental study of supersonic combustion in pure air flow with use of shock tunnell Department of Aeronautics and stronautics, Kyushu University, Japan, Acta Astronautica 57, PP 384 – 389.
[9]. Patankar, S.V. and spalding. D.B. 1974, A calculation for the transient and steady state behaviour of shell- and- tube Heat Exchangerl. Heat transfer design and theory sourcebook. Afghan A.A. and Schluner E.U. Washington. PP. 155-176.
[10] M. Sundararaj and S. Elangovan 2013, Computational Analysis of Mixing Characteristics of Jets from Rectangular Nozzle with Internal Grooves” Supplementary Article, Indian Journal of Science and Technology, Print ISSN: 0974-6846, Online ISSN: 0974-5645. Vol 6 (5S).