ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Flow Analysis of Convergent And Divergent Nozzle Using computational Fluid Dynamics (CFD)

P.Akash, M.Chandu, P.Venkat Sai & V.Bharghav
Under the esteemed guidance of
Mrs. K. DEEPTHI,(Ph.D) M.Tech
Associate professor, Department of Mechanical Engineering
Avanthi Institute of Engineering & Technology
(Afflicted by Jawaharlal Nehru technology university, Kakinada) Cherukupalli, Bhogapuram,
Vizianagaram Dist, Ap

ABSTRACT
A nozzle is used to give the direction to the gases coming out of the combustion chamber. Nozzle is a tube with variable cross-sectional area. Nozzles are generally used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the exhaust stream that emerges from them. The nozzle is used to convert the chemical-thermal energy generated in the combustion chamber into kinetic energy. The nozzle converts the low velocity, high pressure, high temperature gas in the combustion chamber into high velocity gas of lower pressure and low temperature.

Our study is carried out on Convergent and Divergent nozzle for analysis of flow using computational fluid dynamics (CFD). Numerical study is to be conducted to understand the air flow in a Convergent-Divergent Nozzle at different divergence and convergence angles using three- dimensional models. The present study is aimed at examining the supersonic flow in C-D nozzle at various divergence and convergence angles. The flow is simulated using Annoys fluent. The result shows the variation in the pressure and velocity.

Keywords: CFD, C-D nozzle, Pressure, Velocity.

Chapter-1

1.1 NOZZLE
A nozzle is a device designed to control the direction or characteristics of a fluid flow (especially to increase velocity) as it exits (or enters) an enclosed chamber or pipe. A nozzle is often a pipe or tube of varying cross-sectional area, and it can be used to direct or modify the flow of a fluid (liquid or gas). Nozzles are frequently used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that emerges from them. In a nozzle, the velocity of fluid increases at the expense of its pressure energy. Converging nozzles, are tubes with an area that decreases from the nozzle entry to the exit (or throat) of the nozzle. As the nozzle area decreases, the flow velocity increases, with the maximum flow velocity occurring at the nozzle throat. A nozzle whose cross section becomes larger in the direction of flow are called Divergent nozzle. The figure 1.1 shows the convergent and divergent nozzle.

Cite this article as: P.Akash, M.Chandu, P.Venkat Sai, V.Bharghav & Mrs. K. Deepthi, "Flow Analysis Of Convergent And Divergent Nozzle Using computational Fluid Dynamics (CFD)", International Journal & Magazine of Engineering, Technology, Management and Research (IJMETMR), ISSN 2348-4845, Volume 10 Issue 3, March 2023, Page 9-16.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Exhaust gases from combustion are pushed into throat region of nozzle.

- Throat is smaller cross-sectional area than rest of engine; here gases are compressed to high pressure.
- Nozzle gradually increases in cross-sectional area allowing gases to expand and push against walls creating thrust.
- Mathematically, ultimate purpose of nozzle is to expand gases as efficiently as possible so as to maximize exit velocity.

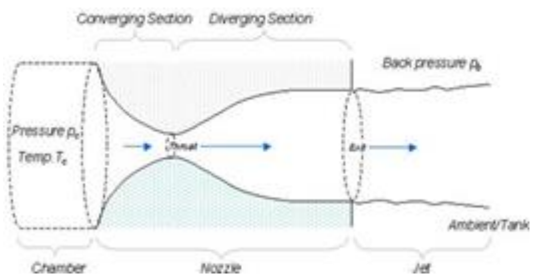

Fig 1.1 C-D nozzle

Chapter-2
Literature review

This chapter provides the information about the works carried out by different authors on flow analysis on various nozzles using CFD.

BijuKuttan P [1] Conducted numerical analysis to determine an optimum divergent angle for the nozzle which would give the maximum outlet velocity and meet the thrust requirements. The inlet dimensions and the boundary conditions are kept constant and the divergent angles are varied in order to understand how the variation in divergent angle affects the flow pattern through the nozzle. Among the various models available in Fluent, the k-ε model was selected for their work. A two-dimensional axi- symmetric geometrical model of the nozzle was used for the analysis purpose. Divergence angles chosen were 4°, 7°, 10°, 13° and 15°

C A Hunter [2] conducted experimental, theoretical, and computational study of separated nozzle flows. Experimental testing was performed at the NASA Langley 16- Foot Transonic Tunnel Complex. As part of a comprehensive static recital examination, moment, force and pressure measurements were made and Schifrin flow visualization was obtained for a subscale, non axi-symmetric, two-dimensional, C-D nozzle. In addition, two-dimensional numerical simulations were run using the computational fluid dynamics code PAB3D with two-equation turbulence closure and algebraic Reynolds stress modeling. For reference, experimental and computational results were compared with theoretical predictions based on one-dimensional gas dynamics and an approximate integral momentum boundary layer method.

NazraMenem Mahmood [3] simulated steady flow of a gas through a C-D nozzle which has a varying cross sectional area and showed that the nature of the flow can be explained by considering how the flow and its characteristics in the nozzle changes as the back pressure decreases. The characteristics of gas flow were simulated using the ANSYS Fluent 12.1 software to solve the quasi-one dimensional nozzle flow. According to the study, the reduction in the back pressure cannot affect conditions upstream of the throat, hence the nozzle is choked. The shock wave increases the pressure, density and temperature and reduces the velocity and Mach number to a subsonic value and as back pressure is further reduced to a certain value, the extent of the supersonic flow region increases, the shock wave moving further down the divergent portion of the nozzle towards the exit plane.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Chapter-3
Problem Statement

The main objective of the present work is to simulate supersonic flow through convergent and divergent nozzle to precisely understand the flow dynamics and variation of flow parameters namely Pressure and velocity using computational fluid dynamics (CFD).

For this study a nozzle is designed of varying convergence angles and divergence angles, analyzed using CFD for various performance parameters. The flow conditions were selected based on the pressure and temperature of the air at the inlet of the nozzle.

Chapter-4
MODELLING AND SIMULATION

4.1 modelling

4.1.1 Geometry:

The geometry of the nozzle was created using the Geometry workbench of ANSYS. A two-dimensional geometry of the nozzle was created.

Table 4.1 dimensions of the nozzle

Length of nozzle	200mm
Inlet width about axis	50mm
Outlet width about axis	80mm
Radius of curvature of throat	15mm
Divergence angle	30°
Convergence angle	30°

Fig 4.1 shows the symmetric fig of a 2-D C-D nozzle

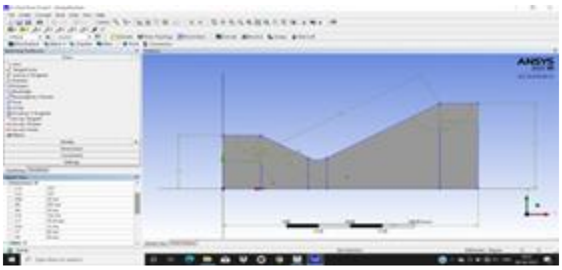

Fig 4.1 2-d Model of C-D nozzle

4.1.2 Meshing

Ansys provides general purpose, high-performance, automated, intelligent meshing software that produces the most appropriate mesh for accurate, efficient multiphasic solutions from easy, automatic meshing to highly crafted mesh. Smart defaults are built into the software to make meshing a painless and intuitive task, delivering the required resolution to capture solution gradients properly for dependable results.

General	Solver type: density based 3D Space: Axi-symmetric
Models	Energy equation: On Viscous model: inviscid
Materials	Air-Ideal gas
Boundary Conditions	Inlet- pressure Total gauge pressure 1.01325 bar Outlet – pressure Gauge pressure 0.028 bar Axis – axis boundary
Operating conditions	Operating pressure 0 Pascal

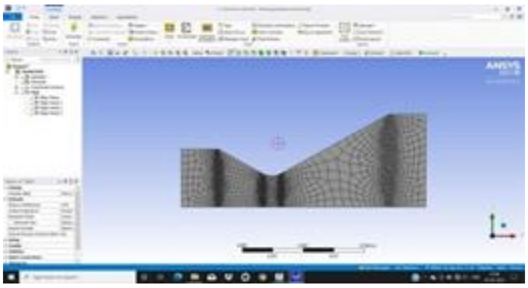

Fig 4.2 Initial mesh of nozzle

4.2 Simulation

The meshed geometry is now imported to the FLUENT workbench Setup module.Certain boundary conditions applied for simulation process, the solver type selected as inviscid to eliminate viscosity and 2-d space as axisymmetric. The figure 4.4 shows the solver setup for ansys simulation.

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Fig 4.4: Solver setup for ansys simulation

Solution initialization	Compute from: inlet
Run calculation	Check case, Enter number ofiterations: 500 Click Calculation

Table 4.3 solver setup Table 4.2 Problem setup

Chapter 5

RESULTS AND DISCUSSIONS

5.1 Case 1: Convergence 30° Divergence 25°

The pressure contour & velocity contour were analysed using CFD by considering convergence 30° and Divergence 25°

5.1.1 : Pressure Contour

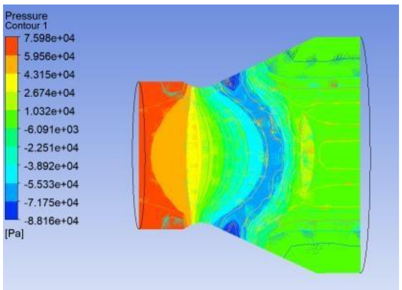

Fig 5.1 PRESSURE CONTUR: 7.598e^-6 MPA

5.1.2 : Velocity Contour

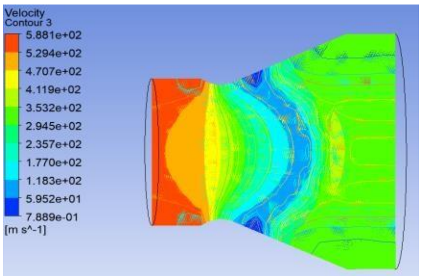

Fig5.2 VELOCITY CONTUR:3.232e^+02ms^-1

5.1.3 : Temperature Contour

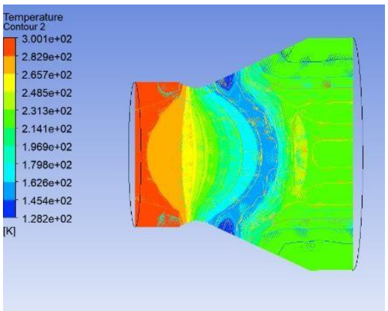

Fig 5.3TEMPERATURE CONTUR :3.001e^+02k

5.2 Case 2:Convergence 30° Divergence 35°

The pressure contour & velocity contour were analysed using CFD by considering convergence 30° and Divergence 35°

5.2.1 : Pressure Contour

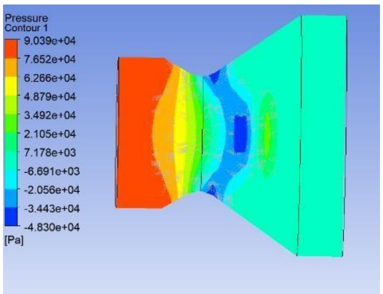

Fig 5.4 PRESSURE CONTUR: 9.039e^-6 MPA

5.2.2 : Velocity Contour

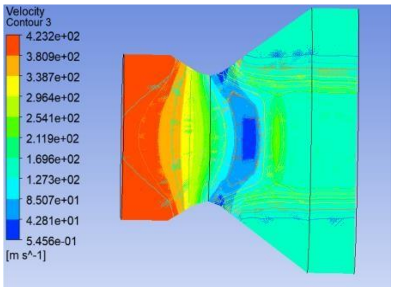

Fig 5.5VELOCITY CONTUR: 4.232e^+02ms^-1

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

5.2.3 : Temperature Contour

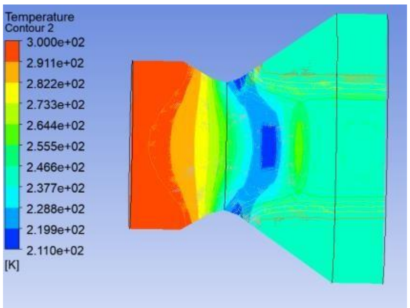

Fig 5.6TEMPERATURE CONTUR :3.000e^+02k

5.3 Case 3: Convergence 25° divergence 30°

The pressure contour & velocity contour were analysed using CFD by considering convergence 25° and Divergence 30°

5.3.1 : Pressure Contour

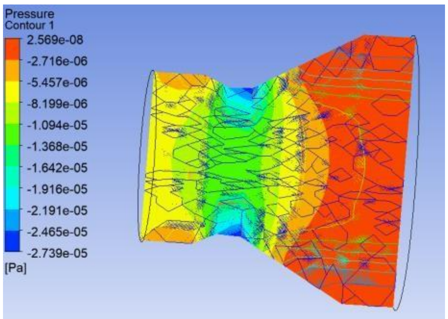

Fig 5.7PRESSURE CONTUR: 2.569e^-6 MPA

5.3.2 : Velocity Contour

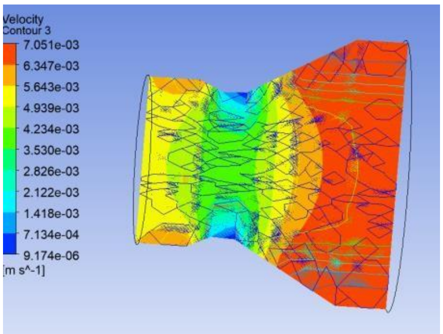

Fig 5.8VELOCITY CONTUR: 7.051e^-03ms^-1

5.3.3 : Temperature Contour

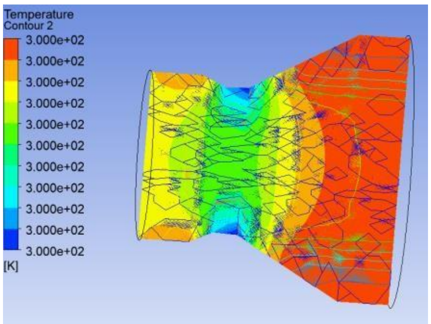

Fig 5.9TEMPERATURE CONTUR :3.000e^+02k

5.4 Case 4: Convergence 35° Divergence 30°

The pressure contour & velocity contour were analyzed using CFD by considering convergence 35° and Divergence 30°

5.4.1 : Pressure Contour

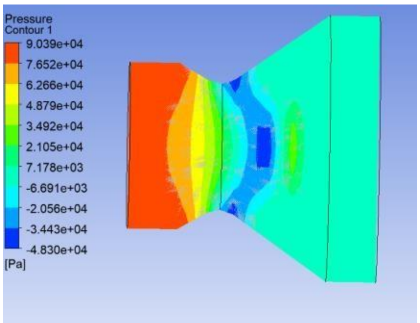

Fig 5.9 PRESSURE CONTUR: 9.039e^-6 MPA

5.4.2 : Velocity Contour

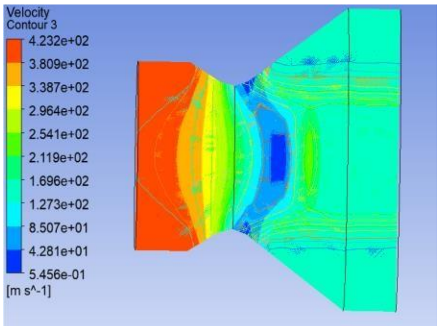

Fig 5.10 VELOCITY CONTUR: 4.232e^+02ms^-1

5.4.3 : Temperature Contour

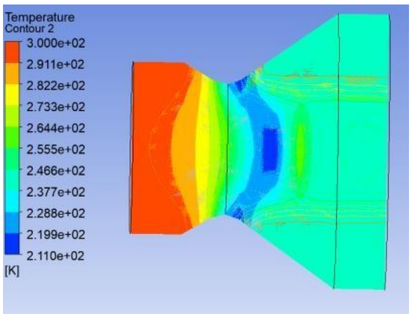

Fig 5.11 TEMPERATURE CONTUR : 3.000e^+02k

5.5 Case 5: Convergence 30° Divergence 30°

The pressure contour & velocity contour were analyzed using CFD by considering convergence 30° and Divergence 30°

5.5.1 : Pressure Contour

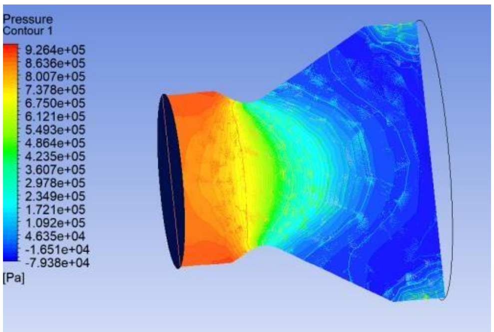

Fig 5.12 PRESSURE CONTUR: 9.264e^-6 MPA

5.5.2 : Velocity Contour

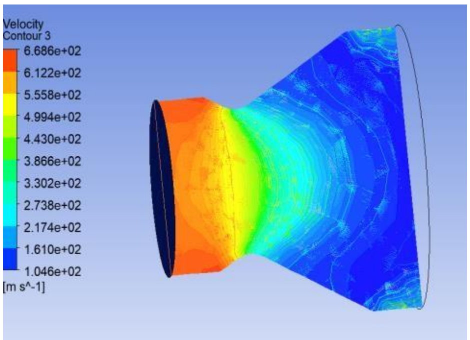

Fig 5.13 VELOCITY CONTUR: 6.686e^+02ms^-1

5.5.3 : Temperature Contour

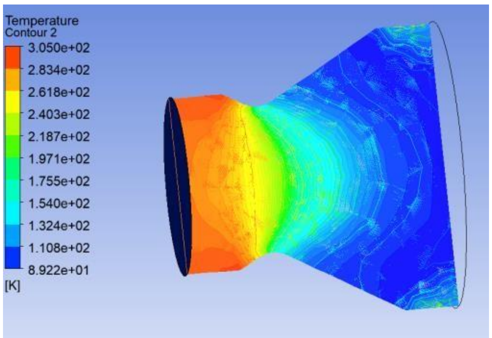

Fig 5.13 TEMPERATURE CONTUR :3.050e^+02k

SR NO:	CONVERGENT DIVERGENT	PRESSURE CONTUR(MPA)	TEMPERATURE CONTUR(K)	VELOCITY CONTUR(m^-1)
1	30-30	9.264e^-6	3.050e^+02	6.686e^+02
2	25-30	2.569e^-6	3.000e^+02	7.051e^-03
3	35-30	9.039e^-6	3.000e^+02	4.232e^+02
4	30-25	7.598e^-6	3.001e^+02	3.232e^+02
5	30-35	9.039e^-6	3.000e^+02	4.232e^+02

RESULT TABLE

GRAPHS

GRAPH FOR PRESURE COUNTUR:

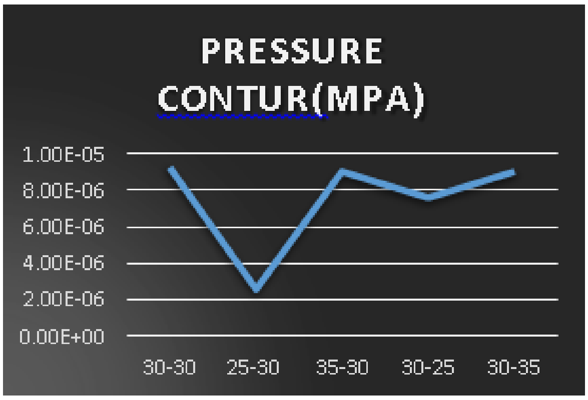

Fig 1 PRESSURE CONTUR(MPA)

GRAPH FOR TEMPERATURE COUNTUR:

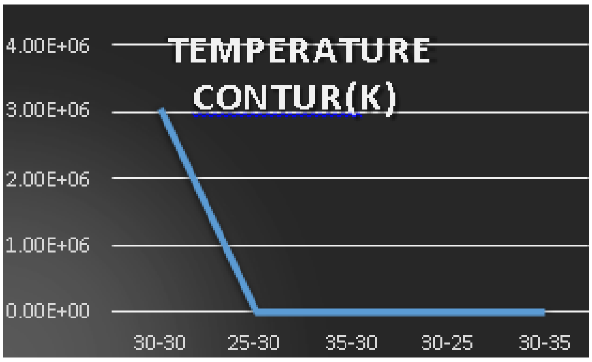

Fig2 TEMPERATURE CONTUR(K)

GRAPH FOR VELOCITY COUNTUR

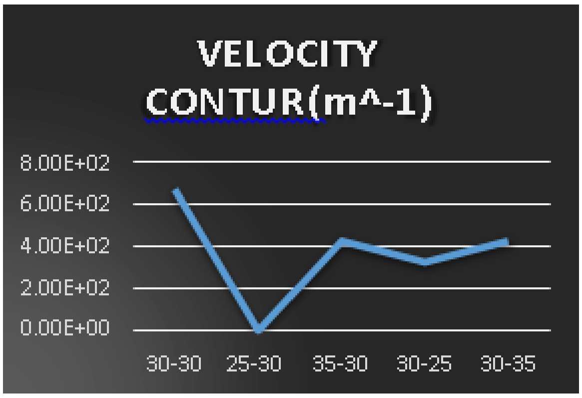

Fig 3VELOCITY CONTUR(m^-1)

Chapter-6

CONCLUSIONS

The following observations were found in the nozzle by varying convergence and divergence angles

- When we increase the divergence angle keeping the convergence angle as constant, pressure decreases and velocity increases in great magnitude.
- By keeping convergence angle 30° and varying divergence angle, the maximum velocity is obtained at 35° divergence angle.
- When we increase the convergence angle keeping the divergence angle constant, we observed there is decrease in pressure and increase in velocity but the variation is less compared to earlier case.
- 35° convergence angle is giving maximum velocity when compared to other angles by keeping divergence angle constant at 30°.

Therefore, we can conclude that in a Convergent-Divergent nozzle, variation in divergence angle affects the velocity and pressure effects compared to variation in convergence angle.

FUTURE SCOPE OF WORK:

1. The present work may be extended to optimization of convergence and divergence angle for more outputs.
2. The analysis may be extended using ANN & Fuzzy Techniques.

List of references

1. BijuKuttan P: "Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics" The International Journal Of Engineering And Science (Ijes),Volume 2,Issue2, Issn: 2319 – 1813 Isbn: 2319 – 1805, 2013, pp 196-207.
2. C.A. Hunter: "Experimental, Theoretical, and Computational Investigation of Separated Nozzle Flows-AIAA 98-3107".
3. Nazr Menem Mahmood Simulation Of Back Pressure Effect On Behavior Of Convergent Divergent Nozzle Diyala Journal of Engineering Sciences ISSN 1999- 8716Vol. 06, No. 01, March 2013 pp. 105-120
4. Sibendu Soma, Anita I. Ramirez , Douglas E. Longman , Suresh K. Aggarwal, "Effect of nozzle orifice geometry on spray, combustion, and emission characteristics under diesel

engine conditions" journal homepage: www.elsevier.com/locate/fuel

5. P. Padmanathan, Dr. S. Vaidya Nathan, "Computational Analysis of Shockwave in Convergent Divergent Nozzle" International Journal of Engineering Research and Applications, Vol. 2, Issue 2,Mar-Apr 2012, pp.1597-1605

6. 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, vol. 13, iss. 4, pp. 58–65, 2016.
7. B.V.V.N. Sudhakar, B.P.C. Sekhar, P.N. Mohan, M.D.T. Ahmad, "Modeling and simulation of Convergent-Divergent Nozzle Using Computational Fluid Dynamics," International Research Journal of Engineering and Technology, vol. 3, iss. 8, pp. 346-350, 2016.
8. David Getrix, Dutton, J.C "Swirling Supersonic Nozzle Flow," Journal of Propulsion and Power, vol.3, July 1987.
9. TrongBui, Bogdan-AlexandruBelega, "CFD analysis of flow in Convergent-divergent nozzle", international conference of scientific paper AFASES 2015 Brasov, 28-30 May 2015
10. Gustaf De-Laval, "Concepts and CFD analysis of De- Laval nozzle" in International Journal of Mechanical Engineering and Technology (IJMET) Volume 7, Issue 5, September 1991
11. Nikhil D. Deshpande, Suyash, S. Vidwans, "Theoretical and CFD analysis of De-Laval nozzle" in International Journal of Mechanical and Production Engineering, Volume- 2, Issue- 4, April-2014
12. Nathan spotts "A CFD analysis of compressive flow through convergent conical nozzle " first AIAA Propulsion Aerodynamics Workshop, Atlanta Georgia, July 29, 2012