Computation of C-D Nozzle flow using Finite Element Method

Y. Phani Chandra
Department of Mechanical Engineering
P.V.P. Siddhartha Institute of Technology, Kanuru, Vijayawada-520007

V. Sai Manasa
Department of Mechanical Engineering
P.V.P. Siddhartha Institute of Technology, Kanuru, Vijayawada-520007

K. Chanti
Department of Mechanical Engineering
P.V.P. Siddhartha Institute of Technology, Kanuru, Vijayawada-520007

N. Spadana
Department of Mechanical Engineering
P.V.P. Siddhartha Institute of Technology, Kanuru, Vijayawada-520007

P. Phani Prasanthi
Department of Mechanical Engineering
P.V.P. Siddhartha Institute of Technology, Kanuru, Vijayawada-520007

Abstract - Convergent-divergent (C-D) nozzles found several applications in wide fields of engineering. The finite element method is a very useful approach for the engineers to forecast the behavior of the nozzle under designed conditions. The main motto of the present work is to study the properties of flow through C-D nozzles, using the finite element analysis software ANSYS. The analysis performed by considering the density based solver which is effective for supersonic compressible flows. Later the work is continued by considering the pressure based solver which is preferred for solving the flows of low speed incompressible type. The variation of velocity, pressure, density and temperature obtained from the both the analysis are compared under the same working conditions and the differences between the two solvers are highlighted.

Key words: Convergent-divergent nozzle, Finite element method, Pressure, Density.

I. INTRODUCTION

Nozzles are used to regulate the flow of substances. They are found at the tip of a hose or pipe from which fluids are discharged. Different nozzles are made available for various applications. Nozzles are used in lubrication systems for proper supply of lubrication; injectors are used to inject the fluid with the help of nozzles. Like this so many applications are found in several areas with liquid as well as air media. Finite element software is a wonderful tool which provides precised information with prerequisite knowledge.

Density based and Pressure based approaches are two popular supporting methods available in ANSYS to quantify the output parameters of pressure, density, temperature, Mach number etc. In the density-based approach, the continuity equation is used to obtain the density field while the pressure field is determined from the equation of state. On the other hand, in the pressure-based approach, the pressure field is extracted by solving a pressure or pressure correction equation which is obtained from continuity and momentum equations.

Many authors made their contribution towards the nozzle analysis with finite element method. Analysis of pressure vessel is very crucial with the attachment of nozzles in the aspect of stress concentration. Pravin Narale et al., [1]

made stress analysis on pressure vessels with nozzle to determine the stress at different sections. The stress level in the nozzle N1, N4, N12 and N13 category of hydro carbon vessels are analyzed by Navnath V. at al., using finite element method.

A finite element formulation is developed by Mehta R.C. et al., [3] with axisymmetric, transient, anisotropic approaches on throat nozzle for solid rocket motor. A method is developed for a precise structured modeling and for estimating the stress intensities at the junction of nozzles and pressure vessels by Devaraju et al., [4]. An investigation of the shakedown behavior of axi- symmetric nozzles under internal pressure is presented by Martin Muscat and Donald Mackenzie [5] using finite element method. The effect of material, pressure, temperature on hot runner nozzle assembly is investigated by ensuring proper nozzle design by Sanjay Rao P, Subba Rao D.V. [6] using finite element method. Franco C. Frate and James E. Bridges [7] designed extensible rectangular nozzle using finite element software.

Manuel et al., [8] presented work on how the deformation of fire hose is done when it is used for fire extinguishing. In their work experimental researches were conducted on the fire hoses deformation and, output was validated by modeling this phenomenon using numerical simulation software. Alan Vincent E V [9] performed CFD Analysis of Supersonic Exhaust Diffuser System for Higher Altitude Simulation. The variation of temperature, pressure and velocity at various sections of the de Laval nozzles are performed by using CFD software by NIKHIL at al., [10] Numerical analysis of airflow inside and behind the convergent-divergent nozzle with supersonic exhaust, performed by Olivera P. Kostic [11] considering without, and with a mechanical obstacle in exit section of the nozzle. The objective of the present work is to determine the fluid parameters velocity, pressure, temperature, Mach number of convergent divergent nozzle using finite element software Ansys.

II. FINITE ELEMENT MODELING

Geometry: The details of the finite element model geometry are provided in Fig.1.a. The dimensions of the FE model are maintained in mm. The finite element mesh to carry out the present analysis in also presented in Fig.1.b.

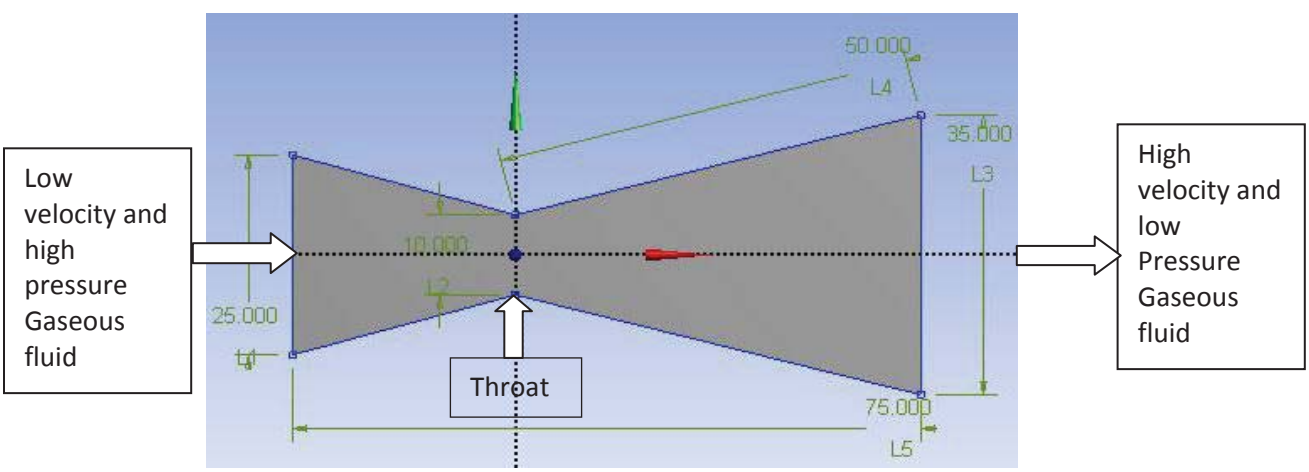

Fig.1.a Geometrical details of C-D Nozzle

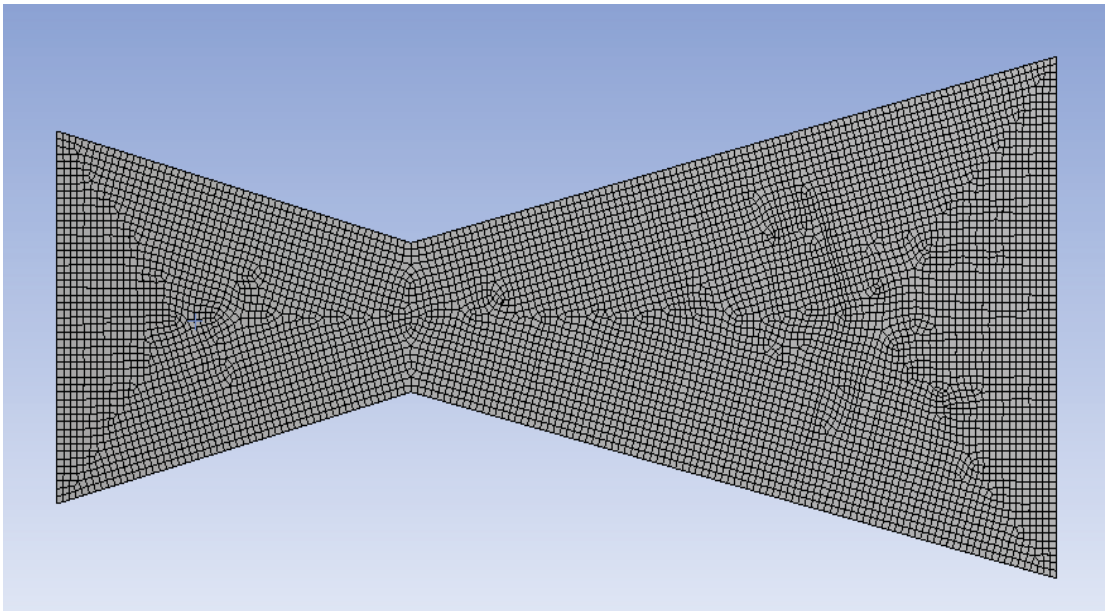

Fig.1.b Finite element mesh representation of C-D Nozzle

Material Properties: The materials taken in the study are as mentioned:
Fluid is considered to be air whose properties are predefined in software as:
Density of fluid is taken to be same as of ideal gas.
Viscosity is taken based on Sutherland law where reference viscosity is 1.716e-05 kg/m-s and reference temperature is 273.11K
Specific heat of the air is 1006.43 J/Kg-K
Thermal conductivity is 0.0242 W/m-K

III. DISCUSSION OF RESULTS

The variation of density and contour of considered nozzle is represented in Fig.2-3 for density based and pressure based solvers. No considerable differences in density are observed from the both solvers. The density of the fluid is high at the input as it passed through the nozzle; the magnitude of the density is decreased.

A clear difference is observed in Mach number of density and pressure based solvers. Even though from the both the solvers the flow becomes supersonic (Mach number >1) at the output of the nozzle. But the Mach number obtained from density based approach is more than pressure based approach. This indicated that the flow at the divergent portion of the C-D nozzle is supersonic (Fig.4-5). Similar to density, the variation of static pressure and static temperature is almost same in both the approaches (Fig.6-9).

The most important application of the nozzle is to increase the velocity of the fluid, this objective is simulated in ANSYS software and the magnitude of velocity is more in density approach. (Fig.10-11).

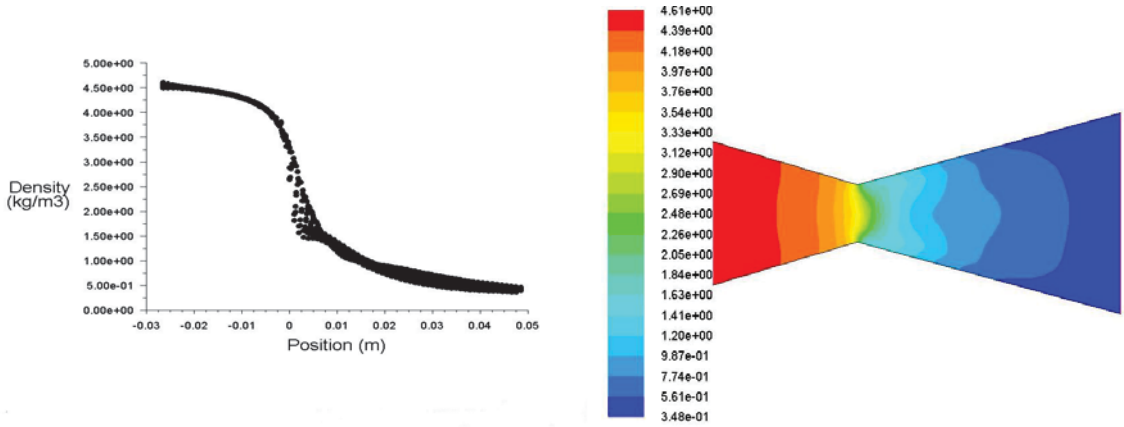

Figure 2*:* Variations in density along the length of CD nozzle in density based solver

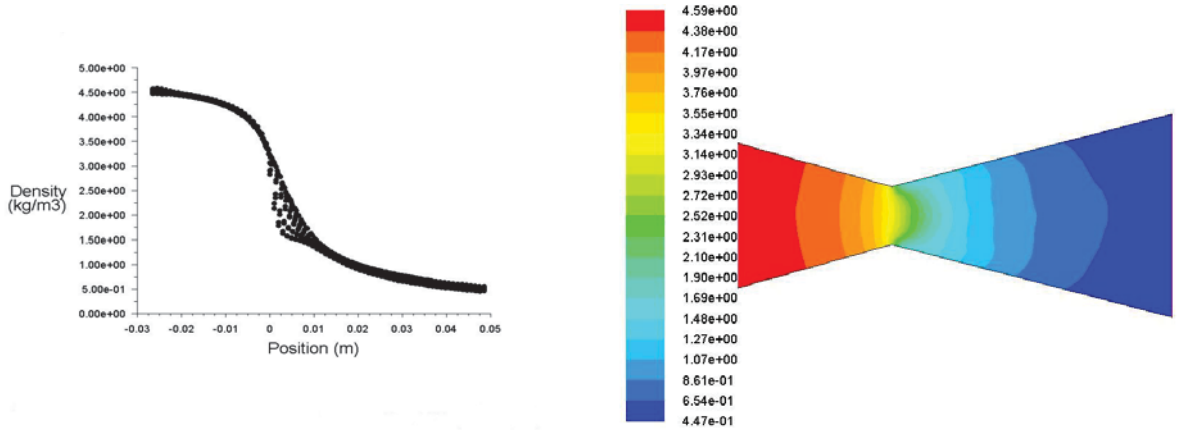

Figure 3: Variations in density along the length of CD nozzle using pressure based solver

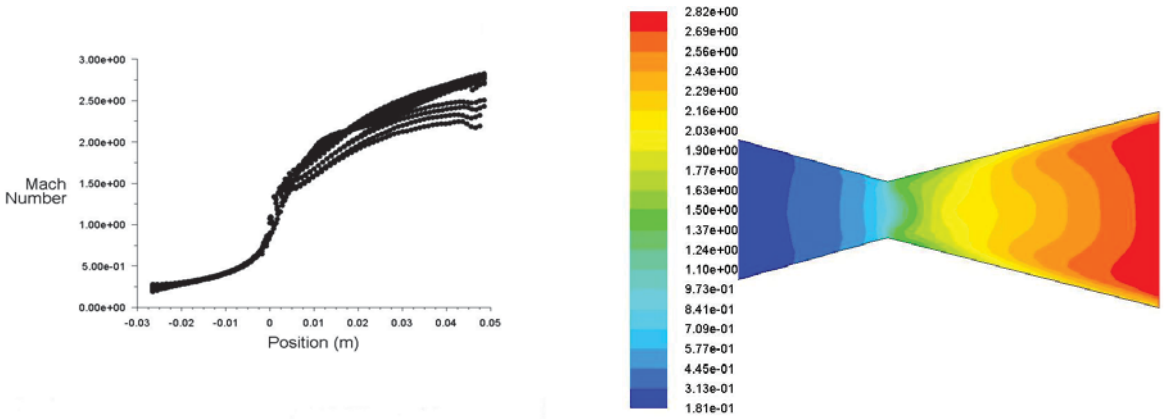

Figure 4: Variations of Mach number along the length of nozzle using density based solver

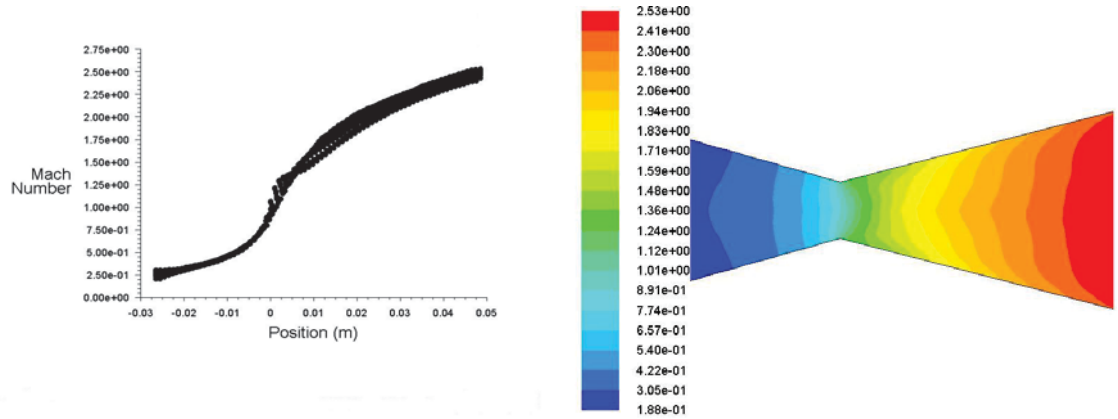

Figure 5: Variations of Mach number along the length of nozzle using pressure based solver

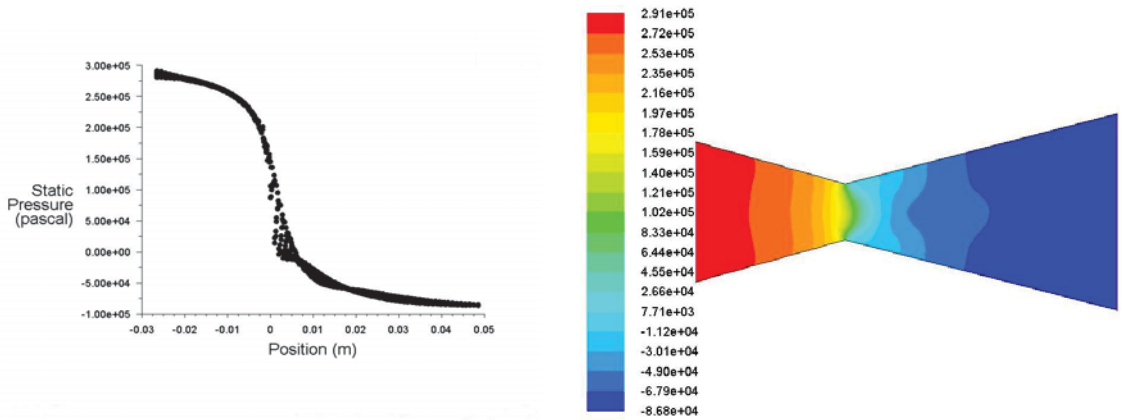

Figure 6: Variations in Static Pressure along the length of nozzle using the density based solver

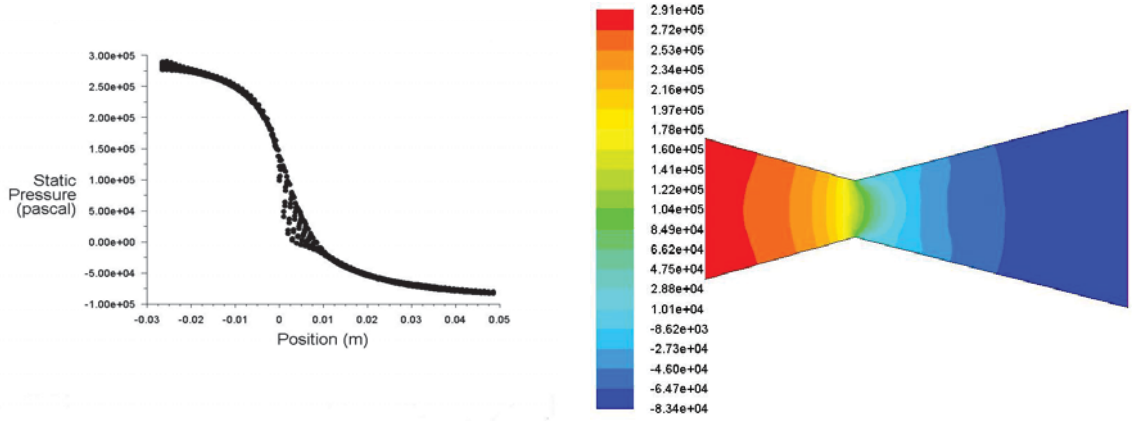

Figure 7: Variations in Static pressure along the length of nozzle using pressure based solver

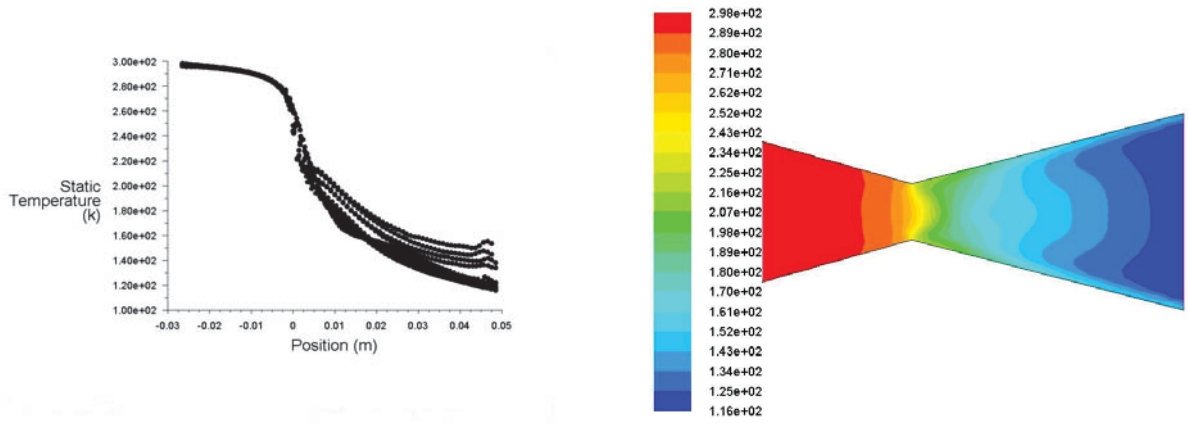

Figure 8: Variations in static temperature along the length of nozzle using density based solver

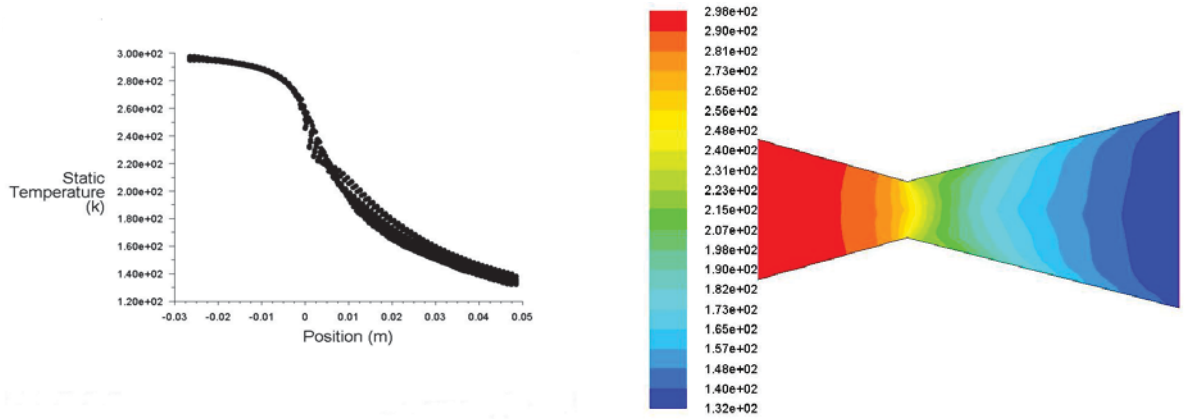

Figure 9: Variations in Static temperature along the length of nozzle using pressure based solver

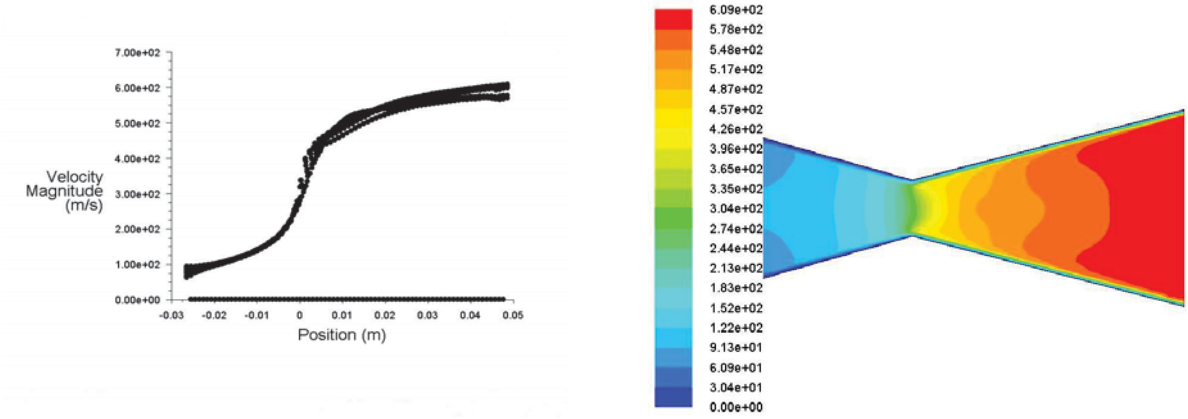

Figure 10: Variations in magnitude of velocity along the length of nozzle using density based solver

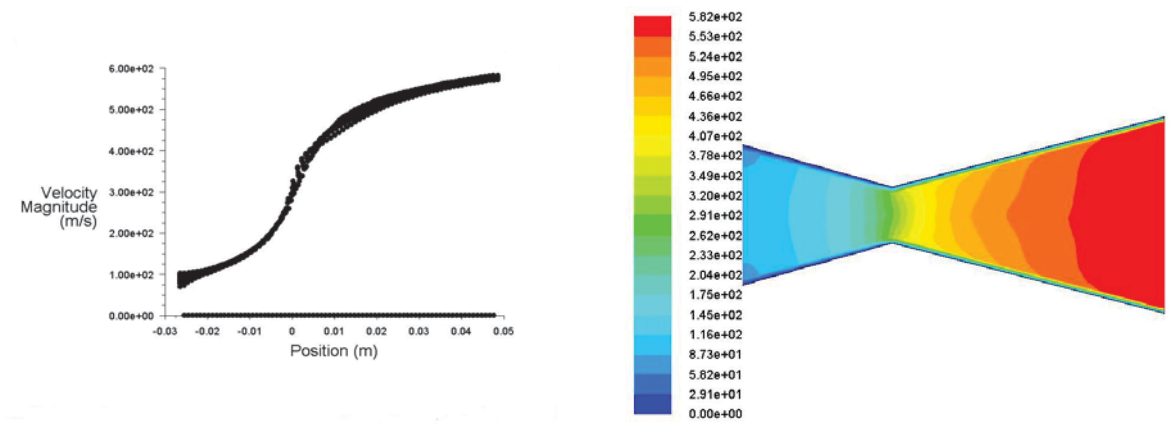

Figure 11: Variations in magnitude of velocity along the length of nozzle using pressure based solver

IV. CONCLUSIONS

For the considered geometry of the C-D nozzle the maximum Mach number is in the same zone of minimum pressure in two approaches. The velocity of the C-D nozzle is supersonic at the outlet; Mach number is greater than 1 after the throat region, whereas the Mach number is 1 in throat in both approaches, but the density approach leads to a higher average value of Mach number than pressure based solver.

REFERENCES

[1] Pravin Narale , Prof. P. S. Kachare, " Structural Analysis of Nozzle Attachment on Pressure Vessel Design", International Journal of Engineering Research and Applications (IJERA)Vol. 2, Issue4, July-August 2012, pp.1353-1358.

[2] Navnath V. Avhad, Vijay G. Bhamre, " Investigating Stress Level through FEA of Nozzles of Carbon Drain Vessel by ASME Sect-VIII", International Research Journal of Engineering and Technology, Volume: 02 Issue: 05 | Aug-2015.

[3] Mehata R.C., Suresh K and. Narayanana Iyer R, "Thermal stress analysis of a solid rocket motor nozzle throat insert using finite elementt method, Indian journal of engineering and Material science, Volume 5, PP 271-277, 1998.

[4] Devaraju U.R1, Manjunath K2, Mahesh T.S, "Design Investigation into the Stress at the Base of a Nozzle of a Pressure Vessel", International Journal of Engineering Sciences & Research Technology, July, 2014.

[5] Martin Muscat and Donald Mackenzie, "Elastic-Shakedown Analysis of Axisymmetric Nozzles", Journal of Pressure Vessel Technology, Vol. 125, NOVEMBER 2003.

[6] Sanjay Rao P., D.V.Subba Rao," Design and Analysis of Hot Runner Nozzle Using Fem", International Journal of Mechanical and Industrial Technology, Vol. 3, Issue 1, pp: (228-241), Month: April 2015.

[7] Franco C. Frate, James E. Bridges," Extensible Rectangular Nozzle Model System", 49th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition 4 - 7 January 2011, Orlando, Florida.

[8] Manuel SERBAN, Ionel Alin MOCIOI, and Dan-Victor CAVAROPOL, "Flow Modeling of the Fire Extinguishing Liquids Inside the Fire Hoses", Development, Energy, Environment, Economics.

[9] Alan Vincent E V, "CFD Analysis of Supersonic Exhaust Diffuser System for Higher Altitude Simulation", International Journal of Engineering studies and technical approach, Volume 01, No.6, June, 2015.

[10] Nikhil D. Deshpande, Suyash S. Vidwans, Pratik R. Mahale, Rutuja S. Joshi, K. R. Jagtap, "Theoretical & CFD Analysis of De Laval Nozzle", International Journal of Mechanical And Production Engineering, Volume- 2, Issue- 4, April-2014

[11] Olivera P. Kostić, Zoran A. Stefanović, Ivan A. Kostic, "CFD Modeling of Supersonic Airflow Generated by 2D Nozzle with and Without an Obstacle at the Exit Section", FME Transactions (2015) 43, 107-113.