ISSN 2394-9716

International Journal of Novel Research in Interdisciplinary Studies
Vol. 2, Issue 6, pp: (16-27), Month: November-December 2015, Available at: www.noveltyjournals.com

Modelling and Simulation of Supersonic Nozzle Using Computational Fluid Dynamics

[1]Venkatesh .V, [2]C Jaya pal Reddy

Department of Aeronautical Engineering, MLR Institute of Technology and Management, Hyderabad

Abstract: **Advances in rocket performance depend heavily upon improved and properly integrated propulsion system. This project provides a discussion about the design procedure of supersonic convergent-divergent (C-D nozzle). The C-D nozzles both conical and contour are designed on an assumption of the isentropic flow of the perfect gas. The computer code which uses the method of characteristics and the stream function to define high efficiency nozzle profile for isentropic, inviscid, ir-rotational supersonic flows of any working fluid for any user-defined exit Mach number. The designed nozzle area ratio is compared to theoretical area ratios for the selected fluid and desired exit Mach number. The nozzle geometry obtained from the code is independently checked with the commercial Computational Fluid Dynamics (CFD) code. ANSYS-FLUENT has been used to simulate flow on nozzle to verify the isentropic flow.**

Keywords: **commercial Computational Fluid Dynamics, Simulation of supersonic nozzle, ANSYS.**

I. INTRODUCTION

The work outlined in this report is to design a supersonic convergent-divergent (CD nozzle). The C-D nozzles are designed on assumption of the isentropic flow of the perfect gas. A design procedure which can determine the configuration of C-D nozzle is shown by arranging the experimental results using CFD (FLUENT). The primary design of a rocket propulsion system order is to produce maximum thrust. Nozzle is an important and basic piece of engineering hardware associated with propulsion and the high speed flow of gases. In this chapter, the basic functions of a nozzle and a brief description of the kinds of nozzle are discussed. It also gives an overview of the basic concepts and the definition of CFD.

II. LITERATURE SURVEY

Presented in this section are the previous works done in supersonic nozzle design pertinent to the current investigation. Since the nozzles designed in this paper are for irrotational, inviscid, isentropic flows, only previous works dealing with these types of nozzles will be discussed. The first part of this section will deal with annular nozzles and the second part will deal with the previous work done on the nozzles.

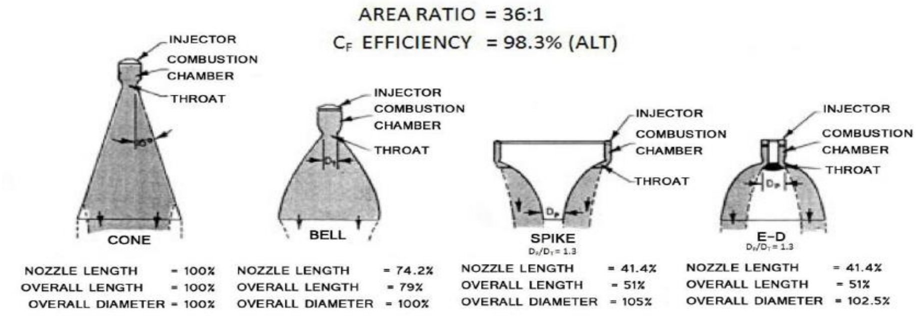

Fig 1 Rocket Nozzle Profiles

III. NOZZLE DESIGN CONCEPTS

To design a nozzle the major requirement is the magnitude of thrust to be produced by the nozzle, the altitude at which nozzle operates and properties of propellant the used. In the design of the nozzle the main constraints of the adiabatic flame temperature and the total temperature at the inlet of the nozzle. The flame temperature is known by the type of propellant used and the pressure is obtained by rate at which the propellant is burned. The properties of the propellant to be known are its molecular weight and any one of the its specific heat at constant pressure or constant volume, specific heat ratio

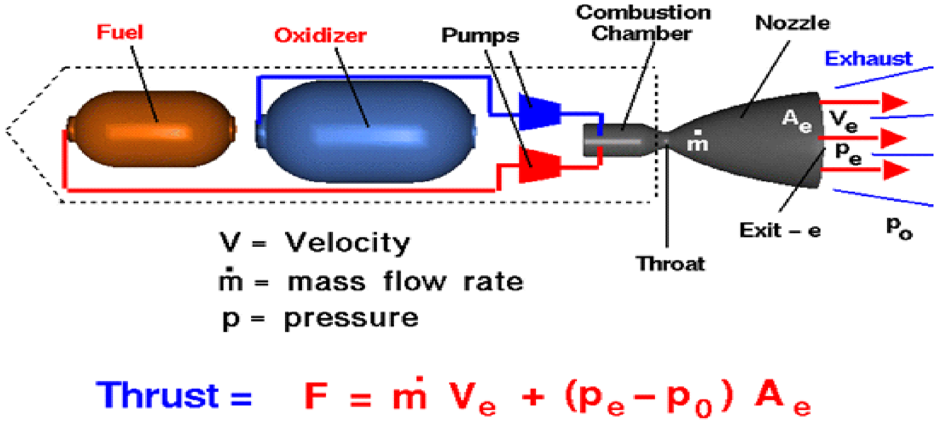

Fig 2 Thrust equation

These equations can be used to find the properties of non-isentropic flows without error because the total or stagnation properties at a state point depends only on the local temperature and the local Mach number and not upon the flow process

$$\frac{p_0}{p} = (1 + \frac{\gamma - 1}{2} M^2)^{\frac{\gamma}{\gamma - 1}}$$

$$\frac{T_0}{T} = \left(\frac{a_0}{a}\right)^2 = 1 + \frac{\gamma - 1}{2} M^2$$

$$\frac{\rho_0}{\rho} = (1 + \frac{\gamma - 1}{2} M^2)^{\frac{1}{\gamma - 1}}$$

IV. DESIGN METHODOLOGY

After getting familiarized with the concepts of the nozzle, let us now get into detail of the design procedure. Therefore this chapter gives a main focus on the design procedure of the different kinds of nozzles. This chapter relates to the application of the above mentioned thermodynamic relations and the parameters required to design nozzle. It mainly consists of designing of a Conical and Contour nozzle.

Design of complete Nozzle: Supersonic nozzles are generally specified in terms of the cross sectional area of final uniform flow A and the final mach number M. The nozzle-throat area is obtained by the 1D flow equation, the shortest nozzles that may be designed by the method of reported are those without a straight-walled section. The straightening part immediately follows with the expanding part. The purpose of method of characteristics is to illustrate the design of a supersonic nozzle by the method of computation with the weak waves.

Consider a supersonic nozzle as shown in figure. The subsonic flow in the convergent portion of the nozzle is accelerated to sonic speed at the throat. Generally, because of the multi-dimensionally of the converging subsonic flow, the sonic line is gently curved. We assume the sonic line to be straight at the throat in most of the applications. In the divergent portion downstream of the throat, let ϴw be the angle at any point on the duct wall. The portion of the nozzle with increasing ϴw is called the expansion section, where expansion waves are generated and propagate in the downstream direction, reflecting from the opposite wall. At a particular point where the ϴw is maximum, there is an inflection of the duct in the wall contour.

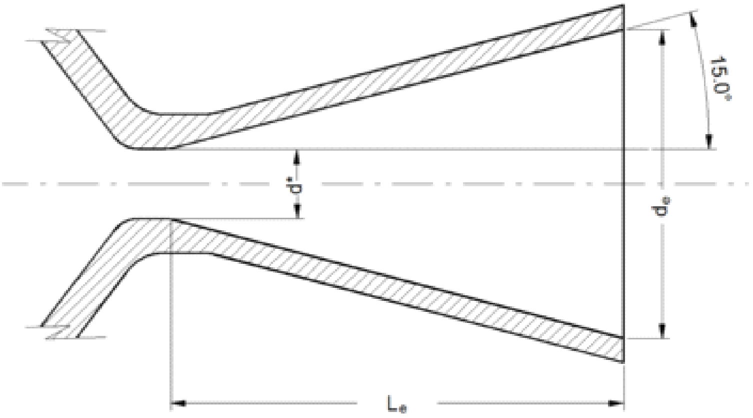

Fig 3. Conical nozzle

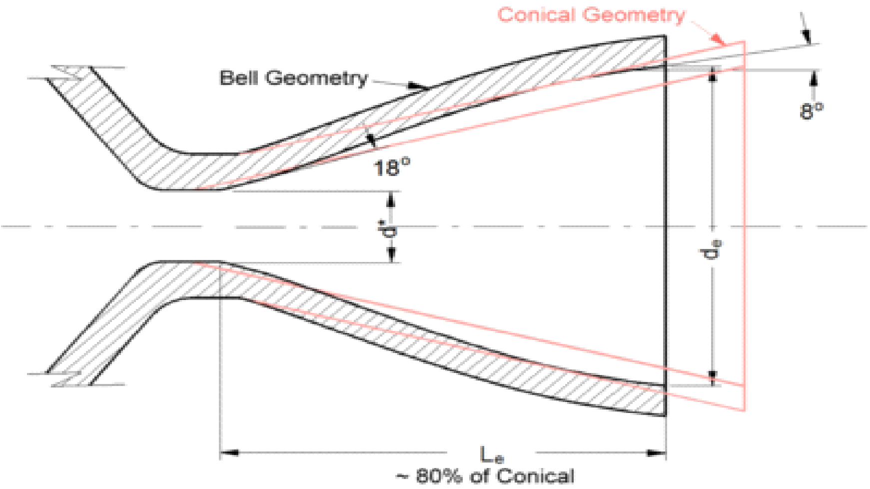

Fig 4. Contour nozzle

V. COMPUTATIONAL STRUCTURE

This chapter primarily describes the assumptions made while designing. It also gives an outline of the kinds of direction of flows .These are the analytical results using the isentropic flow equations of pressure, temperature, density and mach number.

Assumptions:

- For simplicity, the combustion gas is assumed to be an ideal gas.
- The gas flow is isentropic (i.e., at constant entropy), frictionless, and adiabatic (i.e., there is little or no heat gained or lost)
- The gas flow is constant (i.e., steady) during the period of the propellant burn.
- The gas flow is along a straight line from gas inlet to exhaust gas exit (i.e., along the nozzle’s axis of symmetry
- The gas flow behavior is compressible since the flow is at very high velocities.

Meshing:

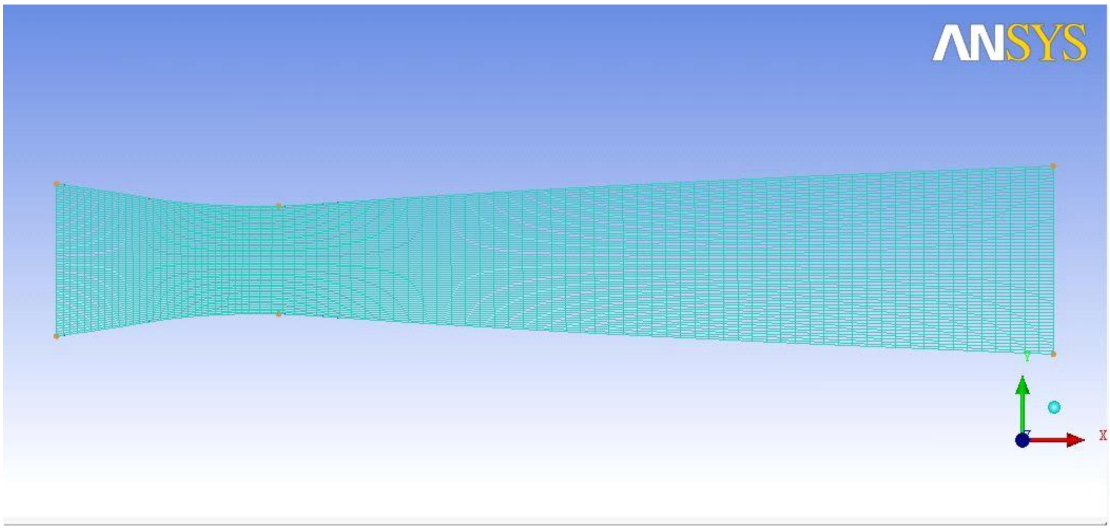

Fig 5 Mesh for Conical nozzle

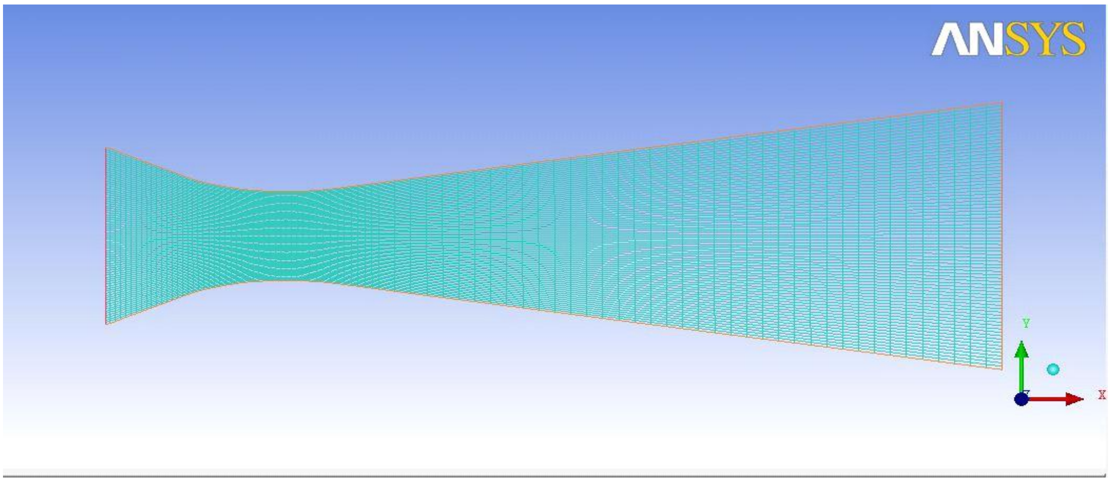

Fig 6. Mesh for Contour nozzle

After the meshing is done we have to specify the boundary condition to the flow domain. In order to do that, move to the zones command where the boundary types are assigned such as inlet, outlet, wall, axis and so on. After the boundary conditions are specified, the mesh is imported to FLUENT in 2D-mesh form.

VI. RESULTS AND DISCUSSION

If we compare the analytical results with the numerical results at exit of both the nozzles conical and contour, the resulted values are as shown below.

Table 1. Comparison of results

Analytical results	Numerical results
Me (for conical) = 2.526 Me (for contour) = 3.154	Me (for conical) = 2.04e Me (for contour) = 2.64e
Pe (for conical) = 89457 Pe (for contour) = 98547	Pe (for conical) = 90000 pascals Pe (for contour)=99955 pascals
Te (for conical) =165 Te (for contour) = 125	Te (for conical) = 1.6e+02 Te (for contour) =1.25e+02
Density (for conical) = 2.2 Density (for contour) = 1.2	Density (for conical) =2.54e-01 Density (for contour) = 1.3e-01

Conical Nozzle: The pressure contours as shown in 5.8 gives us the variation of static pressure across the nozzle. The pressure decreases from inlet to outlet of the nozzle, during which pressure energy is converted into kinetic energy. In converging section the velocity increases and mach number reaches 1 at the throat and it increases in the divergent section until the exit of the nozzle at the expensive of pressure and temperature. We can also use the variation of static pressure along the nozzle. The temperature at the inlet is maximum because the combustion gases are high temperature and it decease along the nozzle due to expansion.

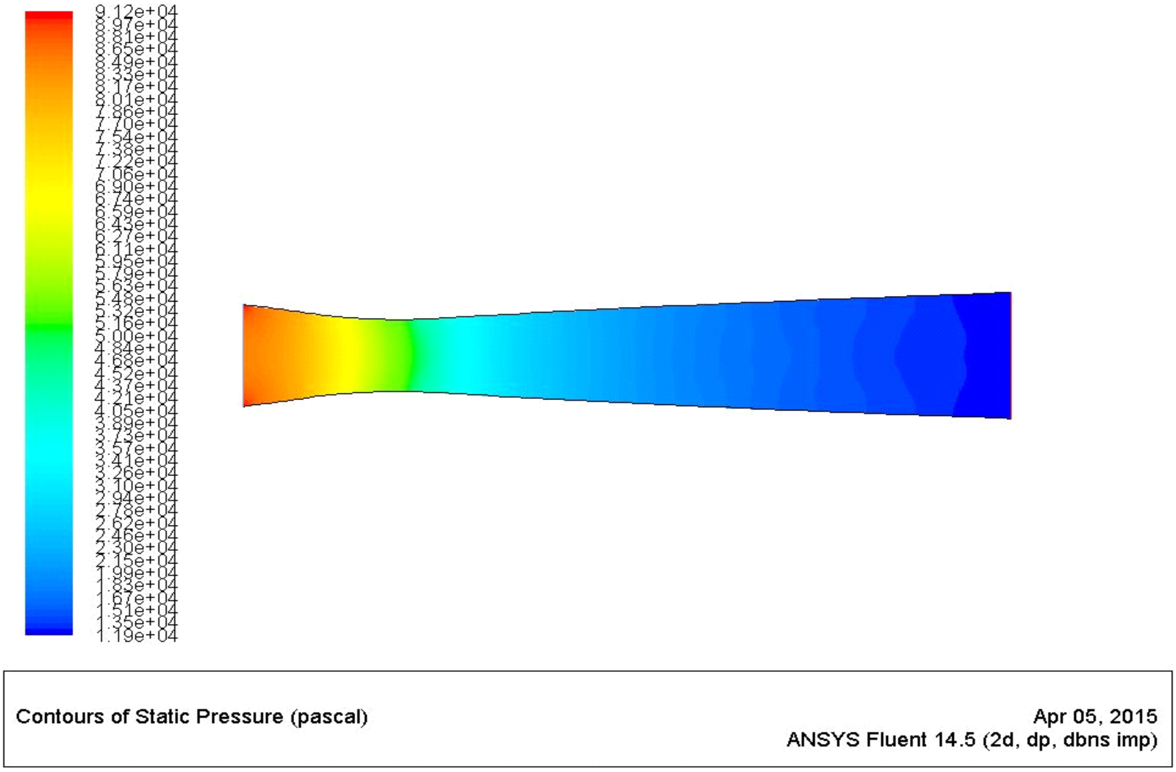

Fig 7. Variation of static pressure

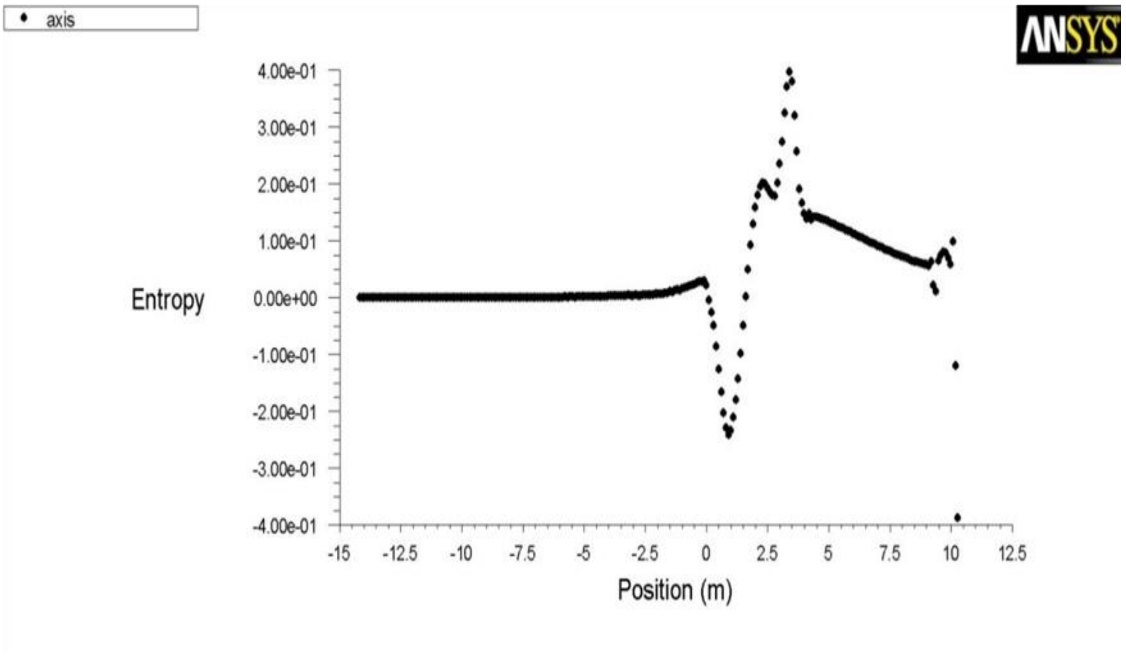

Fig 8 Entropy Plot

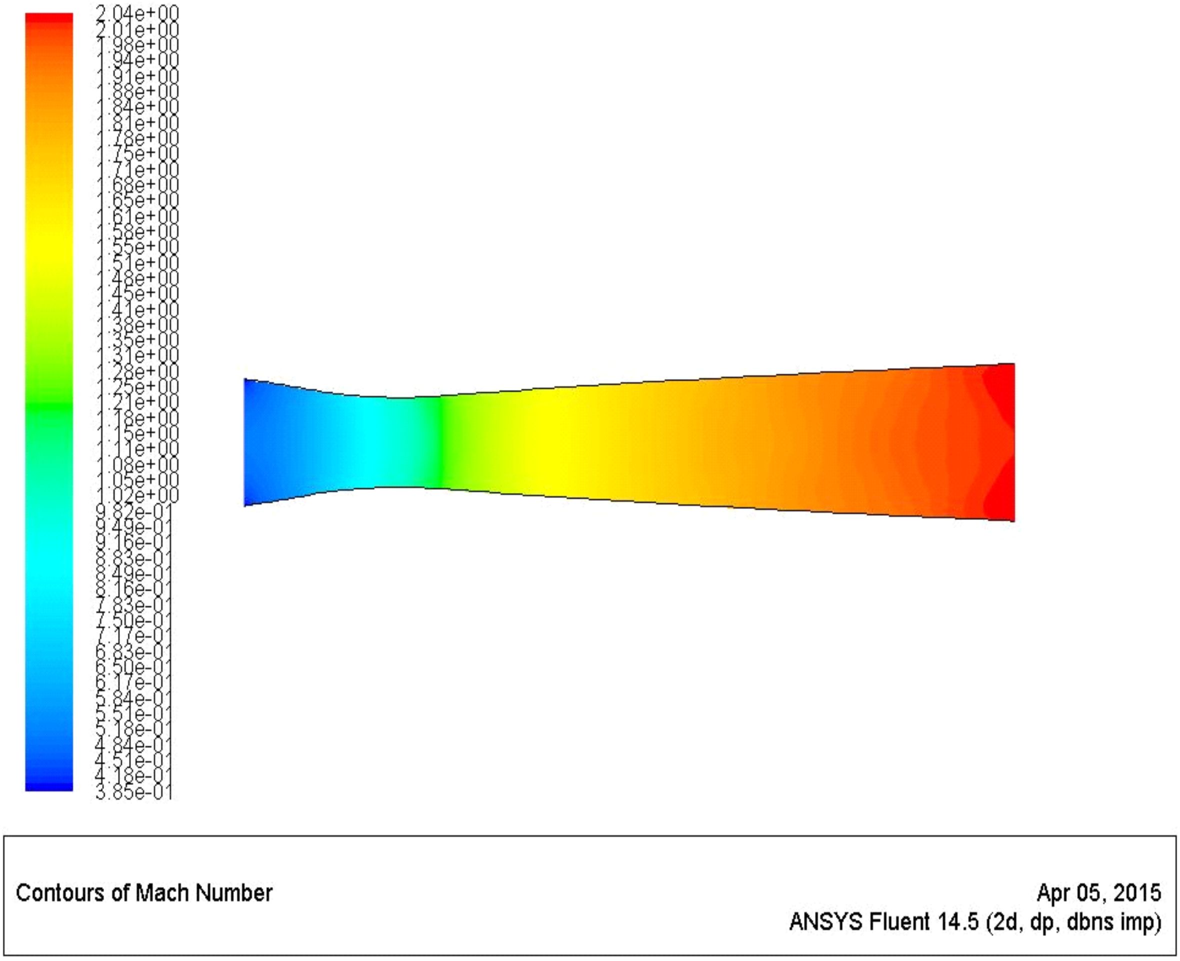

Fig 9. Contours of Mach number

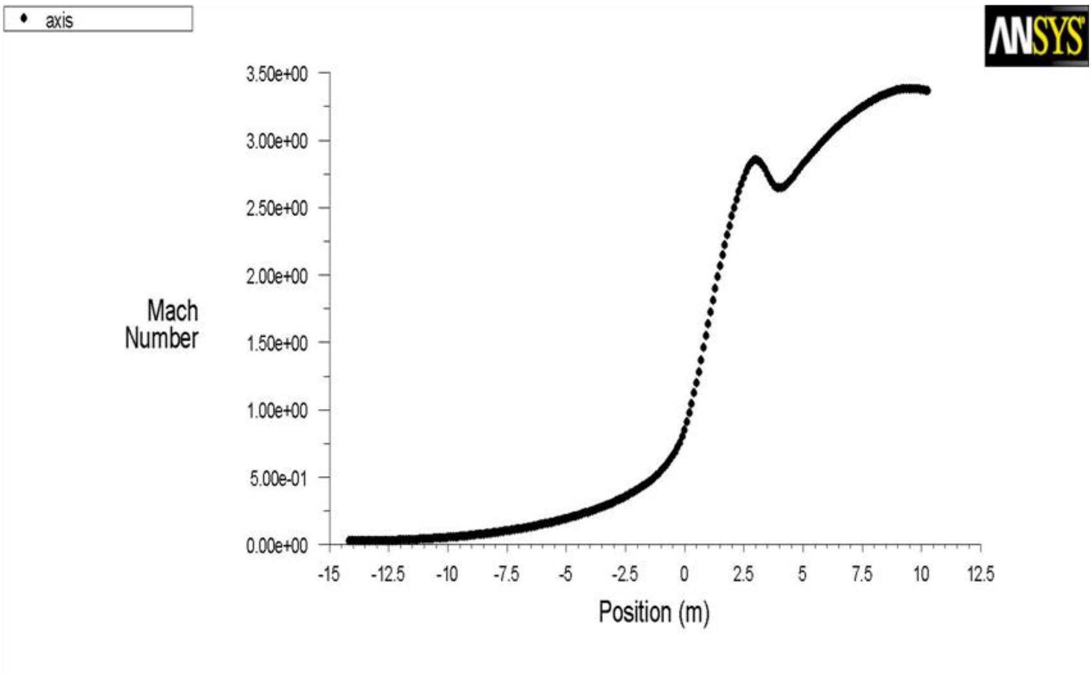

Fig 10. Mach number Plot on axis

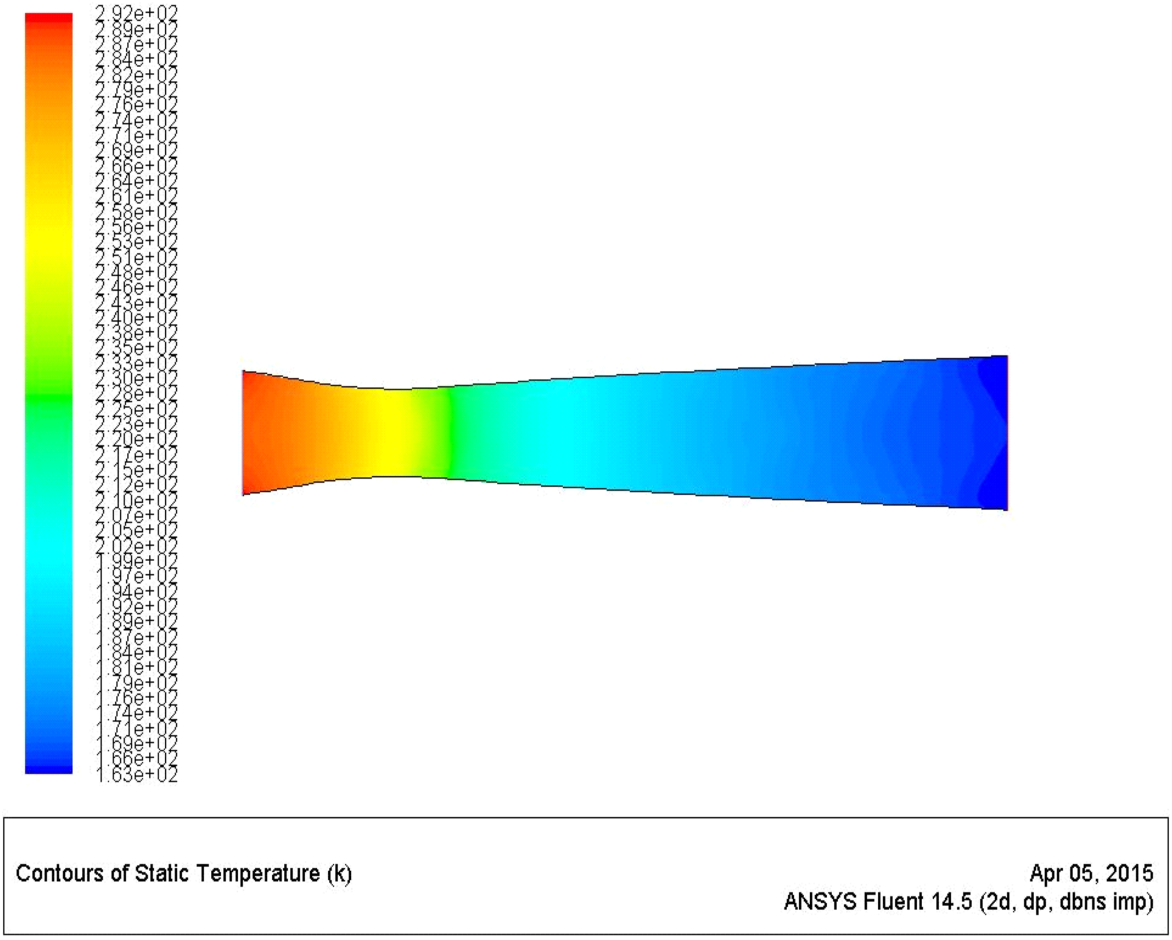

Fig 11 Contour of static temperature

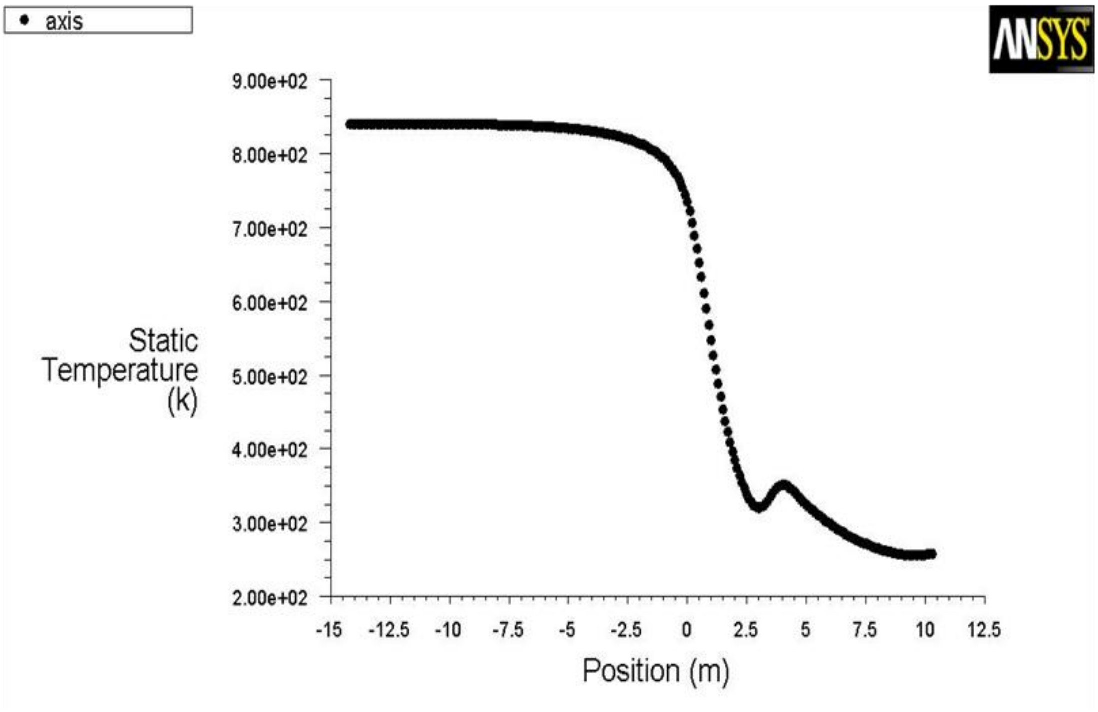

Fig 12 Static Temperature Plot

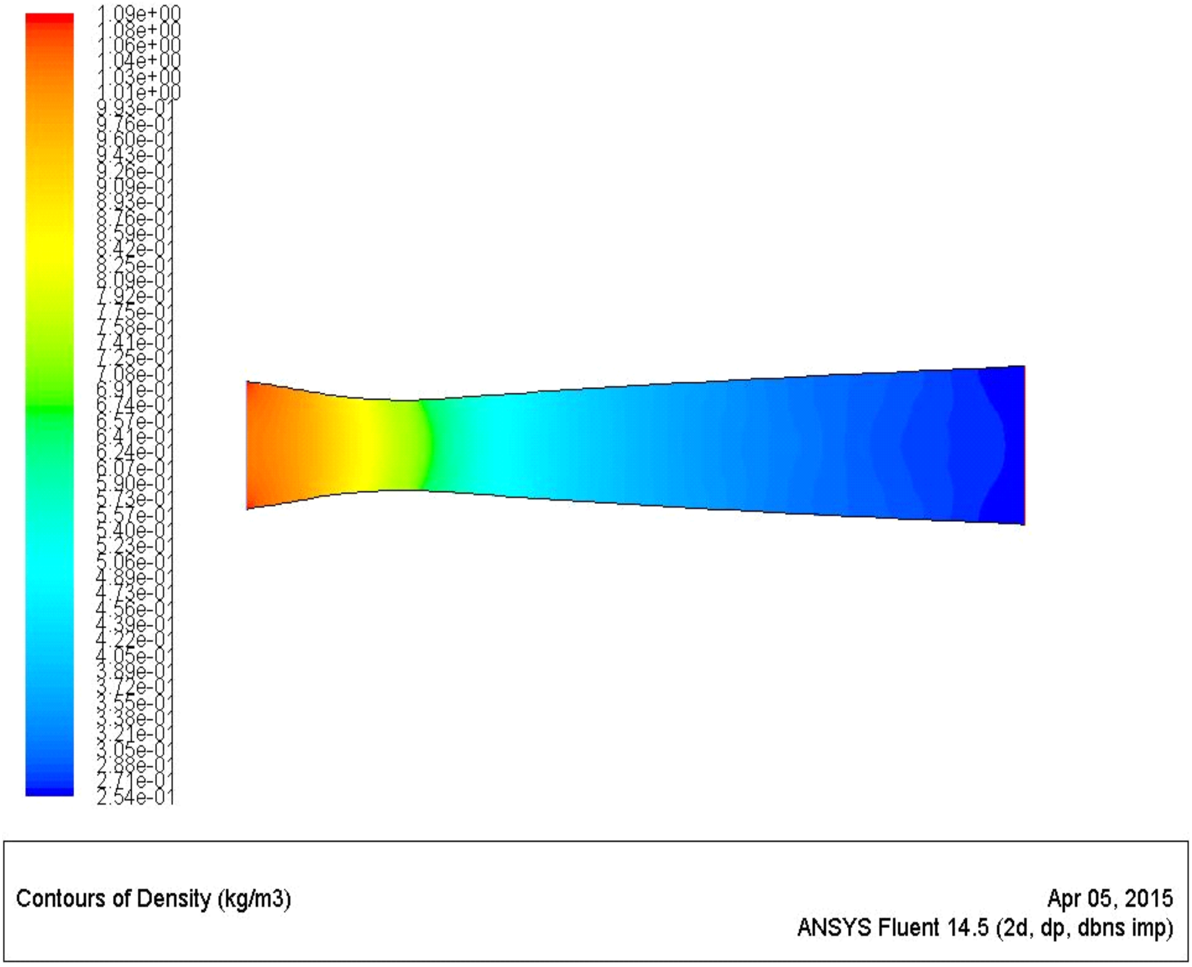

Fig 13. Contours of density

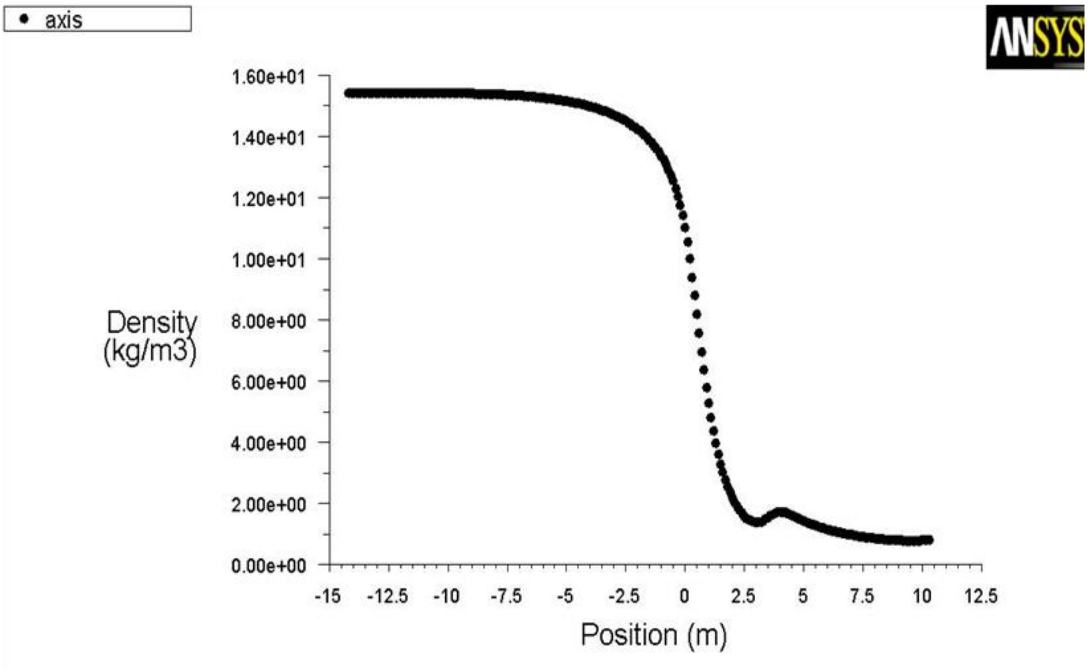

Fig 14. Density Plot

Contours: Bell nozzle: The variation of static pressure, temperature and mach number are shown in figures below. In contour nozzle, the loss of thrust component is less when compared to conical nozzle and this can be seen in mach number contour that mach number is maximum at axis of exit section. The velocity is maximum at the axis and it decreases as we move towards wall. The variation of static temperature is minimum at the axis of exit section than the wall.

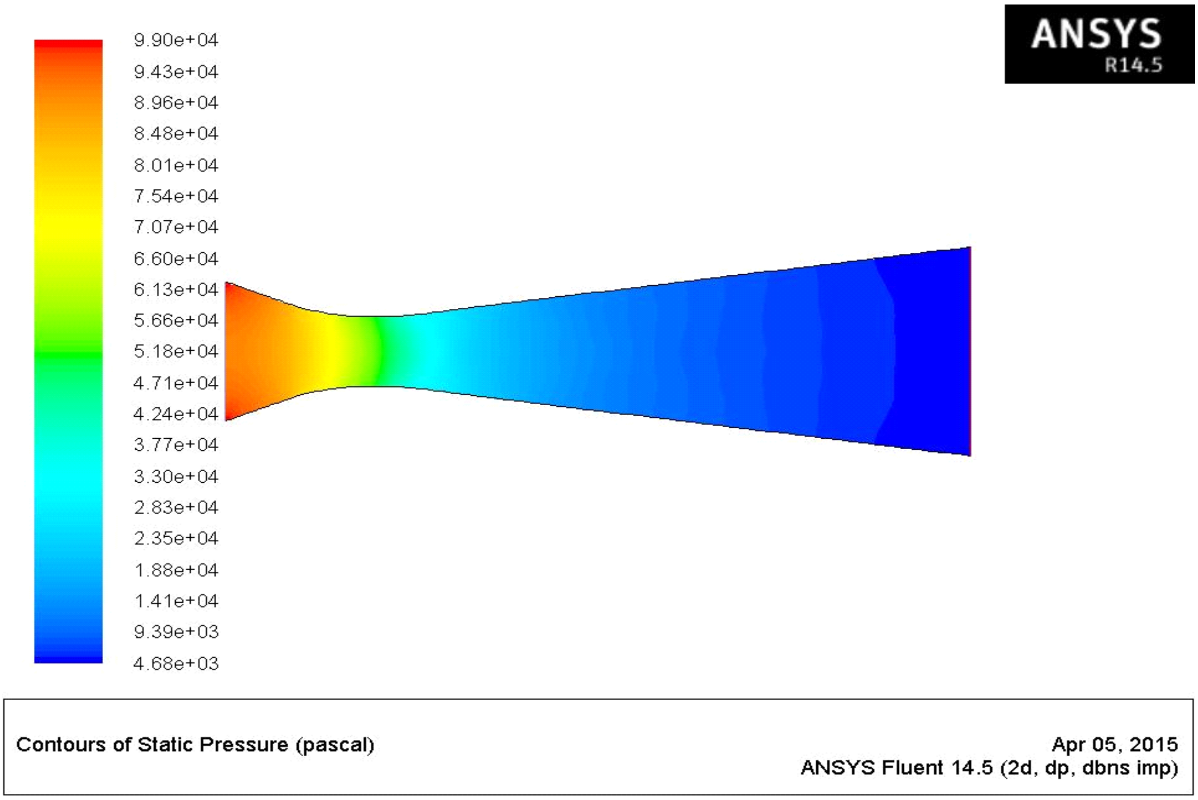

Fig 15 Pressure contour

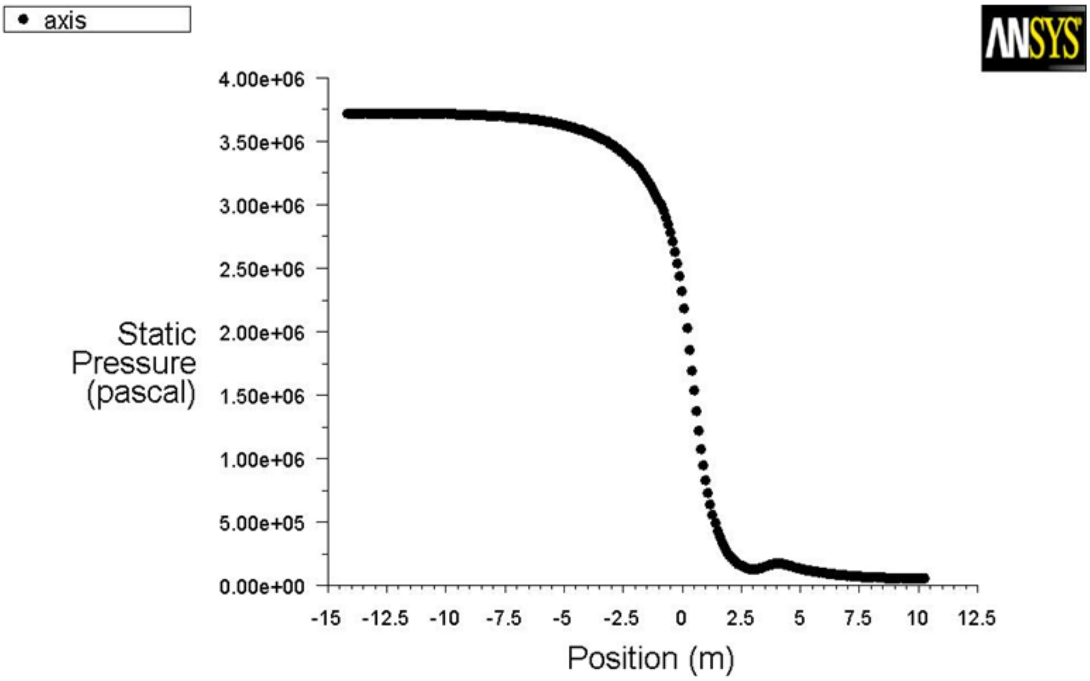

Fig 16. Pressure Plot

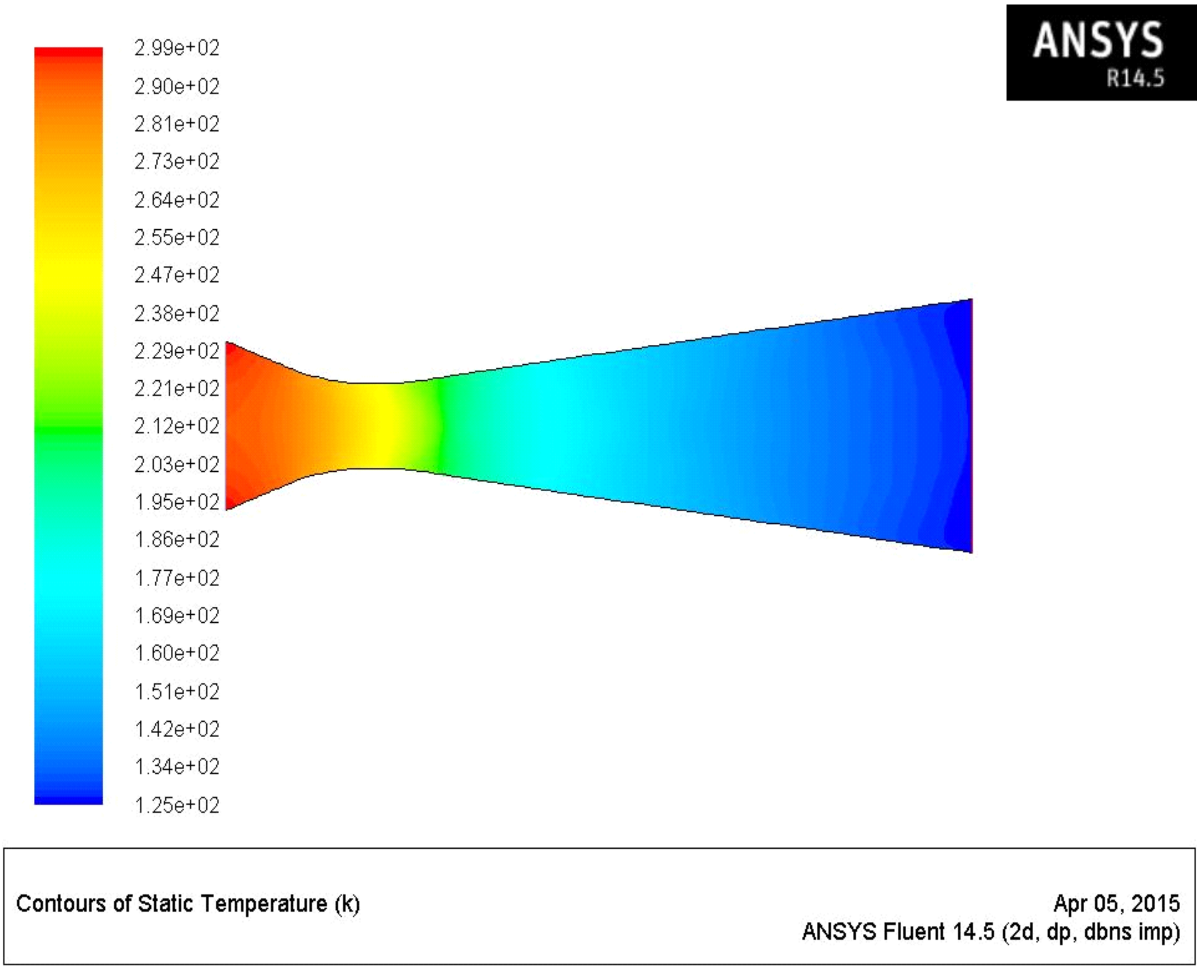

Fig. 17 Temperature contour

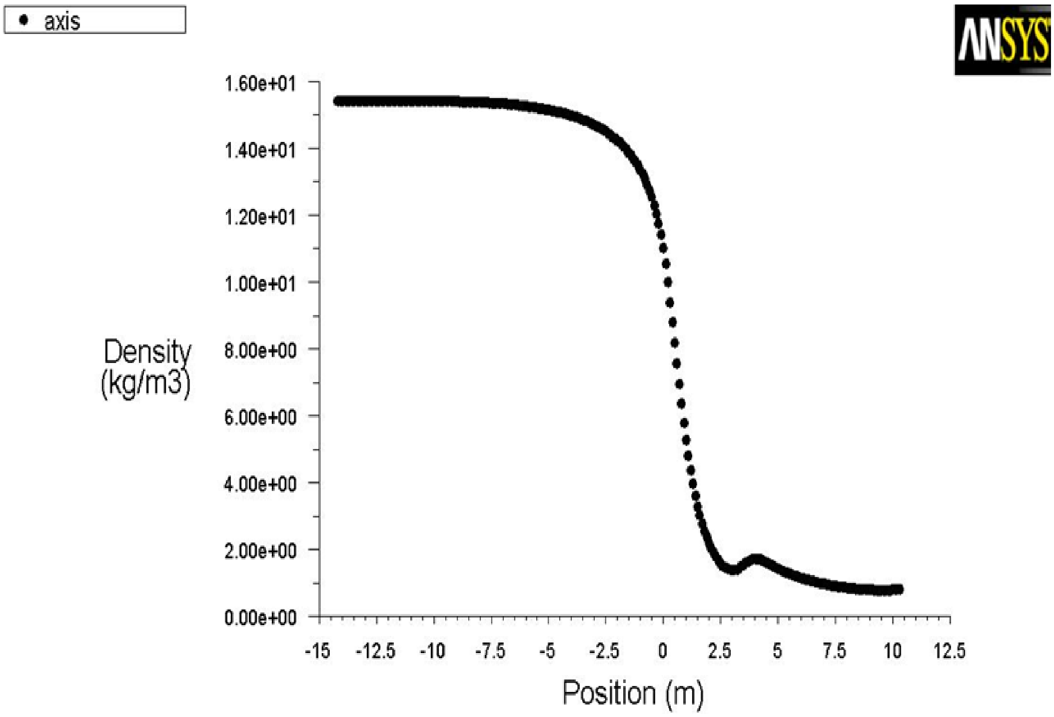

Fig 18 Static Temperature Plot

VII. CONCLUSIONS

1. A Convergent-Divergent nozzle is designed on an assumption of Quasi-One dimensional isentropic flow. Conical nozzle has been designed in the modelling software. Along with this conical nozzle, a designed contour nozzle has been analyzed using CFD (FLUENT).

2. Fluent is utilized to simulate the transient gas flow by a coupled explicit solver and it gives a 2-D result. An overall first order and second order scheme is employed spatially and temporally. Simulated Pressure histories, Temperature histories and Mach number distributions agree well with the corresponding reported static and pilot pressure measurements. Comparison of the Pressure ratio, Density ratio, Temperature ratio and Mach numbers are done between the analytical and fluent output.

3. From the report, it can be observed that the Contour nozzle gives a greater expansion ratio comparatively to a conical nozzle. Thus a Conical nozzle has to be used at sea-level and a Contour nozzle has to be used at a higher altitude since greater expansion ratio is required at a higher altitude for a given length. A conical nozzle has a simple geometry and easy to fabricate, whereas a Contour nozzle has a complex geometry and is difficult to fabricate.

ACKNOWLEDGEMENT

I owe a debt of gratitude to Dr. JayaPal Reddy at MLR Inst of Tech & Management for his valuable suggestions, vision and foresight which inspired me to conceive this project.

I express my sincere gratitude to Dr. K V Reddy, Principal and Dr. D. Muppala, Head of the department of Aeronautical department at MLR Institute of Technology and Management, for their encouragement in pursuing the project. Also I am very much thankful to T. Naganna, and Nirmith Kumar Mishra, Associate Professors for Aeronautical department for their support through and through.

REFERENCES

[1] Anderson JD, [2001], Fundamentals of Aerodynamics, 3rd Edition, pp.532-537, pp.555-585.

[2] Anderson JD, [1982], Modern Compressible Flow with Historical, pp.268-270, pp.282-286.

[3] Shapiro, AH, [1953], The Dynamics and Thermodynamics of Compressible Fluid Flow, Vol.1, pp. 294-295.

[4] Shapiro, AH, [1953], The Dynamics and Thermodynamics of Compressible Fluid Flow, Vol.2, pp. 694-695.

[5] Farley John M, Campbell Carl E, [1960], Performance of several characteristics exhausts nozzles, NASA Lewis Research center.

[6] Angelino, G., Oct. [1964], "Approximate Method for Plug Nozzle Design", AIAA Journal, Vol. 2, No. 10, pp. 1834-1835.

[7] J.Reid, [1964], An Experiment on Aerodynamics nozzles at Mach 2, Aeronautical research council reports & memoranda, R & M. No 3382.

[8] G T Galesworthy, JB Robert and C Overy [1966], the Performance of conical convergent divergent nozzles of area ratio 2.44 and 2.14 in external flow. Aeronautical Research Council CP No 893.

[9] TV Nguyen and JL Pieper [1996], Nozzle separation prediction techniques and controlling techniques, AIAA paper.

[10] Gerald Hagemann, [1998], Advanced Rocket Nozzles, Journal of Propulsion & Power, Vol.14, and No.5.

[11] Dale, D., Kozak, J., Patru, D., [2006], "Design and Testing of a Small Scale Rocket for Pico- Satellite Launching", Rochester Institute of Technology METEOR Project, Senior Design Project 06006.

[12] Nicholas J Georgiadis, Teryn W DalBello, Charles J Trefny, and Albert L. Johns,[2006],Aerodynamic Design and Analysis of high performance nozzles for Mach 4 Accelerator Vehicles.

[13] Taro Shimizu, Manatoshi Kodera, Nobuyunki Tsuboi, [2008], Internal & External flows of rocket nozzle, Journal of Earth Simulator, Vol 9, pp.19-26.

[14] T.S.Leu, C.T.Wang, J.M.Sun, [2010], Optimal design & operation on convergent divergent nozzle type no moving parts valves in micro channel, Journal of Mechanics, Vol 26 No.3.

[15] G Satyanarayana, Ch Varun, SS Naidu, [2013], CFD Analysis of Convergent Divergent Nozzle, ACTA Technica Corviniensis Bulletin of Engineering Tome VI Fascicule 3, ISSN 2067 – 3809.

[16] Bijju Kuttan P, M Sajesh, [2013], Optimization of divergent angle of a rocket engine nozzle using CFD, THE IJES Vol 2 issue 2 pp. 196-207.

[17] Balaji Krushna P, P Srinivas Rao, B Bala Krishna, [2013], Analysis of Dual Bell Rocket Nozzle using CFD, IJRET, Vol.2 issue 11.

[18] Gutti Rajeswara Rao, [2013], Flow analysis in a convergent divergent nozzle using CFD, IJRME, ISSN Online: 2347-5188.

[19] Nazar Muneam Mahmood, [2013], Simulation of back pressure effect on behaviour of convergent divergent nozzle, Diyala Journal of Engineering Sciences, Vol. 06 No. 01 pp 105-120.

[20] Nikhil d Deshpande, [2014], Theoretical & CFD Analysis of De Laval Nozzle, IJMPE, ISSN 2320-2092

[21] Ralf H.Stark, Flow Seperation in Rocket Nozzles-A simple criteria, AIAA, pp.1-8.

[22] Sutton GP, Rocket Propulsion Elements, 7th Edition.