International Journal of Innovative Research in Science and Engineering
Vol. No.3, Issue 01, January 2017
www.ijirse.com
ISSN (O) 2454-9665
ISSN (P) 2454-0663

FLOW ANALYSIS OF ROCKET NOZZLE USING METHOD OF CHARACTERISTICS

Dipen R. Dangi[1], Parth B. Thaker[2], Atal B. Harichandan[3]

[1, 2, 3] Department of Mechanical Engineering, Marwadi Education Foundation, (India)

ABSTRACT

The present research work has been carried out for the analysis of bell type nozzle by using method of characteristic with different Mach number conditions. A C-code has been developed to define the nozzle geometry by using method of characteristics usually designed for De-Laval nozzle. ANSYS 14.0 has been used for the present flow analysis by considering hear Stress Transport k-ω (SSTKW) turbulence model. In this paper CFD is used to develop best geometry of rocket nozzle with consider of aero - thermodynamic property like pressure, velocity, temperature and Mach number.

Keywords: Design Of Supersonic Nozzle, Method Of Characteristic And Minimum Length Of Nozzle.

I INTRODUCTION

Performance of rocket nozzle is always less than the theoretical approach in all practical approaches. It is always essential to improve the performance of rocket nozzle. Previous research works show the De- Laval nozzle is the most efficient nozzle among all other types of nozzles. The De-Laval nozzle gives the best performance rocket nozzle as Mach number and velocity of fluid flow is continuously increasing while at same time pressure is continuously decreasing. But still some research gap is there to develop best geometry of rocket nozzle. Even today most of the engineers are work on how to develop best geometry of rocket nozzle and to achieve higher Mach number and higher velocity of fluid flow. Due to that reason authors works on the Method of Characteristics (MOC) and developed best geometry which gives possible optimum values of Mach number, velocity, pressure distribution and density distribution. Flow separation and shock occurred inside the nozzle if the geometry of rocket nozzle is improper. 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 a high velocity gas of lower pressure and temperature. A rocket nozzle is a propelling nozzle (usually of the de Laval type) used in a rocket engine to expand and accelerate the combustion gases produced by burning propellants. The exhaust gases exit the nozzle at hypersonic velocities. It is used to give the direction to the gases coming out of the combustion chamber.Pandey et al. [1] analysed rocket nozzle computationally with single, two and four inlets. It is concluded in this paper that four inlet rocket nozzle having better performance compare to single and double outlet. We can easily visualize the figure that. There is decrease in stagnation pressure near the nozzle wall due to viscous effect.

International Journal of Innovative Research in Science and Engineering
Vol. No.3, Issue 01, January 2017
www.ijirse.com
IJIRSE
ISSN (O) 2454-9665
ISSN (P) 2454-0663

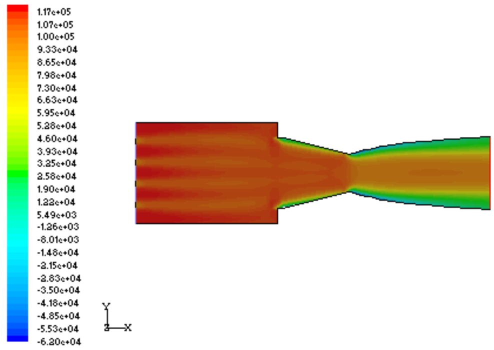


Fig. 1 Total Pressure Contour at 4 inlets

Lijo et al. [2] have performed a numerical investigation of transient flows in an axisymmetric over-expanded thrust-optimized contour nozzle is presented. Authors concluded two types of shock waves generated inside the nozzle. One of them is free shock separation and second one is restricted shock separation.

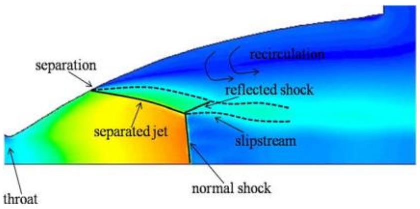


Fig. 2 Free shock separation pattern.

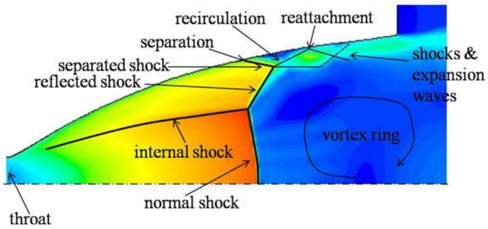


Fig. 3 Restricted shock separation.

Kbab et al. [3] performed the numerical method to develop the bell type nozzle. The method of characteristics has been used to draw the base nozzle profile. Once the profile is generated, an analysis of the thermodynamic-parameters evolution (such as: pressure, Mach number. The simulations were performed by using the k-ω SST turbulence model.

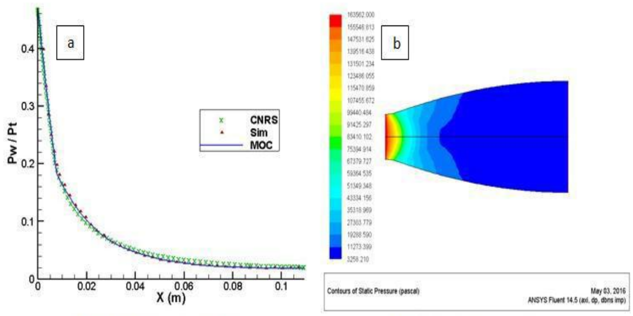


Fig. 4 Total Pressure Contour at 4 inlets

Natta et al. [4] perform the flow analysis on rocket nozzle computationally for Mach number 3 at various divergence degree of angle. Different Mach numbers are achieved at different divergence angles at different specific condition. It has been observed that geometry of nozzle has not been designed by method of characteristics for Mach number beyond 3.2. In the present research work, authors have set forth dual objectives

International Journal of Innovative Research in Science and Engineering
Vol. No.3, Issue 01, January 2017
www.ijirse.com
ISSN (O) 2454-9665
ISSN (P) 2454-0663

of generating better geometry of rocket nozzle for different Mach numbers of 3.2 or beyond it using method of characteristics and to study shock wave boundary layer interaction and flow separation behaviour.

II GOVERNING EQUATION

1. Conservation of mass:

$$\dot{m} = \rho A V = constant$$

$$\frac{d\rho}{\rho} + \frac{dV}{V} + \frac{dA}{A} = 0 \qquad (1)$$

2. Conservation of momentum:

$$\rho V dV = -dp \qquad (2)$$

3. Energy equation:

$$h_o = h + V^2/2 \qquad (3)$$

For a given stagnation state, the static properties such as Temperature, Pressure and Density at any section in the flow passage is determined using following equations, if the flow Mach number at the section is known.

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

$$P_0 = P(1 + \frac{\gamma-1}{2}M^2)^{\gamma/\gamma-1} \qquad (5)$$

$$\rho_0 = \rho(1 + \frac{\gamma-1}{2}M^2)^{1/\gamma-1} \qquad (6)$$

The flow parameters at the critical state are:

$$\frac{T^*}{T_0} = \frac{2}{\gamma+1} \qquad (7)$$

Area ratio in terms of Mach number

$$\frac{A}{A^*} = \frac{1}{M}(\frac{2}{\gamma+1} + \frac{\gamma-1}{\gamma+1})^{\gamma+1/2(\gamma-1)} \qquad (8)$$

Non dimensional mass flow rate in terms of mach number

$$\frac{\dot{m}}{a} = \sqrt{\frac{\gamma}{R}} \times \frac{P_0}{\sqrt{T_0}} M(1 + \frac{\gamma-1}{2}M^2)^{\frac{\gamma+1}{2(\gamma-1)}} \qquad (9)$$

Mach number in terms of Pressure ratio:

$$M = \left\{\frac{2}{\gamma-1}\left[(\frac{P_0}{P})^{\gamma-1/\gamma} - 1\right]\right\}^{1/2} \qquad (10)$$

III PROBLEM DEFINITION

If the length of nozzle is long then flow separation and shock wave occur inside the nozzle. Therefore, authors have used MOC to create geometry of rocket nozzle which have minimum length and maximum Mach number.

ISSN (O) 2454-9665
ISSN (P) 2454-0663

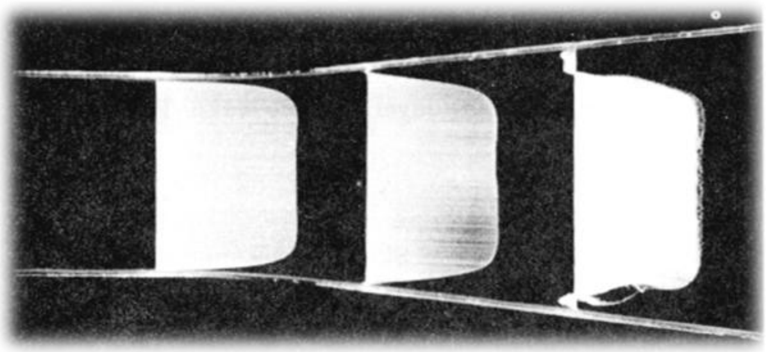


Fig. 5 Flow Separation in a Nozzle

IV NUMERICAL CONSIDERATION AND METHODOLOGY

In supersonic flow the Euler equations are hyperbolic i.e. the flow is only determined by the upstream conditions. In this case, the MOC can be used to calculate the nozzle flow field. This method is the most commonly used in the rocket nozzle for generating nozzle contours and determining loads and performances. Further for supersonic convergent divergent nozzle, it is essential to have wave free and parallel flow in the test section at the desired Mach number. An improper contour will result in presence of weak waves, which may coalesce to form a finite shock and prevent a uniform flow in the test section. Therefore, it is imperative to have a proper design of nozzle contours for generation of uniform supersonic flows.

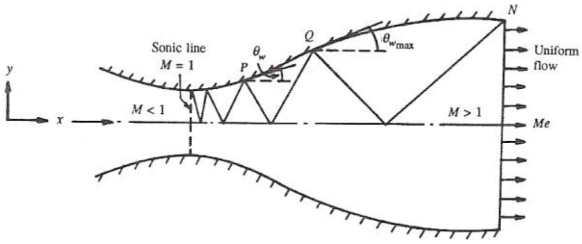


Fig. 6 Supersonic Nozzle [5]

The MOC provides technique for properly designing the contour of supersonic nozzle for shock-free, isentropic flow, taking into account the multidimensional flow inside the duct. The length L shown in Figure 3.2 is the minimum length possible for shock free, isentropic flow. If the contour is made within a length shorter than L, shocks will develop inside the nozzle.

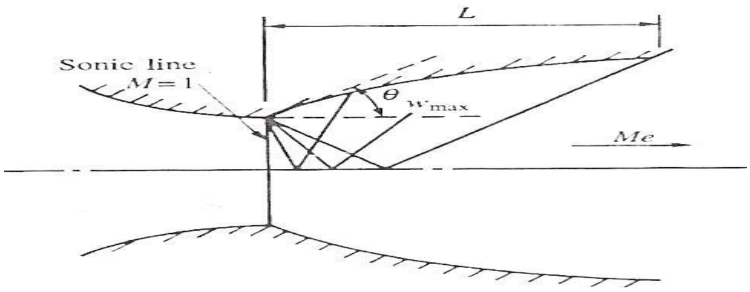


Fig. 7 Minimum length nozzle [5]

International Journal of Innovative Research in Science and Engineering
Vol. No.3, Issue 01, January 2017
www.ijirse.com
ISSN (O) 2454-9665
ISSN (P) 2454-0663

ANSYS 14.0 with certain numerical assumption has been considered for the standard nozzle geometry. A two-dimensional density based solver with sst k- turbulence model has been used for the simulation. The convergence criteria for the residuals were set to be 10^{-5}. The contours of pressure, temperature, velocity and Mach number are considered for the analysis.

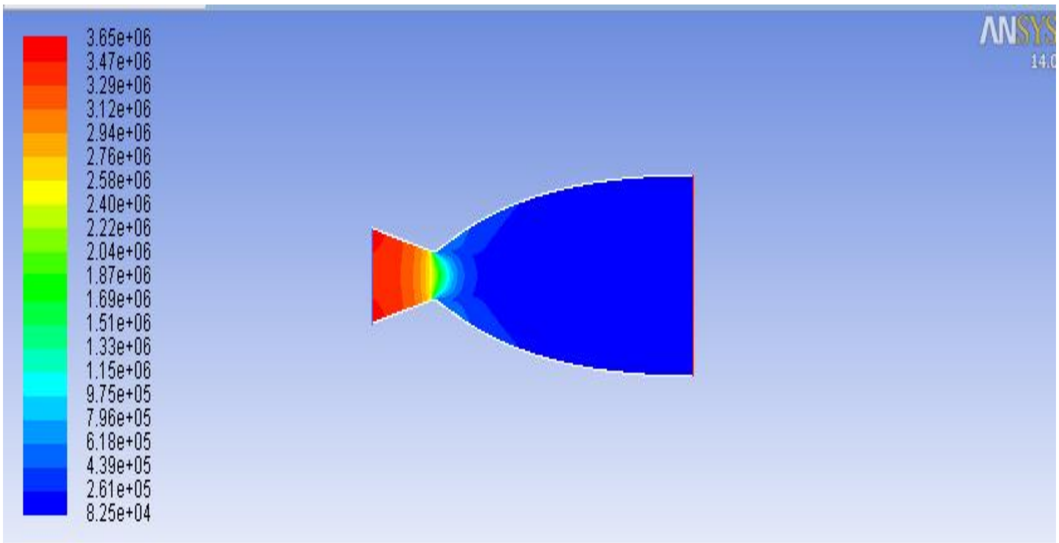
Boundary conditions are,

(Stagnation pressure)	(Stagnation temperature)
(Inlet)	(Inlet)
(Outlet)	(Outlet)

Above values are obtained from using equations (4), (5), (6) and (7).

V RESULTS AND DISCUSSION

In a rocket nozzle, pressure continuously decreases in nozzle convergent-divergent section while at same time velocity and Mach number continuously increases from convergent section to divergent section. Contour shows the behaviour of fluid flow rocket nozzle. Higher Mach number achieved at end of divergence section is the requirement to produce higher thrust from the rocket nozzle engine. In this paper, contours of different flow properties and Mach number are plotted and discussed below. To get better idea of flow property variations, the graphs are plotted throughout the nozzle length. Fig. 8 depicts the iso-pressure contours. As can be noted, there is no disturbance or pressure fluctuations, which corresponds to a typical isentropic flow along the diverging portion.



International Journal of Innovative Research in Science and Engineering
Vol. No.3, Issue 01, January 2017
www.ijirse.com
ISSN (O) 2454-9665
ISSN (P) 2454-0663

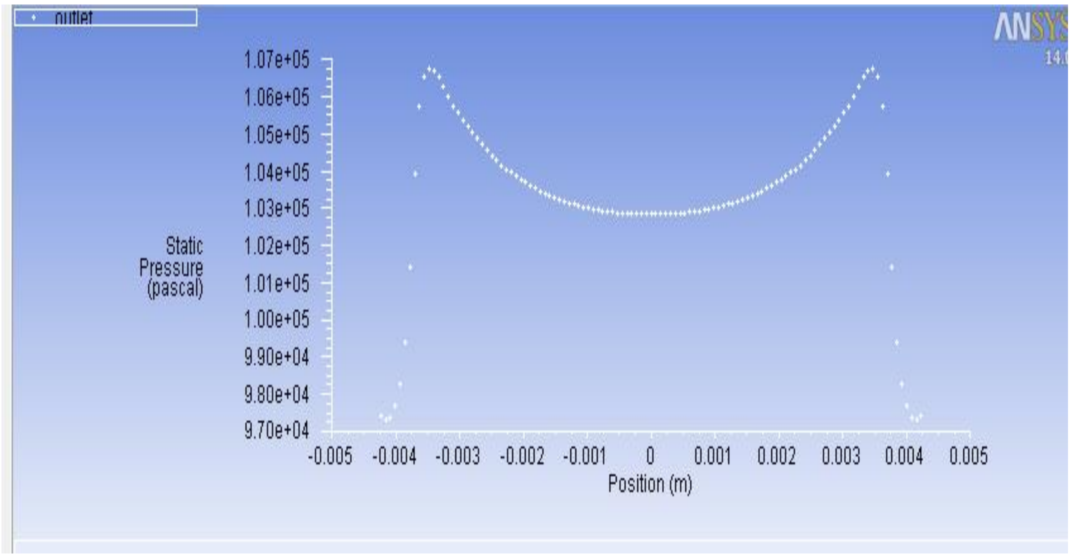
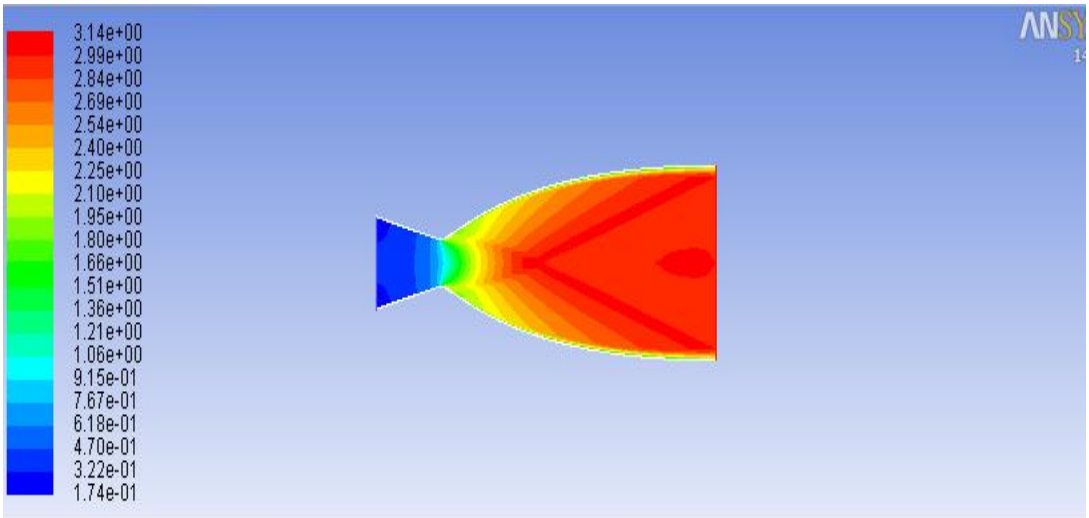
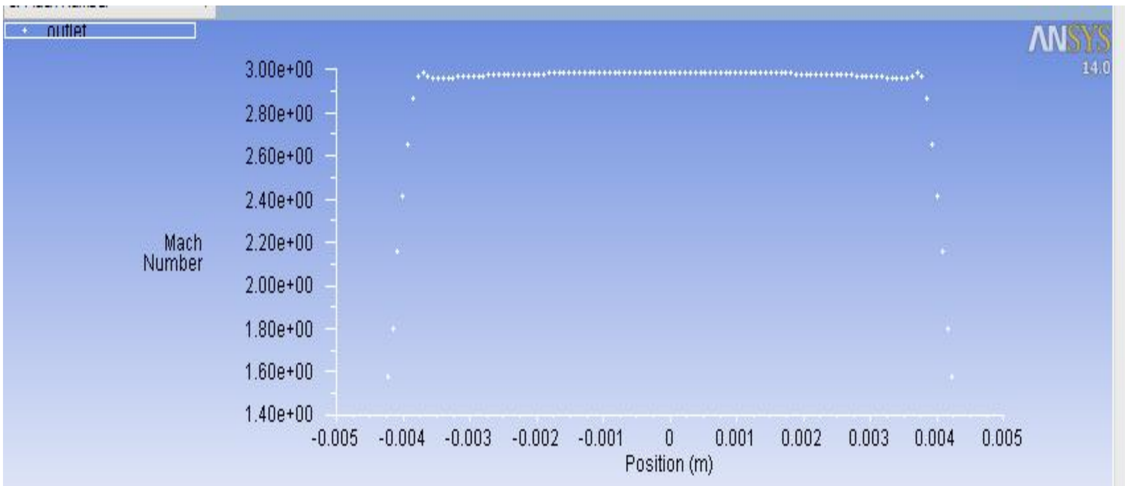


Fig. 8 Static pressure distribution in Bell Nozzle along the length of nozzle at =300 k

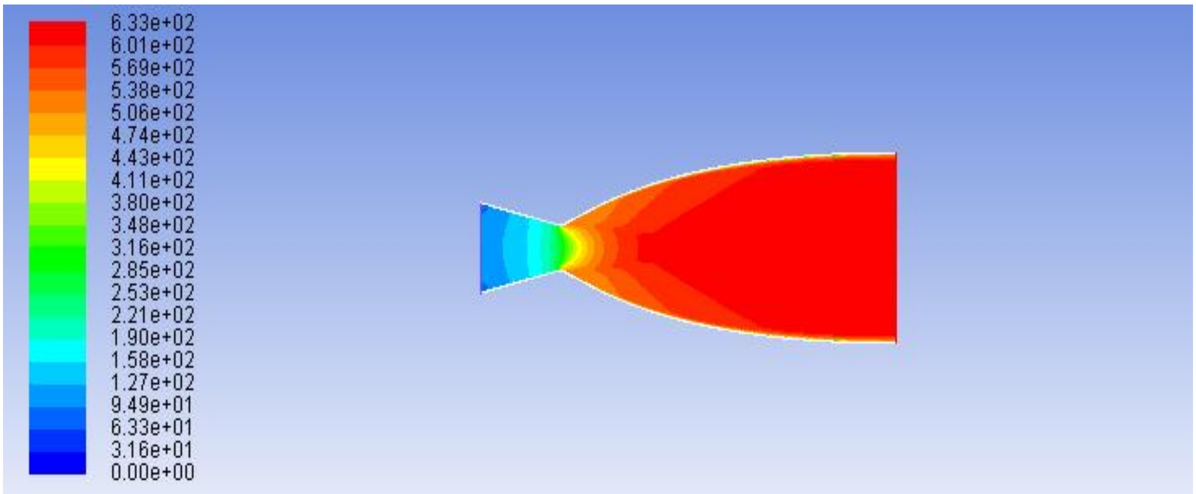




ISSN (O) 2454-9665
ISSN (P) 2454-0663

Fig. 9 Mach number distribution in Bell Nozzle along the length of nozzle at =300 k

Fig.9 depicts the evolution of the Mach number along of the base nozzle wall. It is found that Mach number increases quickly at the nozzle throat expansion region. However, in the divergent part, the Mach number grows continuously until reaching the value of the nozzle designing Mach number at the exit. In this case Mach number achieved 2.89.



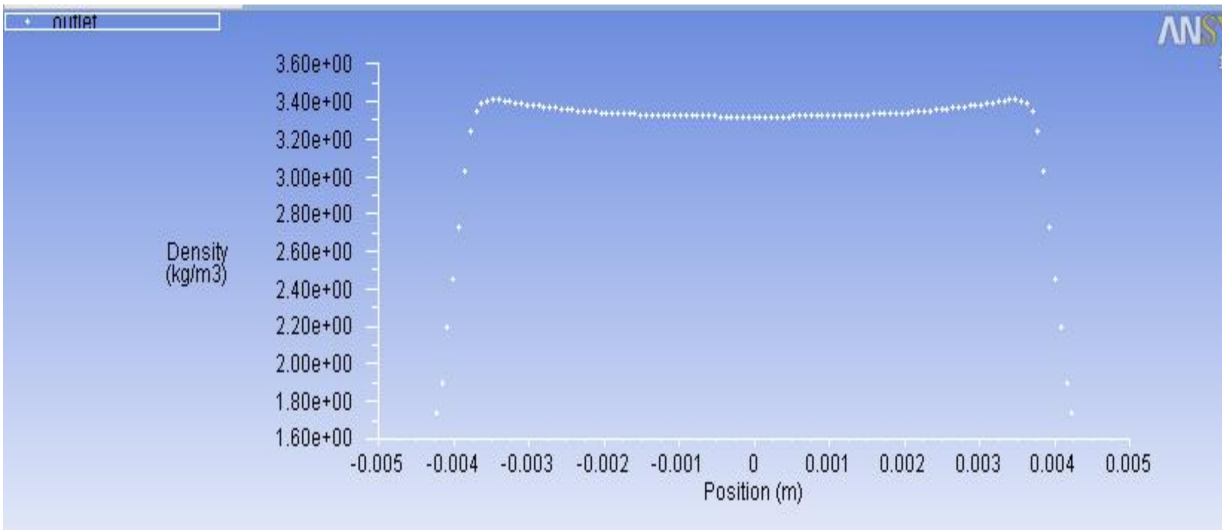


Fig. 10 Velocity contour of Bell nozzle along length of nozzle at T_0 =300K

The figure 10 shows the velocity variation of bell nozzle. The ideal gas at 105.94 m/s enters into the nozzle and leaves at 600.64 m/s at the exit. The velocity graph shows at outlet of rocket nozzle.

VI CONCLUSION

The present research work shows a judicious approach to design better geometry of bell type rocket nozzle using method of characteristic (MOC). The fluid flow is parallel to the nozzle wall immediately after the flow struck with nozzle wall. The fluid flow does not reflect for anymore times on another side of nozzle wall. As a result, the side-loads reduces on rocket nozzle wall and the performance of rocket nozzle increases. In case of Mach number 3.0 maximum Mach number and velocity achieved by simulation are 2.89 and 600.64 m/s respectively. However, Authors have observed that, by using Modified Method of Characteristic (MMOC), Mach number is achieved beyond 2.89 in case of designing of rocket nozzle at Mach number 3.0.

REFERENCES

[1] K.M.Pandey and S.K.Yadav, CFD analysis of a rocket nozzle with four inlet at mach 2.1, *International Journal of Chemical Engineering and Application, 1(4)*, 2010, 319-325.

[2] V. Lijo, H. Dong Kim, T. Setogichi and S. Matsuo, Numerical simulation of transient flow in a rocket nozzle propulsion nozzle, *International Journal of Heat and Fluid Flow, 31*, 2010, 409-417.

[3] H.kbab, M Sellam, T. Hemitouche, S.Bersheul and L. Lagab, Design and evaluation of a dual bell nozzle, *Acta Astronautica*, 2016.

[4] P. Natta, V.R.Kumar and Y.V.H. Rao, Flow analysis of rocket nozzle using computational fluid dynamics(CFD), *International Journal of Engineering Research and Applications, 2(5)*,2012, 1226-1235.

[5] E. Rathakrishnan, 2010, PHI Learning Private Limited, New Delhi.

[6] S.M.Yahya, *Fundamental of compressible flow with aircraft and Rocket propulsion*, New Age international publishers, 1982.

[7] J.D.Anderson Jr. *Computational Fluid dynamics with basic applications* McGraw-Hill company, 2002.

[8] Jan Ostlund *Flow process in rocket engine nozzle with focus on flow separation and Sides-loads*, Licentiate Thesis, Stockholm, 2002.