iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 5 **Issue:** I **Month of publication:** January 2017

DOI:

www.ijraset.com
IC Value: 13.98

Volume 5 Issue I, January 2017
ISSN: 2321-9653

International Journal for Research in Applied Science & Engineering Technology (IJRASET)

CFD analysis of De Laval Nozzle Geometry & Reverse Flow Cavitation Phenomenon

Sree Harsha Bandaru[1], Arjun Singh[2]
School of Engineering & Technology, Indira Gandhi National Open University, New Delhi

Abstract*: De laval nozzle is a convergent divergent type nozzle which has the ability to convert the chemical energy with high pressure in to kinetic energy with high velocity and low pressure. In other words the device takes in minimal power and delivers tremendous amounts of output power used in jet and rocket engines. The pressure and velocity distribution across the nozzle are very important and they entirely depend on the cross section or the geometry. Reverse flow is caused in the negative pressure gradient region and reverse flow Cavitation is very much useful in under water applications. The present study is focussed on analysis of de laval nozzle geometries and reverse flow phenomenon. The nozzle is designed and analysed in Ansys fluent. Pressure distribution and velocity values are noted down and these are compared with the theory of gas dynamics. The case of reverse flow is also simulated and contours are drawn to the available geometry.

***Keywords*: De laval nozzle , Gas Dynamics , Ansys Fluent , Reverse flow , Jet engines**

I. INTRODUCTION

The de laval nozzle contains throat where the multiplication of velocity takes place due to the minimum flow area and inlet area and exit area. The velocities attained by the jet engines are expressed in terms of Mach number and de laval nozzle should facilitate minimum pressure drop and high velocity. The flow goes from subsonic (M- Mach number <1) to Sonic (M=1) and then to the supersonic (M>1). The back pressure when lowered to a value equal to the pressure at the exit of the nozzle offers a uniform super sonic flow. When the air enters the inlet the high pressure fluid expands over the chamber and reduce its pressure to the pressure existing outside the nozzle . This is design consideration for the nozzles.

A. GOVERNING THEORY

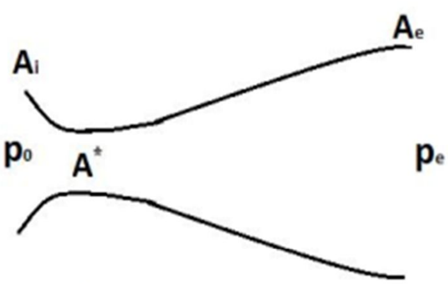


Fig. 1 Flow through convergent divergent duct.

$$\dot{m} = \rho^* u^* A^*$$

Where, ρ^*, A^* and u^* are geometric and flow properties at the throat.

For the steady flow, mass flow rate at any cross-section having geometric and flow properties as ρ, A and u will be equal to the mass flow rate of the throat. Hence,

$$\rho u A = \rho^* u^* A^*$$

$$\frac{A}{A^*} = \frac{\rho^* u^*}{\rho u} = \frac{\rho^*}{\rho_0}\frac{\rho_0}{\rho}\frac{u^*}{u} = \frac{\rho^*}{\rho_0}\frac{\rho_0}{\rho}\frac{a^*}{u}$$

$$\left(\frac{A}{A^*}\right)^2 = \left(\frac{\rho^*}{\rho_0}\right)^2\left(\frac{\rho_0}{\rho}\right)^2\left(\frac{a^*}{A}\right)^2$$

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

$$\frac{\rho_0}{\rho^*} = \left[\frac{(\gamma + 1)}{2}\right]^{\frac{1}{(\gamma-1)}}$$

$$\left[\frac{u}{a^*}\right]^2 = M^{*2} = \frac{\frac{(\gamma+1)}{2} M^2}{1 + \frac{(\gamma-1)}{2} M^2}$$

Hence the area relation can be written as,

$$\left[\frac{A}{A^*}\right]^2 = \frac{1}{M^2}\left[\frac{2}{(\gamma+1)}\left(1 + \frac{(\gamma-1)}{2} M^2\right)\right]^{\frac{(\gamma+1)}{(\gamma-1)}}$$

COMPUTATIONAL MODELLING USING ANSYS

Volume 5 Issue I, January 2017
ISSN: 2321-9653

B. Cad modelling

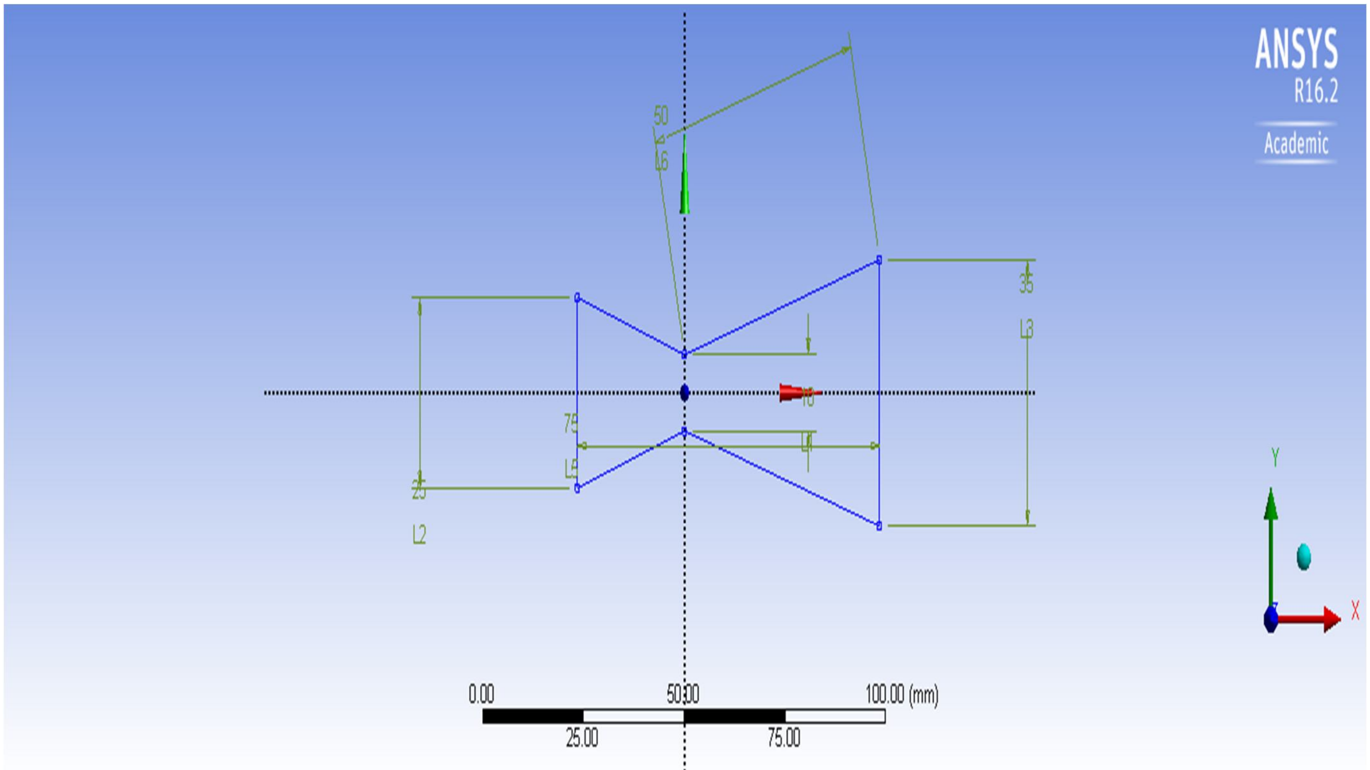


Fig 2. Cad modelling of nozzle

C. MESHING

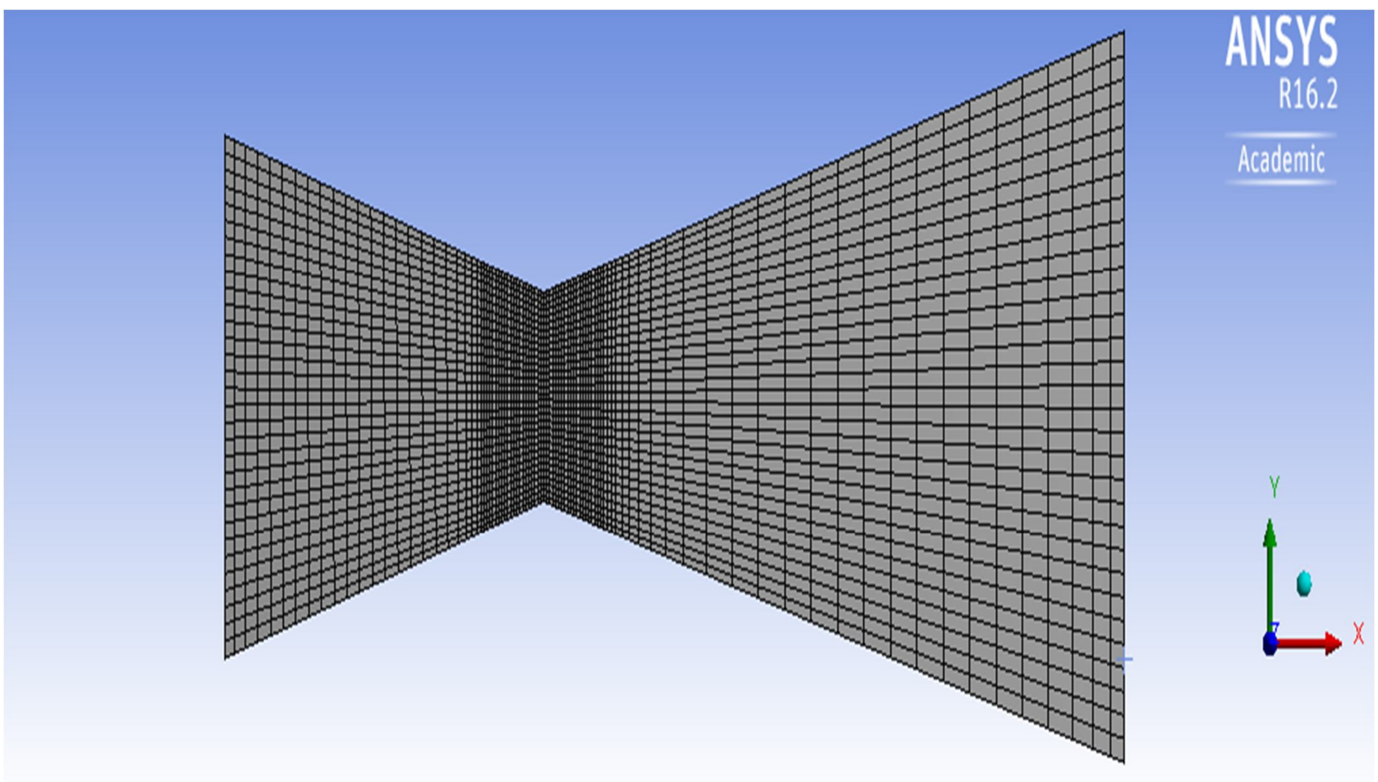


Fig 3 Meshing of C D nozzle

Volume 5 Issue I, January 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

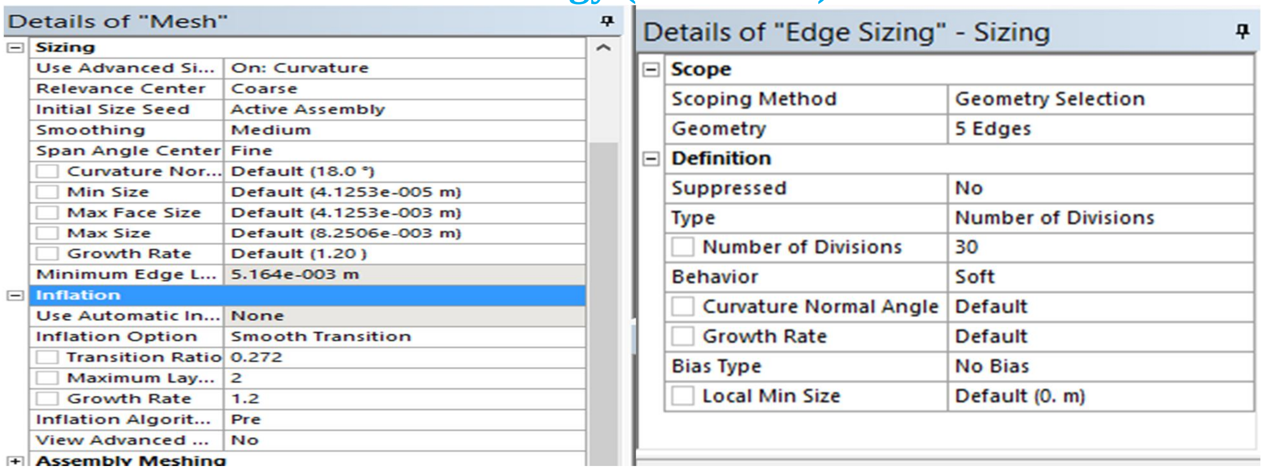


Fig 4 Meshing attributes of the nozzle

D. Pre-procesing

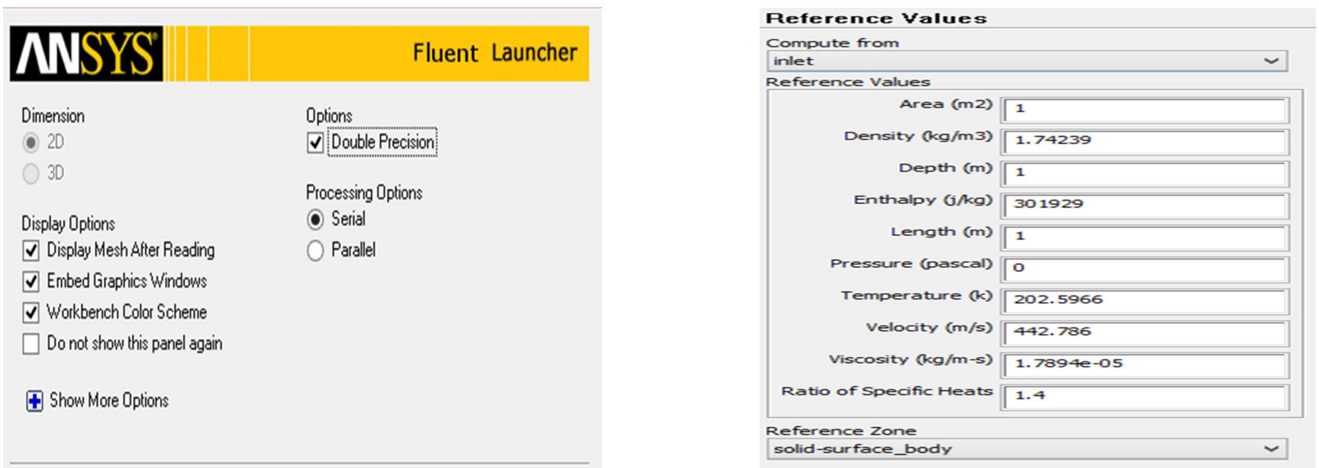


Fig 5 Input values to the model

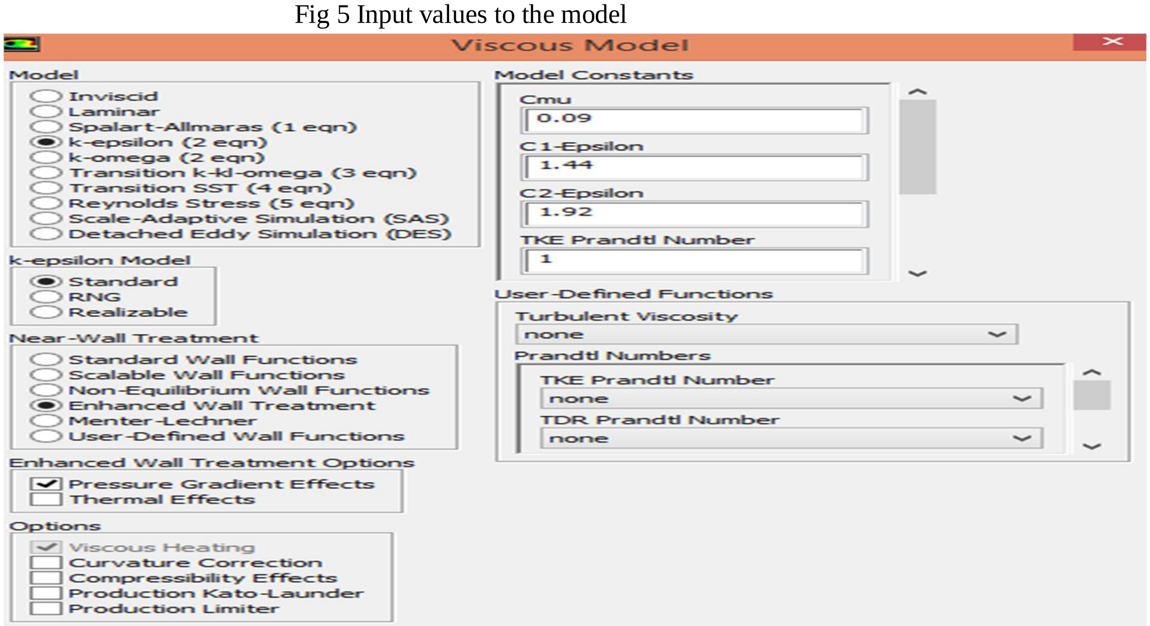


Fig 6 Mathematical modelling

Volume 5 Issue I, January 2017
ISSN: 2321-9653
International Journal for Research in Applied Science & Engineering Technology (IJRASET)

II. SIMULATION RESULTS

A. Convergence

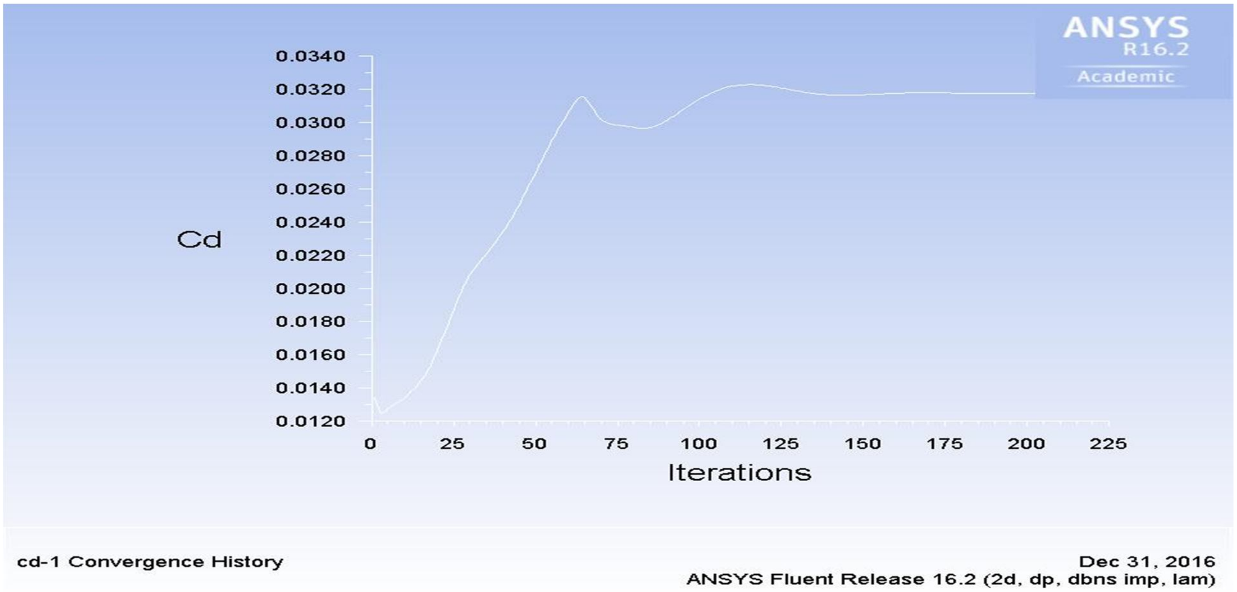


Fig 7 cd-1 Convergence plot

B. Velocity distribution

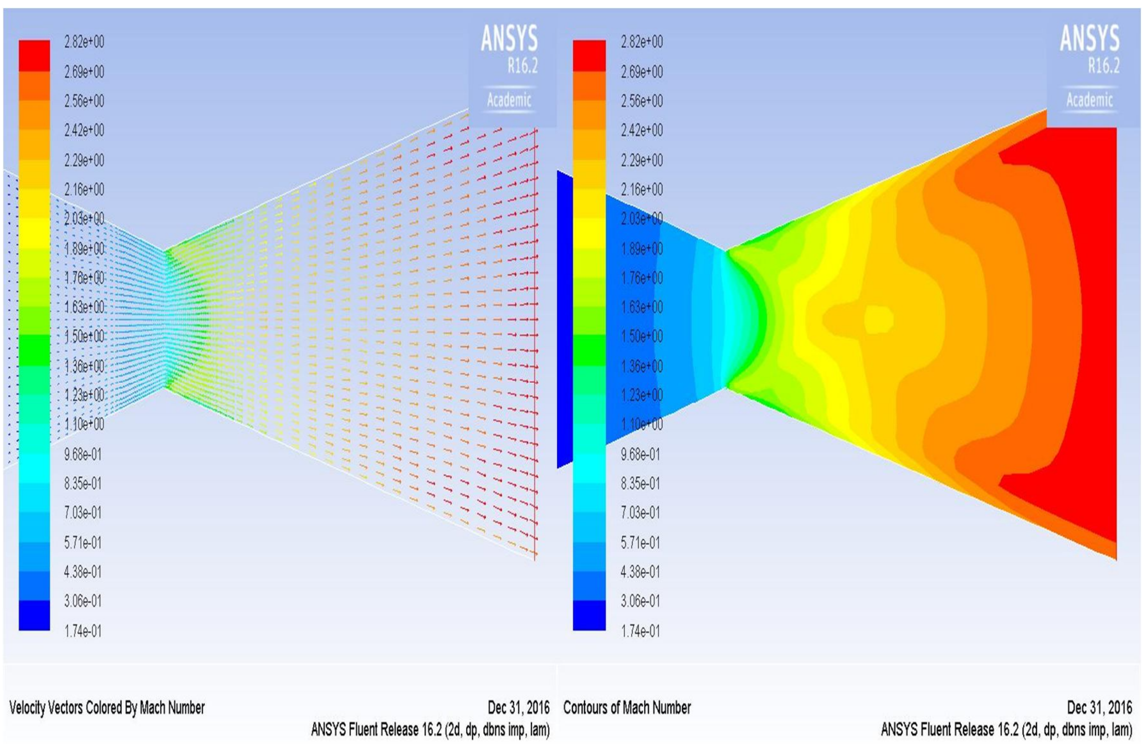


Fig 8 Contours and vectors of Mach number – Velocity

The velocity is maximum with Mach no = 2.82 at the exit of the nozzle. The throat has a Mach – 0.98 approaching the sonic flow and. This is also called as choking at the throat. The red color contour indicates the exit region velocity and the blue region indicates the low velocity profile at the inlet.

C. Pressure distribution

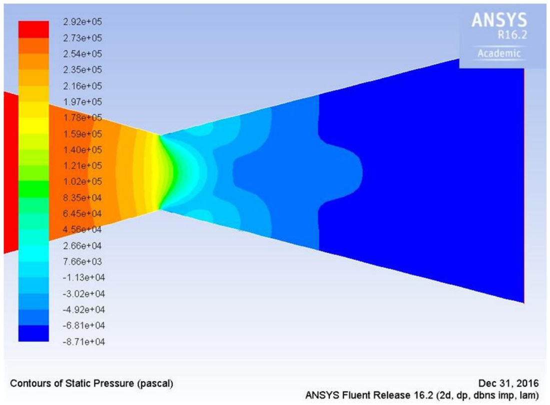


Fig 9 Static pressure contour

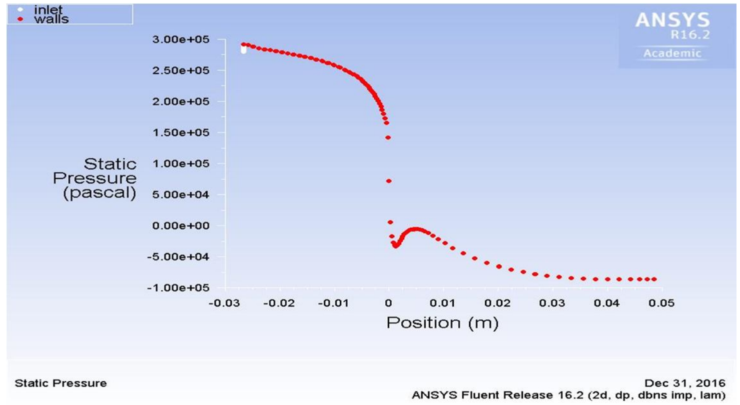


Fig 10 static pressure plot

The inlet pressure is 3 bar and outlet pressure is almost equal to the atmospheric pressure indicated by the blue contours that the pressure is decreased drastically. The plot also indicates a fall in the pressure. This is the design consideration and this is met successfully.

D. Wall shear and Reverse flow phenomena

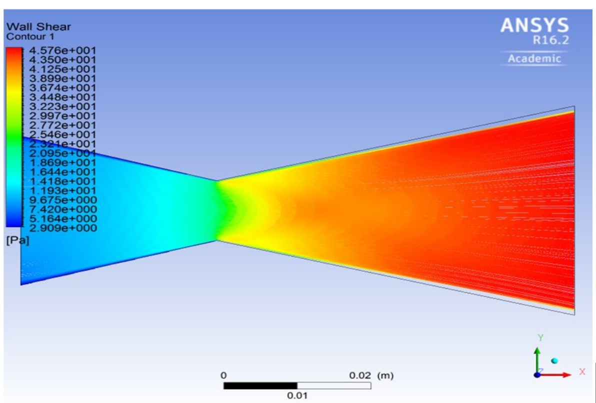


Fig 11 Wall shear contour

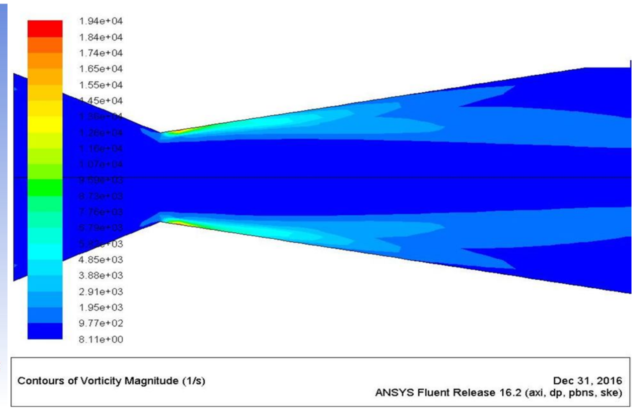


Fig 12 Reverse flow

III. CONCLUSION

The results obtained suggest that the geometry considered is fail safe and can produce maximum power from basic input. The design is also suitable for submerged conditions which is analysed from the reverse flow contours. The design behaves accordingly to the theoretical assumptions with subtle deviations which can be considered negligible in the present scenario. The meshing was robust and separate sections for the chamber and throat are designed keeping the focus entirely on the throat region. The flow conditions considered here are standard and still there is scope for testing the nozzle in extreme real time conditions with appropriate parameters.

IV. ACKNOWLEDGMENT

I feel ecstatic to be part of this work along with Arjun Singh who delivered his best towards completing this analysis on time. The Ansys platform user guide provided all the necessary concepts needed to perform simulations matching the real time conditions. I thank each and everyone for providing various inputs and their valuable time in to this paper.

REFERENCES

[1] Yunus A. Çengel and John M. Cimbala, “Fluid Mechanics”, Tata McGraw-Hill New York, Second edition, (pp 853- 910).

[2] 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, ISSN: 2320-2092

[3] ASME MFC-7M, Measurement of Gas Flow by Means of Critical Flow Venturi Nozzles. New York: American Society of Mechanical Engineers; 1987

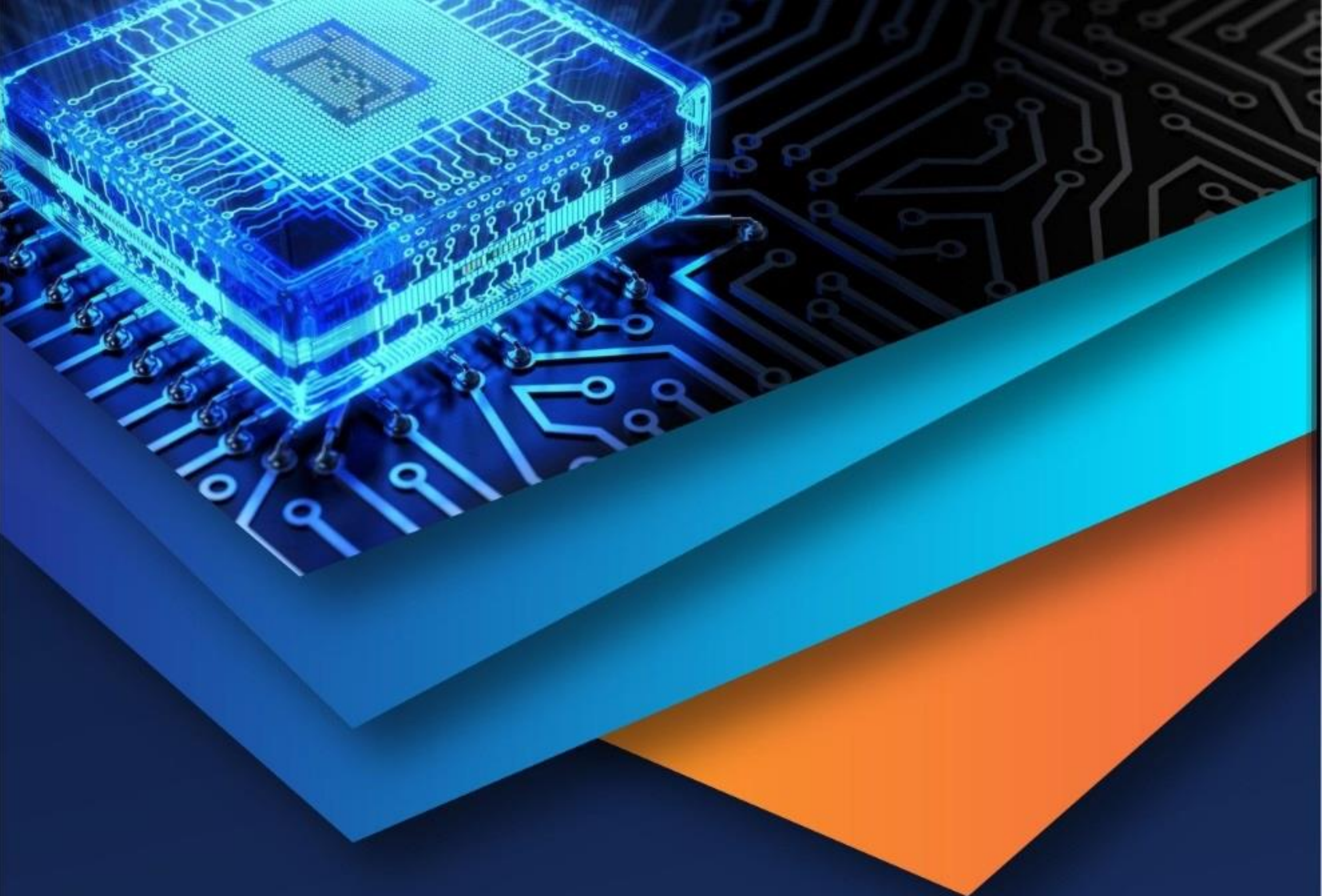
[4] Venturi description – Flow through a Venturi Available at: en.wikipedia.org/wiki/Venturi_effect, Consulted on – 20 Jan 2013

[5] Jack Philip Holman, Experimental Methods for Engineers 6th edition, September 1993, McGraw-Hill Higher Education.

[6] P. Padmanathan, Dr. S. Vaidyanathan "Computational Analysis of Shockwave in Convergent Divergent Nozzle" International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622 Vol. 2, Issue 2, Mar-Apr 2012, pp.1597- 1605 .

[7] A. Balabel et al "Numerical Simulation of Turbulent Gas Flow in a Solid Rocket Motor Nozzle" 13th International Conference on Aerospace Sciences & Aviation Technology, ASAT- 13, May 26 – 28, 2009 Cairo, Egypt.

[8] Abdulnaser Syma, "Computational Fluid Dynamics," 1st edition. Available:http://bookboon.com/en/textbooks/energyenvironment/com putational-fluid-dynamics



10.22214/IJRASET

INDEX COPERNICUS

45.98

IMPACT FACTOR: 7.129

TOGETHER WE REACH THE GOAL

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY