CFD Analysis Of Rocket Engine Nozzle

Prosun Roy[1], Abhijit Mondal[2], Biswanath Barai[3]

[1,2] B.Tech, Department of Mechanical Engineering, Techno India College of Technology, WBUT, India.
[3]Master of Engineering in Design, IIEST, Shipbur, India.

Abstract— *The main objective is to analyze a rocket engine nozzle to understand the phenomena of various design conditions under different convergent angle, divergent angle and throat radius by Computational Fluid Dynamic (CFD). There have also mentioned about inlet boundary conditions with specification according to the experimental information. The paper also addresses static pressure optimization and Mach number optimization. The values on the basis of results along by optimal values of nozzle design parameters obtained from optimization techniques of Taguchi Design. Convergent angle, Divergent angle and Throat radius are considered. Also response of static pressure and Mach number values of CFD analysis in two types of inlet pressure value applied for optimal parameters of nozzle attained.*

Keywords— Computational Fluid Dynamics (CFD), Mach number, Divergent Angle, Convergent Angle and Static Pressure.

I. INTRODUCTION

Computational Fluid Dynamics (CFD) is an engineering tool that assists experimentations. Its scope is not limited to fluid dynamics; CFD could be applied to any process which involves transport phenomena with it. To solve an engineering problem we can use of various methods like the analytical method, experimental methods using prototypes. The analytical method is very complicated and difficult. The experimental methods are very costly. If any errors in the design were detected during the prototype testing, another prototype is to be made clarifying all the errors and again tested. This is a time-consuming as well as a cost-consuming process. The introduction of Computational Fluid Dynamics has overcome this difficulty as well as revolutionized the field of engineering. In CFD a problem is simulated in software and the transport equations associated with the problem is mathematically solved with computer assistance. Thus we would be able to predict the results of a problem before experimentation. Rocket engine nozzle is a propelling nozzle used in a rocket engine to expand and accelerate the combustion gases produced by burning propellants so that the exhaust gases exit the nozzle at hypersonic velocities. The current work aims at determining an optimal convergent angle, divergent angle and throat radius of the nozzle which would give the maximum outlet velocity and meet the thrust requirements. Flow instabilities might be created inside the nozzle due to the formation if shocks which reduce the exit mach number as well as thrust of the engine. This could be eliminated by varying the divergent angle. Here analysis has been conducted on nozzles with divergent angles 4°,7°, 10°, 13°, 15° . Experimentation using the prototypes of each divergent angle is a costly as well as a time consuming process. CFD proves to be an efficient tool to overcome these limitations. Here in this work the trend of various flow parameters are also analyzed.

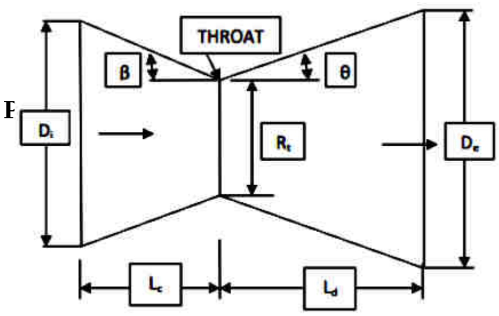


Fig. 1: Geometrical Parameters of Nozzle 2D Model

Table 1: Design Parameters (Nozzle dimensions & Boundary conditions)

Inlet Diameter (m)	1.0
Throat Diameter (m)	0.40
Exit Diameter (m)	0.75
Total Pressure (bar)	42.2 and 51.5
Total Temperature (K)	3600

Table 2: Process Parameters

Stage	Convergent Angle (β) in degree	Divergent Angle (θ) in degree	Throat Radius (R_t) (mm)
I.	30	7.5	0
II.	45	15	130
III.	60	30	225

II. TAGUCHI DESIGN

The three factors (θ, β, R_t) involving in CFD analysis in configuration design for 3 levels of values along by attains Taguchi design. In the design involving in nine configuration of nozzle analysis consist of altered parameters. Also the nine configurations of nozzle analysis conduct from 45.6e5Pa and 52e8Pa of inlet pressure.

Table 3: Taguchi 3x3 Design

Convergent Angle (β) in degree	Divergent Angle (θ) in degree	Throat Radius (R_t) (mm)
30	7.5	0
30	15.0	130
30	30.0	225
45	7.5	130
45	15.0	225
45	30.0	0
60	7.5	225
60	15.0	0
60	30.0	130

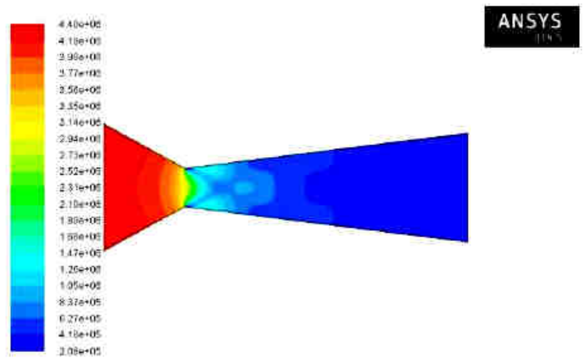
Table 4 : Analysis Procedure

Procedure	**Details**
Solution Setup-General	Solver type- Density Based Velocity Formation-Absolute Time- Steady 2D Space- Planer (symmetric)
Models	Energy- On Viscous model- Inviscid
Materials	Fluid- Air Density- Ideal gas
Cell Zone Conditions	Operating Condition- 1Pa
Boundary Condition	Pressure Inlet- 45.6e5 Pa and 52e8 Pa Temperature- 3600 K
Solutions	Solution Controls – courant number = 6 Solution initialization – Standard Compute – Inlet Run Calculation: Enter the Number of iteration, click calculation.
Result	Graphic and Animations – Contours – Mach number static pressure contour. Plots – XY plot – Mach number vs positions, Static pressure vs positions

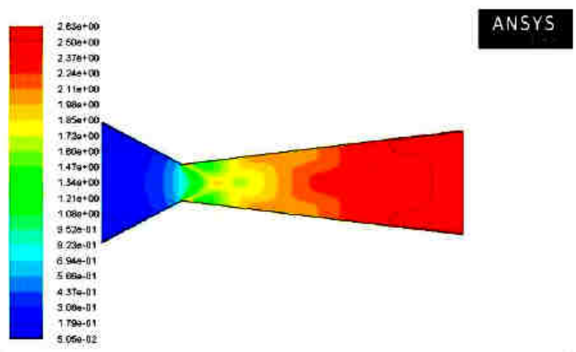
Various Configurations of nozzle is conducted in this procedure.

III. RESULT

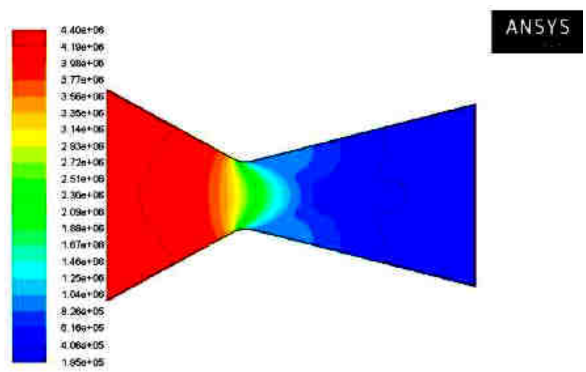
1.Inlet Pressure at 45.6e5 Pa based



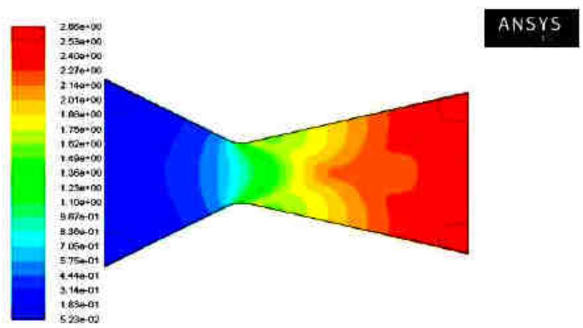
Static Pressure: (β=30, θ= 7.5, R_t =0)

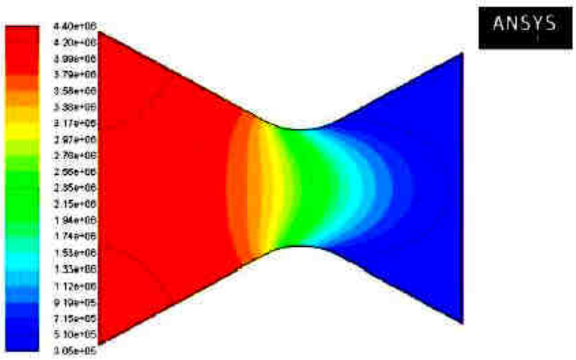


Mach number: (β=30, θ= 7.5, R_t =0)

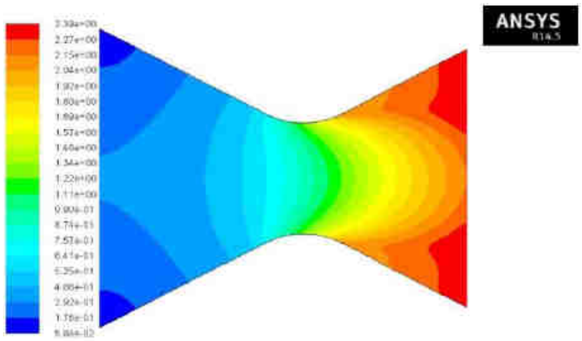


Static Pressure: (β=30, θ= 15, R_t =130)

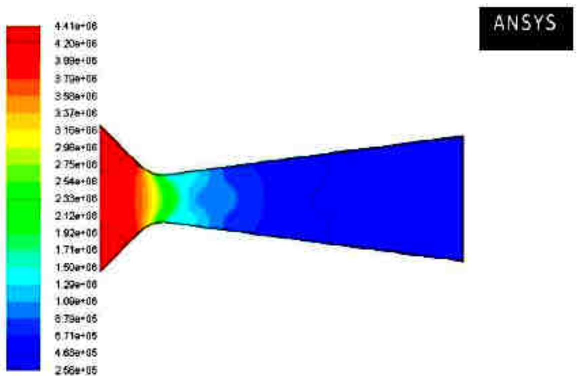




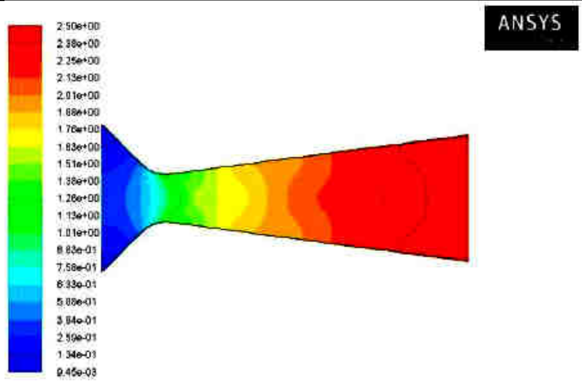
Static Pressure: (β=30, θ= 30, R_t =225)



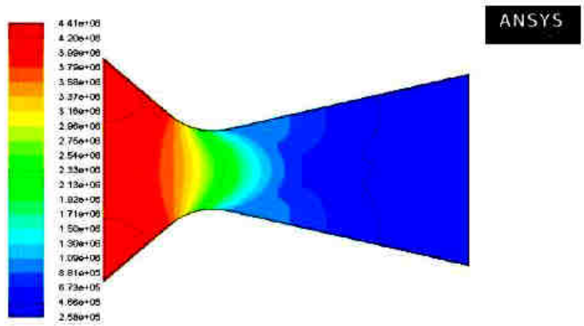
Mach number: (β=30, θ= 30, R_t =225)



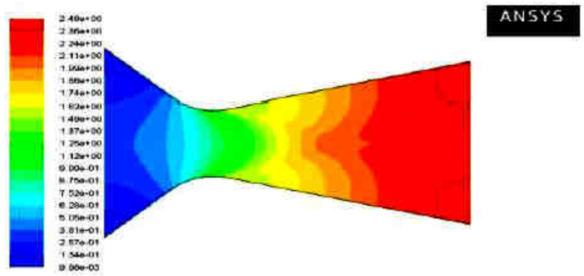
Static Pressure: (β=45, θ= 7.5, R_t =130)



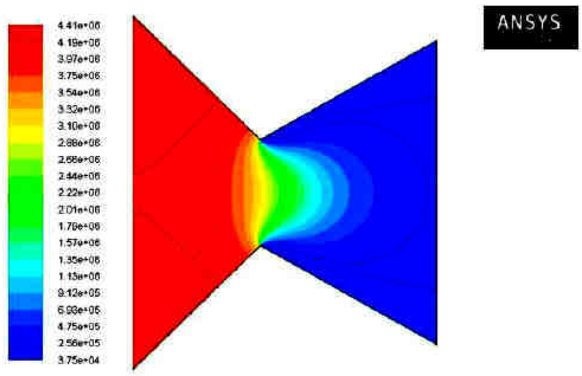
Mach number: (β=45, θ= 7.5, R_t =130)



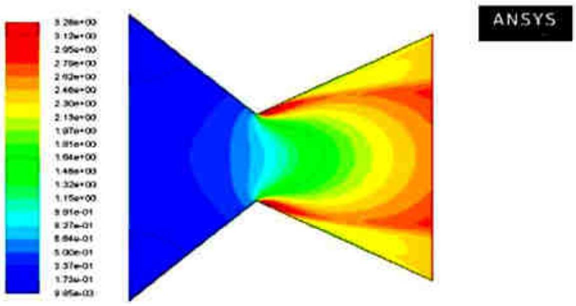
Static Pressure: (β=45, θ= 15, R_t =225)



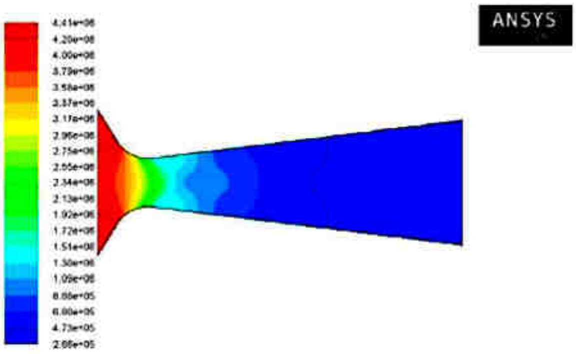
Mach number: (β=45, θ= 15, R_t =225)



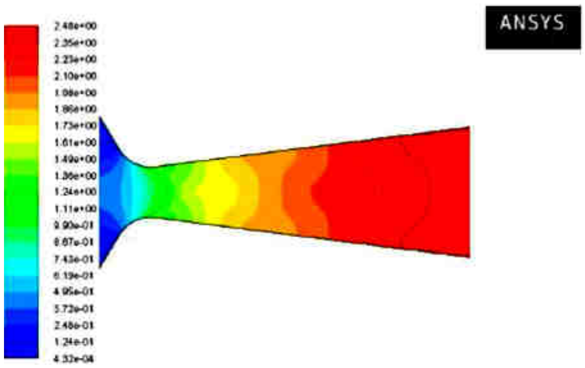
Static Pressure: (β=45, θ= 30, R_t =0)



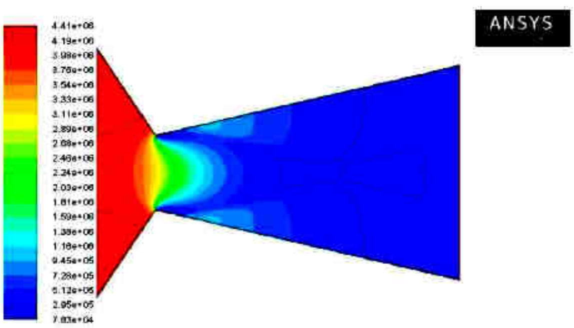
Mach number: (β=45, θ= 30, R_t =0)



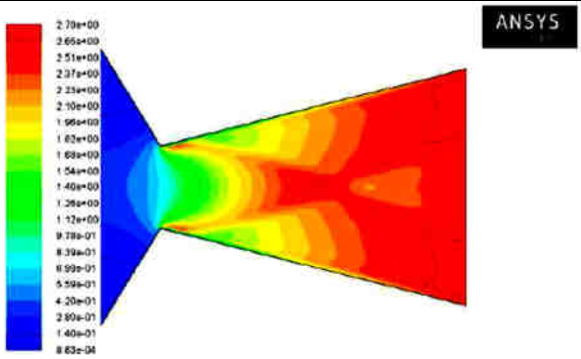
Static Pressure: (β=60, θ= 7.5, R_t =225)



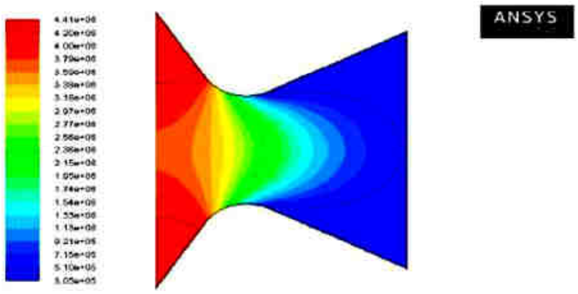
Mach number: (β=60, θ= 7.5, R_t =225)



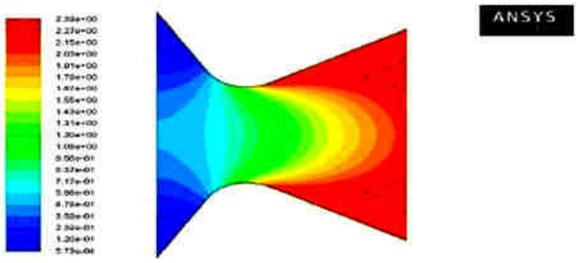
Static Pressure: (β=60, θ= 15, R_t =0)



Mach number: (β=60, θ= 15, R_t =0)

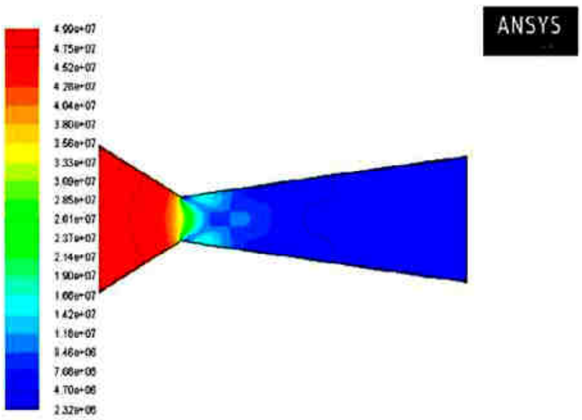


Static Pressure: (β=60, θ= 30, R_t =130)

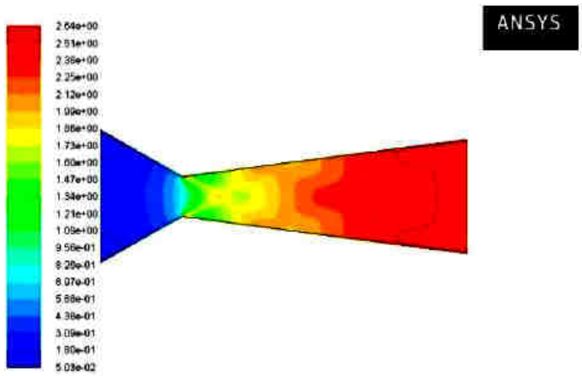


Mach number: (β=60, θ= 30, R_t =130)

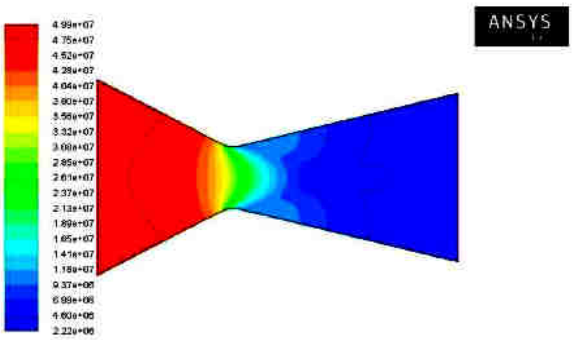
2.Inlet Pressure at 51.5e6 Pa based



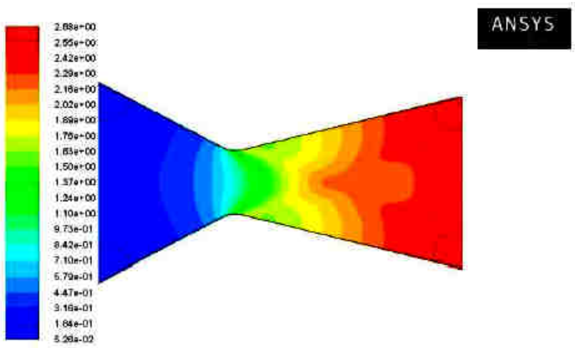
Static Pressure: (β=30, θ= 7.5, R_t =0)



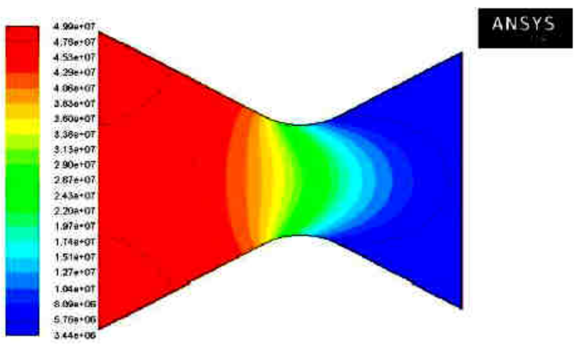
Mach number: (β=30, θ= 7.5, R_t =0)



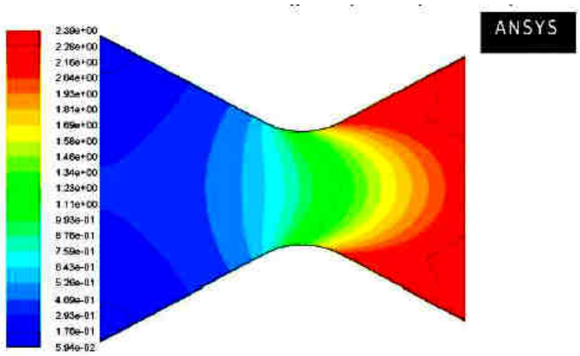
Static Pressure: (β=30, θ= 15, R_t =130)



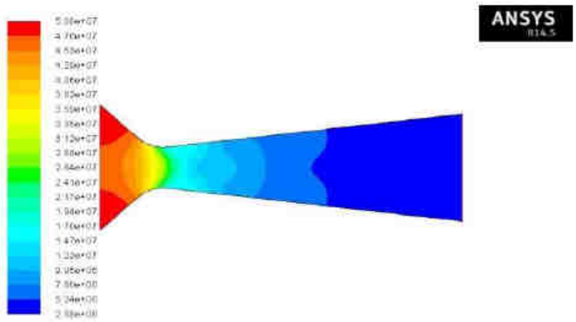
Mach number: (β=30, θ= 15, R_t =130)



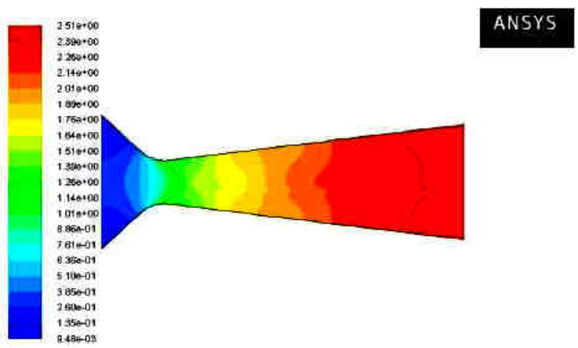
Static Pressure: (β=30, θ= 30, R_t =225)



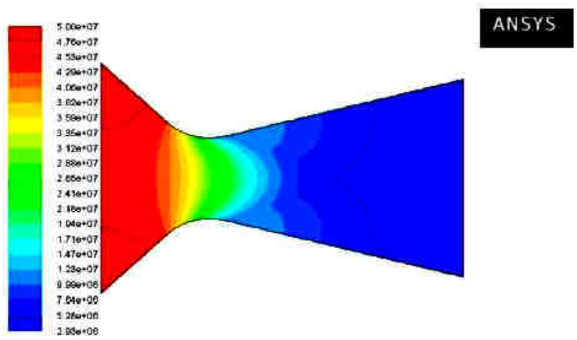
Mach number: (β=30, θ= 30, R_t =225)



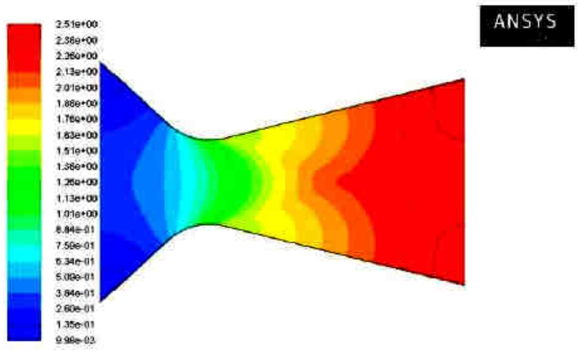
Static Pressure: (β=45, θ= 7.5, R_t =130)



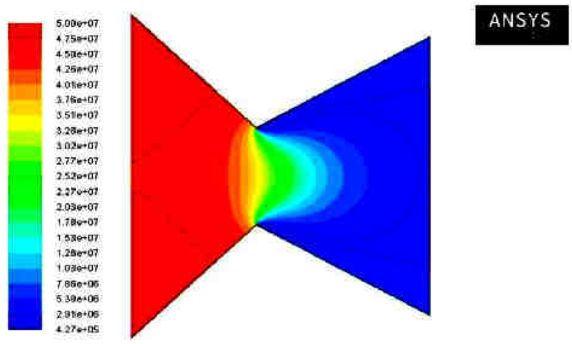
Mach number: (β=45, θ= 7.5, R_t =130)



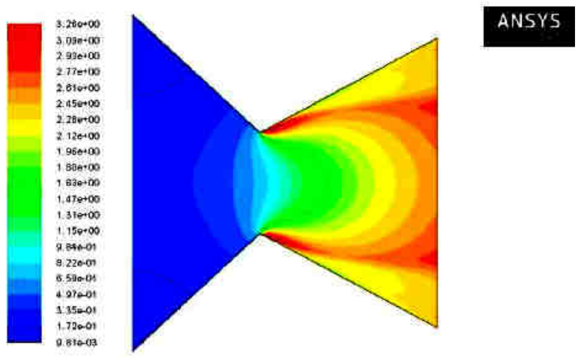
Static Pressure: (β=45, θ= 15, R_t =225)



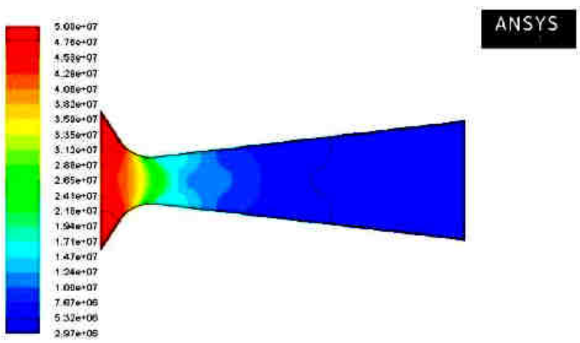
Mach number: (β=45, θ= 15, R_t =225)



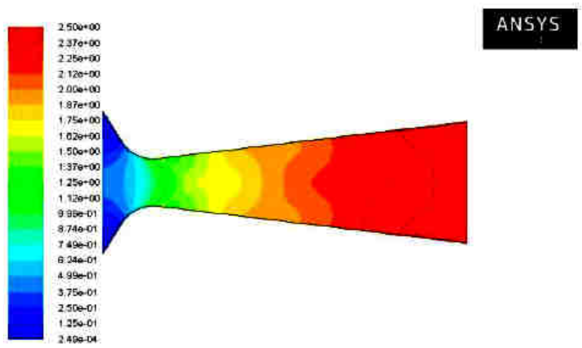
Static Pressure: (β=45, θ= 30, R_t =0)



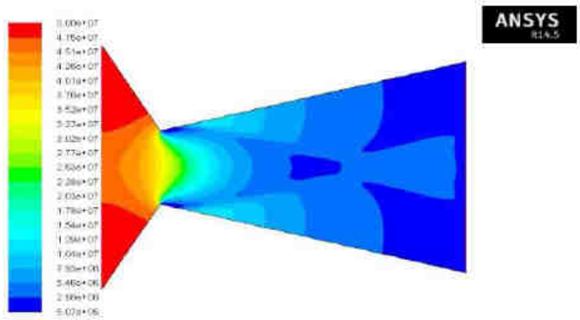
Mach number: (β=45, θ= 30, R_t =0)



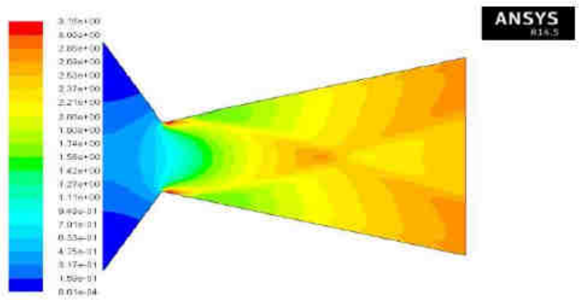
Static Pressure: (β=60, θ= 7.5, R_t =225)



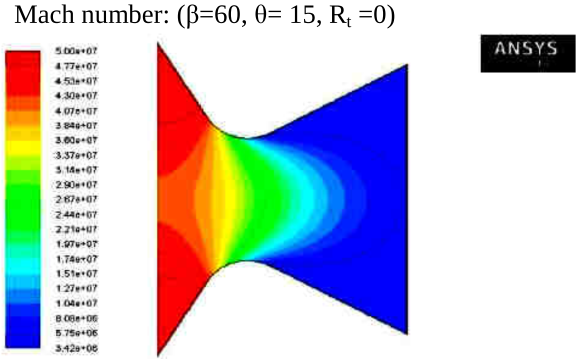
Mach number: (β=60, θ= 7.5, R_t =225)



Static Pressure: (β=60, θ= 15, R_t =0)

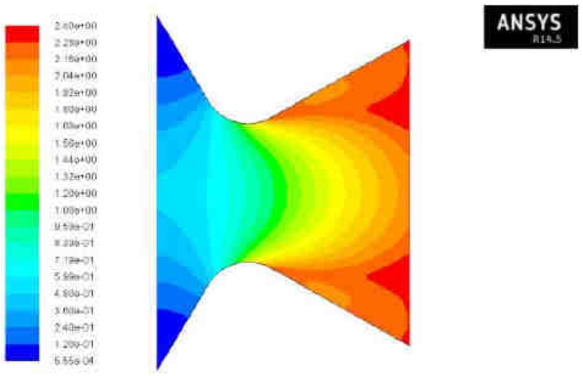


Mach number: (β=60, θ= 15, R_t =0)



Static Pressure: (β=60, θ= 30, R_t =130)

ISSN: 2349-6495



The analysis results in the nozzle exit section are produced by the static pressure and mach number contour of the CFD has to be corresponding design parameters..

Table 5:Static pressure optimization

No	Convergent Angle	Divergent Angle	Static pressure (45.6Pa)	Static Pressure (52Pa)	Mean for static pressure (Pa)
1.	30	7.5	2.04e+06	2.28e+05	1156000
2	30	15.0	2.01e+05	2.14e+05	1307200
3	30	30.0	3.01e+05	3.32e+05	1562500
4	45	7.5	2.96e+05	2.70e+05	1548000
5	45	15.0	2.76e+06	2.81e+05	1791000
6	45	30.0	3.91e+05	4.11e+05	2923000
7	60	7.5	2.33e+06	2.56e+06	1214000
8	60	15.0	6.99e+06	4.98e+06	2354000
9	60	30.0	3.21e+05	3.24e+06	1965700

Table 6: Mach number optimization

No	Convergent Angle	Divergent Angle	Mach number (45.6Pa)	Mach number (52Pa)	Mean for Mach number
1.	30	7.5	3.02e+01	3.92e+05	3.158
2	30	15.0	2.89e+03	3.89e+05	3.002
3	30	30.0	2.18e+01	3.01e+05	2.579
4	45	7.5	2.86e+01	2.73e+05	2.541
5	45	15.0	2.32e+01	2.35e+05	2.379
6	45	30.0	3.56e+01	3.64e+05	3.405
7	60	7.5	2.69e+02	2.48e+06	2.782
8	60	15.0	3.73e+01	2.37e+06	2.993
9	60	30.0	3.28e+01	3.13e+06	3.004

V. CONCLUSION

The optimal static pressure at the exit section is 4.98e+05 and 3.24e+05 the value of static pressure is based on Convergent angle, Divergent angle and Throat radius. Mach number goes on increasing with increase in divergent angle. The range of increment of Mach number is 3.13e+01 to 3.24e+06.

REFERENCES

[1] P. Parthiban, M. Robert Sagayadoss and T. Ambikapathi 'Design and Analysis of Rocket Engine Nozzle by using CFD and Optimization of Nozzle Parameters'. International Journal of Engineering Research. Vol 3, Issue 5. (2015) http://www.ijoer.in

[2] Biju Kuttan P and M Sajesh 'Optimization of Divergent Angle of a Rocket Engine Nozzle using Computational fluid Dynamics'. The International Journal of Engineering and Science (IJES). Vol 2, Issue 2, pp 196-207 (2013)

[3] Varun, R.; Sundararajan,T.; Usha,R.; Srinivasan,k.; Interaction between particle-laden under expanded twin supersonic jets, Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering 2010 224: 1005.

[4] Pandey,K.M.; Singh, A.P.; CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software, International Journal of Chemical Engineering and Applications, Vol. 1, No. 2, August 2010, ISSN: 2010-0221.

[5] Natta, Pardhasaradhi.; Kumar, V.Ranjith.; Rao, Dr. Y.V. Hanumantha.; Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (Cfd), International Journal of Engineering Research and Applications (IJERA), ISSN: 2248-9622,Vol. 2, Issue 5, September- October 2012, pp.1226-1235.

[6] K.M. Pandey, Member IACSIT and A.P. Singh. K.M.Pandey, Member, IACSIT and S.K.YadavK.M.Pandey and S.K.Yadav, —CFD Analysis of a Rocket Nozzle with Two Inlets at Mach2.1, Journal of Environmental Research and Development, Vol 5, No 2, 2010, pp- 308-321.

[7] Shigeru Aso, ArifNur Hakim, Shingo Miyamoto, Kei Inoue and Yasuhiro Tani " Fundamental study of supersonic combustion in pure air flow with use of shock tunnel" Department of Aeronautics and

Astronautics, Kyushu University, Japan , Acta Astronautica 57 (2005) 384 – 389.

[8] 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.

[9] A damson, T.C., Jr., and Nicholls., J.A., “On the structure of jets from Highly under expanded Nozzles into Still Air,” Journal of the Aerospace Sciences, Vol.26, No.1, Jan 1959, pp. 16-24.

[10] Lewis, C. H., Jr., and Carlson, D. J., “Normal Shock Location in under expanded Gas and Gas particle Jets,” AIAA Journal, Vol 2, No.4, April 1964, pp. 776-777.

[11] Anderson, John D.Jr.; Modern Compressible Flow with Historical Perspective, Third edition, 2012.

[12] Vipul Sharma, Ravi Shankar, Gaurav Sharma, -CFD Analysis of Rocket Nozzle – University of Petroleum and energy studies, Department of Aerospace Engineering, Dehradun, India, 2011.

[13] Balaji Krushna. P, P. Srinivasa Rao, B. Balakrishna. ‘Analysis of Dual Bell Rocket Nozzle Using Computational Fluid Dynamics’. Vol 2, Issue 11. Nov. 2013. http://www.ijret.org

[14] Taro Shimuzu, Masatoshi kodera, Nobuyuki Tsuboi. ‘Internal and External Flow of Rocket Engine’ Journal of Earth Simulator. Vol 9, pp 19-26, March 2008.

[15] Mayur Chakravarti - Design analysis of flow in convergent divergent rocket nozzle – University of Petroleum and energy studies, Department of Aerospace Engineering, Dehradun, India, 2011.

[16] C.Satheesh, A. Arulmurugu., “Design and analysis of C-D Nozzle increase the efficiency using CFD” IJMTER – 2015 Pages 490 - 494.