A CFD ANALYSIS & STUDY ON EFFECT OVER MACH OF ROCKET C-D NOZZLE BY VARYING ITS GEOMETRIC PARAMETERS

NIKHIL D. RAUT

UG Student, Department of Mechanical Engineering, BNCOE, PUSAD.
E-mail: nikhil.raut21@hotmail.com

Abstract- CFD analysis of Rocket engine nozzle has be conducted to study the effect over Mach number by varying Convergent-Divergent (CD) nozzle geometric parameters such as providing curvature at throat section, varying inlet, outlet & throat diameter and changing length of divergent section by using ANSYS Fluent Workbench 16.0. And it is found that every CD nozzle shows good result in this limit.

Keywords- Rocket, Convergent- Divergent Nozzle, Mach, CFD.

I. INTRODUCTION

A Nozzle is a mechanical device designed to control the characteristics of a fluid as it enters/exits an enclosed chamber or pipe. Nozzle are used to modify/direct the flow of a fluid, it also controls the rate of flow, speed, mass, and pressure of fluid. Its goal is to increase the kinetic energy of fluid to provide thrust at the expense of its pressure and internal energy. One of the most important application of nozzle is, it used in rocket engine. Nozzles have a great variety in its shape & size according to its use. Convergent Divergent (CD) nozzle is a kind of nozzle which mostly used in rocket engines.
In CD nozzle gas flows through nozzle from high pressure (chamber) to low pressure (ambient). Gas flows from chamber into convergent portion of nozzle, passes through throat, through divergent portion and then exhaust into ambient as a jet. Pressure of ambient is referred as back pressure or thrusts.

II. OBJECTIVE OF PRESENT STUDY

The objective of the present work is to simulate supersonic flow through rocket CD nozzle to study over Mach at exit of nozzle. This simulation is carried out using ANSYS Fluent Workbench 16.0.

III. METHODOLOGY

A numerical method is used to solve and simulate this work. With relative boundary condition and approximate governing equation, a system of linear algebraic equation is known as discretization method. Thus, a problem including calculus is transformed in an algebraic problem which is solved by solution methodology with the help of computer system. This technique used to solve fluid flow problems. There are so many methodology used to solve such problems but most common methods used are Finite Difference Method (FDM), Finite Volume Method (FVM) and Finite Element Method (FEM). Now a day Computational Fluid Dynamic (CFD) which is FDM was most popular. This is algorithmically simple, most efficient and has great accuracy. FLUENT is use to simulate the fluid flow which is based upon finite volume discretization method and is best software for fluid work.
For CFD analysis, computational model is designed in geometric modular with standard dimension. Then meshing is done in mesh modular, a fine structured mesh is done with automatic meshing method. Portion near throat is refined with 3 refinement factor; it is done because there are so many chances of error in that particular region. Practical boundary conditions are applied to rocket nozzle for achieving back thrust nearly Mach 3. By using energy equation and assuming density of air is ideal and viscosity as Sutherland. Pressure input and output are provided as inlet and outlet boundary condition. Then it is solved by using FLUENT solver with reference input as inlet and velocity counter is plotted for study of Mach at exit of rocket nozzle.

IV. RESULT AND DISCUSSION

Velocity counters for Mach is plotted and studied effect of varying geometric parameters for rocket CD nozzle.

4.1. Mach counter of actual designed CD Nozzle.

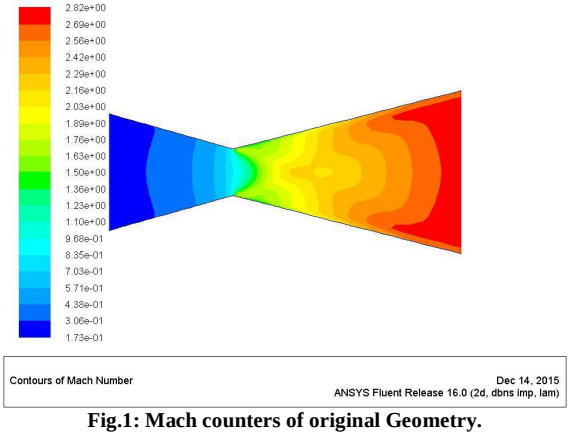


Fig.1: Mach counters of original Geometry.

4.2. Effect of curvature at Throat section of CD Nozzle.

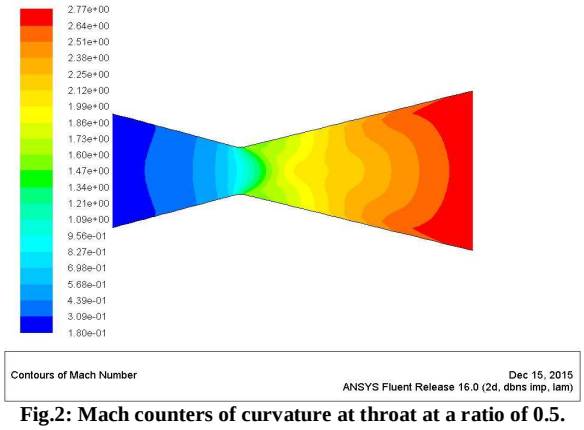


Fig.2: Mach counters of curvature at throat at a ratio of 0.5.

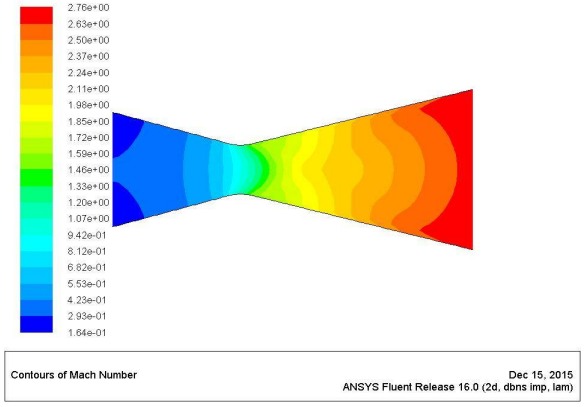


Fig.3: Mach counters of curvature at throat at a ratio of 1.

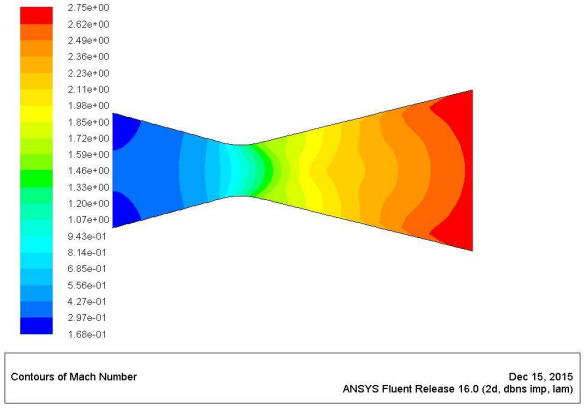


Fig.4: Mach counters of curvature at throat at a ratio of 1.5.

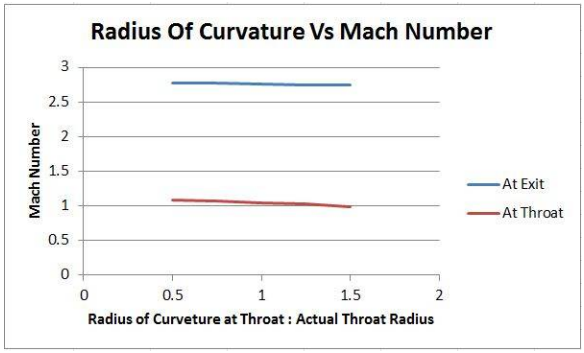


Fig.5: Effect of curvature at throat on Mach.

Fig. 5 shows the effect of curvature at throat section on Mach number, at exit of CD nozzle Mach is constant but up to limited ratio of 1.5 after that it choked up. The Mach at throat section slightly decreases. If a curvature of ratio 1 is provided at throat section sonic condition can easily maintain and also it reduces losses due to variation in cross section. At exit Mach of 2.76 can be obtained.

4.3. Effect due to varying slant length of divergent section.

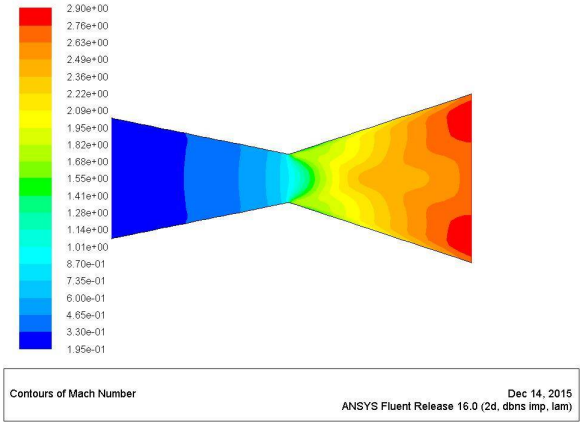


Fig.6: Mach counters of length of divergent section in a ratio of 0.8.

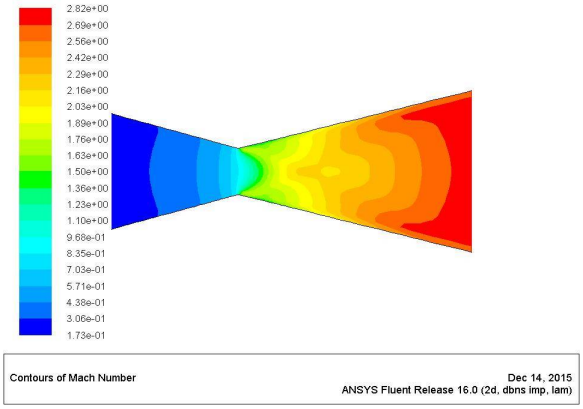


Fig.7: Mach counters of length of divergent section in a ratio of 1.

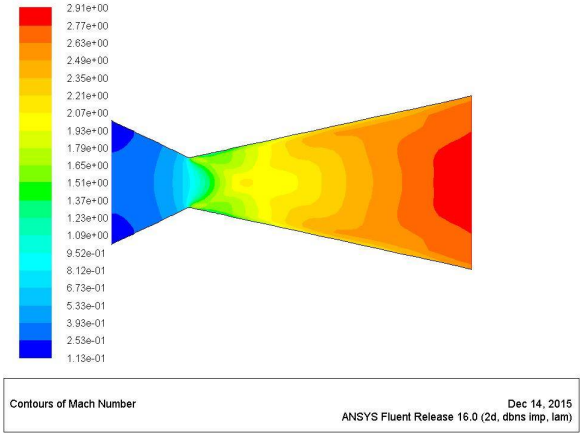


Fig.8: Mach counters of length of divergent section in a ratio of 1.2.

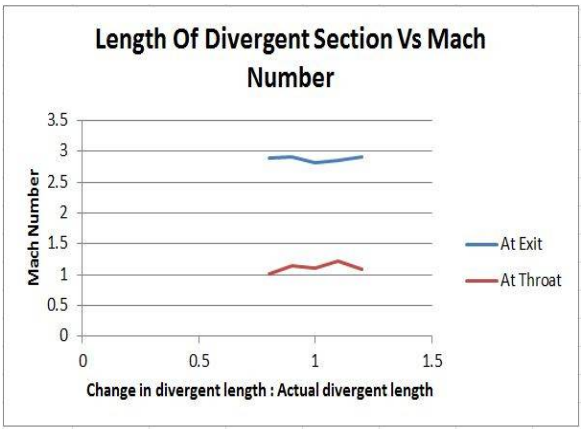


Fig.9: Effect of length of divergent section on Mach.

Fig. 9 shows effect of divergent section length over Mach at exit as well as at throat section it is fluctuating. It is desirable to vary in divergent length in the ratio 0.75-1.2 above and below it, it shows undesirable effect and it also inefficient. At a ratio 0.8 Mach at throat is in sonic condition and Mach at exit is up to 2.92.

4.4. Effect due to varying throat diameter.

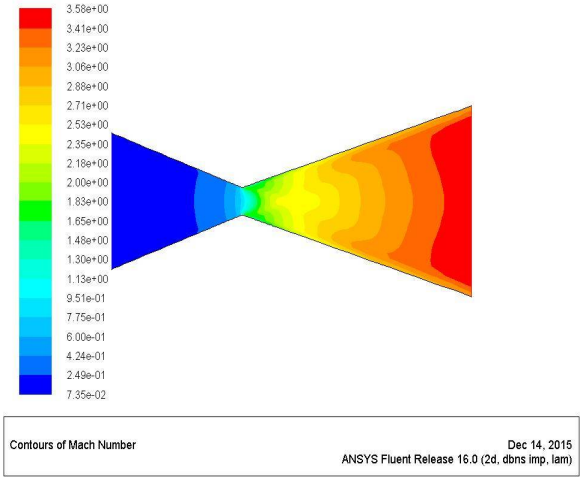


Fig.10: Mach counters of varying throat in a ratio of 0.5.

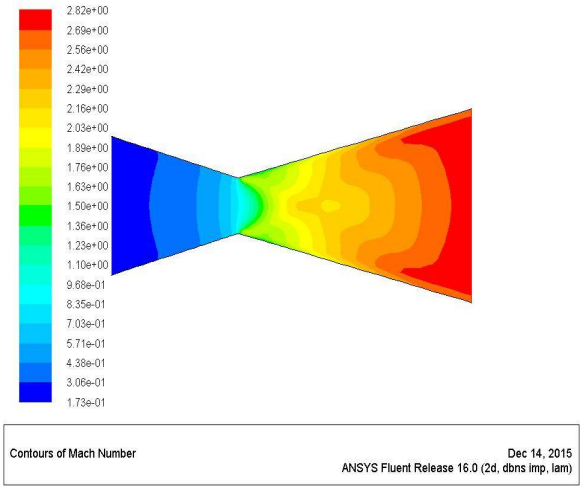


Fig.11: Mach counters of varying throat in a ratio of 1.

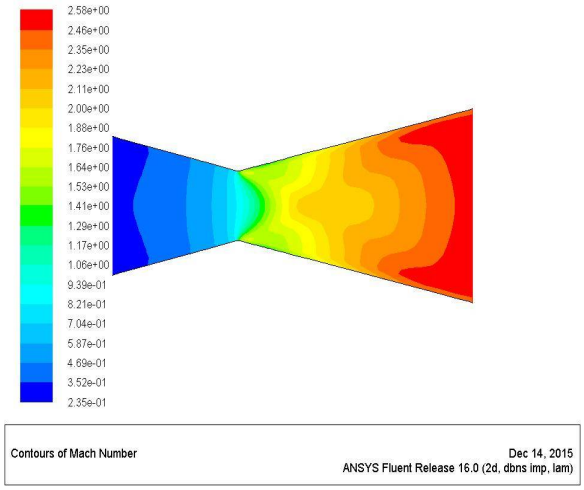


Fig.12: Mach counters of varying throat in a ratio of 1.25.

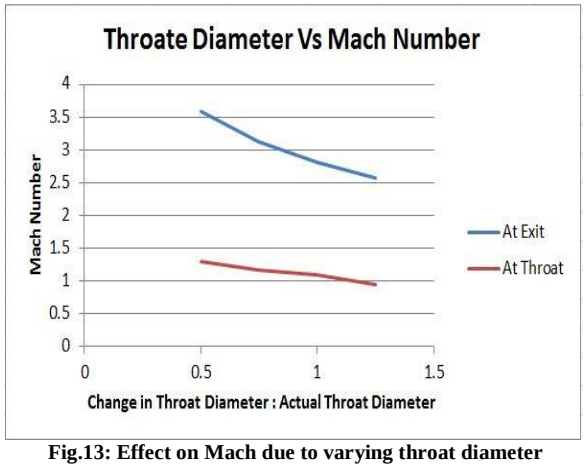


Fig.13: Effect on Mach due to varying throat diameter

Fig. 13 shows the effect of varying diameter of throat on Mach. At a ratio of 0.5 we can achieve 3.5 Mach at exit but it is very undesirable, as nozzle is choked up at throat section. At a ratio of 1.125 Mach at throat is nearly in sonic and also at exit of nozzle Mach is up to 2.75. In the range of 0.5-1.25 desirable to achieve high Mach.

4.5. Effect due to varying outlet diameter.

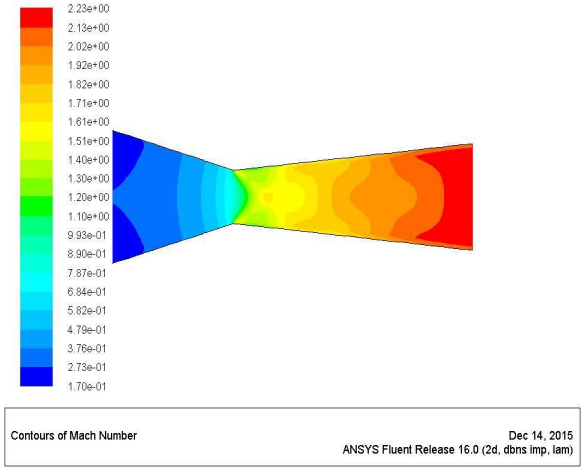


Fig.14: Mach counters of varying exit diameter in a ratio of 0.5.

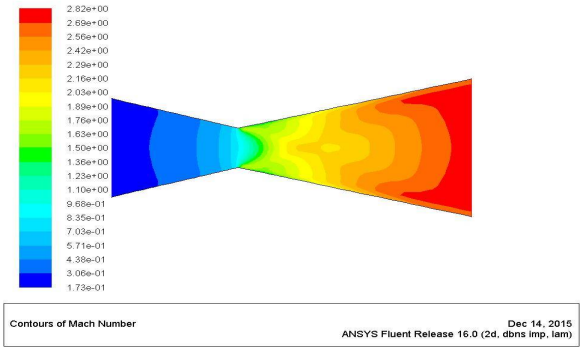


Fig.15: Mach counters of varying exit diameter in a ratio of 1.

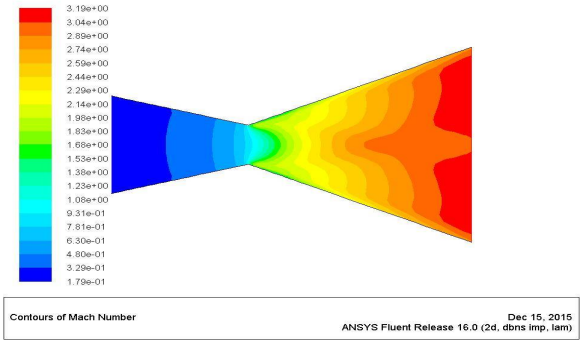


Fig.16: Mach counters of varying exit diameter in a ratio of 1.5.

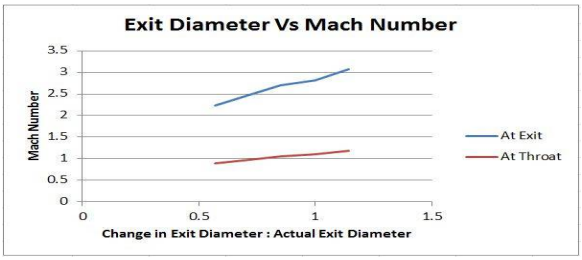


Fig.17: Effect due to variation in exit diameter.

Fig.17 shows effect due to variation in exit diameter, at a ratio of 0.8, throat have sonic Mach and Mach at exit is 2.75. Further increase in ratio, above 3 Mach speed can be obtain. But ratio is limited in range of 0.5 -1.5 and desirable result can be obtained.

CONCLUSION

Following conclusion can be drawn to achieve Mach 3 speed.

- By providing curvature at throat of 0.5 times the throat diameter, about 2.77 Mach speed can be achieved.
- Divergent length is reduced by 0.9 can provide speed nearly 2.92 Mach.
- Reducing throat diameter by 0.75, 3.12 Mach speed at exit can be obtained.
- Increasing exit diameter by 1.4 times the actual exit diameter 3.19 Mach speed can be achieved.

REFERENCES

[1] Dr. Y. V. Hanumantha Rao, “Flow Analysis of Rocket Nozzle Using Computational Dynamics”, IJERA vol. 2, Issue 5, Sept-Oct 2012, ISSN: 2248-9622.

[2] K. M. Pandey, “CFD Analysis Of Conical Nozzle For Mach 3 At Various Angle Of Divergent With Fluent Software.”

[3] Gerald Hagemann, “Advance Rocket Nozzles”, Journal of Propulsion and Power, Vol. 5, Sept-Oct 1998.

[4] Vasiliy zubanov, Vitaliy Egorychev, “Design Of Rocket Engine for Spacecraft using CFD-Modeling”, ScienceDirect Elsevier, 2015

[5] ANSYS FLUENT Solver Modeling guide, Canons-burg, Ansys, Inc., 2006 pp 550-556.

[6] K. Ramamurthi, “Rocket Propulsion”, Macmillan Publication (India), Edition 2012, pp 54-89.

★★★