Design & Simulation of Internal Flow of Nozzle

Kaushik Iyer[1], Saurabh Khandelwal[2], Akshay Malani[3], Hrushikesh Mantri[4], V.H. Bansode[5]
[1]*Student, Mechanical Department, Sinhgad Technical Istitute,* ***iyerkaushik0@email.com***
[2] *Student, Mechanical Department, Sinhgad Technical Istitute,* ***saurabhkhandelwal770@email.com***
[3] *Student, Mechanical Department, Sinhgad Technical Istitute,* ***akshaymalani1@email.com***
[4] *Student, Mechanical Department, Sinhgad Technical Istitute,* ***hrushikeshm26@email.com***
[5]*Assistant Professor, Mechanical dept., Smt. Kashibai Navale College of Engg.*

ABSTRACT

Aerodynamics is a branch of fluid dynamics concerned with studying the motion of air, particularly when it interacts with a solid object, such as an airplane wing. Aerodynamics is one of the sub-field of fluid dynamics and gas dynamics, and many aspects of aerodynamics theory are common to these fields. In this paper we are going to study rocket nozzle i.e C-D Nozzle. Convergent-Divergent nozzles are used to increase the flow of gas to supersonic speeds (as in the case of rockets). Classification is done on the basis of (M<1 or M=1 or M>1). M is the mach number (which means ratio of local speed of flow to the local speed of sound) A is area and V is velocity. Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyse problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. Comparison between different speeds (Mach No) at different Divergent Angles is mentioned.

Keywords: ***Aerodynamics, Nozzle, CFD, Mach No.***

---

1. INTRODUCTION

Aerodynamics is a branch of fluid dynamics concerned with studying the motion of air, particularly when it interacts with a solid object, such as an airplane wing. Aerodynamics is one of the sub-field of fluid dynamics and gas dynamics, and many aspects of aerodynamics theory are common to these fields. The term aerodynamics is often used with gas dynamics, with the difference being that "gas dynamics" applies to the study of the motion of all gases, not limited to air. Recent work in aerodynamics has focused on issues related to compressible flow, turbulence, and boundary layers and has become increasingly computational in nature. Understanding the motion of air around an object (often called flow field) enables the calculation of forces and moments acting on the object. In many aerodynamics problems, the forces of interest are the fundamental forces of flight: lift, drag, thrust, and weight. Of these, lift and drag are aerodynamic forces, i.e. forces due to air flow over a solid body. Calculation of these quantities is often founded upon the assumption that the flow field behaves as a continuum. Continuum flow fields are characterized by properties such as flow velocity, pressure, density, and temperature, which may be functions of spatial position and time. These properties may be directly or indirectly measured in aerodynamics experiments or calculated from equations for the conservation of mass, momentum, and energy in air flows. Density, flow velocity, and an additional property, viscosity, are used to classify flow fields. Flow velocity is used to classify flows according to speed regime. Subsonic flows are flow fields in which the air speed field is always below the local speed of sound. Transonic flows include both regions of subsonic flow and regions in which the local flow speed is greater than the local speed of sound. Supersonic flows are defined to be flows in which the flow speed is greater than the speed of sound everywhere. A fourth classification, hypersonic flow, refers to flows where the flow speed is much greater than the speed of sound. Aerodynamicists disagree on the precise definition of hypersonic flow. Compressibility refers to whether or not the flow in a problem can have a varying density. Subsonic flows are often assumed to be incompressible, i.e. the density is assumed to be constant. Transonic and supersonic flows are compressible, and neglecting the changes in density in these flow fields will yield inaccurate results when performing calculations. Viscosity is associated with the frictional forces in a flow. In some flow fields, viscous effects are very small, and approximate solutions may safely neglect viscous effects. These approximations are called in viscid flows. Flows for which viscosity is not neglected are called viscous flows. Finally, aerodynamic problems may also be classified by the flow environment. External aerodynamics is the study of flow around solid objects of various shapes (e.g. around an airplane wing), while internal aerodynamics is the study of flow through passages inside solid objects (e.g. through a jet engine) Continuum assumption. Unlike liquids and solids, gases are

ISSN: 2321-8169
1318 – 1323

composed of discrete molecules which occupy only a small fraction of the volume filled by the gas. On a molecular level, flow fields are made up of many individual collisions between gas molecules and between gas molecules and solid surfaces. However, in most aerodynamics applications, the discrete molecular nature of gases is ignored, and the flow field is assumed to behave as a continuum. This assumption allows fluid properties such as density and flow velocity to be defined everywhere within the flow. Validity of the continuum assumption is dependent on the density of the gas and the application in question. For the continuum assumption to be valid, the mean free path length must be much smaller than the length scale of the application in question. For example, many aerodynamics applications deal with aircraft flying in atmospheric conditions, where the mean free path length is on the order of micrometres. In these cases, the length scale of the aircraft ranges from a few meters to a few tens of meters, which is much larger than the mean free path length. For these applications, the continuum assumption is reasonable.

2. DESIGN & SIMULATION

2.1 Design

Design of Nozzle consists of one convergent nozzle and one convergent divergent nozzle and blast tube. This is show in fig. below

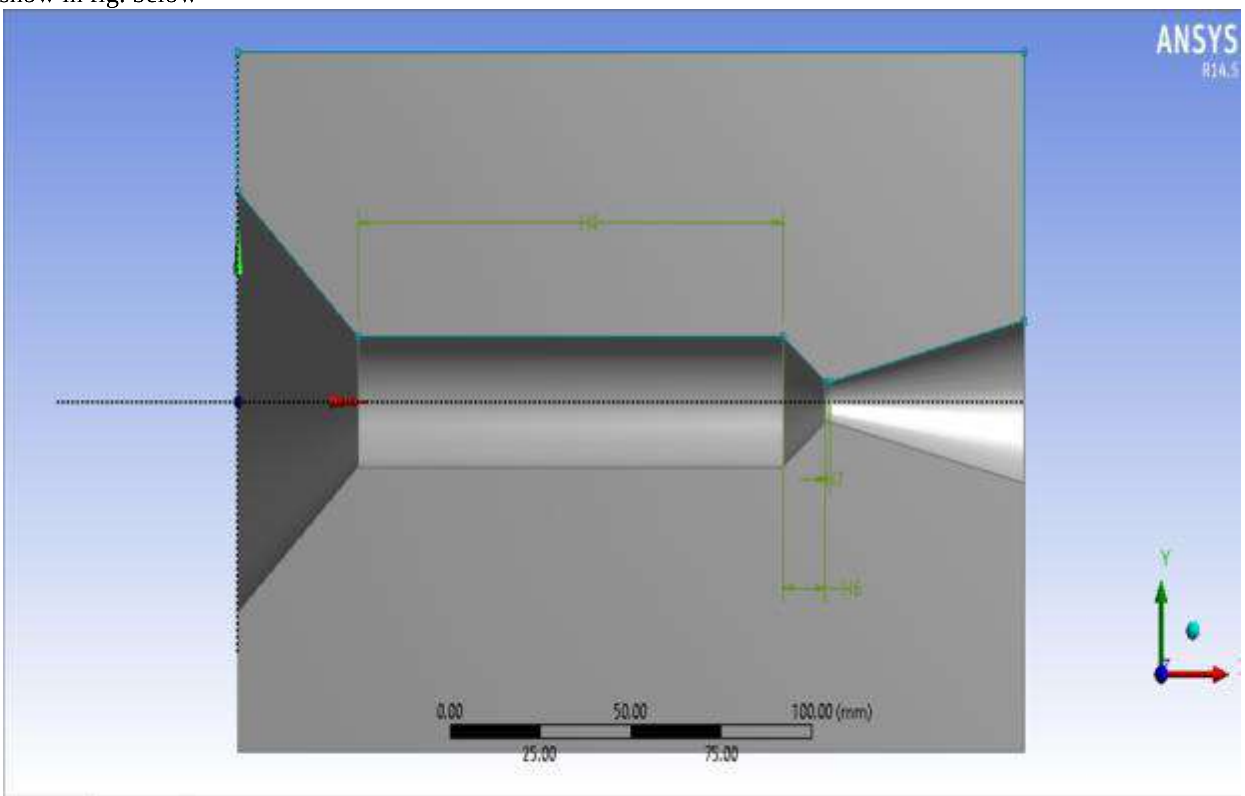


Fig.1 Design of Nozzle

Functions-

1. Nozzle-

A nozzle is a device designed to control the direction or characteristics of a fluid flow (especially to increase velocity) as it exits (or enters) an enclosed chamber or pipe. Nozzles are frequently used to control the rate of flow, speed, direction, mass, shape, and/or the pressure of the stream that emerges from them. In a nozzle, the velocity of fluid increases at the expense of its pressure energy.

2. Convergent-

As the solid fuel burns in the core of missile it has to be converted from Potential energy to Kinetic energy i.e. Velocity. This is done be Convergent nozzle. Convergent is used to convert pressure into velocity as explained before.

3. Blast Tube-

Blast tube connect the convergent nozzle to convergent-divergent nozzle.

ISSN: 2321-8169
1318 – 1323

4. Convergent-Divergent-

It converts sonic velocity to supper sonic. Velocity obtained in this part in 2.5-3 Mach.

2.2 Meshing

Meshing is discretization. It is the most important part of an analysis and can determined the efficiency and effectiveness of an analysis.

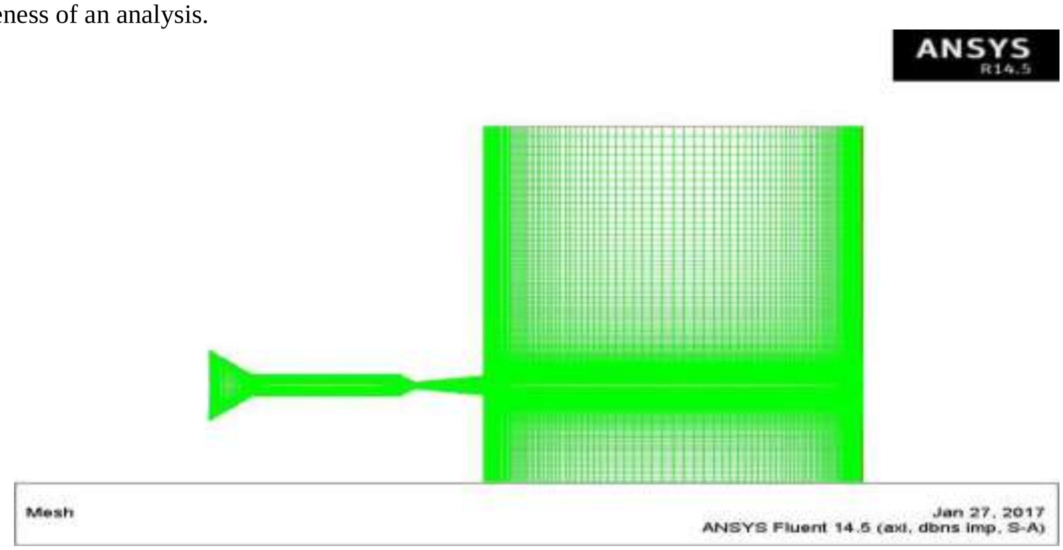


Fig.2 Meshing Of Nozzle 3

3 Simulation in Ansys Fluent

Simulation is testing the designed nozzle under certain boundary conditions. Selecting proper Models, Materials & Boundary conditions comes under this part.

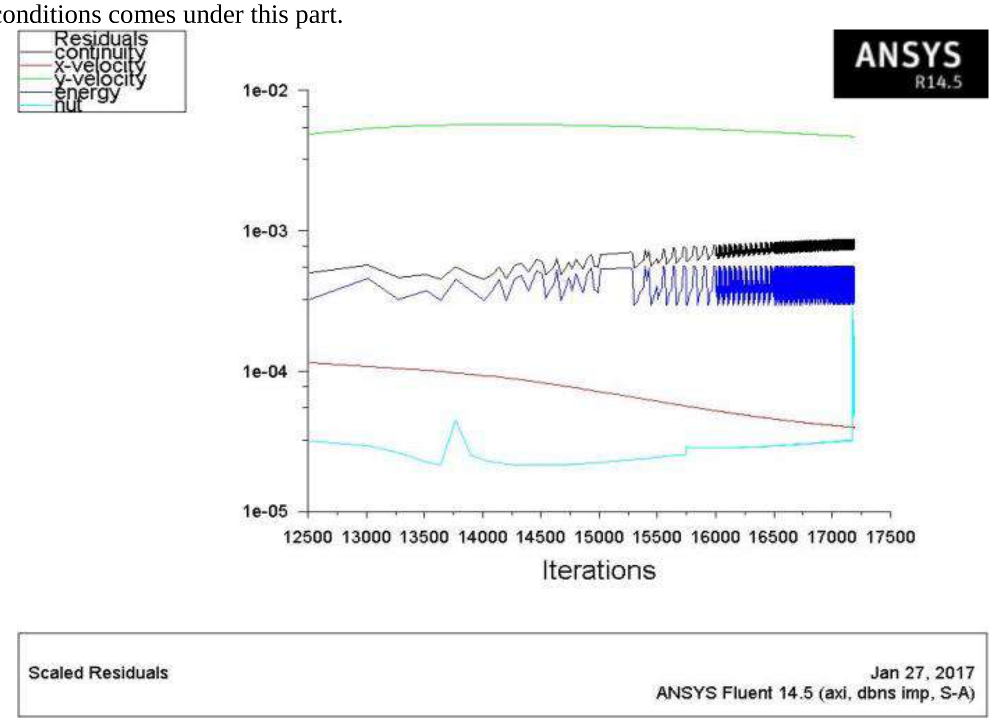


Fig.3 Residuals of Simulations

3. RESULTS

The results obtained by simulating internal flow through nozzle are mentioned below. Different velocities are obtained at different divergent angles by changing the Blast tube length. Table given below shows the comparative study between different divergence angles.

1) Divergence angle 6.5°

The velocity obtain at this Divergent angle is 3.125 Mach. Contours of velocity is shown below.

2) Divergence angle 8°

The velocity obtain at this Divergent angle is 3.35 Mach. Contours of velocity is shown below.

3) Divergence angle 8.38°

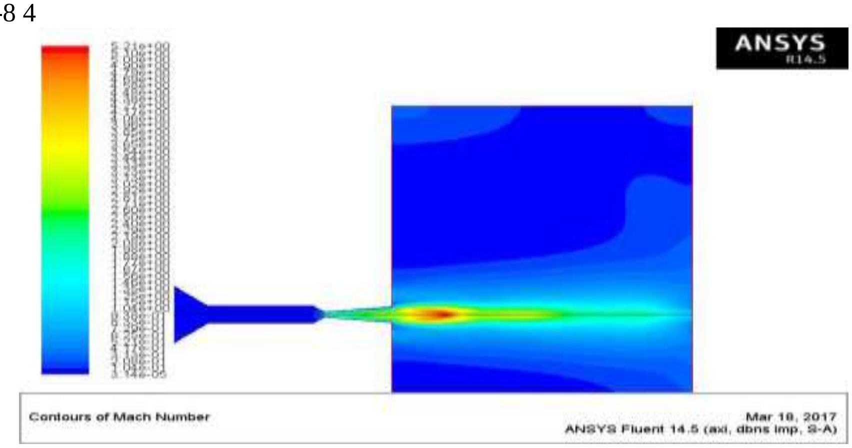
The velocity obtain at this Divergent angle is 3.11 Mach. Contours of velocity is shown below.

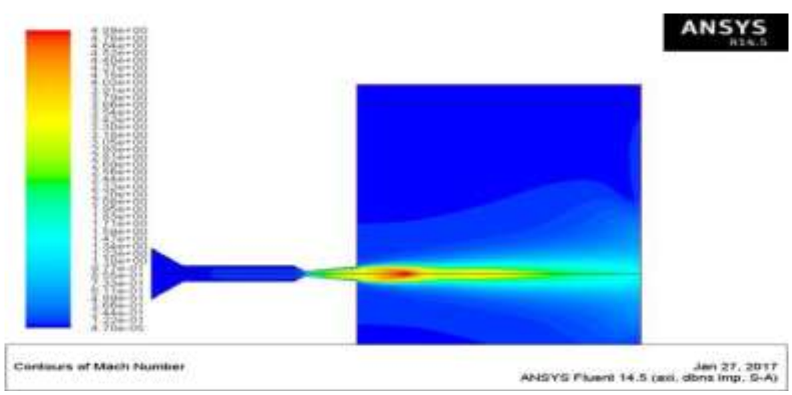
4) Divergence angle 9.28°

The velocity obtain at this Divergent angle is 3.35 Mach. Contours of velocity is shown below.

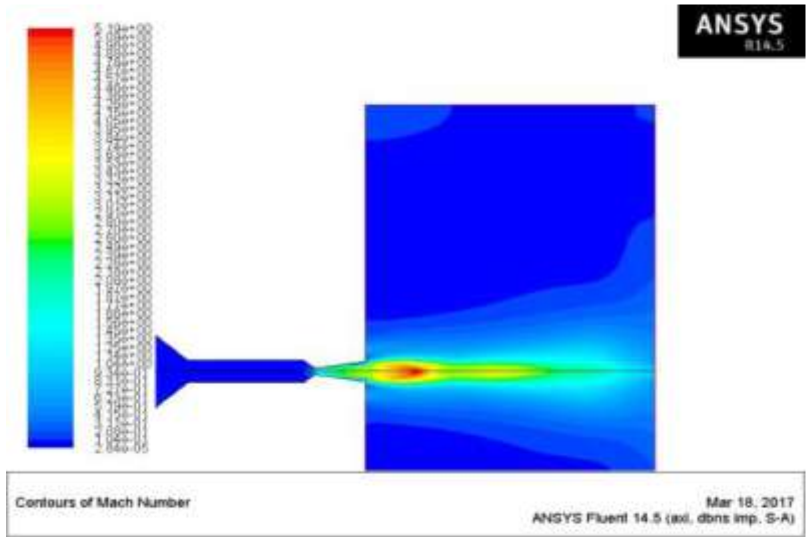
Divergent Angle- 6.5

Divergent Angle-8 4

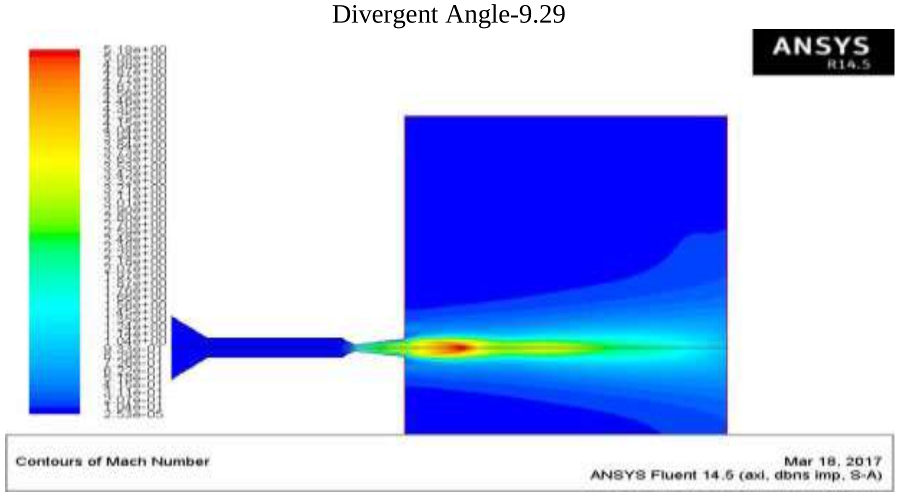




Divergent Angle-8.38



Divergent Angle-9.29



4 CONCLUSION

From the above data it can be seen that maximum velocity is attained at divergence angle of 8°. Comparison table is as shown below.

Divergent Angle	Mach No.
6.5°	3.125
8.0°	3.35
8.38°	3.11
9.28°	3.15

Table no. 1: Result Table

ACKNOWLEDGMENT

The authors are sincerely grateful of ARDE for presenting them with a rare opportunity to undertake such a highly coveted project. They also acknowledge the valuable guidance received from Sci. D, Dr. R. S. Chopade, who is a source of inspiration for the authors. They also wish to thank Prof. Vaibhav Bansode for his continuous support.

REFERENCES

[1] Flow Analysis of Rocket Nozzle Using Computational Fluid Dynamics (Cfd). Pardhasaradhi Natta*, V.Ranjith Kumar**, Dr.Y.V.Hanumantha Rao***. International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622.

[2] CFD Analysis of C-D Nozzle at two divergent angles for prediction of Mach no(M<1, M=1 & M>1) and flow parameters of Newtonian fluid by using ANSYS-FLUENT 14.0**.** A.V.Ramana Rao, I.Akhil. International Journal of Engineering Sciences & Research Technology. ISSN: 2277-9655

[3] Theoretical & CFD Analysis of DE LAVAL Nozzle. Nikhil D. Deshpande, Suyash S. Vidwans, Pratik R. Mahale, Rutuja S. Joshi , K. R. JAGTAP. International Journal of Mechanical And Production Engineering, ISSN: 2320-2092.

[4] Flow Analysis in a Convergent-Divergent Nozzle Using CFD**.** Gutti Rajeswara Rao, U.S. Ramakanth, A. Lakshman. International Journal of Research in Mechanical Engineering , ISSN Online: 2347-5188.