CFD ANALYSIS OF CONVERGENT AND DIVERGENT NOZZLE HAVING CONTOUR EFFECT

RAJAT MISHRA[1], DEVENDRA LOHIA SIR[2]

[1,2]*(DEPARTMENT OF MECHNICAL ENGINEERING RAMA UNIVERSITY, KANPUR)*

ABSTRACT: A nozzle is a device that is used to provide the way and direction to gases to come out of the combustion chamber. The nozzle is a device that is used to convert pressure and thermal energy into useful kinetic energy. A rocket nozzle can be used to manage and control the speed flow of pressure and steam being exhausted .rocket nozzle purpose is to convert high pressure, high-temperature gas into a high velocity, low temperature, and pressure. This paper had an analysis of the flow in the nozzle using the software ANSYS 16. The study is being carried out to have knowledge about flow inside the nozzle when the divergent section of the nozzle is being diverted or change to a certain angle using 2-d modeling to carry out the result and effects of the nozzle. The angle will make the effect on all the aspects of nozzle such as Mach number, static pressure, velocity, and temperature. The throat and inlet diameter of supersonic begin remain the same.

Keyword: Mach number, ANSYS16, supersonic nozzle

INTRODUCTION

Nozzle being present in a different device such as rocket. The supersonic nozzle is present in a rocket that works on the principle of Newton's third law (every action has an equal and opposite reaction). Rocket engines obtain thrust mainly because of the combustion inside the combustion chamber of the nozzle. Rocket engine ejecting mass outward by the theory of Newton's third law. The fluid is generally gas that makes the high-pressure propellant inside the combustion chamber. The high-pressure propellant is a pass to the nozzle which passes heat energy at an accelerated speed which against the ground (which is a reaction) and therefore it pushes the rocket against the direction on the ground (upward direction). The main function of the nozzle to accelerate gases that coming out of the exhaust. A vehicle propelled by a rocket engine is known as rockets. The divergent angle of a nozzle optimally considered to overcome the shock and flow separation at the wall. Therefore in this paper, we overcome the problem numerical value using ANSYS 16.

The fig1.4 given below explains the different parts of the supersonic nozzle such as

a)combustion chamber, b)injection (inlet), c) outlet (exhausting the heat and gas), d) Thorat (where mach number value is zero)

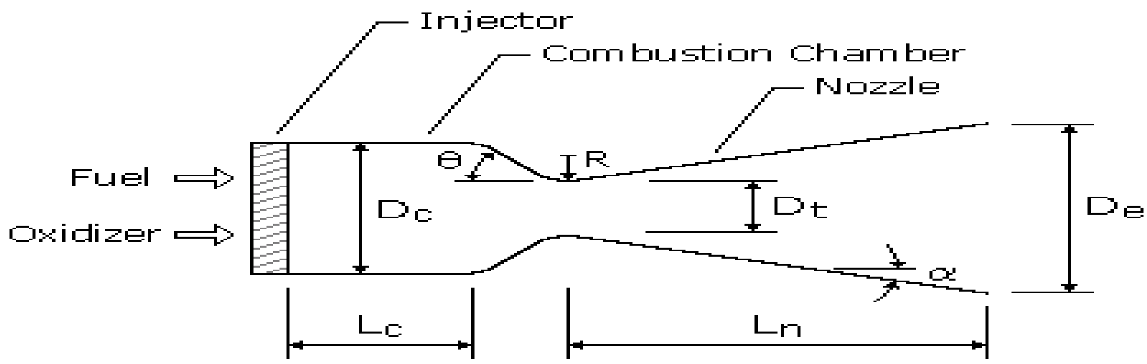
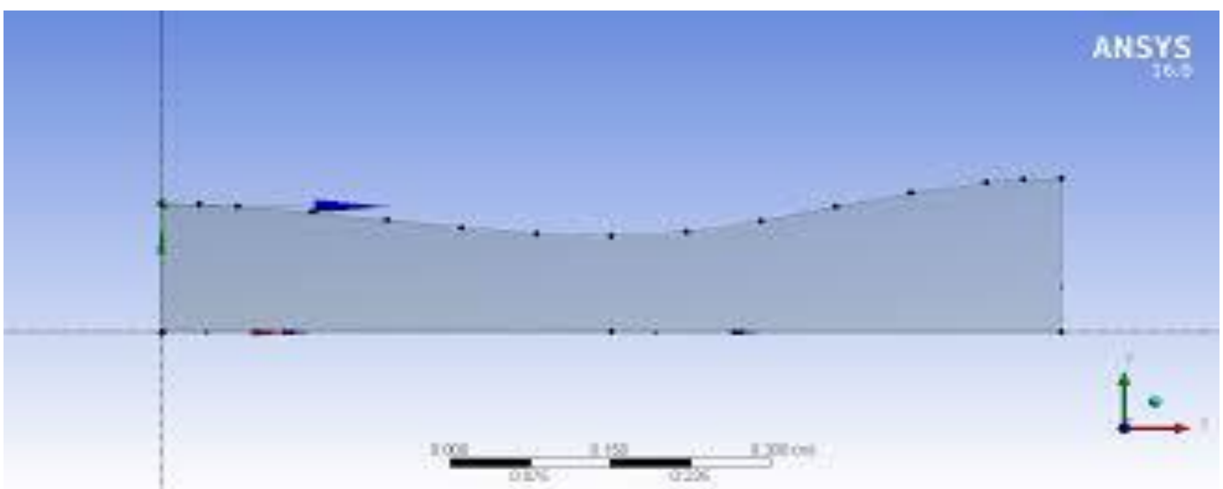


Figure 1.4

Literature review of rocket nozzle and basic design

Literature review of rocket nozzle and basic design

Laval nozzle is one of the most efficient nozzles presents which is the first narrow increasing the speed of the gas present in it as area decrease the pace of propellent increase .the shape of the nozzle is an important process in the expansion process. Here we have used the optimal divergent part with a different angle to maximize the speed and velocity of gas which exhausts out of the rocket nozzle. the inlet and boundary condition is kept at the constant position where it variable is the divergent part of the nozzle .paper has done do study the behavior fluent coming out nozzle affect Mach number, temperature, static pressure, and velocity, etc



Measurements of unit and dimension

1. Inlet diameter: 60mm, 2ition.Outer diameter: 80 mm, 3.convergent Angle: 35 degree, 4. Length of Nozzle: 225mm, 5.Mass flow Rate: 820, 6. Divergent angle: 15 degree, 7. Inlet temperature: 3300

Boundary condition

1. Axis - Axis boundary condition Wall - Wall 2.boundary condition –No-slip condition 3. idel gas as fluent

The Principle, Procedure, and Design

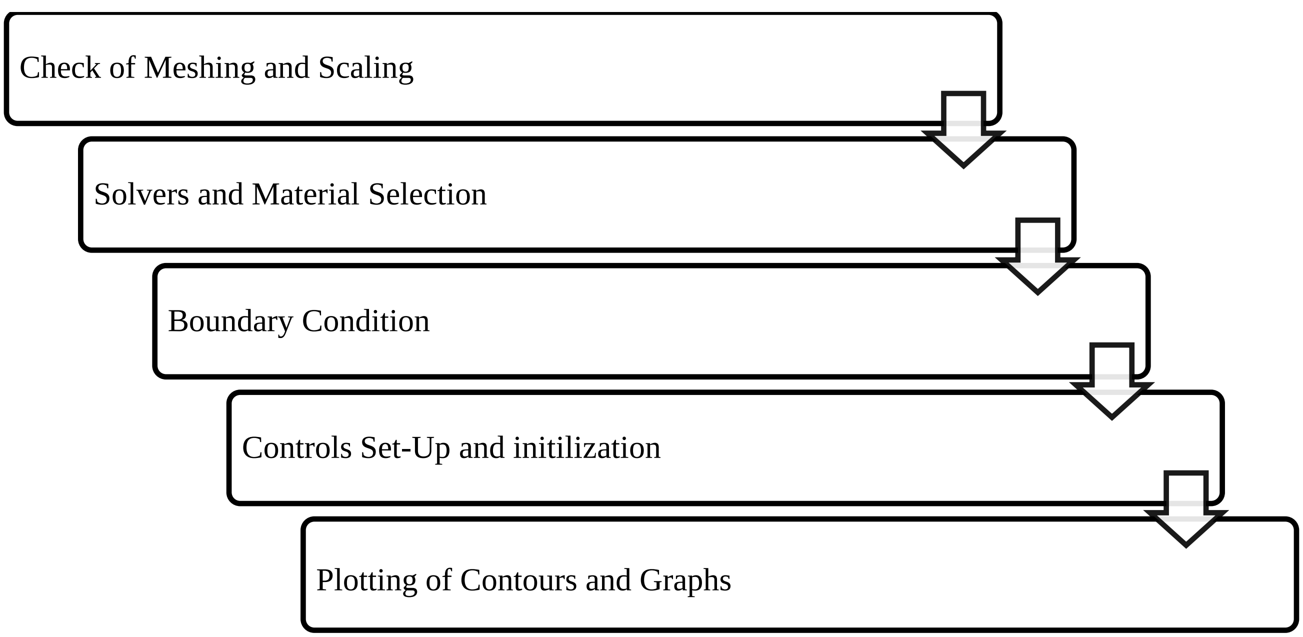
Study of the being carried on numerical and experiment analysis using the Navier stroke equation. Expansion ratio and other boundary conditions are being taken on a basis literature survey. The process is done using the computation work on ANSYS 16. A convergent-divergent nozzle contour can be explained by following the equation

$A=0.1+2X$; $(-0.5<X<0.5)$ where A=cross sectional Area of nozzle(square meter) and X=axial distance from throat that varies

Nozzle contour has drawn using the above equation for analysis the ANSYS 16 software is being considered. The mathematical model consists of the differential equation which governs the character of the system along with the boundary condition to start ANSYS 16 it is compulsory to check the meshing is correct so we have chosen the defined model and the boundary condition we also deal with energy equation which is used govern the nozzle and the fluent to flow. Nozzle show flow to convergent to the throat and from thorat to the divergent area where changing the angle used to get a different result. As nozzle diverges flow will increase due to design and once the flow exit from divergent part the flow become supersonic .supersonic flow is being calculated using equation

$dA/A=(M^2-1)\ du/u$

The procedure of the whole analysis is as follow:



Result and discussion

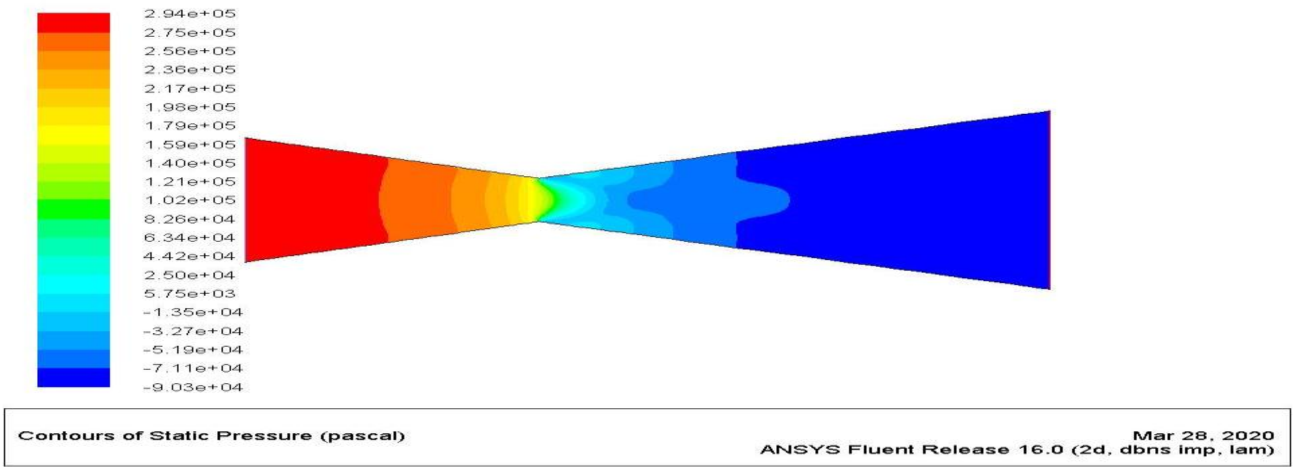


Fig 2

Static pressure (fig2) showing a contour provides information as we move left to right part(convergent to divergent part) of nozzle we will see a decrease of static pressure .At inlet ,static pressure is 1.59e+0.5 and exit 2.50e+0.5 whereas at throat it is 1.02e+0.5.

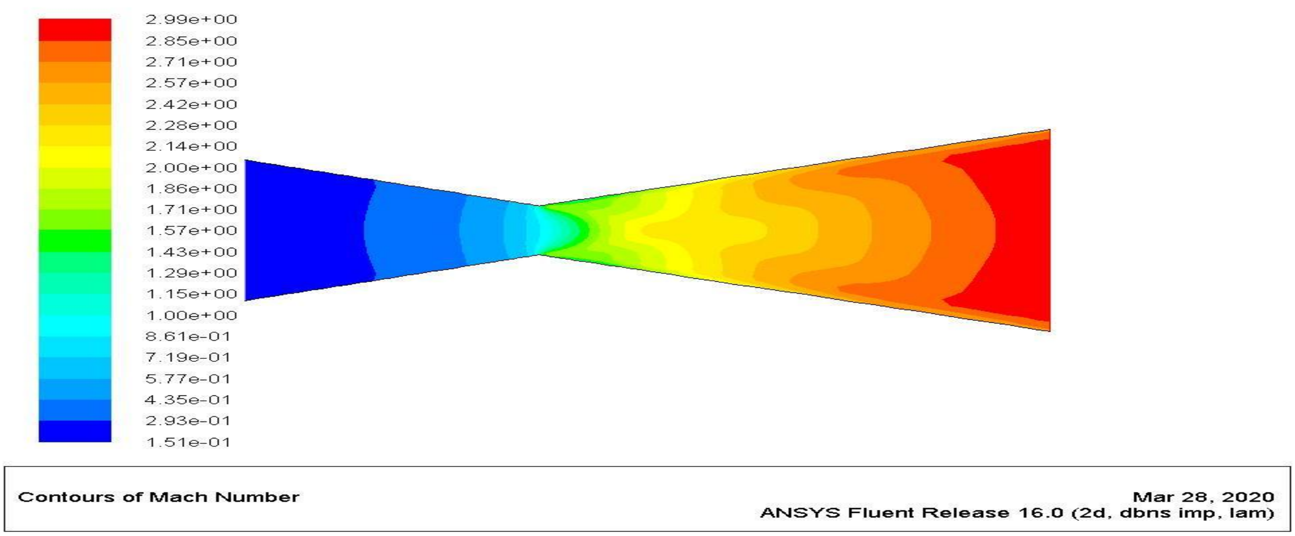


Fig 3

As we see above the Mach number has a marginal increase of 0.28 from the inlet of the nozzle to the outlet of nozzle, which helps the device to increase the velocity at exit due shock. At the inlet, velocity is without the shock, due which velocity remain less as compare to outlet speed of the fluent

Conclusion

With help of the model, we can see different parameters showing the same characteristic due to shock the velocity increase of fluent inside the nozzle when it moves out of the divergent part of the nozzle and become supersonic .mach number contour effect simulation also shows an increase of velocity at exit nozzle. And it gives validation by even restriction the nozzle show acceptable result.

References

1. Biju Kuttan P e.tal:"Optimization of Divergent Angle of a Rocket Engine Nozzle Using Computational Fluid Dynamics" The International Journal Of Engineering And Science (Ijes), Volume 2, Issue2, Issn: 2319 – 1813 Isbn: 2319 – 1805, 2013, pp 196-207.
2. Launder and Sharma, Letters in Heat and Mass Transfer, 1974, P131-138
3. KM Pandey and S K Yadav, "CFD analysis of a rocket nozzle with two inlets at Mach 2.1" JERD Vol No 2, 2010
4. Nazar Muneam Mahmood Simulation of Back Pressure Effect On Behavior Of Convergent Divergent Nozzle Diyala Journal of Engineering Sciences ISSN 1999- 8716Vol. 06, No. 01, March 2013 pp. 105-120,
5. Xiao, Q., Tsai, H.M. and Papamoschou, D., Numerical Investigation of Supersonic Nozzle Flow Separation. AIAA Journal, Vol 45, No 3, March 2007.