CFD ANALYSIS OF CONVERGENT AND DIVERGENT NOZZLE

Rajat Mishra, Devendra Lohia

Department Of Mechnical Engineering, Rama University, Kanpur

Abstract: A nozzle is a very essential device that is used to control character of the fluid. The main purpose of the nozzle is to increase the velocity in one way or another. De Laval nozzle is a converging-diverging nozzle which has the ability to convert the chemical energy (high pressure) into kinetic energy (high velocity and low pressure). De - Laval nozzle has mainly 3 parts such as throat, diverging part, and converging part. Expansion in C-D nozzle has been studied and analyzed by experimentation moreover as numerically by numerous researchers with an objective to optimize the performance beneath given conditions. Within the gift work, supersonic flow through the rocket nozzle has been simulated mistreatment numerical methodology. The analysis has been performed keeping the same input and according to the shape of the nozzle. Our objective is to investigate the best suit nozzle which gives high exit velocity among the different cross-sections considered. The main aim of this paper is to a proper comparison with theoretical data to determine the behavior of fluid during the movement of fluid inside the nozzle. Therefore CFD analysis is being done using ANSYS 16. The paper contains a proper analysis of the convergent-divergent nozzle. Analysis of Mach number and velocity is done inside the nozzle

Keywords: Mach numbers, ANYSY 16, C-D Nozzle

Introduction

The nozzle is an important device that is used in many areas such field of aerospace, a field of a power plant, etc from control the rocket velocity. Convergent and Divergent nozzle is used to accelerate the speed of fluid (present inside the nozzle) to supersonic speed according to pressure at exit. The nozzle converts the low velocity, high pressure, high-temperature gas in the combustion chamber into high-velocity gas of lower pressure and low temperature. Our analysis is carried out using by using software such as ANSYS 16.

Let a nozzle is used to get supersonic speed by the low speed at the inlet. The Mach number which gets an increase from inlet M=0 to M>0 at the outlet .nozzle always converges at the subsonic part and diverges at the supersonic part. This is pretty much clear that the Mach number is equal to unity (M=1) at the throat. Therefore sonic speed is always achieved at the throat. Fig 1 given below give a clear concept of it. The design of this nozzle is obtained from the area-velocity relation (DA / DV) = -(A/V)(1-M^2) where MA is the Mach number.

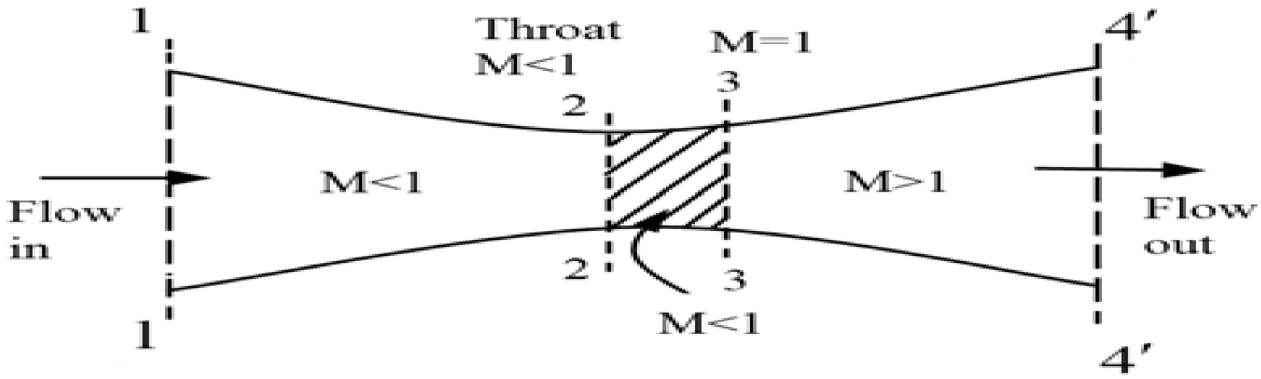
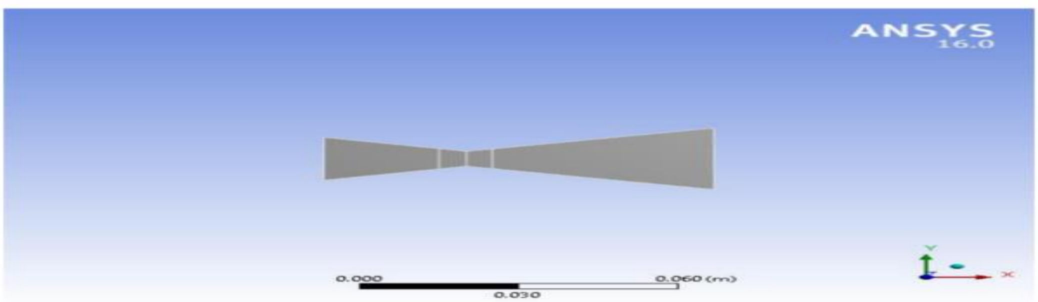


Fig 1

In this paper, the flow has been modeled and analyzed at supersonic Mach number from the CD Nozzle. A decrease in the area mean increase of pressure along with velocity and increase the area act as vice a versa. We have to maintain favorable pressure throughout the nozzle.



Unit and measurement

1. Inlet Diameter- 60 mm , 2. Outer Diameter- 80 mm

3. c-d length-180mm , 4.Mach number-1

Principal, Design process Analysis and Equation

The analysis and design process did in ANSYS 16 given in the below figure explain in a structural manner. The clear picture is providing how the whole process is being performed. Fig 2 explains below how the whole process of this paper is being performed.

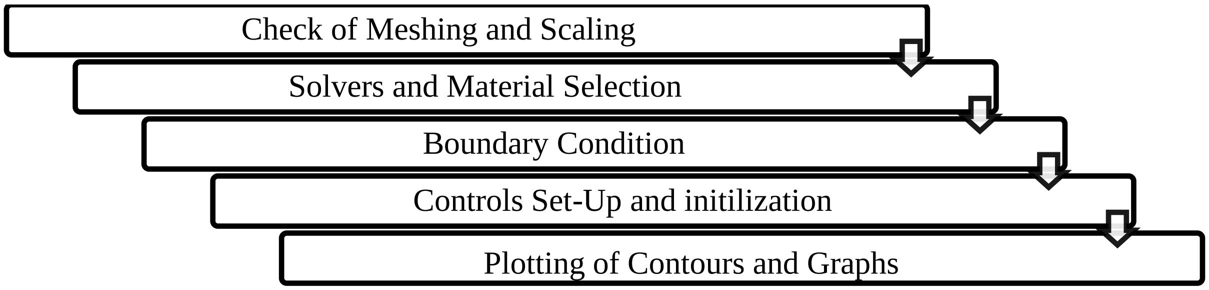
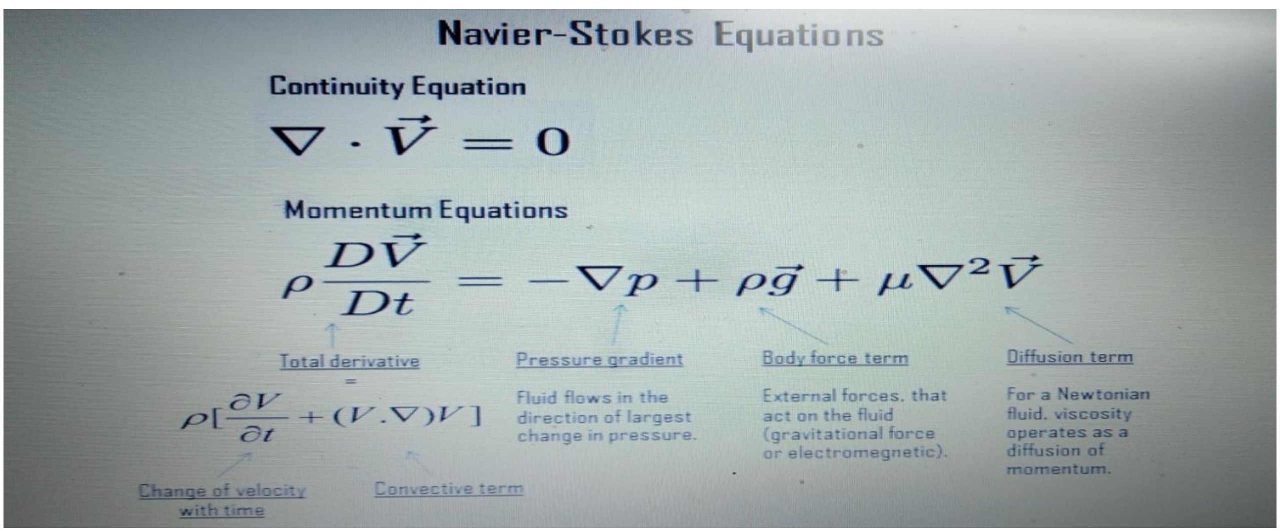


Fig 2

First, the modeling is done then followed by meshing, and then all the pre-processing is done. Meshing is being after modeling in ANSYS 16.module which is generally used to perform meshing is fluid flow .Automatic process of meshing is being selected to perform the task. Navier- Stroke equation being used to calculate Mach number and velocity calculated.



Boundary conditions

1. Shear condition: no-slip, 2. At Inlet : Axial velocity: 46.85m/s , temperature:300K

3 Pressure Outlet : a) Backflow Temperature : 300K , b)gauge pressure:2785 Pascal

4. Initial gauge pressure: 100000 Pascal

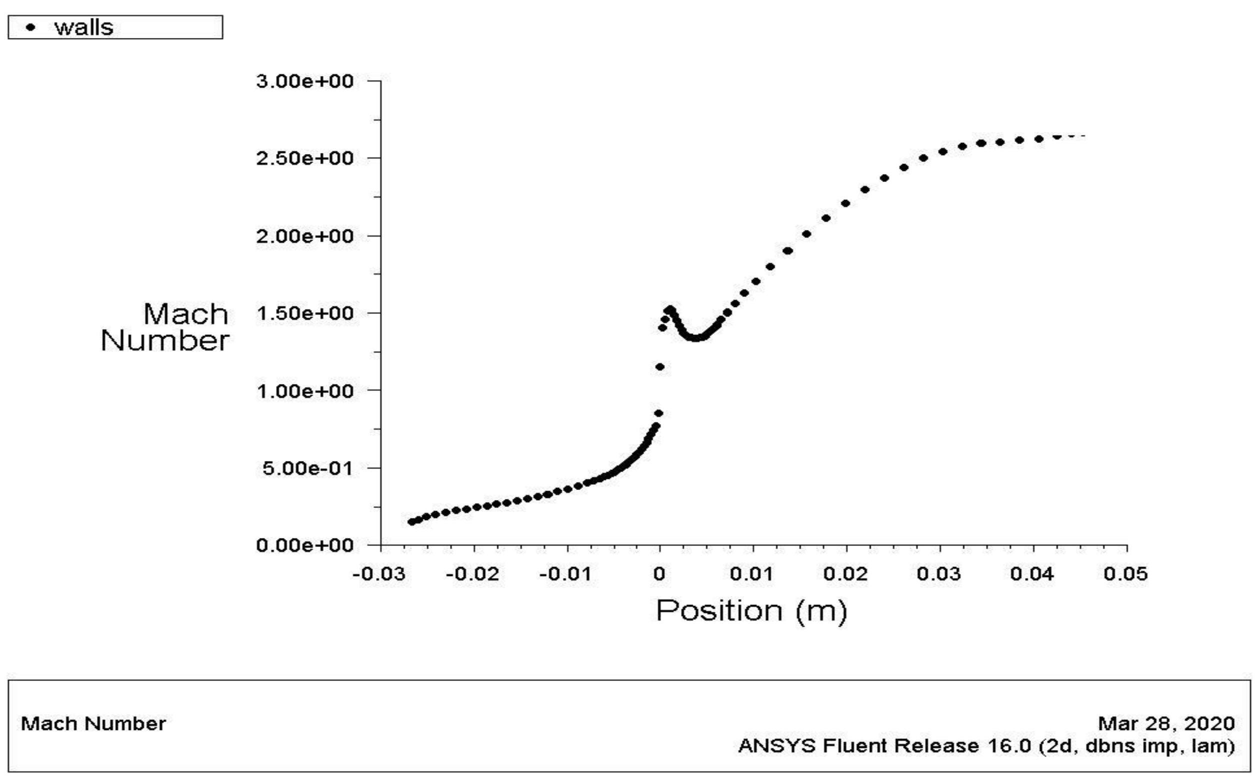
5. Outlet gauge pressure: 101785 Pascal

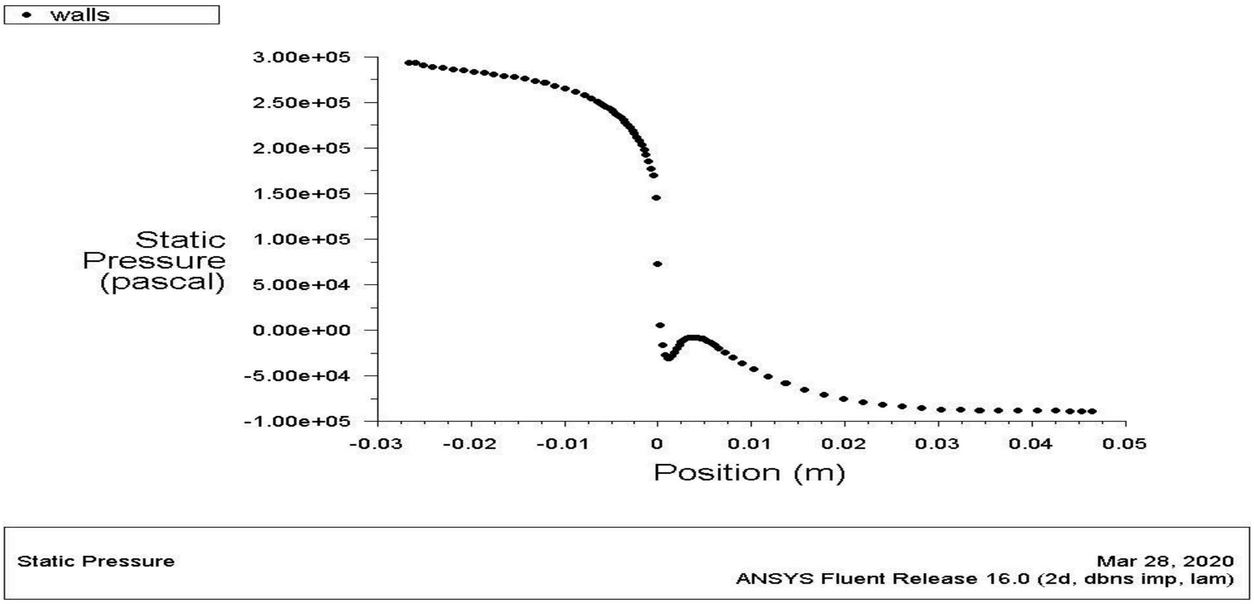
Two dimensional C-D Nozzle

2D model of nozzle is being designed using ANSYS workbench is being implemented in the model. The above-given boundary condition is being defined considering the edge of the model to help the fluent to get flow easily and smoothly in the nozzle. The process of meshing is done using the workbench is used and creates the structural mesh, and the number of elements is used to create a mesh in a closed edge about 4824 binary code is generated using the two-dimensional model. This helps out giving a logical conclusion to the model.

Results

In the given graph position VS Mach numbers show that Mach number is 1.50e+00 when a position is at o. Afterward, the movement of the graph shows the constant growth which gave the conclusion that there is a continuous increase in the velocity from inlet to outlet. This also shows there is no shock, while at outlet Mach number is 2.50e+00.





The static pressure VS position also shows the continuous drop of static pressure from inlet to outlet section.

Conclusion

The nozzle model is being used to find out-static pressure and velocity. From the Mach number graph, above show the velocity of fluid inside nozzle increase from inlet to outlet .constant growth shows no shock at the throat section of the nozzle. And static pressure graph which continuous drop that fluid accelerate inside the nozzle.

Reference

1. Wen, Chuang, Xuewen Cao, Yan Yang, and Jing Zhang. "Supersonic swirling characteristics of natural gas in convergent-divergent nozzles." *Petroleum Science* 8, no. 1 (2011): 114-119.
2. Ali, A., Neely, A., Young, J., Blake, B., & Lim, J. Y. (2010). Numerical Simulation of Fluidic Modulation of Nozzle Thrust. In 17th Australasian Fluid Mechanics Conference (pp. 5–8).
3. Anderson, J. S., & Williams, T. J. (1968). Base pressure and noise produced by the abrupt expansion of air in a cylindrical duct. Journal of Mechanical Engineering Science, 10(3), 262–268.
4. A. K Pandey , K. and A. Singh. Singh, CFD analysis of conical nozzle for mach 3 at various angles of divergence with fluent software. International Journal of Chemical Engineering and Applications, 2010. 1(2): p. 17.