CONVERGENT-DIVERGENT NOZZLE USING ANSYS FLUENT

Visualization of Mach number, Velocity and static pressure

Dinesh Poluru Baskar
dineshpb0812@gmail.com

Table of Contents

Figure of Contents

Abstract

This report provides an information about the simulation of Convergent – Divergent nozzle using Ansys Fluent software. This project is purely based on self-motivation in-order to gain in-depth knowledge over the software. This report discusses about the setup, simulation results and observations done on the same.

Introduction

The simulation of Convergent – Divergent nozzle is a common problem where engineers study to understand the behavior of Mach number in different regions. In this project, the study of Mach number at different section of the CD nozzle as well as the velocity and static pressure has been studied and observed and provided with the explanation for each section. The CD nozzle are drawn with already available dimensions in the internet and the gauge pressure are taken from the isentropic flow table.

Convergent – Divergent Nozzle

CD nozzle is most commonly used for aeronautics and combustion purposes. This is popularly known as De Laval nozzle which was used by Gustaf De Laval for his impulse turbine engines. The layout of the CD nozzle has three parts namely: Convergence section, the throat and the divergence section. The gas passes from the convergence section and goes through the throat section and exits through the divergence section.

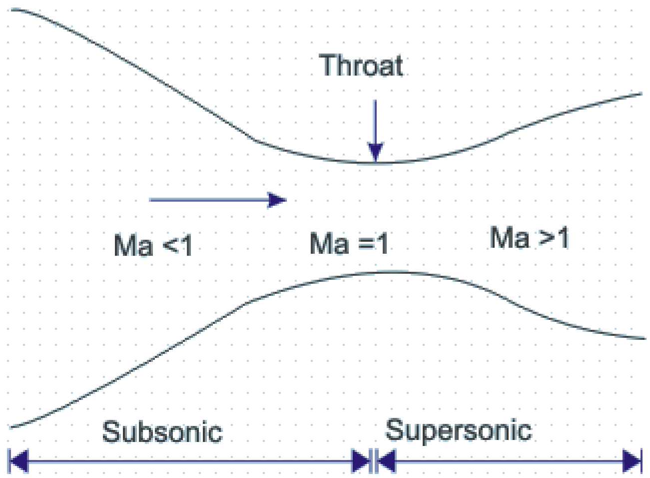


Figure 1: CD nozzle [(nptel, n.d.)].

The gas flow from the convergence to the divergence side, these sections are divided into three types namely:

- Subsonic, where Mach number is less than 1; M<1.
- Sonic or transonic, where Mach number is equal to 1; M=1.
- Supersonic, where Mach number is greater than 1; M>1.0
- Hypersonic, where Mach number is greater than 5; M>5.0

Mach number is the quantity of measuring the speed of the airflow or velocity of the object with respect to the speed of the sound. It is denoted by 'M' or 'Ma'.

Ma = $\frac{v}{c}$

Where, v = velocity of the object or flight

C = speed of sound, 343m/s.

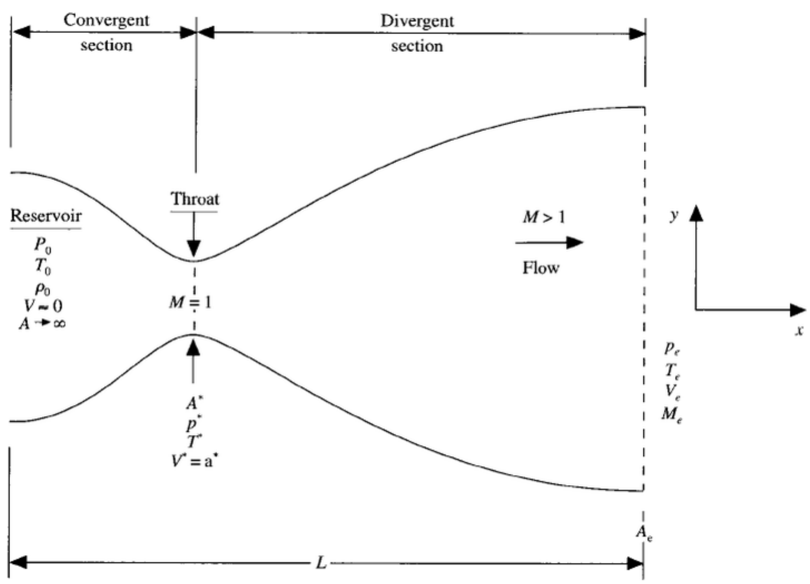


Figure 2: CD nozzle with parameters..

From the above diagram,

P_0 = Stagnation pressure at the reservoir.

L = Distance throughout the nozzle.

A^* = Throat area.

P_e = Exit pressure.

When there is flow of gas, above the point of P_0 the pressure increases as a result the velocity decreases so as the Mach number remains less than 1. As the flow reaches the throat area, where the area is decreased the pressure drops down and the velocity increases. The highest velocity and lowest pressure will occur at the throat. At the divergence section, if we assume the whole CD nozzle to be in subsonic condition the area in the divergence nozzle increases as a result velocity decreases and pressure increases. This is same as the case with convergence side, which is Mach number still at less than 1. In a CD nozzle the supersonic flows occur at the divergence side only when there is a chocked flow at the throat. This chocked flow can be achieved when the throat pressure to the stagnation pressure is 0.5283 for $\gamma = 1.4$.

Procedure

The procedure involves creating the geometry with specified dimensions and creating a face split and face mesh for the geometry and setup the physics in the Ansys fluent tab. The process flow process has been discussed detailed in the upcoming sections.

Creation of Geometry:

The geometry for CD nozzle is created by using simple line tools available in the Ansys designer modular tab. The required dimensions are given for the CD nozzle and required angle for the convergence and divergence has been specified.

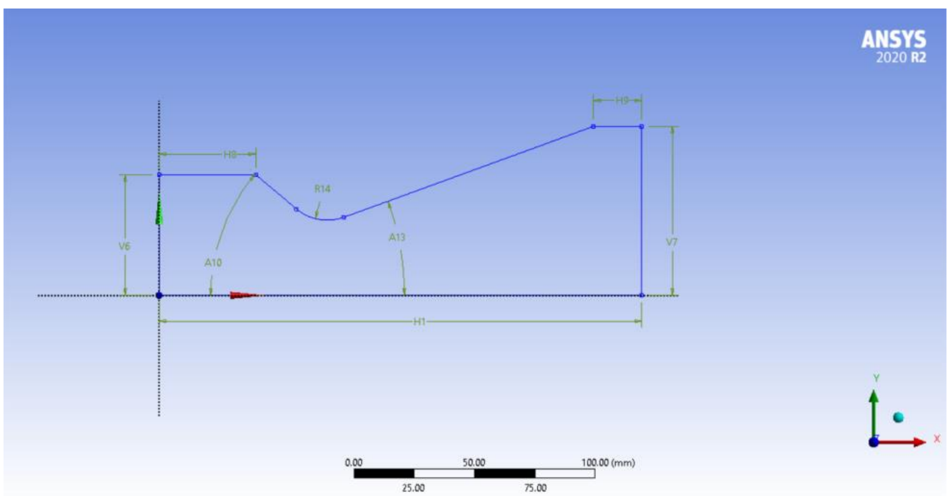


Figure 3: Geometry.

Details View	
Show Constraints?	No
Dimensions: 8	
A10	40 °
A13	20 °
H1	200 mm
H8	40 mm
H9	20 mm
R14	20 mm
V6	50 mm
V7	70 mm
Edges: 8	

Figure 4: Dimensions for Geometry.

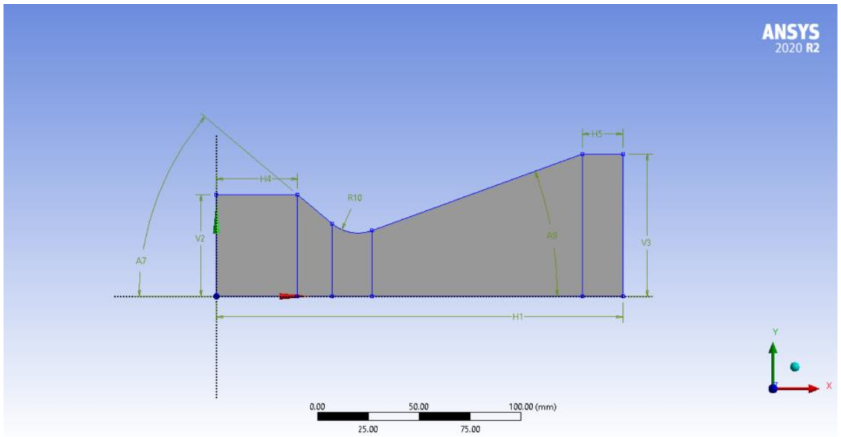


Figure 5: Face split.

In the above shown picture there are some lines draw for the separation of the CD nozzle into different section. **Face split** option is used to separate every section of the nozzle into desired meshing type. This is done because the flow the happening at the face of the CD nozzle since this is a 2D nozzle and the face has to be split and the boundary has to be sized according to the type of geometry. At the throat section the sizing is done with maximum number of divisions in order to solve the solution accurately by the solver. The solver here we are using is a compressible flow with heat equation solver.

Mesh:

In this method Face split is used splitting up the geometry into 5parts which connects the convergence and divergence section. The edge sizing option is used for providing the edge sizing for every edge as required. At the throat section the edge sizing is maximum and at the curvature the option for curvature is set on so that the solver type can identify the curvature.

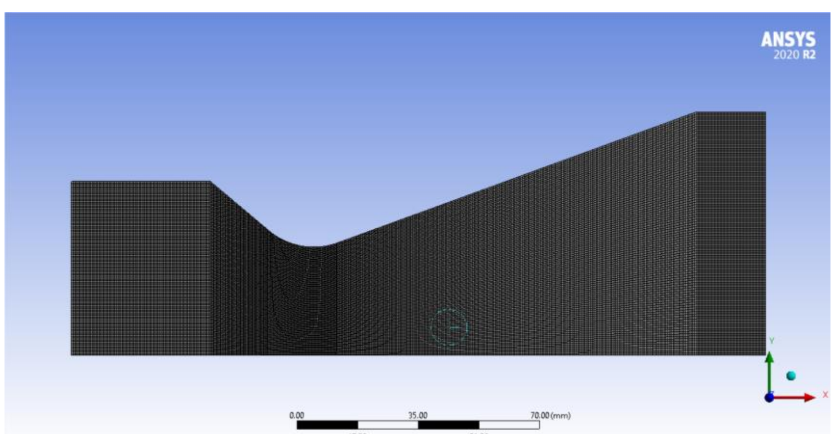


Figure 6: Face Mesh.

At the throat section the meshing is maximum than the other section, this is because at the critical area like throat section the solver has to solve very efficiently. The axis, wall, inlet and outlet names have been specified for the solver to identify the position where it is being solved.

Setup:

In this setup **Density-based** solver is chosen and **axisymmetric** is applied since the geometry we created is an axisymmetric one. The **energy equation** is switched on as well as the solver is kept as **inviscid flow** because the flow is a compressible flow (variable density).

For the materials the ideal gas is chosen with the default specific heat and molecular constant. In the boundary condition, the inlet has been chosen as pressure inlet and the gauge pressure has been provided with the atm pressure 101325 pascals with initial gauge pressure 100000. For the outlet condition the gauge pressure has been set to 2814 by referring the isentropic flow table for gamma = 1.4. Then the initialization is done from the inlet and calculation is set to run.

Results and Discussion:

The simulation is setup and the solution is run for about 1500 iterations and the results are provided and observed.

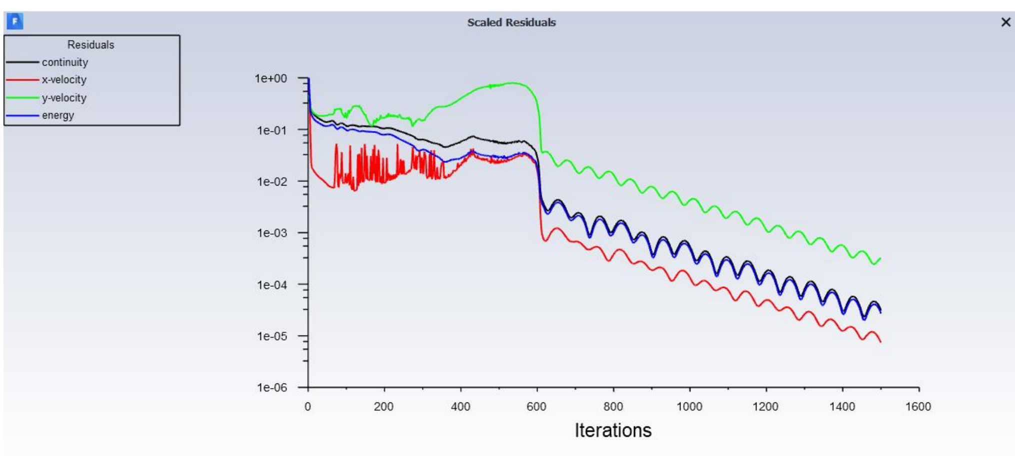


Figure 7: residual.

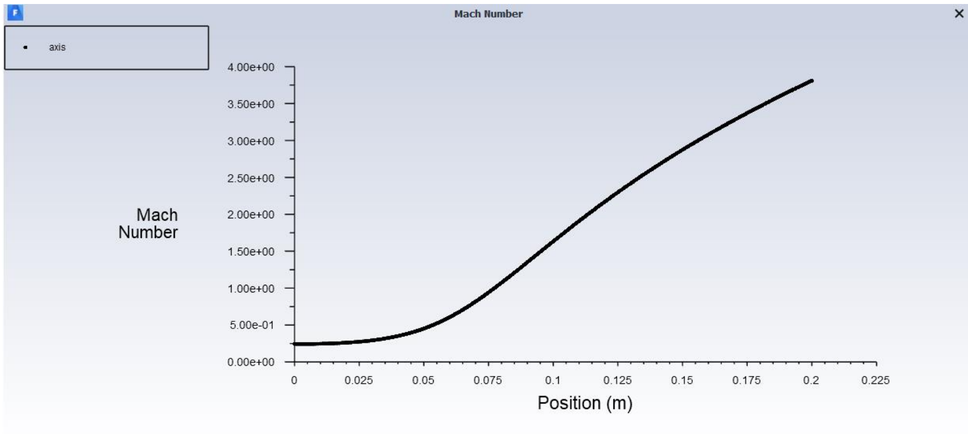
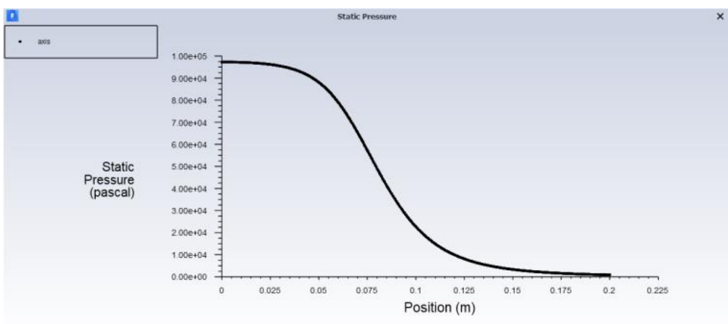


Figure 8: Mach number.

The Mach number is less than one at the starting of the graph which represents the subsonic condition in the nozzle. As it moves forward the Mach number reaches the throat with sonic condition and supersonic condition.



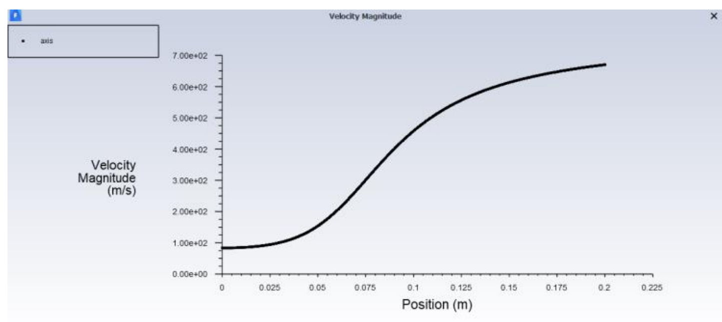


Figure 9: Pressure gradient & velocity gradient.

The pressure at the throat is maximum while the velocity is minimum and when the pressure at the convergence divergence section is minimum the velocity is maximum.

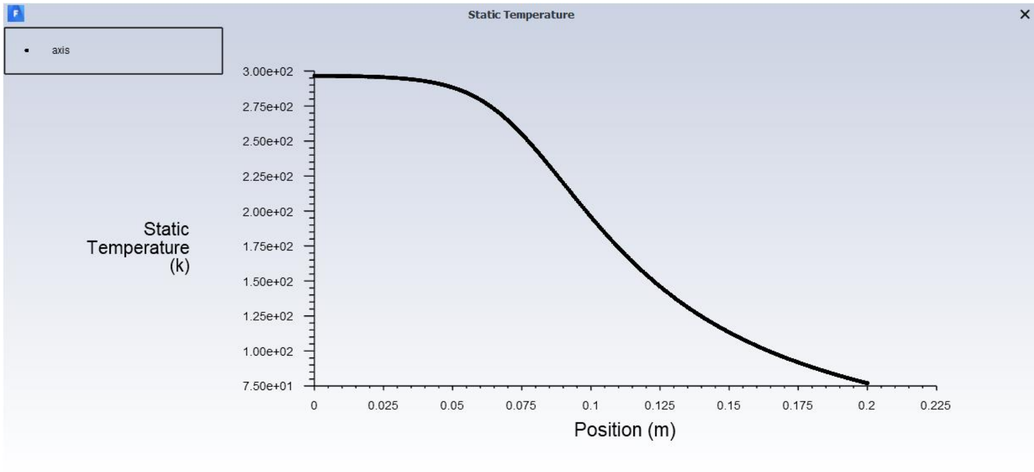


Figure 10: Temperature gradient.

The temperature is maximum at the convergence section and it reduces gradually at the throat and the divergence section. The contour images are shown below for better understanding.

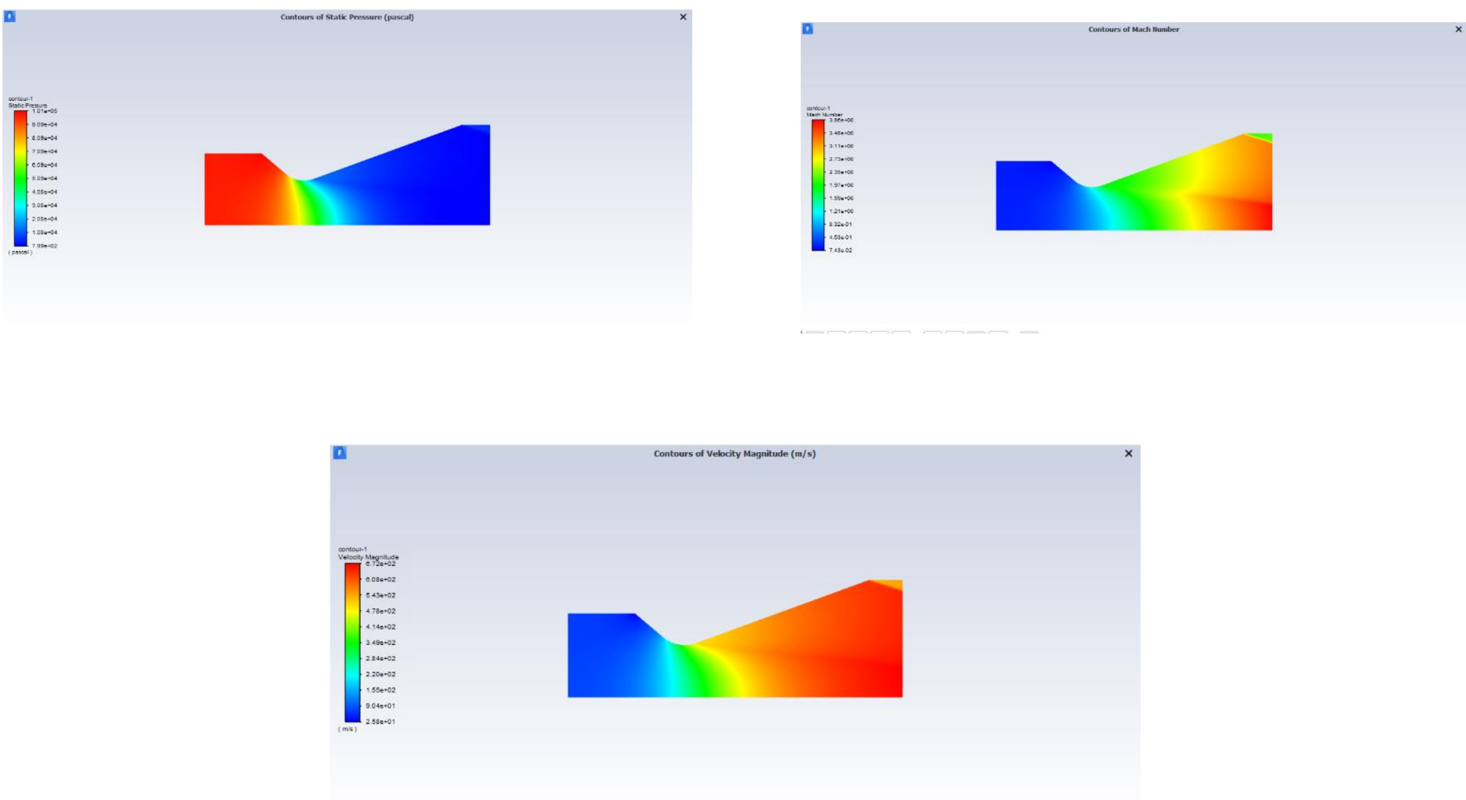


Figure 11: Different types of contour.

Conclusion

This project is an explanation of creating a simulation for Convergence – Divergence nozzle using Ansys fluent. Different types of contours and graphs have been displayed and explained. This project is a demonstration of Mach number, velocity and pressure gradient acting on the different section of the CD nozzle.

References

nptel. (n.d.). Retrieved from https://nptel.ac.in/content/storage2/courses/112104118/lecture-40/40_4_nozzle.htm