CFD Analysis of Convergent Divergent of Supersonic Nozzle

Vipul Shakya , Dr Devendra lohia

Department of Mechanical Engineering, Rama University; Mandana, Kanpur

---

Abstract: 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 parameters like physicist number, static pressure and shocks square measure discovered for cone-shaped and contour nozzles using axisymmetric model in ANSYS FLUENT 16® computer code. The occurrences of shocks for the cone-shaped nozzles were discovered alongside the opposite parameters for various divergent angles. The parameters beneath observation square measure compared there upon of contour nozzle for individual divergent angles by maintaining the water, outlet and throat diameter and lengths of oblique and divergent parts as same. The convergent portion and throat diameter square measure unbroken constant across the cases. The phenomenon of shock was pictured and also the results showed shut similitude in formation of physicist disk and its reflection patterns as reportable in numerous experimental studies on growth in cone-shaped C-D nozzles with lower divergent angles. No occurrence of shocks is discovered with higher divergent angles. Results delineated higher exit speed and better degree of flow separation with contour nozzles compared to it with corresponding cone-shaped nozzles.

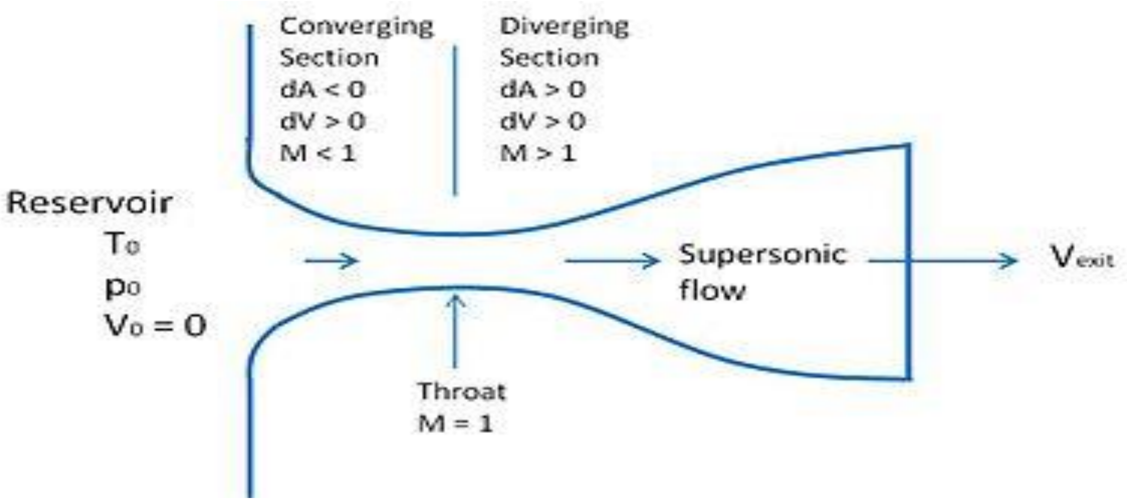
Keywords: C-D nozzle, ANSYS FLUENT 16

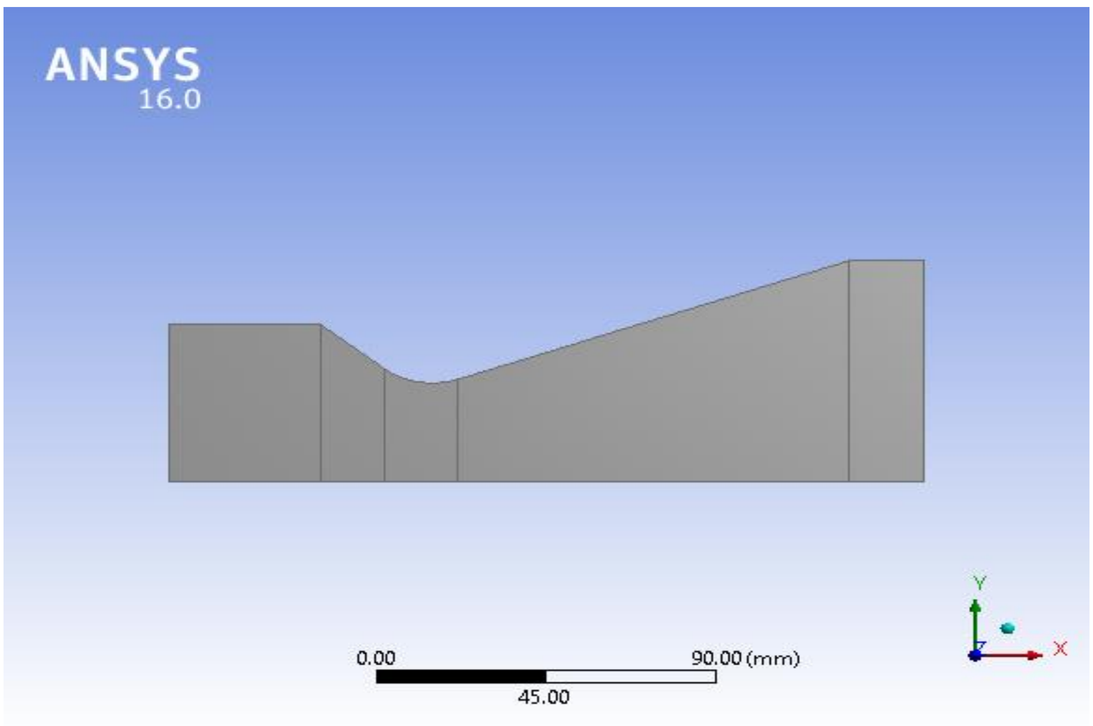
Introduction

The nozzle is widely used in various areas, from rocket to control the velocity, direction, and required parameters of Nozzles are designed to operate in all flow regions the supersonic nozzle remains a challenging task in fluid mechanics. In a supersonic nozzle, not only the physical parameters of the nozzle play an essential role, but the thermodynamic parameters of the flow also play a crucial role Nozzle known as de Laval nozzle is the most common and produced in the combustion chamber (rocket motor) is converts high pressure, high temperature, and low velocity(subsonic) gases into low pressure, low temperature, and high velocity (supersonic) gases, hence producing high thrust [1]. To achieve the desired objectives, de Laval found that the most efficient conversion occurred when the nozzle area converges until the throat area, where the flow travels at sonic velocity, followed by a divergent section of the nozzle which accelerates Studies have been carried out by keeping the Mach number constant and varying the divergence angle to observe its effect on various parameters like velocity, temperature, and pressure Authors in designed a nozzle by varying the exit Mach number and observed its effect on the length of the nozzle, variation in pressure and velocity while keeping the divergence angles and observed their effect on Mach number, angles up to a specific limit provide better results. In supersonic nozzles, the sudden expansion of gases can produce design must be optimized to achieve maximum thrust without simulations have been carried out to choose the best nozzle. A convergent-divergent nozzle is meant for attaining speeds that square measure larger than speed of sound. the planning of this nozzle is obtained from the space-velocity relation (district attorney / dV) = -(A/V)(1-M^2) wherever M is that the ratio (which means that the quantitative relation of native speed of flow to the native speed of sound) A is area and V is speed. From the fig given below we are able to observe that

(a). The decrease in space leads to the rise of pressure and reduce in velocity as seen within the on top of figure at the entry of the nozzle.

(b). The rise in space leads to increasing the rate at the exit of the nozzle by decreasing the pressure. Also we are able to verify that M1 leads to supersonic speeds .One vital purpose is that to achieve supersonic speeds there's a desire to keep up favorable pressure ratios across the nozzle.





Units and parameters

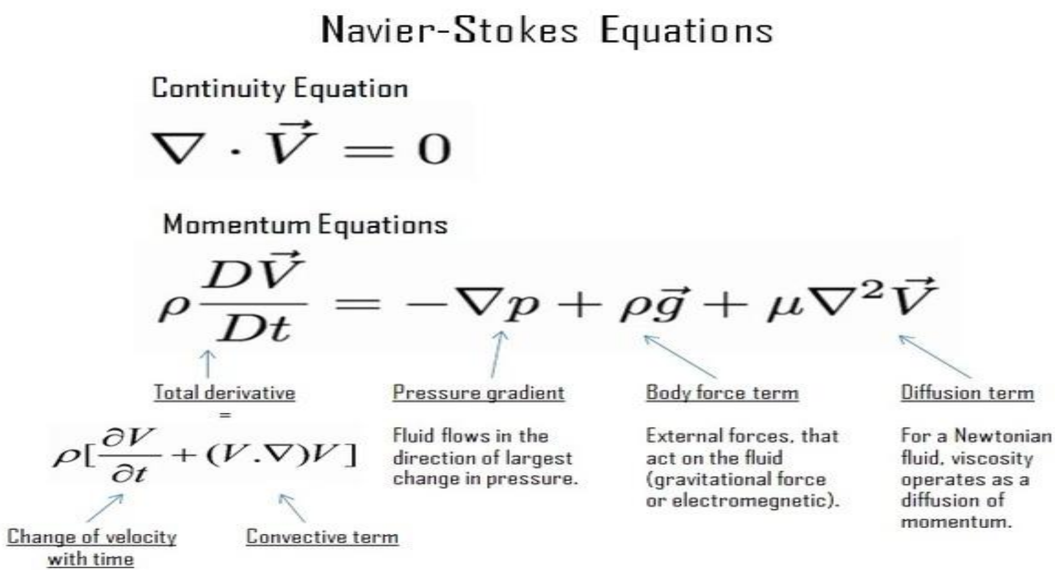
1. Mach number - 1
2. Inlet Diameter - 50mm
3. Outlet Diameter - 70mm
4. C-D length - 200mm

Two-dimensional CD nozzle

For two-dimensional model analyses ANSYS workbench is implemented to model the CD nozzle in case of 2-D planar body. Boundary conditions defined considering edges of the planar body to initiate the solution with perfect ideal gas flow. The 2D planar model of CD nozzle is depicted and closed form of finite element meshing is depicted. For meshing, ANSYS workbench used and created structural mesh, number of elements has used very high to create fine mesh in closed area at edge of the planar body. Total, 42521 binary nodes, were generated for 2D planar model.

Basic equation used

Meshing In ANSYS 16 After the modelling is completed the meshing is to be done. The module used to perform meshing is Fluid Flow (Fluent). The meshing method used here is Automatic Method and the mesh type is selected as All Quad. The overning equations used in mesh are as follows. It is assumed that there is a unique, single valued relationship between the generalized co-ordinates and the physical co-ordinates which can be expressed.



Boundary conditions

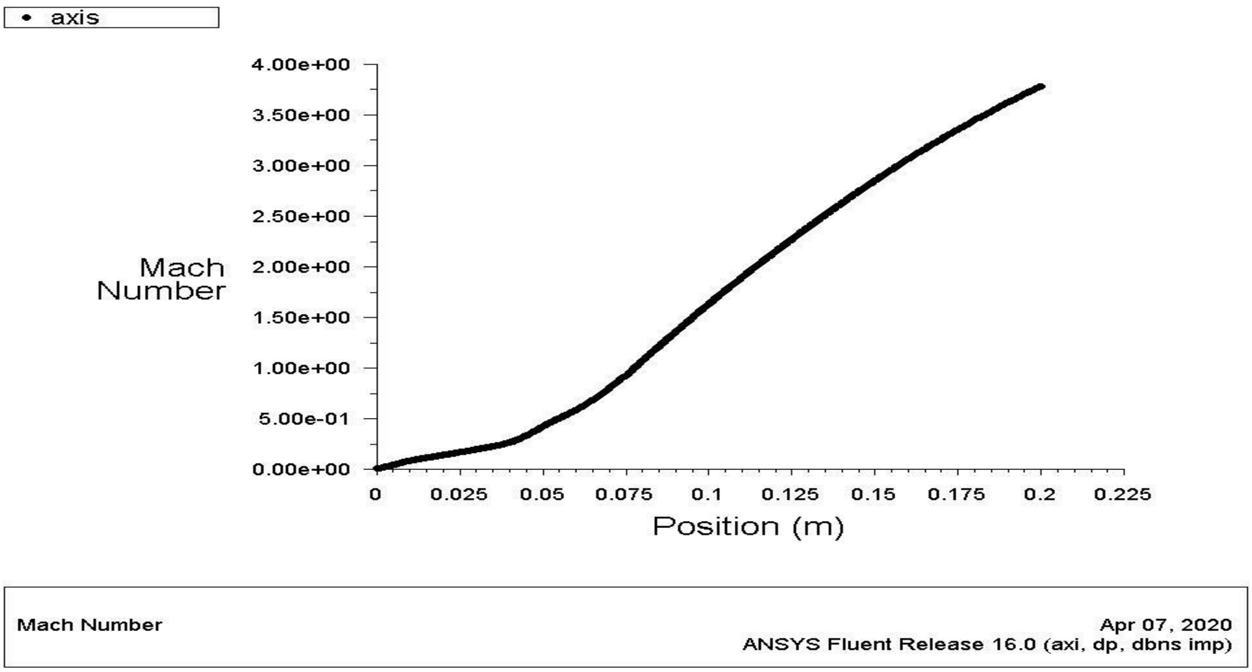
- **At velocity inlet**

1. Initial gauge pressure -100000 Pascal
2. Outlet gauge pressure -101325 Pascal
3. Temperature - 300K
4. Axial velocity -47.56m/s

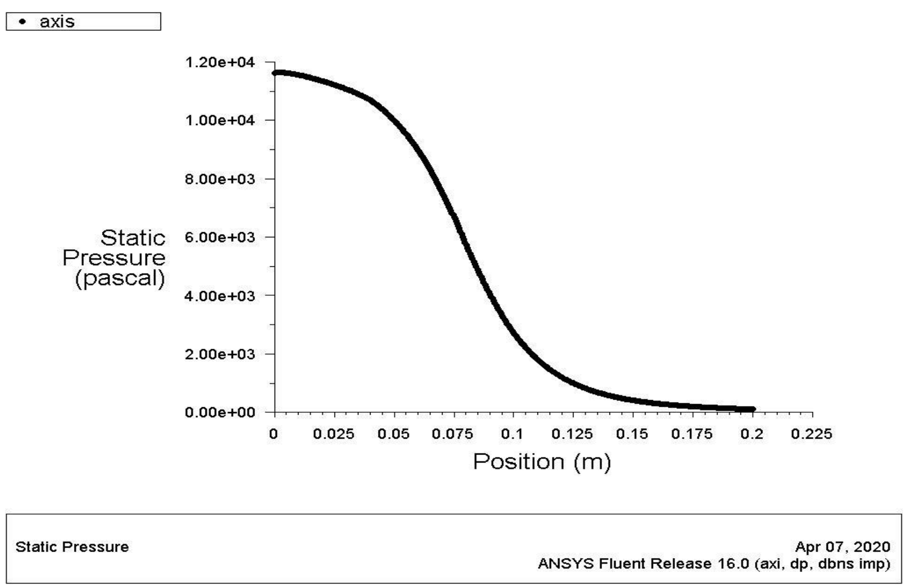
- **At pressure outlet**

Gauge pressure -2814Pascal
Backflow total temperature-300K

Result and Analysis



The Mach number V/s position plot also shows a continuous increase in the velocity from inlet section to the outlet section. And there is no sudden drop in velocity which shows that no shock is occurring in the nozzle. The exit Mach number of the nozzle is 3.90e+00 Mach at the axis.



The static pressure V/s position plot also shows a continuous drop in the velocity from inlet section to the outlet section. And there is sudden drop in velocity and the position of the nozzle is 0.2m at the axis.

Conclusion

A nozzle model was developed to determine the pressure. Various steps of the model were validated with good accordance with the experimental data and numerical results found in the literature.

From the mach number graph it shows the velocity increase from inlet to outlet.

References

[1] Madhu B P, Syed Sameer, Kalyana Kumar M and Mahendra Mani G:" CFD ANALYSIS OF CONVERGENT DIVERGENT AND CONTOUR NOZZLE" International Journal of Mechanical Engineering and Technology (IJMET) Volume 8, Issue 8, August 2017, pp. 670–677, Article ID: IJMET_08_08_073.

[**2**] Wikipedia contributors. (2018, August 16). De Laval nozzle. In Wikipedia, the Free Encyclopedia. Retrieved 18:53, October 20, 2018,

[3] Lijo, V., Kim, H. D., Setoguchi, T., & Matsuo, S. (2010). Numerical simulation of transient flows in a rocket propulsion nozzle. International Journal of Heat and Fluid Flow, 31(3), 409-417.

[**4**] Stevens, T., & Hobart, H. M. (1906). Steam turbine engineering. Whittaker and Company, pp. 24-27.