CFD Analysis of Convergent Divergent and Contour Nozzle

Vipul Shakya[1], Rajat Mishra[2], Aditya Gautam[3] and Mr. Devendra Kumar Lohia sir[4]

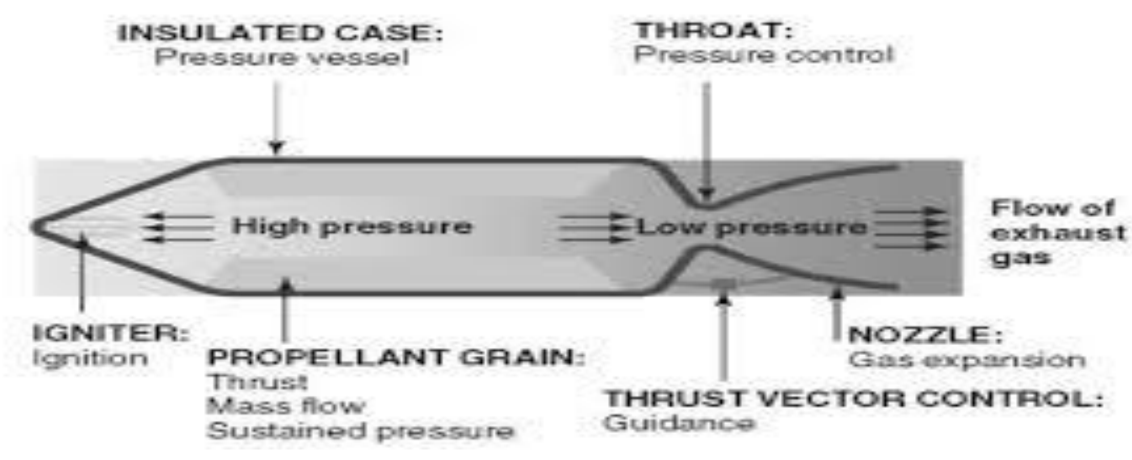
[1-4](Department of Mechanical Engineering, Rama University; Mandana, Kanpur)

Abstract: A rocket nozzle could be a automation that is meant to manage the speed of flow, speed direction and pressure of stream that exhaust through it. There area unit varied kinds of rocket nozzle that area depending upon the mission of the rocket. This paper contains analysis over a convergent divergent rocket nozzle that is performed by varied quantity of divisions in mesh.The parameters like philosopher number, static pressure and shocks are determined for cone like and contour nozzles using axis-symmetric model in ANSYS FLUENT 14® software system. The occurrences of shocks for the cone like nozzles were determined together with the opposite parameters for various divergent angles. The parameters underneath observation are compared therewith of contour nozzle for individual divergent angles by maintaining the recess, outlet and throat diameter and lengths of focused and divergent parts as same. The convergent portion and throat diameter are unbroken constant across the cases. The phenomenon of shock was unreal and therefore the results showed shut alikeness in formation of philosopher disk and its reflection patterns as according in varied experimental studies on growth in cone like C-D nozzles with lower divergent angles. No occurrence of shocks is determined with higher divergent angles. Results pictured higher exit rate and better degree of flow separation with contour nozzles compared to it off with corresponding cone like nozzle.

Key words: C-D nozzle, ANSYS FLUENT 14®, rocket nozzle, Mach disk

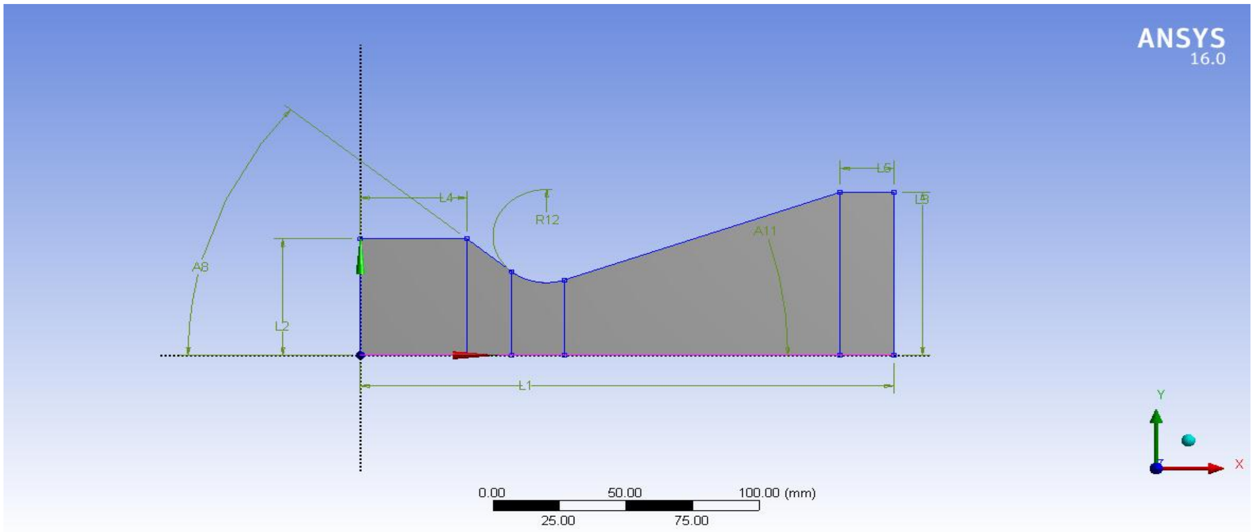
Introduction

A jet engine uses hold on rocket propellants because the reaction mass for forming a high-speed propulsive jet of fluid, typically high-temperature gas. Rocket engines are reaction engines, manufacturing thrust by ejecting mass rearward, in accordance with Newton's third law. Most rocket engines use the combustion of reactive chemicals to provide the mandatory energy, however non-combusting forms like cold gas thrusters and nuclear thermal rockets additionally exist. Vehicles propelled by rocket engines are normally referred to as rockets. Rocket vehicles carry their own oxidizing agent, not like most combustion engines, thus rocket engines is employed in a vacuum to propel satellite and flight missiles.



Rocket engine nozzle

A rocket nozzle may be a propellant nozzle (usually of the De Laval type) employed in a rocket to expand and accelerate the combustion gases created by burning propellants so the exhaust gases exit the nozzle at hypersonic velocities. Simply: the rocket (pumps and a combustion chamber) generates high, a couple of hundred atmospheres. The nozzle turns the static high hot temperature gas into apace moving gas at near-ambient pressure.



Dimensions

L_1 (C-D Length) -	210mm
L_2 (Inlet diameter) -	55mm
L_3 (outer diameter) -	75mm
A_{11} (Divergent angle) -	20♂
A_8 (convergent angle) -	40♂

Nozzle Design Methodology

A convergent divergent nozzle was considered air flowing high speed circular cross section area , A (square meter), that varies with axial distance from the throat, x (meters).

$$A = 0.1 + x\ 2; (-0.5 < x < 0.5)$$

Firstly, a convergent-divergent nozzle contour has drawn mistreatment the on top of equation. Then the meshing half is completed mistreatment ANSYS. For the analysis of this model FLUENT software is employed. On the nozzle flow behavior is obtained. Once face mashing and body filler at the mash half coordinate name was elite on the boundary named: Pressure recess, Pressure outlet, Nozzle wall, and also the axis. At the setup portion thinker sort was elite as density primarily based, time was steady and second house was elite as axisymmetric. Then in model section energy equation was on and viscous as elite as inviscid. For the nozzle, air is taken as a working medium and density was elite as a perfect gas. Mistreatment the physical property flow relations space ratio of the dual-bell nozzle makes up my mind. At velocity inlet section, the supersonic gauge pressure at the recess was given 100100 Pa, stagnation temperature at the recess was 300 K and outflow gauge pressure 101330Pa also the static pressure at the exit was three,2800 Pa. an answer methodology section abstraction discretization gradient was given as statistical procedure cell primarily based. Hybrid format was elite as resolution format and the knowledge was computed from the pressure recess. Once running calculation taking 1500 range of iteration the physical property parameter was calculated. Since the nozzle intends to diverge the flow will continue increasing. Once this flow reaches the exit, it becomes supersonic. This result was calculated using the following equation:

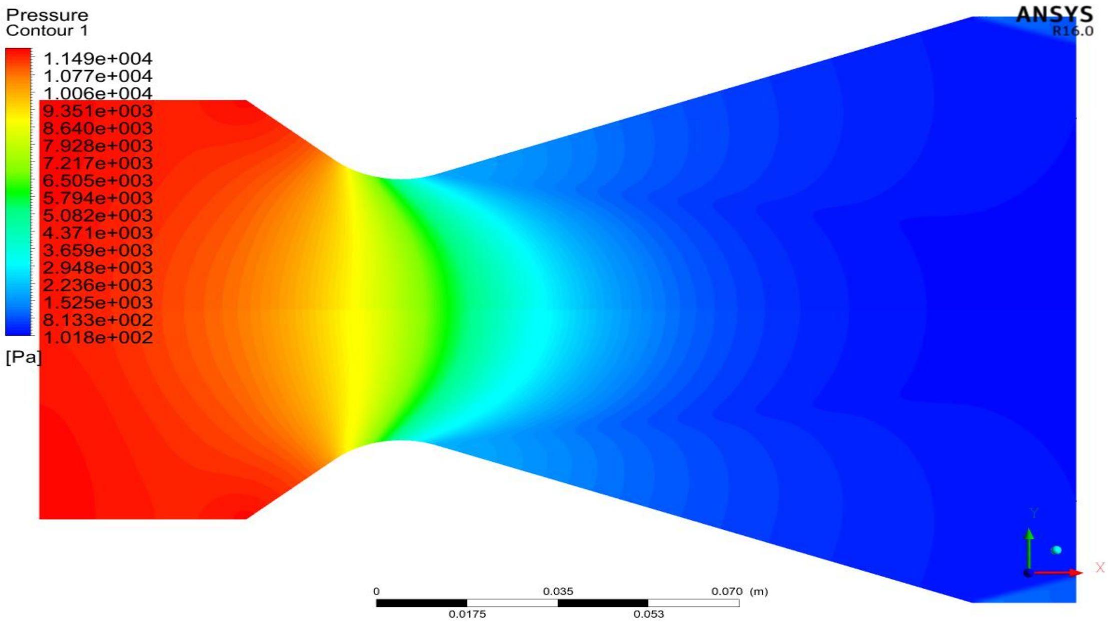
$$dA/A=(M^2-1)du/u$$

Results and discussion

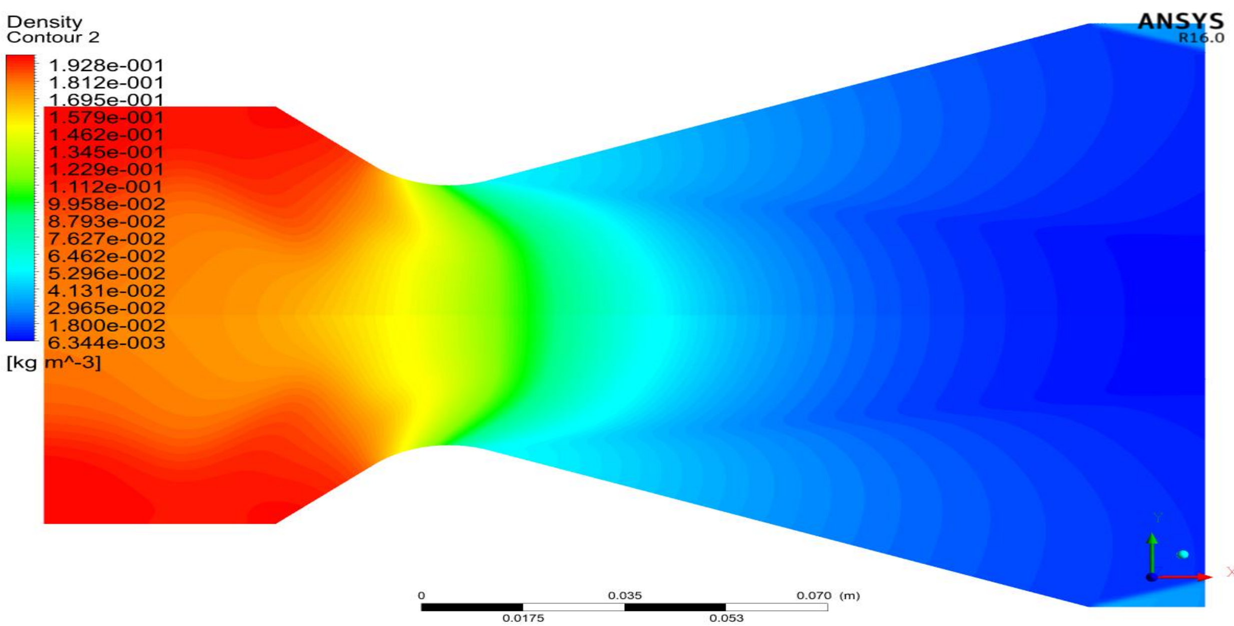
Static pressure contour

The static pressure contour shows a reduction in the static pressure throughout the nozzle.

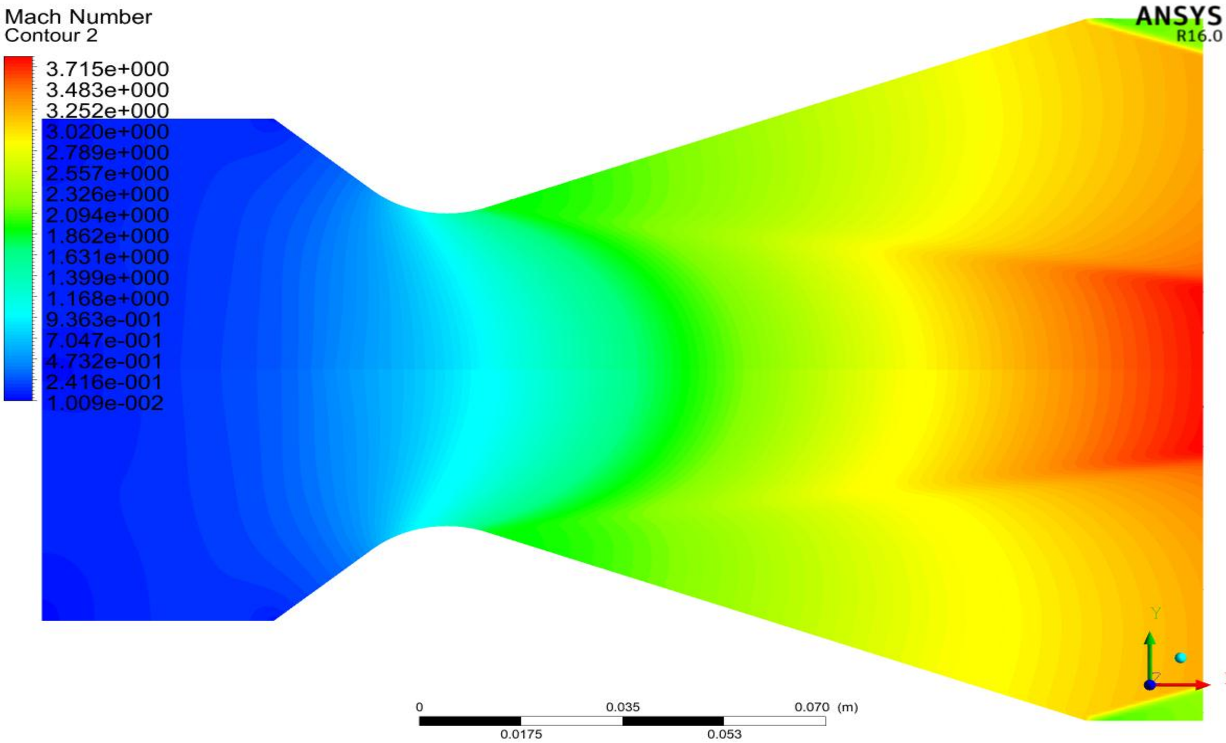
At the inlet, the static pressure is found to be 9.351e+003Pa. At the throat it has reduced to 1.149e+004Pa. This value again reduces to a value of -1.018e+002Pa and remains constant till the exit section.



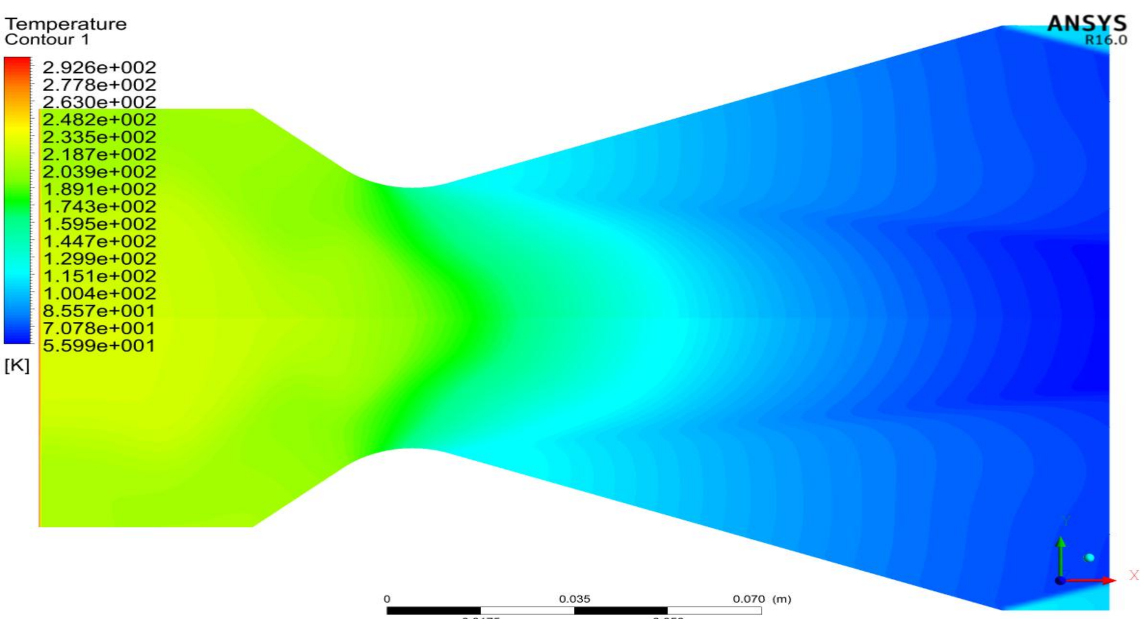
Density contour



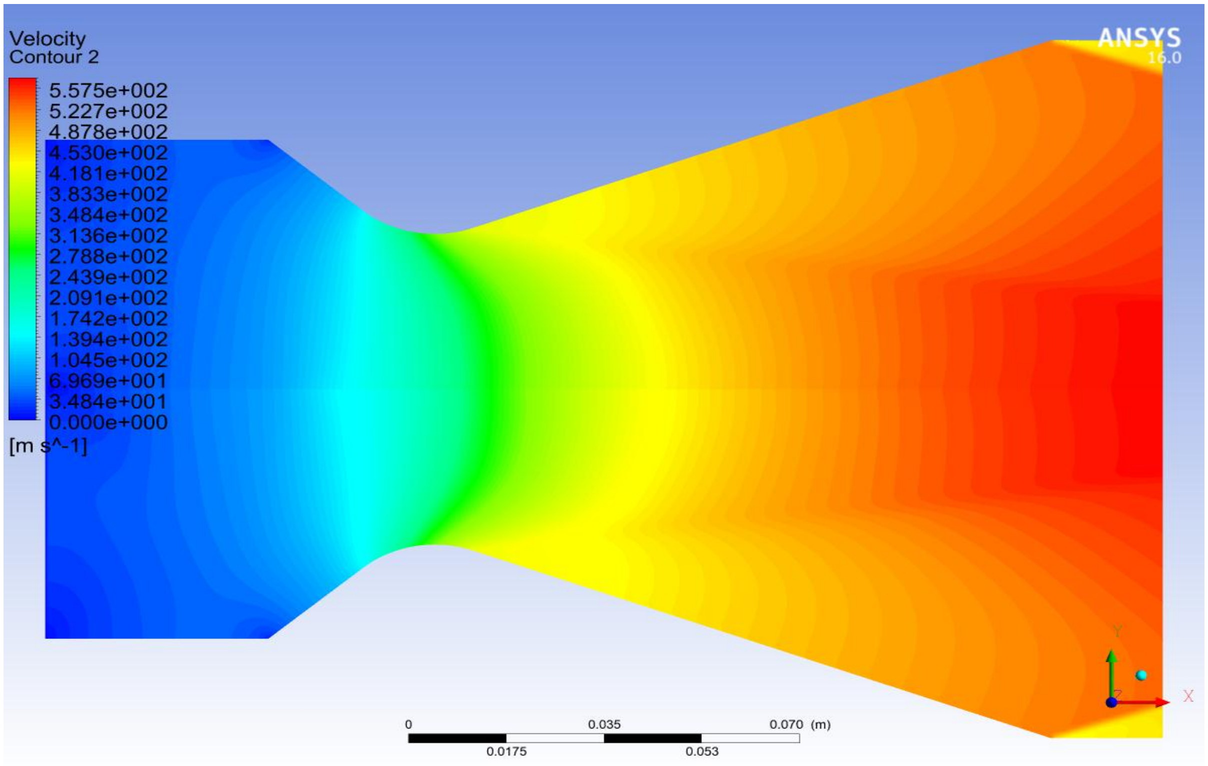
Mach number contour



Temperature contour



Velocity contour



Conclusion

A nozzle model was developed to see the pressure, Mach number, cell equi angle skew, cell Reynolds range, molecular prandtl range and orthogonal quality in it. Various steps of the model were validated with sensible accordance with the experimental knowledge and numerical results found within the literature. The contours of the higher than mentioned parameters square measure found when analyzing the model with success within the thinker.

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). DeLaval 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.

Computational Fluid Dynamics" The International Journal Of Engineering And Science (Ijes),Volume 2,Issue2, Issn: 2319 – 1813 Isbn: 2319 – 1805, 2013, pp 196-207.