Available online: www.ijsrmes.com

International Journal for Scientific Research in Modern Engineering and Science

International Journal for Scientific Research in Modern Engineering and Science, 3(4): 78- 83 (2022)

# ANALYSIS OF CONTROL FLOW SEPARATION IN ROCKET NOZZLE ON VARIATIONS OF PRESSURE RATIO AND CONVERGENT-DIVERGENT NOZZLE ANGLE

**Pallavi Chelak [1]*, Vinay Kumar [2]**

*[12]Department of Department of Mechanical Engineering, SSTC, Bhilai, C.G. India.*

## Abstract

The exhaust nozzle is a very important component of the entire engine propulsion system. Convergent-divergent nozzles are used to accelerate the fluid passing through the throat of such nozzles to supersonic speeds. In this case, a shock wave may be generated in the flow field depending on the ratio of the average outlet pressure to the inlet dynamic pressure. Nozzles provide thrust and give momentum to aerospace aircraft. Nozzle design should be developed in relation to the application based on various factors that are considered when developing the system design. The required specific impulse is one of the main factors in defining the nozzle configuration with minimal variation in the thrust supplied. Numerical simulations are currently being performed using the available Ansys Fluent software to study and understand the performance of conical bell-type nozzles. CFD studies were performed to select the appropriate mesh and turbulence model for calculating the jet flow field. The SSTk-ε turbulence model is applied to simulate the flow of compressive gas in a nozzle and its plume. Numerically study the effects of back pressure, convergence and divergence on nozzle performance. The performance parameters of the exhaust nozzle were calculated and compared with the available analytically resolved data to demonstrate the validity of the simulation. For this purpose, different nozzle back pressures are considered for different operating conditions, including overinflation, underexpansion, and design conditions. Investigate the effects of nozzle shape (convergence half-width) on the number of mechanisms, pressure ratio, and thrust coefficient. In general, the proposed numerical methods applied in this study show excellent ability to predict the physical phenomena and flow characteristics encountered in such types of complex turbulence.

***Keywords:*** *Convergent – Divergent, Nozzle, Expansion Ratio, Pressure Ratio, Thrust Coefficient.*

** Corresponding author*

## 1. INTRODUCTION

A rocket will perform in the best way only if the pressure at the nozzle's end is equal to the ambient pressure. If the exhaust's pressure is higher, then the pressure will not convert into thrust, and energy is not fully utilized. In the same way, if pressure is lower, then because of the difference in pressure between engine top and exit, the vehicle will slow down. Therefore, for making exhaust pressure remain equal to the ambient pressure, the nozzle diameter would change (increase) when going to higher altitudes. A longer nozzle would be available for the pressure to act, thus reducing exit pressure and temperature. A lightweight nozzle is generally used in a rocket, and when throttling occurs, the atmospheric performance is compromised. For rectification, different designs of nozzles are used. Each designed as per the atmospheric conditions giving extra higher thrust and velocity at high altitudes [6].

But certainly, there are two problems:

- The performance is compromised by the weight of the nozzle (heavyweight).
- Freezing of chemicals in engine because of the adiabatic expansion of exhaust gases resulting in snow production inside the jet.

A nozzle is used to eliminate these problems in which exhaust gas detachment occurs in an over-expanded nozzle. The detachment point is not axial. This non-axial point detachment will create a side force over the engine. This side force will vary with time resulting in controlling problems inside the vehicle.

## 2. FEA MODELING AND ANALYSIS OF NOZZLE

The engineering problem to be investigated and solved as selected as a converging diverging nozzle. Therefore, a geometrical nozzle model is used for a flow analysis in CFD software (Ansys Fluent). The modeling, dimensioning and design of the nozzle is carried out using Ansys Design Modeler (ADM) module. CFD Analysis is broadly used to carry out the simulation works involving fluid flows. This makes use of procedures and numerical methodologies to solve for various aerodynamic, marine and fluid flow problems.

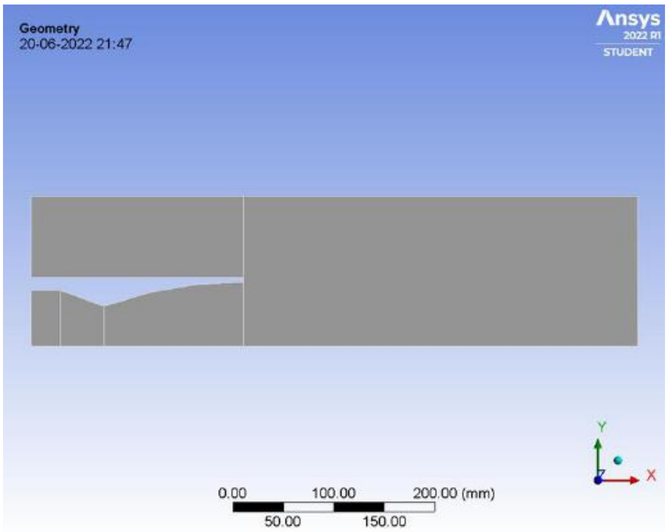



Figure 1. Two-Dimensional geometry modeled in Ansys

The grid generation or meshing of computational domain is carried out in Ansys Fluent module and meshed model as shown in Figure 4.4. Present analysis the structured non-uniform quadrilateral grids are generated. Three different grids were generated to simulate the gas flow inside the baseline geometry of CD nozzle and its exhaust plume. A total number of cells in the nozzle are 23,000, 36,000 and 56,000 cells for coarse mesh, medium mesh and fine mesh respectively.

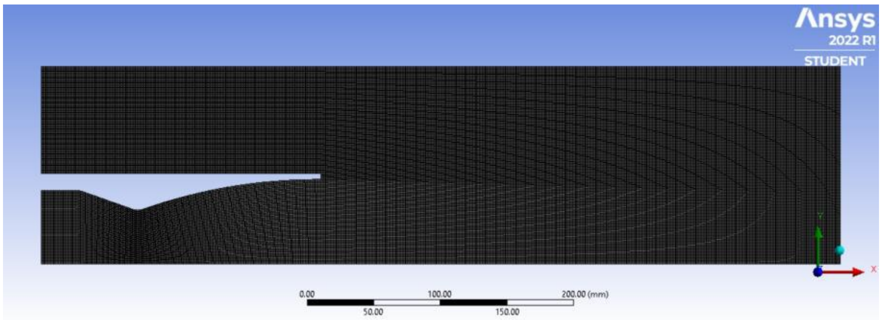
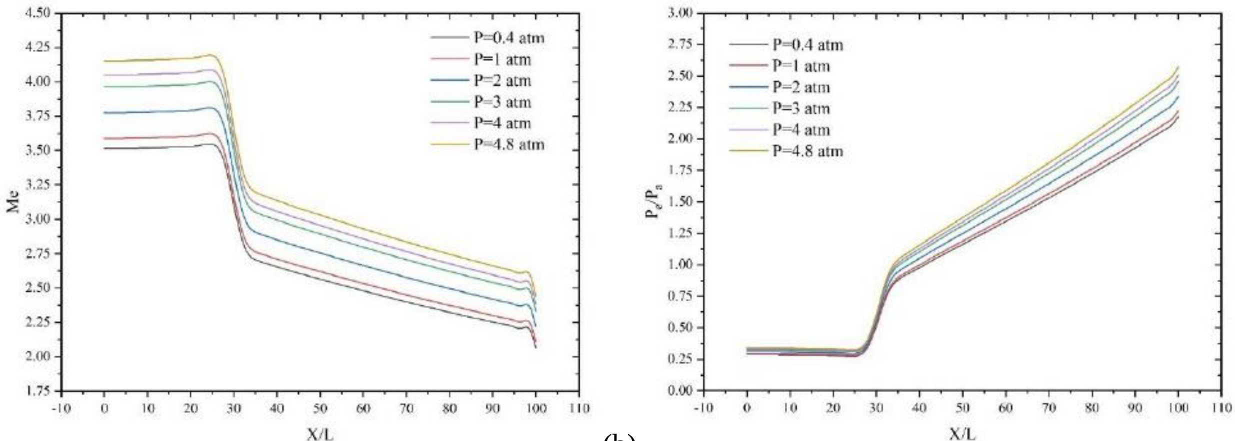



Figure 2. Mesh generated for gas flow computation of the nozzle

## 3. RESULTS AND DISCUSSION

Contour of computed for base line geometry of convergence and divergence half angle 20° and 25° are plotted in Figure 5.2 – 5.6. These contour plots demonstrated that the influence of the nozzle gas flow characteristics inside the nozzle and its exit plume. For details discussion, variation of back pressure (Pb) is performed i.e., P = 0.4, (under-expansion), 1 atm (full-expansion), 2 atm. (over-expansion), 3, 4 and 4.8 atm (normal shock inside of the divergent section and at the outlet area of the nozzle) are considered in the simulations as shown in Figure 4.7 – 4.9 along the nozzle exit radius for the nozzles. These Figures shows that if back pressure increases Me, Pe/Pa and CF also increases from the throat to the exit area of the nozzle.




(a) (b)

Figure 3. Comparison of dimensionless (a) Mach number and (b) nozzle pressure ratio along the nozzle exit radius for the nozzles with various back pressure

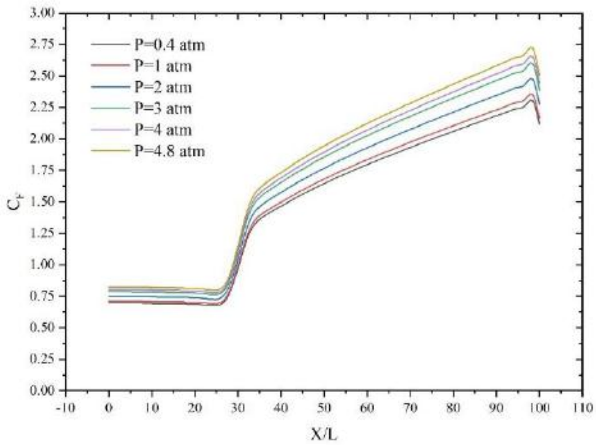



Figure 4. Comparison of dimensionless Thrust coefficient along the ratio of nozzle exit radius for the nozzles with various back pressure

## 4. CONCLUSION

Compressible gas flow inside a converging–diverging nozzle at different back pressure has been numerically studied by SST k - x turbulence model. Effects of the changes in convergence half-angle (θ) and divergence half angle (β) on the nozzle performance, including Mach number (Me), Pressure ratio ($P_e/P_a$), and Thrust coefficient ($C_F$), have been numerically investigated. For validation of the numerical method, the predicted results of Thrust (F) were compared with good agreement with the analytical solution. Some important findings of this work can be present blow:

- Different of back pressure ($P_b$) is performed i.e., P = 0.4, (under-expansion), 1 atm (full-expansion), 2 atm. (over-expansion), 3, 4 and 4.8 atm (normal shock inside of the divergent section and at the outlet area of the nozzle) are considered in the simulations along with the nozzle exit radius for the nozzles and found that if back pressure increases Me, $P_e/P_a$ and $C_F$ also increases from the throat to the exit area of the nozzle.
- Convergence angle (θ= 20°, 25°, 30°, 40°, 45°) have been considered in simulation. The divergence half angle (β=25°) is kept constant, and obtained using CFD analysis. In this parametric changes the Me, $P_e/P_a$ and $C_F$ are rapidly decreases towards the ratio of nozzle exit radius and found that the maximum Me, $C_F$ and minimum $P_e/P_a$ at θ= 45° and β= 25°.
- Convergence and Divergence half-angle significantly effects (about more than 10%) on the Me, $C_F$ and minimum $P_e/P_a$.

## References

[1] A. T. Olson, "Nozzle discharge coefficients-compressible flow". Journal of Fluids Engineering, vol. 96 issue, 1, pp. 21–24, 1974.

[2] B. P. Krushna, P. S. Rao, B. Balakrishna, "Analysis of dual bell rocket nozzle using computational fluid dynamics", International Journal of Research in Engineering and Technology, vol 2, issue 11, pp. 412-417, 2013.

[3] P. K. Kundu, I. M. Cohen, D. R. Dowling, "Fluid mechanics". Academic Press. 2012.

[4] J. D. Mattingly, K. Boyer, "Elements of Propulsion: Gas Turbines and Rockets", Second Edition (AIAA Education). Amer Inst of Aeronautics. 2016.

[5] G. P. Sutton "Rocket propulsion elements" 2nd edition, John Wiley and Sons, New York. 2010.

[6] [1] K.M. Pandey, "CFD Analysis of Rocket Nozzle with 4 inlets", International Journal of Chemical Engineering and Applications, Vol. 1, No. 4, 2010.

[7] [2] K. M. Pandey, S. K. Yadav, "CFD Analysis of a Rocket Nozzle with Two Inlets at Mach 2.1", Journal of Environmental Research and Development, Vol 5, No 2, pp. 308-321, 2010.

[8] [3] B. Kuttan, M. Sajesh, "Optimisation of Divergent angle of Rocket Engine Nozzle using Computational Fluid Dynamics", The International Journal of Engineering and Science, vol 2, Issue 2, pp. 196-207, 2013.

[9] [4] K. P. S. Surya Narayana, K. S. Reddy, "Simulation of Convergent Divergent Rocket Nozzle using CFD Analysis", IOSR Journal of Mechanical and Civil Engineering, vol. 13, Issue 4, pp. 58-65, 2016.

[10] [5] B. P. Madhu, S. Syed, M. K. Kalyana, G. M. Mahendra, "CFD Analysis of Convergent Divergent and Contour Nozzle", International Journal of Mechanical Engineering and Technology, vol 8, Issue 8, pp. 670–677, 2017.

[11] A. R. Almeida, "Some Design Aspects for Venturi Gas Lift Valves", SPE Production & Operations, vol 30, issue 4, pp. 321-328, 2015.

[12] [2]. K. A. Park, Y. M Choi, H. M. Choi, T. S. Cha, B. H. Yoon, "The evaluation of critical pressure ratios of sonic nozzles at low Reynolds numbers", Flow measurement and Instrumentation, vol 12, issue 1, pp. 37-41, 2001.

[13] [3]. M. L. Mason, L. E. Putnam, J. R. Richard, "The effect of throat contouring on two-dimensional converging-diverging nozzles at static condition". NASA technical paper, 1704, 1980.

[14] [4]. J. S. Paik, K. A. Park, J. T. Park, "Inter-laboratory comparison of sonic nozzles at KRISS", Flow Meas. Instrum., vol 11, pp. 339–344, 2000.

[15] [5]. N. Spotts, S. Guzik, X. Gaoz, "A CFD analysis of compressible flow through convergent-conical nozzles", American Institute of Aeronautics and Astronautics, New York, 2007.

[16] [6]. L. Geng, H. Liu, X. Wei, "CFD analysis of the flashing flow characteristics of subcritical refrigerant R134a through converging-diverging nozzles", International Journal of Thermal Sciences, vol. 137, pp. 438-445, 2019.

[17] M. H. H. Estakhrsar, H. M. Moghaddam, M. Jahromi, "Investigation of effects of convergence and divergence half-angles on the performance of a nozzle for different operating conditions", Journal of the Brazilian Society of Mechanical Sciences and Engineering, vol. 40, issue 7, pp. 1-12 353, 2018.

[18] S. Mukthiyar, D. R. Sarath, B. V. Kumar, A. Madabhushi, "Design and CFD Analysis of R410a Refrigerant in Convergent Nozzle", Materials Today: Proceedings, vol 5, issue 9, pp. 19463- 19470, 2018.

[19] K. K. Quintao, Design optimization of nozzle shapes for maximum uniformity of exit flow. M.S. Thesis, Florida International University, 2012.

[20] V. N. Sreerag, F. Mohammad, V. Nandan, AnurA.ag Pramod, K. P. Subhajayan, S. Jash, “Parametric study on a method to control flow separation in rocket nozzles, Materials Today: Proceedings, vol 46, Part 19, pp. 9950-9955, 2021.

[21] S. N. Ananthesha, S. D. Vamsidhar, “Numerical investigation and parametric study of fluidic thrust vectoring by shock vector control method”, SAS Tech, vol 5, pp. 58–65, 2007.