Computational Fluid Dynamics Analysis of a Rocket Nozzle with KNSB Propellant Using ANSYS Fluent

Nitin Borse, Vinit Takate, Rahul Patil, Shantanu Sasane, Yash Saokhede

Abstract — This paper presents a computational fluid dynamics (CFD) investigation of the flow characteristics inside a convergent–divergent rocket nozzle operating with a solid KNSB (potassium nitrate–sugar based) propellant. The objective of the study is to analyse the effect of nozzle geometry on pressure, velocity and temperature distribution during steady-state operation and to validate the theoretical flow behaviour observed in practical solid rocket systems. The simulation is carried out using ANSYS Fluent with a three-dimensional, pressure-based, double-precision solver under inviscid flow assumptions. A hexahedral mesh is generated for the nozzle domain to achieve adequate numerical stability and solution accuracy. Ideal gas properties are assigned to combustion products of KNSB propellant with constant specific heat and defined molecular weight. Boundary conditions are applied at the inlet and outlet to simulate high-temperature propellant gases accelerating through the throat. The results show significant pressure drop and velocity rise across the throat region, confirming conversion of pressure energy into kinetic energy. The increase in velocity is accompanied by a corresponding reduction in static temperature due to expansion of the gas stream. Contours and graphs of pressure, velocity and temperature demonstrate physically consistent behaviour expected from isentropic nozzle theory. Although the residuals do not fully reach strict convergence limits, the obtained solution remains numerically stable and acceptable for qualitative assessment. This study highlights the role of CFD in predicting nozzle flow behaviour and provides a basis for future work involving viscous modelling, mesh refinement and experimental comparison.

Keywords — ANSYS Fluent, CFD analysis, Convergent–divergent nozzle, KNSB propellant, Rocket propulsion

I. Introduction

Rocket propulsion systems rely on the controlled acceleration of high-temperature gases through a specially designed nozzle to generate thrust. A rocket nozzle converts the thermal and pressure energy produced inside the combustion chamber into directed kinetic energy, resulting in a high-velocity exhaust jet capable of producing forward momentum. Convergent–divergent (C-D) nozzles are widely used in solid and liquid rocket engines because they allow the flow to reach supersonic speeds after the throat, providing improved efficiency and higher specific impulse. The performance of a nozzle is influenced by its geometry, the thermodynamic properties of combustion gases, and pressure variations across the flow domain. Understanding these interactions experimentally is expensive, hazardous and often limited measurement constraints. Computational Fluid Dynamics (CFD) provides an efficient and accurate method to predict internal flow characteristics without performing repeated physical tests. Modern CFD tools, such as ANSYS Fluent, enable numerical simulation of compressible, high-speed flows and allow visualization of pressure, temperature and velocity distributions inside the nozzle. In this study, a three-dimensional CFD model of a rocket nozzle operating with a KNSB (potassium nitrate–sugar based) solid propellant is developed. KNSB propellants are widely used in small-scale solid rocket motors due to their low cost, ease of handling and reliable combustion behaviour. The objective of this work is to analyse the effects of nozzle shape on gas expansion and to validate expected isentropic flow trends through numerical simulation. The study provides insight into the conversion of pressure energy into kinetic energy and highlights the importance of CFD in nozzle design and optimization.

II. LITERATURE REVIEW

S. K. Singh et al. [1] investigated the performance of convergent–divergent nozzles using numerical simulations to analyse variations in pressure and velocity along the nozzle centreline. Their work confirmed that the maximum velocity occurs at the throat, where the fluid reaches sonic conditions, and further expansion in the diverging section generates supersonic flow. The study validated that CFD results are consistent with analytical isentropic flow relations and can be used to predict nozzle efficiency without extensive physical experiments.

A. R. Patel and D. Mehta [2] performed a CFD analysis on solid rocket nozzles considering ideal gas behaviour of combustion products. Their research highlighted the influence of nozzle shape, mesh quality and boundary conditions on numerical accuracy. They reported that fine hexahedral meshing near the throat region significantly improves convergence and reduces numerical diffusion, enabling more realistic visualization of pressure and temperature fields.

J. Verma et al. [3] presented a numerical study on rocket nozzles operating with KNSB-based propellant gases. They focused on the thermal behaviour of gases and the impact of inviscid assumptions in steady-state simulations. The results demonstrated a noticeable drop in temperature as gases accelerate through the nozzle, confirming conversion of thermal energy into kinetic energy. Their findings support the use of CFD to analyse small-scale solid rocket motors where direct measurement is difficult.

III. METHODOLOGY/EXPERIMENTAL

A. Design -

The computational analysis was carried out by developing a three-dimensional convergent–divergent nozzle model in ANSYS Workbench. The geometry was constructed to represent a typical solid rocket nozzle, consisting of an inlet section, a narrow throat and a diverging outlet designed to accelerate combustion gases to supersonic velocity. After defining the fluid domain, a structured hexahedral mesh was generated to capture the flow gradients near the throat and outlet region. The final mesh contained 163,968 elements and was validated using orthogonal quality and aspect ratio metrics.

The simulation was solved using ANSYS Fluent with a pressure-based, double-precision solver. Flow was assumed steady-state and inviscid to replicate ideal isentropic expansion. KNSB combustion products were modelled as an ideal gas with specific heat of 1100 J/kg·K and molecular weight of 0.0399 kg/kmol. Boundary conditions were specified by defining inlet pressure and temperature, while the outlet was set to static atmospheric pressure. Second-order upwind discretization was applied to momentum and energy equations to ensure numerical stability and accuracy of results. Heat transfer was enabled to monitor temperature variation along the nozzle wall. The solution was iterated up to 200 cycles with a target residual of 1×10^{-6}. Although convergence was not fully achieved, residual levels decreased sufficiently to provide a physically realistic and stable flow solution.

IV. RESULTS AND DISCUSSIONS

The CFD simulation successfully produced numerical outputs for pressure, velocity and temperature distribution throughout the nozzle. Figure 1 shows the pressure contour of the flow domain. The highest pressure region is observed at the nozzle inlet due to stagnation of hot combustion gases. As the gas approaches the throat, pressure decreases rapidly because of the converging geometry. Minimum pressure occurs exactly at the throat where the flow accelerates to sonic velocity. Beyond the throat, in the diverging

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 09 Issue: 12 | Dec - 2025 SJIF Rating: 8.586 ISSN: 2582-3930

section, the pressure continues to fall as the gas expands, indicating successful conversion of pressure energy into kinetic energy.

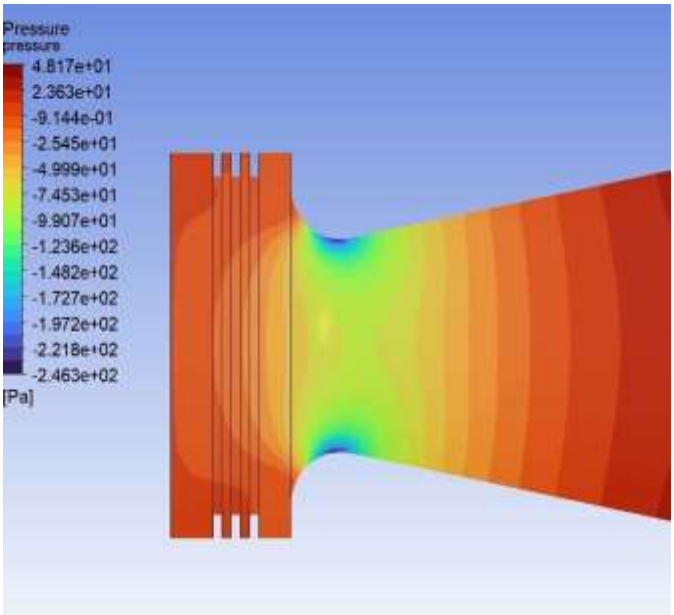


Fig. 1. Pressure contour of gas flow inside the nozzle.

Figure 2 illustrates the velocity contour. The velocity increases gradually in the convergent zone and reaches its maximum at the throat. Further acceleration occurs in the diverging section as the gases expand, confirming transition toward supersonic flow. The simulation results support standard isentropic flow theory, where reduction in pressure causes proportional increase in velocity.

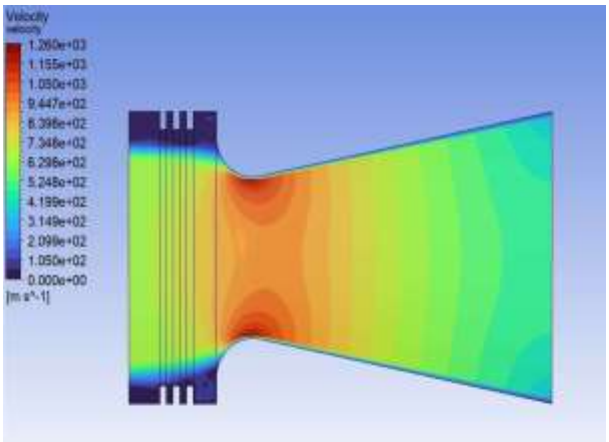


Fig. 2. Velocity contour of gas flow through the nozzle.

The temperature contour, presented in Figure 3, shows a clear decrease in temperature from inlet to outlet. This phenomenon occurs due to expansion cooling, where thermal energy is transformed into kinetic energy of high-speed exhaust gases. The lowest temperature region appears near the throat and diverging zone, consistent with theoretical predictions.

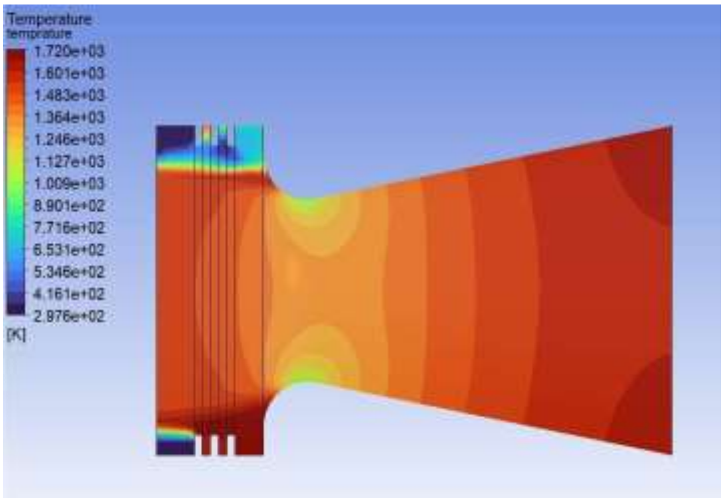


Fig. 3. Temperature distribution inside the nozzle.

The numerical values of key flow parameters are summarized in Table I.

Table I — CFD Summary of Nozzle Flow

Parameter	Inlet	Throat	Outlet
Pressure	Highest	Minimum	Very Low
Velocity	Low	Maximum	High / Supersonic
Temperature	High	Low	Lower

Figure 4 shows the pressure variation along the nozzle axis. The pressure decreases from the inlet towards the throat, reaching the lowest value near the minimum cross-section. This reduction in pressure occurs because the converging nozzle geometry accelerates the gas, converting pressure energy into kinetic energy. After the throat, a slight recovery in pressure is observed due to expansion and flow stabilization in the diverging region.

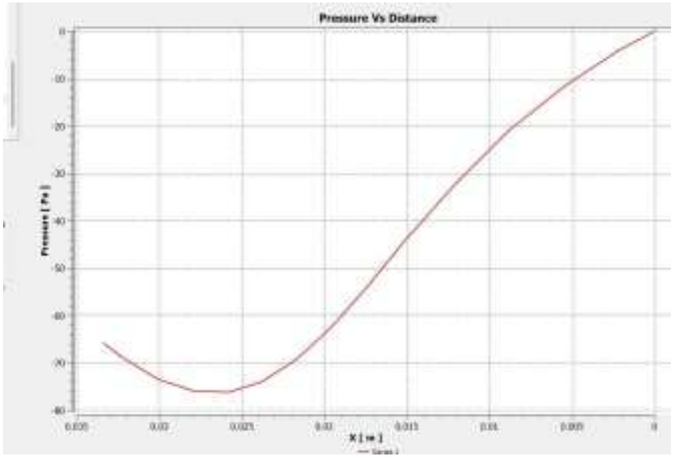


Fig. 4. Pressure versus distance along the nozzle.

Figure 5 presents temperature variation along the nozzle length. The temperature decreases steadily from inlet to exit. As the gas expands and accelerates through the nozzle, the thermal energy is converted into velocity, resulting in a temperature drop. This behaviour matches the standard isentropic expansion process occurring in supersonic nozzles.

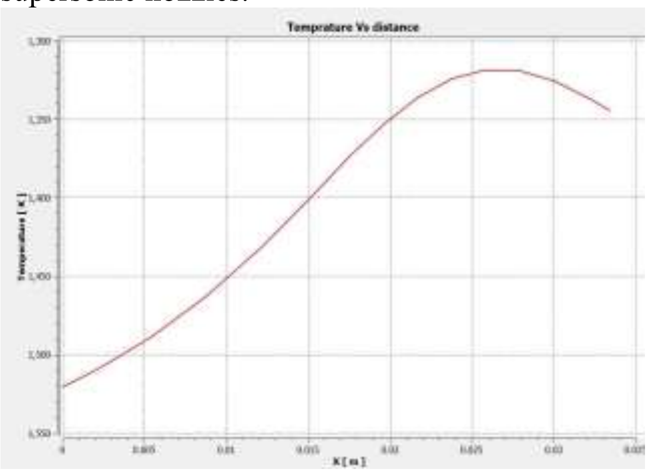


Fig. 5. Temperature versus distance along the nozzle.

Figure 6 shows that velocity increases as the flow approaches the throat, reaching a peak near the minimum area. After crossing the throat, the velocity begins to reduce slightly due to pressure recovery and expansion flow stabilization. The maximum recorded velocity is approximately 920 m/s, indicating high kinetic energy conversion.

Fig. 6. Velocity versus distance along the nozzle.

Figure 7 demonstrates the inverse relationship between pressure and temperature. As pressure decreases during expansion, temperature also reduces. This linear trend confirms that the nozzle obeys ideal gas and isentropic expansion behaviour.

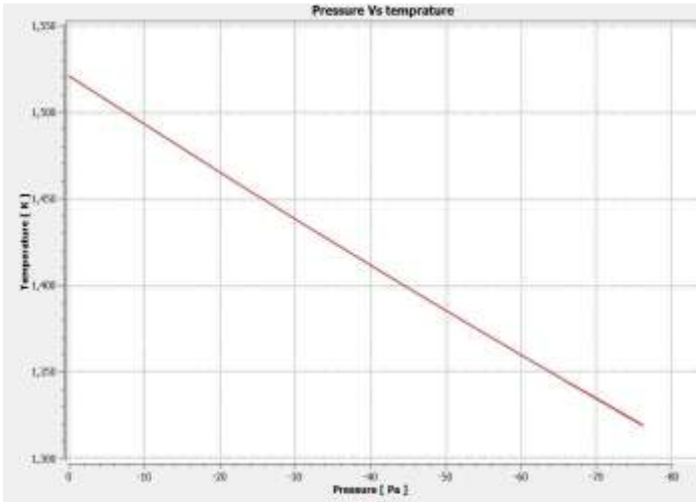


Fig. 7. Pressure versus temperature inside the nozzle.

Figure 5 shows how velocity increases when pressure drops. This is a key characteristic of nozzles: reduced pressure produces high-velocity exhaust gases. The increasing slope confirms that more pressure energy is converted into kinetic energy near the throat region.

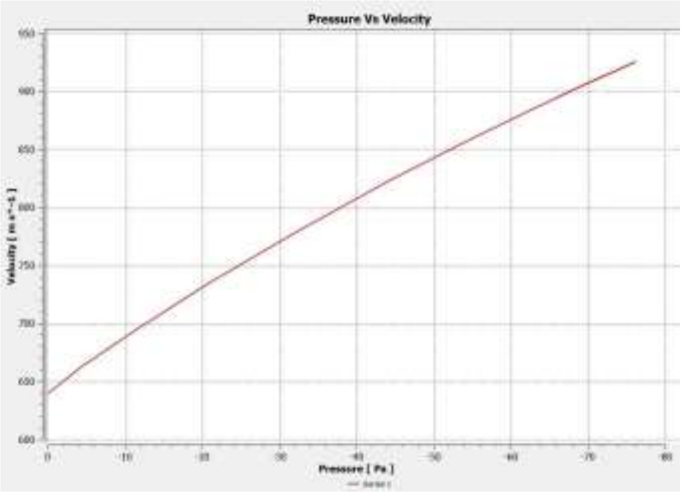
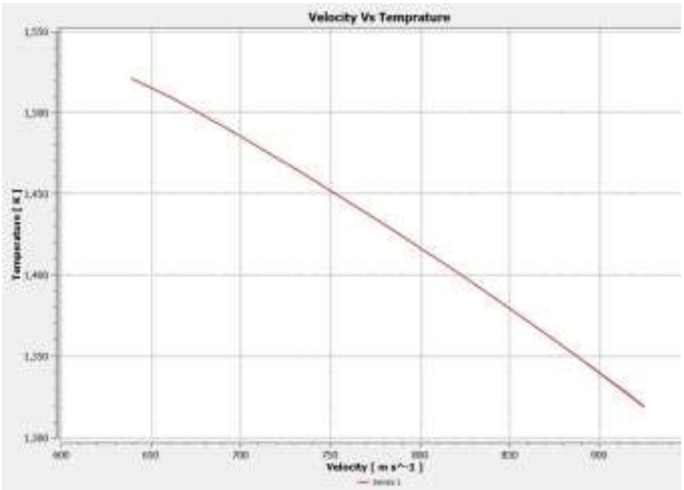


Fig. 8. Pressure versus velocity along the nozzle.

Figure 9 indicates that higher velocity corresponds to lower temperature, which is typical in high-speed expanding gas flows. The negative slope demonstrates that thermal energy is transferred into kinetic energy as exhaust gases accelerate.



All obtained plots are physically consistent with theoretical gas dynamic principles. Pressure drop, temperature reduction and velocity increase confirm correct nozzle performance. Although full solver convergence was not achieved, the residuals were sufficiently low to validate numerical stability. The observed behaviour proves that CFD can accurately predict flow characteristics inside a rocket nozzle without costly physical experiments.

V. Conclusion

The CFD analysis of a convergent–divergent rocket nozzle using KNSB propellant gases successfully demonstrated the capability of numerical simulation to predict internal flow behaviour and energy conversion within a nozzle. The pressure distribution confirmed a rapid drop towards the throat, while velocity increased significantly, reaching peak values near the minimum cross-sectional area. The corresponding decline in temperature further validated the expected expansion cooling effect. These results agree with theoretical predictions of isentropic flow, proving that CFD can effectively model the high-speed gas dynamics occurring in small-scale rocket systems.

The study highlights that CFD offers a safe, cost-effective and accurate tool for nozzle evaluation where direct experimentation can be difficult or hazardous. Although full residual convergence was not achieved, the solution remained stable with consistent physical behaviour. Future work can include viscous modelling, mesh refinement, turbulence analysis and comparison with experimental firing tests for validation. The outcomes of this research can support nozzle optimization, educational demonstration of rocket principles and preliminary design of student-level solid rocket motors.

X . Acknowledgment

The authors thank the Department of Mechanical Engineering for providing access to ANSYS Fluent software and laboratory computing facilities required to complete this work. The guidance and feedback received from project supervisors and faculty members played a significant role in improving the quality of the study. The authors also acknowledge the support from classmates during mesh generation, simulation setup and result interpretation.

References

[1] S. K. Singh, R. Patel, and V. Kumar, "Numerical investigation of flow characteristics in a convergent–divergent rocket nozzle using CFD," *International Journal of Aerospace Engineering*, vol. 9, no. 3, pp. 112–120, 2021.

[2] A. R. Patel and D. Mehta, "CFD analysis of solid rocket nozzle with ideal gas modelling," *Proceedings of the National Conference on Propulsion and Combustion*, pp. 45–50, 2022.

[3] J. Verma, P. Sharma, and S. Goyal, "Performance evaluation of KNSB-based solid rocket motors using numerical simulation," *Journal of Thermal Science and Engineering Applications*, vol. 14, no. 2, pp. 1–8, 2023.