International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 09 Issue: 06 | June - 2025 SJIF Rating: 8.586 ISSN: 2582-3930

Effect of Divergence Angle on Velocity and Turbulence Intensity of the Flow in Rocket Nozzles: A Computational Analysis

Mohamed Abdullah[1], Deepak S[2], Mohanraj N[3], Tarun Jayakumar T J[4]
Corresponding Author Email: mohamedabdullah.aeroin@gmail.com
AEROIN SPACETECH PRIVATE LIMITED

---***---

ABSTRACT - A subfield of fluid mechanics called computational fluid dynamics (CFD) simulates and analyzes fluid flow issues using numerical techniques and algorithms. This investigation focuses on the CFD analysis of the flow characteristics within a supersonic rocket nozzle, specifically examining the impact of varying divergent angles. A two-dimensional axisymmetric model, employing the control volume method, is utilized to solve the governing equations of fluid flow. Key flow parameters, including velocity and turbulence intensity are analyzed to determine the optimal divergent angle. Furthermore, velocity intensity exhibits a consistent increase with larger divergent angles, leading to the identification of an optimal configuration that balances thrust performance. This investigation provides insights into nozzle design, aiming to enhance performance and operational stability in supersonic propulsion systems.

Key Words: *supersonic nozzle, CFD analysis, divergence angle, turbulence intensity, velocity variation, contour based.*

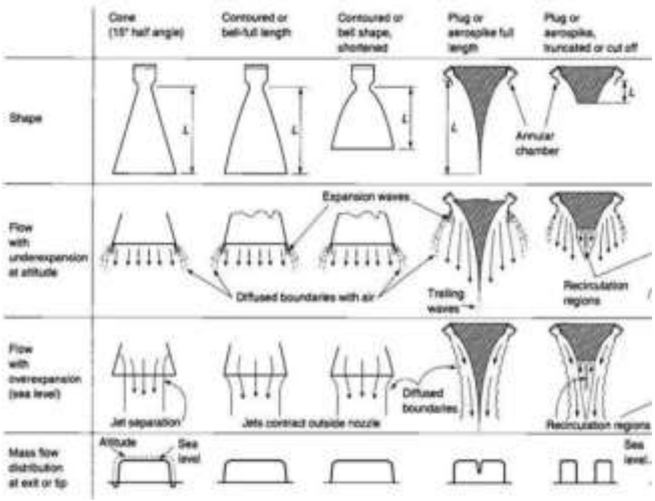
1. INTRODUCTION:

An essential part of propulsion systems, a nozzle's main function is to transform the chemical-thermal energy produced in a combustion chamber into kinetic energy. This is accomplished by converting low-velocity, high-temperature, high-pressure gas into high-velocity, lower-pressure gas. This energy conversion is necessary for turbines and rockets to provide thrust.

The concept of efficient nozzle design can be attributed to Swedish-French engineer Gustaf de Laval, who developed a steam turbine in which a nozzle was a key feature. De Laval's nozzle design incorporated a convergent section to accelerate steam to sonic velocity, followed by a divergent section to further accelerate the flow beyond the speed of sound. This supersonic expansion facilitated an efficient conversion of heat energy into motion, rendering it suitable for high-speed applications such as rockets. Nozzles play a crucial role in achieving desired thrust levels. Their geometry, particularly the divergent angle, significantly influences flow dynamics, pressure distribution, and exit velocity. For instance, in rocket engines, a de Laval nozzle facilitates the maximization of thrust by expanding high-pressure gases efficiently. In contemporary engineering, Computational Fluid Dynamics (CFD) has become an indispensable tool for optimizing nozzle

Figure 1

designs. CFD enables engineers to simulate fluid flow and solve complex transport equations. By utilizing CFD, one can predict the effects of different geometries, such as divergent angles, on the nozzle's performance. This reduces reliance on costly and time-consuming physical prototyping.



The research aims to determine the optimum divergent angle for maximizing outlet velocity and meeting thrust requirements. Utilizing CFD, the study will simulate nozzle performance under various conditions to achieve optimal efficiency. Such studies are critical in advancing propulsion technologies, particularly in aerospace and power generation applications.

2. LITERATURE REVIEW:

B.P. Madhu et, al (2021) [1] The flow characteristics of a supersonic convergent-divergent nozzle with different divergent angles 9°, 11°, and 13°—are examined in this work. While the divergent section varies with angle, the convergent area remains constant. Mach number, velocity, turbulent kinetic energy, and static pressure are among the important flow characteristics that are examined. Utilizing compressible-transient state analysis in a two-dimensional domain meshed with ICEM CFD, ANSYS Fluent is utilized for simulations.

Nabila ALILI, Khacem KADDOURI, Salem MOKADEM et. al (2024) In order to assess rocket engine nozzle exhaust performance, this study carried out a thorough numerical analysis that looked at the interaction between convergent and divergent angles.[3] The parameters analysed included the velocity coefficient (Cv), angularity coefficient (Ca), and gross thrust coefficient (Cfg).

- **Convergent Angles**: Examined in the range of 20° to 45°, with an optimal angle identified at **37.5°**.
- **Divergent Angles**: Divided into small (10°-13°), medium (14°-19°), and large (20°-25°). The **15° angle** performed well overall, while the **23° angle** provided the best compromise between thrust, efficiency, and weight.

Biju Kuttan et. al (2013) [2]To find the ideal divergence angle for the nozzle that would provide the highest outlet velocity and satisfy the thrust requirements, numerical analysis was done. To find out how the divergent angle variation impacts the flow pattern via the nozzle, the intake size and boundary conditions are held constant while the divergent angles are changed. The k-ε model was chosen for their task out of all the models that Fluent offered. For the analysis, a two-dimensional axis-symmetric geometrical model of the nozzle was employed. The selected divergence angles were 4°, 7°, 10°, 13°, and 15°.

3. METHODOLOGY:

The analysis has been conducted on nozzles with divergent angles 10°, 15°,20°. The dimensions of CD nozzle are mentioned in table 1. These simulations are carried out using ANSYS fluent.

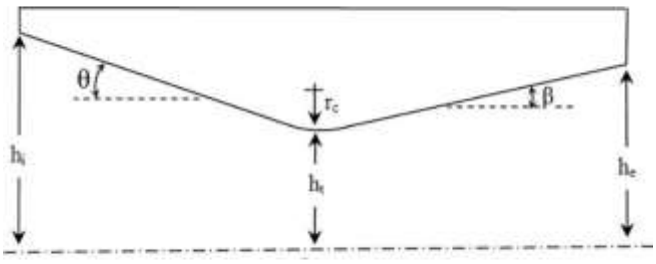
Table 1: Dimension of CD Nozzle

This baseline geometry was modified by changing the convergence half-angle (θ) 20° and divergence half-angle (β) 10°,15°, 20° while the parameters he (exit radius) and hi (throat radius) are kept constant.

3.1. FINITE ELEMNET METHOD:

Figure 3(a): 2D Planar Fluid Body Finite Element

Inlet radius	35mm
Throat Radius	10mm
Exit Radius	65mm
Convergent Length	66mm
Convergent angle	20°
Divergent angle	10°, 15°, 20°.respectively.



Model

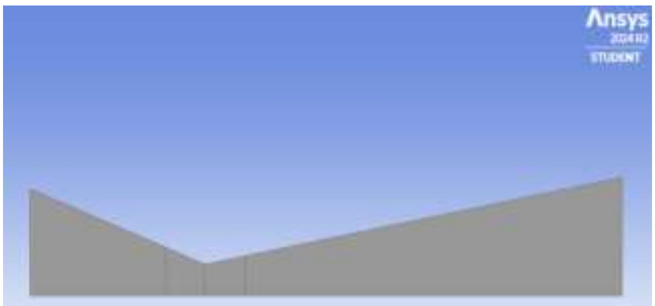


Figure 3(b): 2D Planar Fluid Body Meshing

Figure 3(a) displays the CD nozzle's 2D planar model, whereas Figure 3(b) displays the closed form of the finite element meshing. In order to construct a fine mesh in the enclosed area at the planar body's border, a large number of elements were employed in the ANSYS workbench to create the structural mesh. 65000 nodes in total were produced for the 2D planar model.

3.2. FLUENT STEP FOR SOLUTION INITIALISATION:

Computations of the flow field inside the control volume were done using RANS (Reynolds Stokes) equations with k-ε standard turbulent model. The table 2 shows the most important setting which has been used for simulating the results in the present study.

Table 2: Solution Set up

Solution Method	
Solver	Absolute 2D planar density-Based
Turbulence Model	k-ε standard Wall function
Fluid	Ideal gas, Viscosity by Sutherland law
Boundary Condition	Inlet: Pressure Inlet(4.5bar), Outlet: Pressure Outlet(0bar) Wall, axis
Solution Method	Second order upwind
Solution Initialization	Standard from Inlet
Reference Value	Inlet (Solid surface)

4. RESULTS:

4.1. Nozzle with divergence angle of 10⁰:

4.1.1. Velocity magnitude:

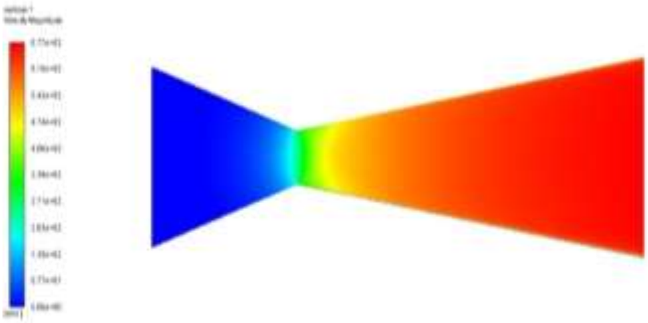


Figure 4(a): Contour of velocity at divergent angle of 10⁰

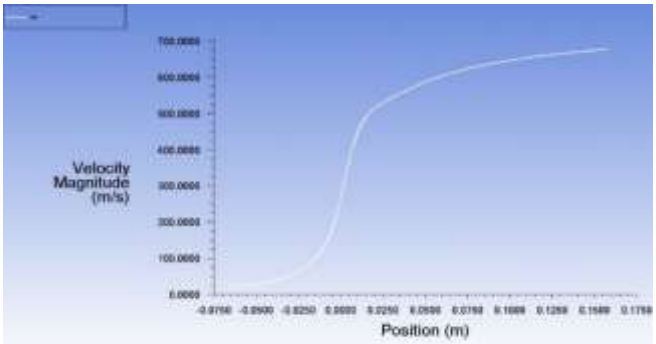


Figure 4(b): Velocity magnitude Vs Position plot at 10⁰ of divergent angle

From obtained contour results in figure 4(a), velocity of the nozzle at 10⁰ divergent angle is observed to be, the maximum velocity is at the end of nozzle with a magnitude of 6.77e+02 m/s. From the figure 4(b), it is inferred that there is sudden increase in velocity magnitude at the throat and after the throat, the velocity gradually attains it maximum value at the nozzle from 2.71e+02 m/s to 6.77e+02 m/s. It is also inferred that the flow reaches a velocity of 3.39e+02 m/s at the throat and makes it a supersonic flow in the divergent section.

4.1.2. Turbulence intensity:

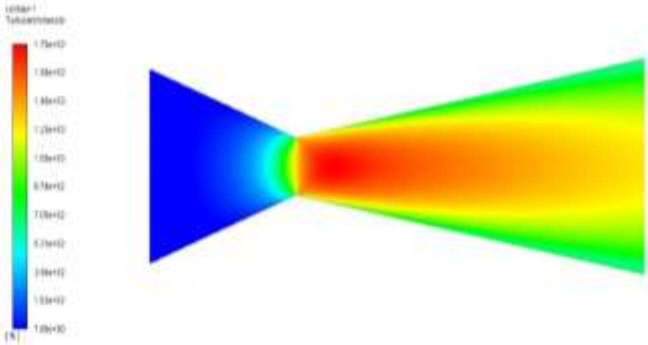


Figure 4(c): Contour of turbulent intensity at divergent angle of 10⁰

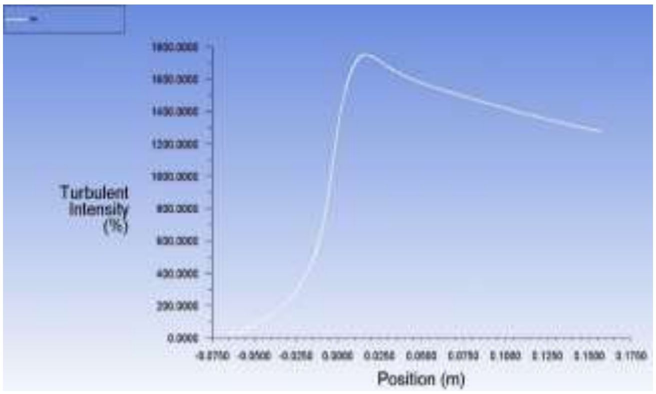


Figure 4(d): Turbulent intensity Vs position plot at 10⁰ of divergent angle

Results obtained in contour and graph in figure 4(c) and 4(d), there is low turbulence in the inlet of about 7.86e+00% to 1.82e+02% and increases rapidly at the

International Journal of Scientific Research in Engineering and Management (IJSREM)
Volume: 09 Issue: 06 | June - 2025 SJIF Rating: 8.586 ISSN: 2582-3930

throat and due to sudden expansion in nozzle the turbulence gets increased further more to a maximum value of 1.75e+03% behind the throat and gets decreased when flow reaches the nozzle exit with value of 1.23e+03%. Notable point is that the maximum value 1.75e+03% is attained at just after the throat. So, it can be concluded that flow has higher turbulent intensity at just behind the throat or at the region of sudden expansion.

4.2. Nozzle with divergence angle of 15⁰:

4.2.1. Velocity magnitude:

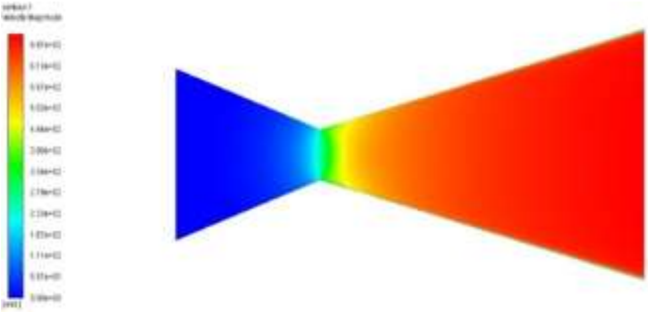


Figure 5(a): Contour of velocity at divergent angle of 15⁰

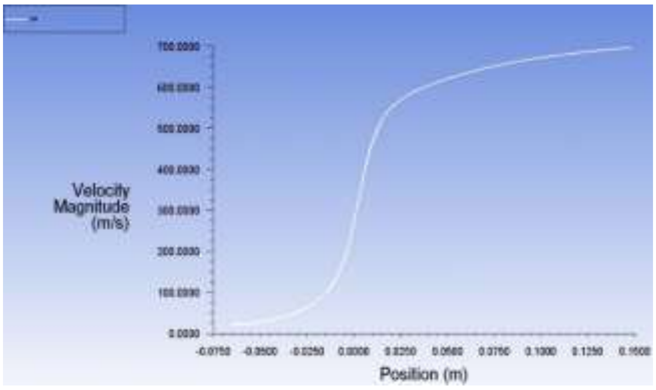


Figure 5(b): Velocity magnitude Vs Position plot at 15⁰ of divergent angle

From figure 5(a), it is inferred that there is a maximum velocity of 6.97e+02 m/s at the end of the nozzle. As observed from figure 5(b), at the throat, the velocity increases from 2.23e+02 m/s to 3.34e+02 m/s and reaches about 4.00e+02 m/s when entering the divergent section. Then the velocity increases gradually and attains a supersonic flow with velocity about 6.97e+02 m/s. Hence, the increase in divergent angle of the nozzle increases the exit velocity of the flow.

4.2.2. Turbulence intensity:

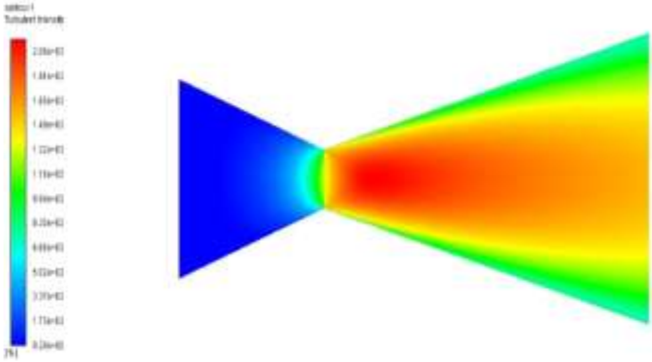


Figure 5(c): Contour of turbulent intensity at divergent angle of 15⁰

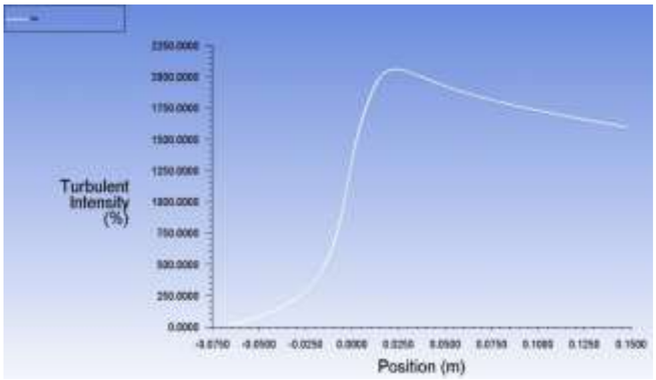


Figure 5(d): Turbulent intensity vs position plot at 15⁰ of divergent angle

From figure 5(c) & 5(d), there is low turbulence in the inlet and before the throat turbulence reaches intensity percentage near to 6.66e+02% to 9.94e+02% and flow gets expanded suddenly after the throat with maximum intensity of 2.06e+03%. The turbulence intensity has slight decrement when the flow gets near to the nozzle exit.

4.3. Nozzle with divergence angle of 20⁰:

4.3.1. Velocity magnitude:

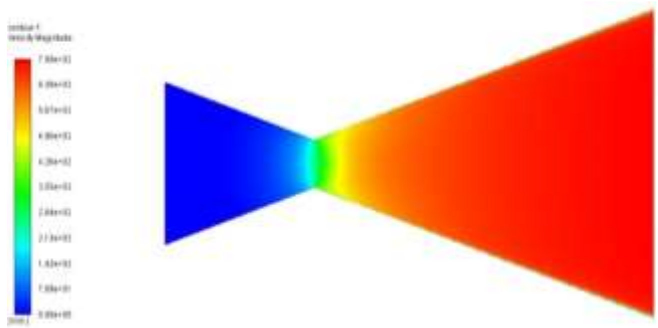


Figure 6(a): Contour of velocity at divergent angle of 20⁰

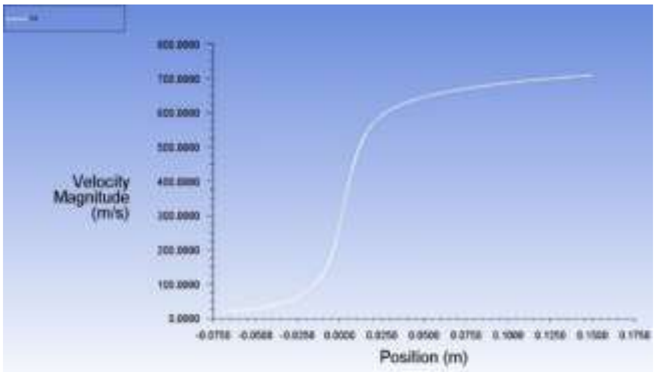


Figure 6(b): Velocity magnitude Vs Position plot at 20⁰ of divergent angle

From figure 6(a) & 6(b), it is observed that at the throat, flow velocity tends to increase suddenly to attain a velocity about 3.55e+02 m/s and then flows through the divergent section with gradual increase in velocity attaining the maximum velocity at the nozzle exit about 7.09e+02m/s. This increase in velocity affects the velocity near the walls of divergent section leading to velocity drop and causing turbulence.

4.3.2. Turbulence intensity:

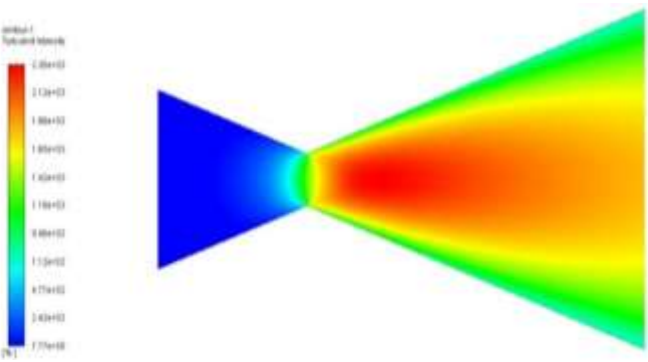


Figure 6(c): Contour of turbulent intensity at divergent angle of 20⁰

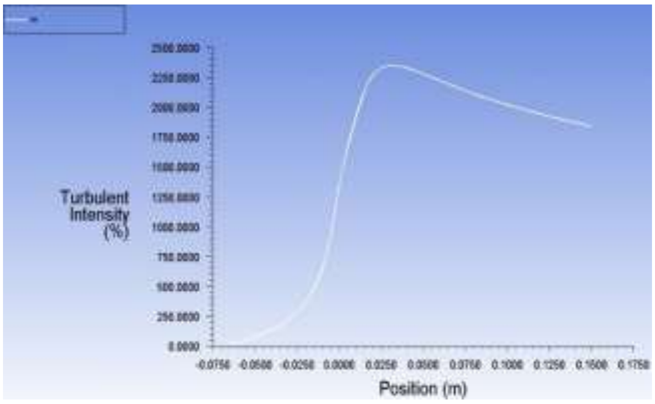


Figure 6(d): Turbulent intensity Vs position plot at 20⁰ of divergent angle

From the figures 6(c) & 6(d), the flow is in minimum turbulence intensity at inlet and the flows gets its rise in intensity before the throat with a value of 1.18e+03% and after the throat there is a maximum value of intensity of about 2.35e+03% and as the flow approaches the nozzle exit, there is decrement in the turbulence intensity. When the divergence angle is increased, there is rise in the turbulence intensity after throat due to sudden expansion and also, the increase in divergence also affects the turbulence intensity region before the throat, near the walls and at the nozzle exit.

5. CONCLUSION:

Influence of variation in divergent angle on different flow property is shown in below table.

Table 3: Properties at exit section

S. No	Divergent angle (degrees)	Velocity (m/s)	Turbulent intensity (%)
1	10	6.77e+02	1.75e+03
2	15	6.97e+02	2.06e+03
3	20	7.09e+02	2.35e+03

At 10⁰ of divergence, flow remains attached to the nozzle walls and velocity increases smoothly along the nozzle, achieving maximum acceleration at the exit. This provides an optimal performance for generating thrust and with minimal energy loss. At 15⁰ divergence angle, velocity near the walls starts to drop because of the moderate adverse pressure gradients making the flow to separate. This leads to the high velocity region is concentrated in the central core, but there will be rise in wall inefficiencies. This divergent angle is suitable for moderate performance applications because, this angle balances thrust efficiency and manages the flow separation. At 20⁰ divergence angle, flow separation becomes significant near the walls, making turbulent wake region and velocity drops. This will affect the performance and stability, and making it less efficient for rocket nozzle applications.

The divergent angle had a substantial influence on the exit flow characteristics. As the angle increases from 10° to 20°, the exit velocity increases from 6.77e+02 m/s to 7.09e+02 m/s, while the turbulence intensity concurrently increases from 17.50% to 23.50%. A more pronounced angle facilitates enhanced flow acceleration and mixing; however, excessive turbulence can result in energy dissipation. Consequently, the optimization of the divergent angle is of paramount importance in achieving an equilibrium between velocity enhancement and turbulence management for the design of efficient flow system

6. REFERENCE:

1. Madhu, B., Mahendramani, G., & Bhaskar, K. (2020b). Numerical analysis on flow properties in convergent – Divergent nozzle for different divergence angle. *Materials Today Proceedings*, *45*, 207–215. https://doi.org/10.1016/j.matpr.2020.10.419

2. Optimization of divergent angle of a rocket engine nozzle using computational fluid dynamics .https://theijes.com/papers/v2-i2/AB02201960207.pdf

3. Alili, N., Kaddouri, K., Mokadem, S., & Alami, A. (2024). Numerical analysis of Convergent-Divergent angles and operating conditions impact on rocket nozzle performance parameters. *INCAS BULLETIN*, *16*, 3–14. https://doi.org/10.13111/2066-8201.2024.16.1.1

4. Pandey, K. M., & Singh, A. P. (2010). CFD Analysis of Conical Nozzle for Mach 3 at Various Angles of Divergence with Fluent Software. *International Journal of Chemical Engineering and Applications*, 179–185. https://doi.org/10.7763/ijcea.2010.v1.31
5. Kumar, R. R., & Devarajan, Y. (2018). CFD simulation analysis of two-dimensional convergent-divergent nozzle. *International Journal of Ambient Energy*, *41*(13), 1505–1515. https://doi.org/10.1080/01430750.2018.1517683
6. Khan, S. A., Ibrahim, O. M., & Aabid, A. (2021). CFD analysis of compressible flows in a convergent-divergent nozzle. *Materials Today Proceedings*, *46*, 2835–2842. https://doi.org/10.1016/j.matpr.2021.03.074
7. Joshi, N. P., Gandhi, N. T., & Parveen, N. S. (2020). Critical Designing and Flow Analysis of Various Nozzles using CFD Analysis. *International Journal of Engineering Research And*, *V9*(02). https://doi.org/10.17577/ijertv9is020208
8. Khalid, M. W., & Ahsan, M. (2020). Computational Fluid Dynamics Analysis of Compressible Flow Through a Converging-Diverging Nozzle using the k-ε Turbulence Model. *Engineering Technology & Applied Science Research*, *10*(1), 5180–5185. https://doi.org/10.48084/etasr.3140
9. Meena, L., Niranjan, Aman, N., Gautam, N., Gagandeep, N., Kumar, G., & Zunaid, M. (2021). Numerical study of convergent-divergent nozzle at different throat diameters and divergence angles. *Materials Today Proceedings*, *46*, 10676–10680. https://doi.org/10.1016/j.matpr.2021.01.432
10. Al, F. a. G. M. E. a. F. a. G. M. E. (2018). Numerical analysis of Convergent-Divergent nozzle using finite element method. *International Journal of Mechanical and Production Engineering Research and Development*, *8*(6), 373–382. https://doi.org/10.24247/ijmperddec201842
11. Agarwal, T. (2023). Computational fluid analysis and optimization of rocket engine nozzles at various divergent angles. *Research Square (Research Square)*. https://doi.org/10.21203/rs.3.rs-3059286/v1