Bulgarian Chemical Communications, Volume 55, Special Issue A (pp. 245-249) 2023 DOI: 10.34049/bcc.55.A.0037

Performance of rocket nozzle with polymer material and its numerical analysis

V. Saxena*, K. K. Soni, D. Kumar, P. Patel, B. P. Singh

Mahatma Jyotiba Phule Rohilkhand University, Bareilly, 243001, India

Received: April 18, 2023; Revised: August 22, 2023

Polymer material modeling is an effective tool for optimizing rocket nozzle design and improving its performance. It provides a means of predicting the behavior of the nozzle under different operating conditions, enabling the evaluation of different polymer materials that can be used in nozzle construction. It also allows for the optimization of nozzle design through the analysis of various parameters, such as shape, thickness, and material composition. Polyimide and polyether ether ketone (PEEK) are excellent materials for use in modeling rocket nozzle design and manufacture. The use of these materials can result in high-quality rocket nozzles that can withstand the rigors of spaceflight. In this study, we will study the rocket nozzle performance by using ANSYS software with polymer materials.

Keywords: Nozzle, Designing of nozzle, Rocket engine.

INTRODUCTION

Rocket nozzles are key components of rockets, as they regulate the flow of exhaust gases during propulsion. The creation of a high-quality rocket nozzle requires the use of materials that can withstand high temperatures and pressures. A nozzle is a tool used to simultaneously manipulate a fluid flow's direction and properties. When it leaves or enters a closed chamber or conduit, it is generally employed to boost velocity [1]. A nozzle is a tool used to change the properties of a fluid flow as it leaves (or enters) a closed chamber, notably to enhance velocity. In order to direct or modify the flow of a fluid, nozzles are frequently pipes or tubes with different cross-sections (liquid or gas). Nozzles are widely employed to adjust the stream's pressure, mass, direction, speed, and/or other characteristics. The energy from burning fuel combined with the inducted air in a jet exhaust generates a net thrust. This heated air is sent into a propelling nozzle at high speed, greatly boosting its kinetic energy. A nozzle's main function is to increase a fluid's kinetic energy while decreasing its pressure and internal energy. Nozzles are also used to increase the velocity of fluid and maximize the thrust. When the mixture of fuel and propellant is burned in a combustion chamber, nozzles are typically pipes or tubes with various cross-sectional areas that can be used to adjust fluid flow. These devices are regularly employed to control flow, speed, direction, and mass [2].

Polymer materials offer several advantages such as flexibility, durability, and minimal thermal expansion which make them ideal for designing high-performance nozzles. These simulations exhaust gases through nozzle geometries and variations in polymer composition, enabling precise optimization of nozzle design to achieve maximum thrust at different altitudes and velocities. Polyimide is an excellent material for use in the fabrication of nozzle components that require a high level of durability and resistance to high temperatures. PEEK, on the other hand, can be used in the manufacture of nozzle components that require a high level of mechanical and thermal properties [3-5].

EXPERIMENTAL

Numerical consideration and methodology

The hyperbolic nature of the Euler equations in supersonic flow means that only upstream conditions can affect the flow. The nozzle flow field in this situation can be calculated using the MOC (method of characteristics). This technique is most frequently used in rocket nozzles to generate nozzle shapes and assess loads and performances. Furthermore, wave-free parallel flow in the test section at the specified Mach number is crucial for supersonic convergent diverging nozzle. [6, 7]. The shortest length L for shock-free, isentropic flow is depicted. If the contour is created within a length smaller than L, shocks will form inside the nozzle.

Computational model

This issue examines the air and combustible gas flow along the turbojet engine's inlet and exhaust passages. Convergent exhaust nozzles and divergent inlets are used. The analysis also displays the mole fractions of each species. Define material attributes and enable physical modeling.

---

* To whom all correspondence should be sent:
E-mail: vishal.saxena@mjpru.ac.in

1. Specify the domain as being two-dimensional, and leave the default (segregated) solver in place.
2. Turn on the k-epsilon turbulence model.
3. Enable the exchange and reaction of chemical species.
4. Make the field variable initial.
5. Enable residual plotting throughout the computation.

RESULTS AND DISCUSSION

How are nozzles designed?

To design a rocket nozzle using polyimide and PEEK, several factors are considered. These include the shape and dimensions of the nozzle, the combustion properties of the fuel used, and the environmental conditions in which the nozzle will operate. The shape and dimensions of the nozzle are crucial factors in determining the thrust and efficiency of the propulsion system. The combustion properties of the fuel used will determine the temperature and pressure in the combustion chamber, which will affect the durability of the nozzle. The environmental conditions in which the nozzle will operate will affect the overall performance of the propulsion system [8].

There are a few variables that affect how a nozzle is designed, including the amount of thrust it should generate, exit velocity, pressure at the engine's exit and mass flow rate. To comprehend the phenomenon of subsonic flow through a rocket engine nozzle at various divergence angles, a CFD analysis of the nozzle was performed. The governing equations were solved using the finite-volume approach in the ANSYS fluent® program employing a two-dimensional axis-symmetric model for the analysis. Various techniques, including the analytical method and experimental methods including prototypes, can be used to solve engineering problems [9]. It's a really challenging and complicated analytical process. Here, analysis was done on divergent-angle nozzles, resulting in changes to the external diameters 32, 34, 40, 44, 50, and 55.

CFD have shown to be a useful technique for getting reliable solution over mentioned restrictions. The trend of several flow parameters is also studied in this work. After that, when the flow begins to diverge, it isentropic and expands until it achieves supersonic velocity, but the ratio of the area at the neck to the area at the exit determines the Mach number.

When the gases expand in the diverging portion, their pressure and temperature also decrease [10]. The expanding process led to this. The quantity of thrust that should be produced, the engine's exit velocity, the pressure there, and the engine's mass flow rate are all factors that affect the nozzle design. The speed of sound at exit depends on exit temperature. Every one of these flow variables is impacted by the rocket design. A rocket is bulky; a nozzle is little [11].

Software used

The programme we employed to create several nozzle types is known as ANSYS Fluent 2021 R1. There are five components to it: geometry, mesh, setup, solution, and result. The geometry section defines the nozzle's dimensions in 2D before converting them to 3D. While working with geometry in the past, extra caution should be used since if it is not appropriately defined, we won't find the needed solutions.

1. To ensure proper heat distribution, the 3D nozzle is further separated into pieces in the mesh section.
2. We include various materials and boundary conditions in the setup and solution parts. Then computations are performed and iterations are performed.
3. The results section, as the name suggests, has a number of contours, including pressure, velocity, kinetic energy, temperature, etc.

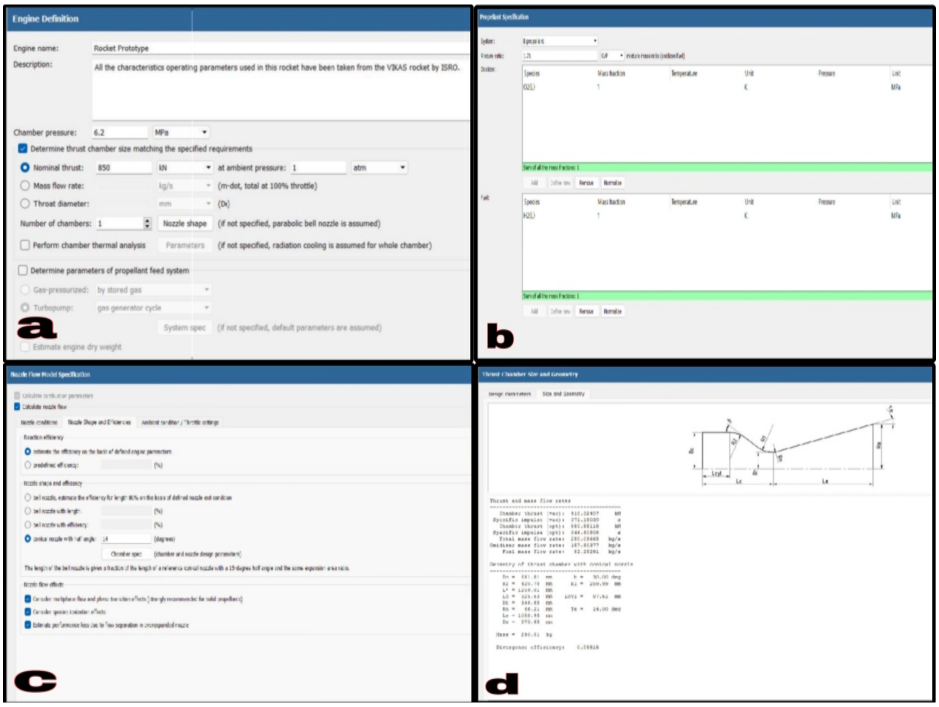
Plotted are the key points: The necessary dimensions must first be created using the provided coordinate points. Producing edges: To construct edges, join these focal points together, we must be considering how important it is to the process, make sure the appropriate key points are connected with one another. Creating faces: The generated 2D figure does not provide enough thickness to be transformed into a 3D figure.

Procedure for analysis

Meshing: We now close the geometry part and launch the mesh section. We divide the figure into various components in the mesh portion, and then accurately size & mesh each piece [12]. Put together the required mesh (Fig. 3).

Applying boundary conditions: We define the boundary conditions specified above, such as temperature, inlet pressure, outlet pressure, and viscosity set to Sutherland technique, in the setup and solution phase because results are precise and plausible.

We will turn on the energy calculations, choose air as the material, and determine its ideal density [13]. Finally, we'll perform calculations and iterations (Figs. 4 and 5).



Engine Definition

Engine name: Rocket Prototype

Description: All the characteristics operating parameters used in this rocket have been taken from the VIKAS rocket by ISRO.

Chamber pressure: 6.2 MPa

☑ Determine thrust chamber size matching the specified requirements

● Nominal thrust: 850 kN at ambient pressure: 1 atm

○ Mass flow rate: kg/s (m-dot, total at 100% throttle)

○ Throat diameter: mm (Dt)

Number of chambers: 1 Nozzle shape (if not specified, parabolic bell nozzle is assumed)

☐ Perform chamber thermal analysis Parameters (if not specified, radiation cooling is assumed for whole chamber)

☐ Determine parameters of propellant feed system

○ Gas-pressurized: by stored gas

○ Turbopump: gas generator cycle

System spec (if not specified, default parameters are assumed)

☐ Estimate engine dry weight

Propellant Specification

Oxidizer: O2(L) 1 K MPa

Sum of all the mass fractions: 1

Fuel: H2(L) 1 K MPa

Sum of all the mass fractions: 1

Nozzle Flow Model Specification

☑ Calculate nozzle flow

● conical nozzle with half angle: 14 (degrees)

Thrust Chamber Size and Geometry

Fig. 1.(a) designing of rocket nozzle using engine definition, (b) propellant specification, (c) nozzle flow model specifications, (d) size and geometry output.

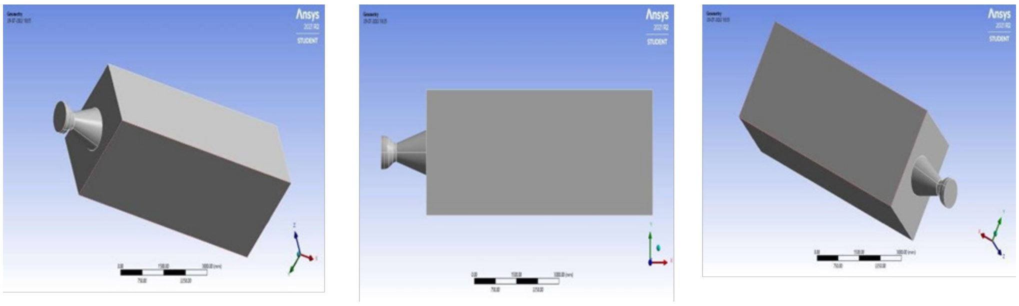


Fig. 2. Designing of nozzle

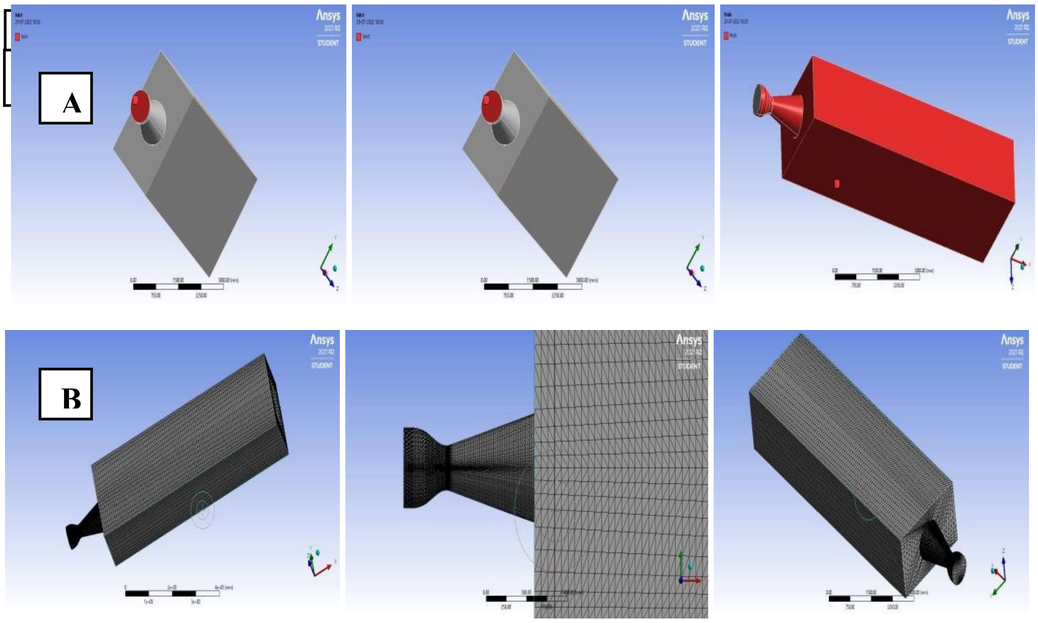


Fig. 3. (A) Named selection in ANSYS mechanical (B) Meshing in ANSYS mechanical

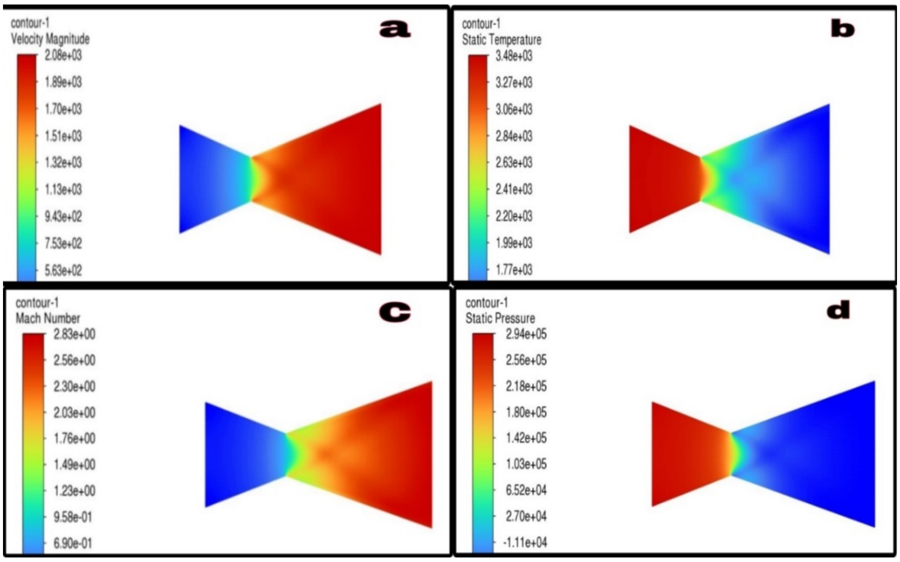


Fig.4.(a) velocity distribution, (b) temperature distribution, (c) mach number variation, (d) static pressure contour

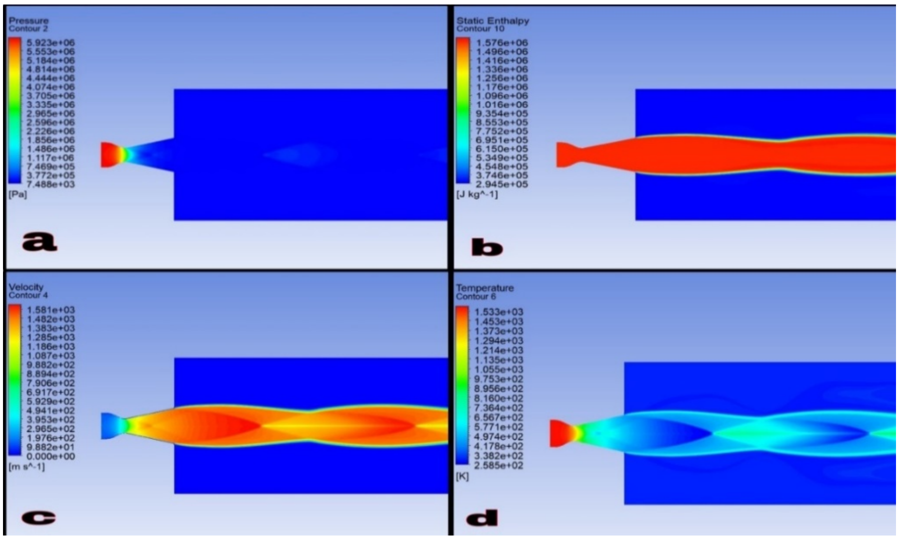


Fig. 5.(a) Pressure contours in far field, (b) Enthalpy contours in far field, (c) Velocity contours in far field, (d) Temperature contours in far field.

Table 1. Results of the conical conventional nozzle

Nozzle type	Average velocity
Conical nozzle	906e+02
Bell nozzle	882e+02
Dual bell nozzle	917e+02

Table 2. Dimensions and boundary conditions for the bell nozzle

Nozzle type	Exit velocity
Conical nozzle	2.18e+05
Bell nozzle	2.56e+05
Dual bell nozzle	2.70e+04

ANSYS software does the CFD analysis of various nozzles to determine the mass's temperature, velocity, and other characteristics as it passes through various-sized nozzles, as well as forms. We consider fluid to be compressible, so the energy approach is activated as well as calculations based on density. The K-epsilon-2 equation model is chosen. The turbulent kinetic energy and turbulent dissipation rate are both selected to be second-order upwind [14].

CONCLUSION

In this review, we have observed that polymer material modeling and simulation is critical in the design of rocket nozzles. Accurate modeling of the thermal behavior and mechanical properties of polymers under extreme conditions is crucial in ensuring optimal performance of rocket nozzles. Multi-scale modeling approaches that account for the polymer's microstructure, chemical composition, and mechanical properties are required to accurately model polymer materials. Advances in polymer material modeling and simulation will lead to the development of more efficient and reliable rocket propulsion systems that can take us to new heights in space exploration. Future work on the design, analysis and simulation of the rocket propulsion system would be CFD calculations, heat transfer calculations and material choices for the nozzle.

REFERENCES

1. A. T. Bolson, *J. Fluids Eng.,* **96**(2), 21 (1974).
2. B. P. Krushna, *Int. J. Res. Eng. Technol.,* **2** (3), 412 (2013).
3. T. Benson, *Nat. Aeron. Space Admin.* (2013).
4. G.V.R. Rao, *J. Jet Propul.,* **28** (6), 377 (1958).
5. J. M. Farley, C. E. Campbell, *Nat. Aeron. Space Admin.* (1960).
6. P. Sajesh, M. Sajesh, *Int. J. Eng. Sci.*, **2** (2),196 (2013).
7. K. Kuo, R. Houim, in: 47th AIAA/ASME/SAE /ASEE. *Joint Propul. Conf. Exhibit.,* **23** (5), 5608 (2011).
8. Davydenko, *Aerosp. Sci. Technol.,***11** (5), 55 (2007).
9. Hagemann, *J. Propul. Power,* **14**(3), 620(1998).
10. M. Frey, Hageman, *J. Propul. Power,* **15**(2), 225 (1999).
11. Hagemann, *Adv. Nozzle Concepts Future Rocket Eng. Appl.*, **4** (5), 3 (2002).
12. D. R. Bartz, *Adv. Heat Transfer*, **2** (1), 43 (1965).
13. R. A. Signorelli, J. R. Johnston. *Nat. Aeron. Space Admin.* 0050 (1963).
14. Shaffer, T. Peter, *Therm. Shock Resis. Refract. Ind. Res.*, **5**(5),36 (1963).