SEMARAK ILMU PUBLISHING
202103268166(003316878-P)

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage:
https://semarakilmu.com.my/journals/index.php/fluid_mechanics_thermal_sciences/index
ISSN: 2289-7879

Optimization of Process Parameters for Flow Nozzle with Different Geometry using Computational Fluid Dynamics Method

Vinayaka Nagarajaiah[1,*], Sara Solomon Raj[2], Ayampalayam Nanjappan Swaminathen[3], Paluru Kiran Kumar[4], Chintala Indira Priyadarsini[4], Kodipaka Mamatha[5]

[1] Department of Aeronautical Engineering, Nitte Meenakshi Institute of Technology, Yelahanka, Bengaluru – 560064, India
[2] Department of Mechanical Engineering, Chaitanya Bharathi Institute of Technology(A), Gandipet, Hyderabad-75, Telangana, India
[3] Department of Civil Engineering, Adi Shankara Institute of Engineering and Technology, Kalad, Kerala, India
[4] Department of Mechanical Engineering, Chaitanya Bharathi Institute of Technology, Gandipet, Hyderabad-500075, Telangana, India
[5] Department of Mathematics, Vardhaman college of engineering, Shamshabad, Telangana 501218, India

ARTICLE INFO

Article history:
Received 17 December 2023
Received in revised form 16 May 2024
Accepted 27 May 2024
Available online 15 June 2024

Keywords:
Fluid mechanics; CFD; bell nozzle; conical nozzle; ANSYS

ABSTRACT

A nozzle is a tubular structure designed to facilitate the flow of hot gases. Rocket nozzle designs typically consist of a stationary convergent section and a stationary divergent section. The term "CD" is an abbreviation for "convergent-divergent" and is used to refer to this specific nozzle type. Upgrades are being made to nozzles and other engine components in order to enhance performance and optimise thrust delivery. Application-specific improvements will be made to rocket nozzles and other current combustion expansion systems. An instance of such progress is the implementation of the bell and twin bell nozzle. This study conducted a comparative analysis of two kinds of nozzles, namely bell and conical, at different Mach speeds to ascertain which one yields the most optimal flow. Subsequently, the flow parameters were modelled with computational fluid dynamics (CFD). Ensuring optimal pressure thrust integral throughout the supersonic zone of the nozzle surface while taking the base pressure into account was the aim of the problem formulation. The research focused on irrotational flow, taking into consideration the impacts of boundary layers.

1. Introduction

The de Laval type of nozzles, often utilised in rocket engines, are used to accelerate and expand the combustion byproducts to achieve high supersonic velocities. In essence, propellants are injected into a combustion chamber using pressures ranging from two to several hundred atmospheres. Either high-pressure ullage gas or pumps may achieve this pressure. Subsequently, the combustion chamber serves as the location where the propellants undergo combustion. The conversion of kinetic energy occurs by the use of a nozzle, which is connected to the combustion chamber, to transfer the energy from the combustion products that possess high pressure and temperature. Increasing the velocity of the gas to a high level and then decreasing it. A nozzle may have several classifications,

* *Corresponding author.*
E-mail address: vn23design.engg@gmail.com

https://doi.org/10.37934/arfmts.118.1.142154

such as conical, bell-shaped, aeroplane, jet, high-velocity, propellant, magnetic, or spray-shaped. This research utilises both a conical and a bell-shaped nozzle. Baidya *et al.,* [1] used computational approaches to assess the three main nozzles of a hypersonic mixed cycle propulsion system. These nozzles included designs for turbo-jets, ramjets, and scramjets. The nozzles underwent testing under several flying situations, namely at altitudes of 10,000 feet with a speed of Mach 1.5, 50,000 feet with a speed of Mach 2.5, and 80,000 feet with a speed of Mach 4. All three optimised nozzles demonstrated superior performance. The thrust performance of the conical nozzle was strong at sea level, but it decreased as the altitude increased. The geometry of the nozzle, which is a crucial factor in determining rocket performance, may be influenced by both structural and thermal stresses.

Kantu [2] conducted research on this subject. A bell nozzle was built via Rao's approach and simulated using the Ansys 2021 R2 Student edition. The research seeks to ascertain if the design can endure stresses without substantial deformation and without notable changes in the shape of the nozzle. Structural failure may happen when the level of stress beyond the allowable limitations. The investigation seeks to ascertain if the design can withstand stresses without significant deformation.

2. Literature Survey

Nagineni and Deepika [3] conduct a comparison of low thrust rocket engines that have constant thrust and those that have variable-thrust. The primary objective of their research is to assess the efficiency of these engines in transitioning between orbits, while optimising the payload mass in the central Newtonian gravitational field. An investigation of the pressure and temperature characteristics of a rocket nozzle with two inlets at a Mach number of 2.1 is carried out using specialised computational fluid dynamics (CFD) software. The performance of the rocket nozzle with two inlets is improved by the rapid temperature changes at the exit section.

Toufik *et al.,* [4] provide a numerical technique for the design of nozzles, using sonic-line computation and the inverse method of characteristics. The software computes the shape of nozzle extensions in order to provide a consistent wall pressure. An analysis is conducted on the thermodynamic parameters and aerodynamic performances, and the findings are then compared to the data obtained by ONERA-France and CNRS. The performance and weight of several configurations (Ideal, TIC, and Dual-Bell) are compared. Sreenath and Mubarak [5] aimed to create and evaluate a contoured convergent diverging bell nozzle. This nozzle will be designed utilising a 2D, parabolic contoured thrust-optimized axisymmetric nozzle that has minimal shock wave effects. The flow is at a temperature of 303K, which is very low. A complete Bell nozzle is constructed using Gambit software, and the process of creating a mesh and doing an analysis is carried out using FLUENT software. Due to symmetry considerations, just one half of the nozzle is simulated, with ambient conditions taken into account. A control volume is created to accurately consider the interplay with the surrounding environment.

Wuye *et al.,* [6] assessed the efficiency of an aerospike nozzle while using solid propellant. Simulation experiments indicate that the thrust of the nozzle first reduces and then rises when subjected to backpressure, which is in contrast to the behaviour seen in typical bell nozzles. The suggested technique introduces a parametric optimisation approach to optimise overall impulse, without constraining it to a given height. This method offers greater performance at both high and low altitudes.

Hagemann *et al.,* [7] explained because of its gradient shape in the near throat region, thrust-optimized contoured nozzles perform better than TIC nozzles, resulting in increased flow expansion. As a result, there is an internal shockwave that raises pressure close to the nozzle's exit plane. Davis *et al.,* [8] specifically designed to provide an exhaust profile that flows in a single direction, truncated idealised contoured (TIC) nozzles are bell-shaped nozzles. Situated close to the throat region, the walls at the plane of departure run parallel to the nozzle's axis. However, it is essential that the nozzle's length be 50 times the throat's radius. A shorter nozzle may result in an early straightening out of the flow, which would reduce nozzle efficiency.

Balaji *et al.,* [9] researched on dual bell nozzles has made limited advancements since its first suggestion. Experimental testing and implementation have been completed; however, the process of large-scale development necessitates the establishment of universal design characteristics. This research uses computational fluid dynamics (CFD) to analyse a specific nozzle and utilise its results as a standard for evaluating the whole approach to investigating dual bell nozzles. In their study, Shiva *et al.,* [10] examined the use of a dual-bell nozzle in booster engines for reusable launch vehicles and found that it can improve performance. In addition, the research performed hot firing experiments to evaluate the thermal stability and structural integrity of the nozzle. The flow of combustion gas exhibited sonic, supersonic, and hypersonic velocities, reaching a Mach number of 5.021 at the exit plane. The thermal layer recession duration was 60 seconds for 100 load steps, and stress analysis indicated minimal stresses caused by aerodynamic pressure and temperature.

Humphreys *et al.,* [11] used the calculus of variations to formulate equations that determine the optimal shape of thrust plug nozzles, taking into account a given intake geometry and imposed limitations. Through parametric study, the ideal injection angle and cowl lip radius are determined. Optimising the pressure thrust integral over the plug surface's supersonic zone while accounting for boundary-layer effects and base pressure is the goal. Examples are provided for a constant mass flow rate and plug length.

Rommel *et al.,* [12] conducted calculations on the flow field of a plug nozzle using various models and ambient pressure settings. These calculations validated the findings of previous tests conducted by DLR. Nevertheless, models demonstrated distinct flow characteristics that resulted in flow separation farther downstream. The work enhanced the wall functions in the turbulence model and devised a novel semi-empirical method to precisely forecast the separation point.

Using an already-existing LOX/GH2 thrust chamber, Stark *et al.,* [13] evaluated the design of a film-cooled dual-bell nozzle before introducing it as a part of a thrust chamber assembly. The nozzle is a scaled-down version of an existing TIC nozzle that includes the injection of a gaseous hydrogen cooling layer. Future analyses will look at variations in the mass flow of cooling film and the rate of firing (ROF).

The study conducted by Pandey and Singh [14] is a numerical investigation that was conducted to analyse gas flows in a conical nozzle at various angles using 2-dimensional axis-symmetric models. The research observed fluctuations in Mach number and pressure ratio, with a lower Mach number seen at the exit for 40 angles and a higher turbulence intensity observed at the exit for 160 degrees. The Mach value at the nozzle exit was 2.91 for angles of 80, while the same Mach number was observed for angles of 120. According to the research, the optimal number of conical nozzles for achieving maximum thrust is either 120 or 160.

Joshi *et al.,* [15] examined the design of nozzles for scientific exploration of celestial bodies, with a specific emphasis on their effectiveness in various atmospheric conditions. The analysis takes into account variables such as Mach velocity, temperature, and pressure, and use computational fluid dynamics (CFD) simulations to comprehend the efficiency across various settings. The objective is to identify appropriate positions for nozzles to release gases such as hydrogen, helium, methane, and carbon dioxide.

Shakya and lohia [16] examined the enlargement of C-D nozzles via the use of experimental and computational techniques in order to enhance performance under specified circumstances. The ANSYS FLUENT 16® computer code was used to determine parameters such as physicist number, static pressure, and shocks for cone-shaped and contour nozzles. At low divergence angles, the physicist observed that the disc and reflection patterns created in cone-shaped C-D nozzles were comparable. Nevertheless, in cases when the divergence angles were high, no shocks were detected. Convex nozzles exhibited superior exit velocities and improved flow detachment.

Nair *et al.,* [17] investigated the use of conical and truncated conical plug nozzles in rocket engines as altitude-compensating nozzles. Numerical simulations are used to verify the performance of the system and to make comparisons with base bleeds. A two-equation shear stress transport k-ω turbulence model and axisymmetric two-dimensional models are used in the work to solve the Reynolds-averaged Navier Stokes equations. Zhang *et al.,* [18] conducted an empirical study of convergent conical nozzles was carried out at the AIAA Propulsion Aerodynamic Workshop. The research included axisymmetric, three-dimensional, and unstable calculations for three specific scenarios. The findings demonstrated strong concurrence with empirical data, while the reliability of the three-dimensional Navier-Stokes conclusions may be compromised. The main frequency at which shedding occurred was 32.2 kHz.

Benderradji *et al.,* [19] investigated several physical phenomena that occur in propellant nozzles, including as supersonic jets, jet separation, unfavourable pressure gradients, shock waves, turbulent boundary layers, and large-scale turbulence. The CFD-FASTRAN search algorithm and Navier-Stokes equations are used to determine fluidic vectorization, and the impact of NPR pressures on shock wave structure is investigated. The research also highlights the separation zone generated by the deflection of the primary jet and fluid jets.

Pillai *et al.,* [20] examined the efficiency of bell nozzles at various altitudes, specifically addressing the problem of flow separation which leads to higher lateral forces, vibrations, and decreased thrust. The research used ANSYS simulation software to analyse changes in flow separation points when secondary flow injection is present or absent.
In their study, Khan *et al.,* [21] discovered that an increase in the area ratio results in a drop in the initial values of base pressure. The efficacy of the control mechanism is constrained as the area ratio increases. Smaller ratios, such as 2.56 and 3.24, are optimal for maintaining control. The microjets are appropriate for these ratios at a Mach number of 2.2, and the control mechanism does not have a detrimental impact on the static wall pressure or duct flow characteristics.

The research conducted by Rosly *et al.,* [22] demonstrates that the Second Order Runge Kutta Scheme is the optimal method for attaining a convergent solution in non-smooth flow issues or flow scenarios including shock waves. Nevertheless, the fourth-order Runge-Kutta scheme is unable to meet the convergence conditions if they are set to a value lower than 0.01. All Total Variation Diminishing (TVD) schemes exhibit convergence, but the majority need a substantial number of iterations. The Modified Runge Kutta-Harten Yee scheme is a suitable numerical method for handling flow problems involving shock waves [22].

Pujowidodor *et al.,* [23] performed a simulation using a two-dimensional model with a constant flow, an impermeable adiabatic wall, and an ideal fluid. The model's predictions were compared with the RNG k-ε and RSM turbulence models. Every model made forecasts for pressure, velocity, temperature, and density. Nevertheless, the conventional k-ε model exhibited an over estimation of turbulent kinetic energy and dissipation rate. The investigation revealed that using adjusted constants for the dissipation equation and eddy viscosity model resulted in enhanced predictions in comparison to the initial constant of the k-ε model.

3. Design and Methodology

A nozzle is a straightforward apparatus, consisting of a specifically designed tube that facilitates the passage of hot gases. Rockets often use a stationary convergent part and a stationary divergent section for the design of the nozzle. Utilising ANSYS software, we conducted a design and simulation process to determine the impact of flow on the nozzle. A wide range of nozzle designs and features have been investigated based on the literature cited in references 1 to 20. The comparison is made between the results obtained from the simulation and meshing of nozzles using ANSYS and CAD software, specifically for the bell and conical type model.

The nozzle in this research was developed using Rao's nozzle and the conical technique. Rao's technique relied on the assumption of inviscid isentropic flow. Conical approaches using lower angles exhibit marginally superior efficiency compared to larger angles, since the latter have a tendency to exert a pulling force on the boundary layer, leading to flow separation. When developing the nozzle, the following assumptions were made: The combustion gases are homogeneous, meaning they have a same composition throughout.

$$P = \rho RT$$

where,
P is the Pressure
ρ is the density
R is the universal gas constant
T is the temperature.

The specific heats of the gases exhibit little variation with changes in temperature and pressure. The flow is intended to be unidirectional and constant. The bell shape design results in a significant expansion angle immediately behind the throat. Afterwards, it is curved in a manner that generates an almost straight flow of gas through the nozzle opening. The contour used is somewhat complex. Figure 1 shows that when an item significantly increases in size, it creates an expansion shock wave at its narrowest point.

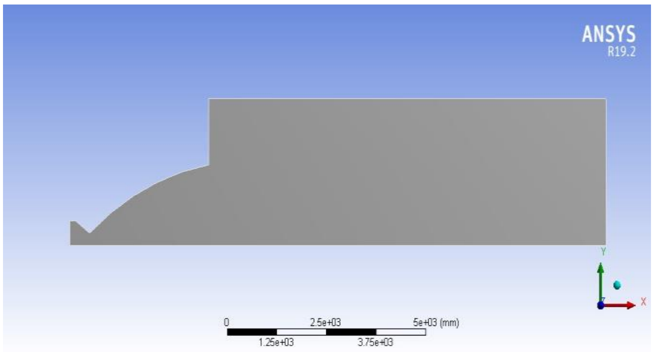

Fig. 1. Bell nozzle profile

The conical nozzle has a linear entrance area that tapers downward, gradually reducing the cross-sectional area until it reaches the throat diameter. This is followed by a linear outlet area that tapers upward, gradually increasing the cross-sectional area. This configuration is seen in Figure 2.

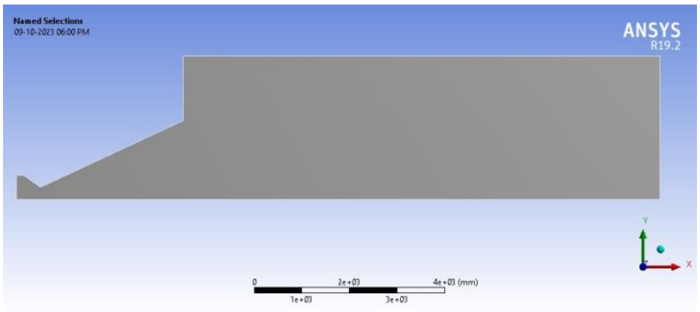

Fig. 2. Conical nozzle profile

4. CFD Modeling

Computer Fluid Dynamics (CFD) is a computer method used in engineering fields like aerodynamics and hydrodynamics to analyse the characteristics of fluid flows, including the interactions between fluids, solids, and gases. It offers data on lift, drag, pressures, and velocities. Computational Fluid Dynamics (CFD) solvers transform the governing physical principles into algebraic equations to enable efficient numerical analysis. Integral equation (IE) solvers are very effective instruments for doing electromagnetic analysis on systems that are both large-scale and complicated. A 2D mesh of components may be created in regions where a more thorough investigation is necessary. The generated mesh is used for simulating two-dimensional fluid dynamics. Modelling 2D flows requires much more processing resources compared to modelling 1D flows. Therefore, it is advisable to restrict mesh development to regions that are of specific relevance. As seen in Figure 3 and Figure 4.

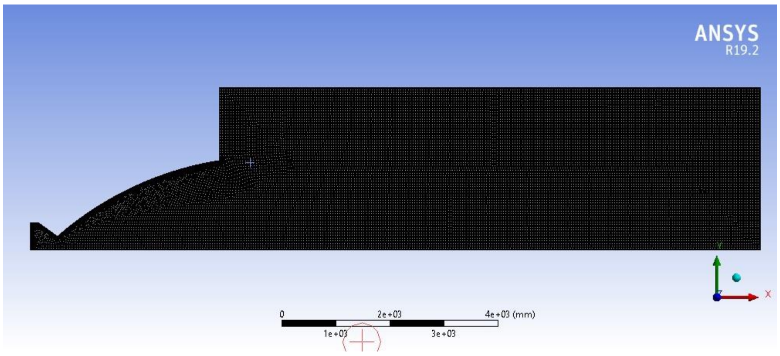

Fig. 3. Geometry and mesh of the CFD in bell nozzle

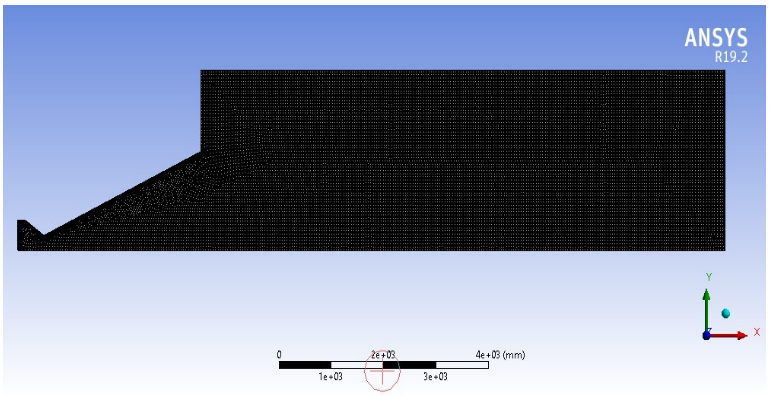

Fig. 4. Geometry and mesh of the CFD in conical nozzle

Table 1 and Table 2 show the parameters based on the research paper [1].

Table 1
Dimensional range of a bell nozzle

Geometrical parameters	Parameter ID	Value range (m)
Nozzle horizontal length	H1	3
Nozzle exit radius	V1	1.65
Nozzle contour radius	R1	6

Table 2
Dimensional range of a conical nozzle

Geometrical parameters	Parameter ID	Value range (m)
Nozzle horizontal length	H1	3
Nozzle exit radius	V1	1.65

4.1 Boundary Condition

Boundary conditions establish the manner in which a system, such as a structure or a fluid, interacts with its environment. Boundary conditions include many factors like as constraints, applied forces, pressures, fluid flow rates, and velocities. The geometry was generated using Ansys. The symmetrical bell and conical nozzle allowed for the creation of half of the geometry for computational economy. The produced geometry is shown in Figure 5. Furthermore, the geometry had a "facial split" feature to facilitate meshing.

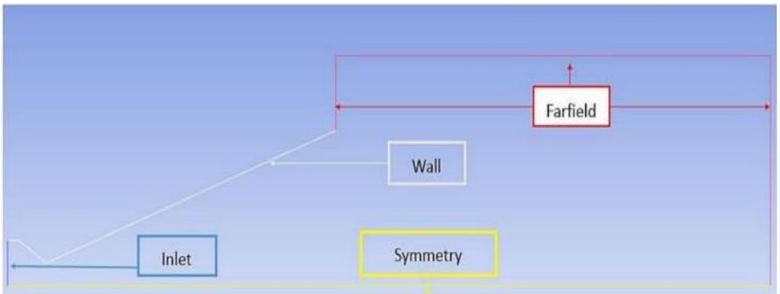

Fig. 5. Nozzle geometry

Table 3 validates the boundary conditions for computational fluid dynamics. The Mach numbers 1 and 2 were assigned to the intake and ambient altitude driving the 10000 m, respectively, followed by a shift in combustion chamber temperature to 2200 K.

Table 3
Validation boundary conditions for computational fluid dynamics (CFD)

Boundary	Temperature (K)	Mach number (M)
Inlet	2200	1,2

5. Result and Discussion

Prior to the simulation, the flow parameters, such as pressure and area, are determined at the nozzle output. The result demonstrates the variations in the Mach number, pressure, temperature distribution, and turbulence intensity. This study examines two distinct Mach numbers, namely 1 and 2.

5.1 Bell Nozzle

The force exerted by a fluid on the bell nozzle is referred to as dynamic pressure. Figure 6 and Figure 7 depict the expansion of the gas near the nozzle exit. The dynamic pressure at Mach 1 and 2 is 5.15 x 10^4 Pa and 2.38 x 10^5 Pa correspondingly as you approach the throat. At the throat, the dynamic pressure is 1.29 x 10^5 Pa for Mach 1 and 4.76 x 10^5 Pa for Mach 2. Following the throat, there is a rapid increase in dynamic pressure along the central axis, suggesting the existence of a shockwave. In contrast to Mach numbers 1 and 2, Mach number 2 has a dynamic pressure that is 3.47 e+05 Pa higher.

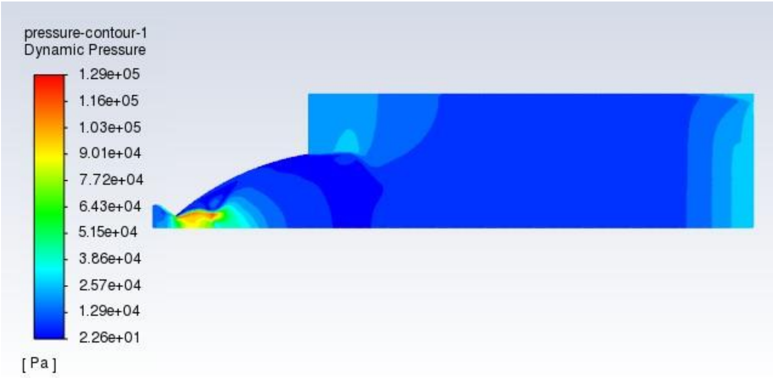

Fig. 6. Dynamic pressure of Mach 1

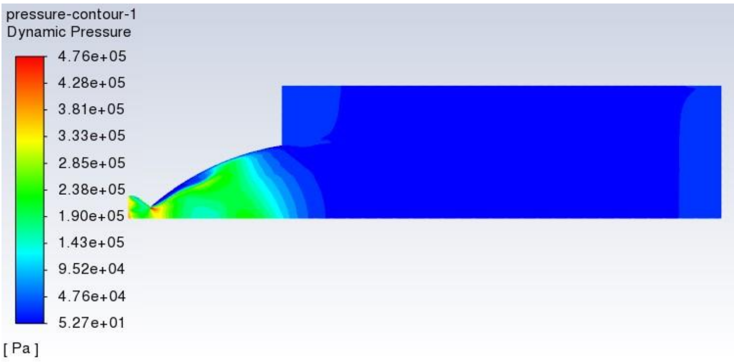

Fig. 7. Dynamic pressure of Mach 2

The static temperature is the temperature measured in a fluid stream when there is no relative motion between the fluid and the measuring device. The provided illustration, labelled as Figure 8 and Figure 9, depicts the expansion of the gas near the output of the nozzle. The temperatures at Mach 1 and 2, when the speed of sound is reached, are 2.41 x 10^3 degrees Celsius and 3.31 x 10^3 degrees Celsius, respectively. As you approach the narrowest part of the flow, known as the throat, the temperature decreases to 1.21 x 10^3 degrees Celsius at Mach 1 and 1.33 x 10^3 degrees Celsius at Mach 2. The static temperature drops precipitously beyond the throat. In comparison to Mach numbers 1 and 2, Mach number 2 has a static temperature that is 9 e+03 higher than Mach number 1.

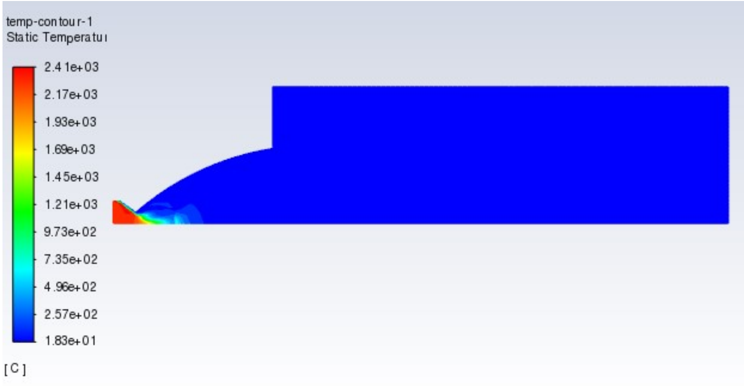

Fig. 8. Static temperature of Mach 1

Fig. 9. Static temperature of Mach 2

The magnitude of the velocity increases as we proceed from the entrance to the exit, as seen in Figure 10 and Figure 11. The intake velocity is 4.96 e+02 at Mach 1 and 6.38 e+02 at Mach 2. The velocity ranges from 1.24 e+03 to 1.60 e+03 in the neck region. Comparing these velocities, Mach 2 velocity is 0.36 e+03 higher than Mach 1.

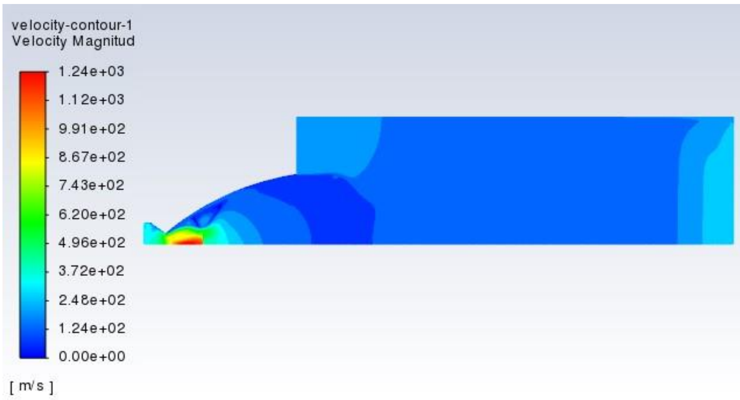

Fig. 10. Velocity magnitude of Mach 1

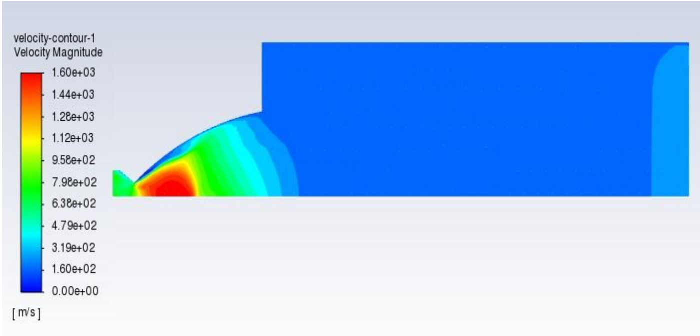

Fig. 11. Velocity magnitude of Mach 2

5.2 Conical Nozzle

Figure 12 and Figure 13 depict the expansion of the gas near the nozzle exit. As you get closer to the neck, the dynamic pressure at Mach 1 and Mach 2 is 16.1 million pascals and 220,000 pascals, respectively. The pressure at the throat is 1.10 x 10^6 Pa at Mach 1 and 8.04 x 10^7 Pa at Mach 2. The dynamic pressure at the axis experiences a rapid increase following the throat, indicating the presence of a shock. When comparing Mach numbers 1 and 2, Mach number 2 exhibits a higher dynamic pressure of 79.3 MPa compared to Mach number 1.

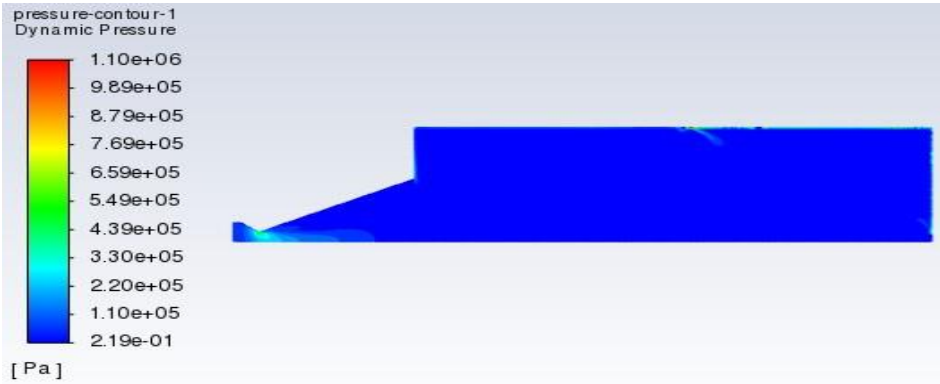

Fig. 12. Dynamic pressure of Mach number 1

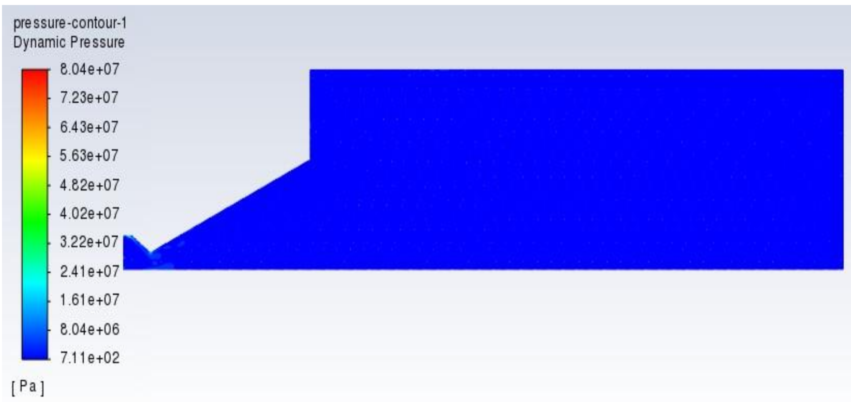

Fig. 13. Dynamic pressure of Mach number 2

Figure 14 and Figure 15 illustrate that the temperature increases as we approach the throat, reaching 2.54 e+03 C at Mach 1 and 4.73 e+03 C at Mach 2. The pressure at the neck is 571 Pa in Mach 1 and 1230 C in Mach 2. There is a significant decrease in static temperature that happens immediately after the throat. When comparing Mach numbers 1 and 2, the static temperature of Mach number 2 is higher than that of Mach number 1.

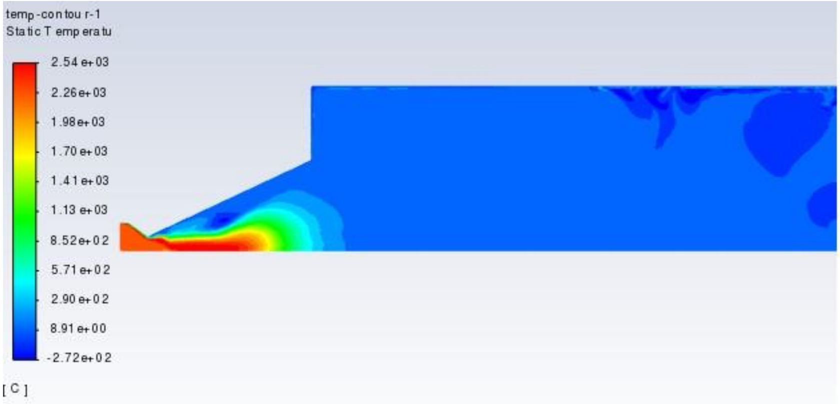

Fig. 14. Static temperature of Mach number 1

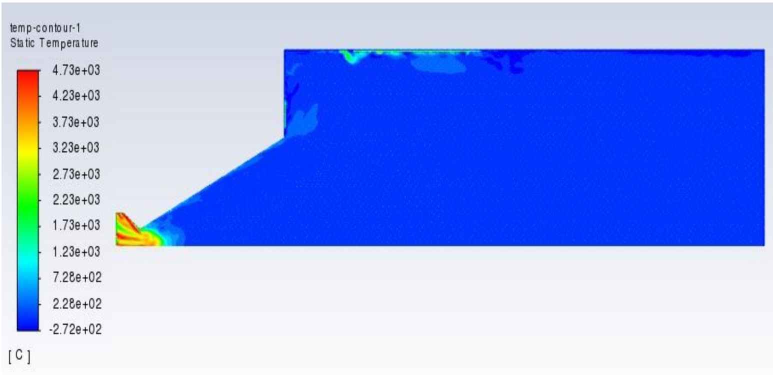

Fig. 15. Static temperature of Mach number 2

Velocity magnitude is observed to increase as one moves from the inlet to the exit, as shown in Figure 16 and Figure 17. The velocity at the inflow is 298 and 3430 in Mach 1 and 2, respectively. The velocity at the throat section varies between 9.93 x 10^2 and 1.14 x 10^4. When comparing these velocities, it can be concluded that Mach 2 velocity is superior to Mach 1 velocity.

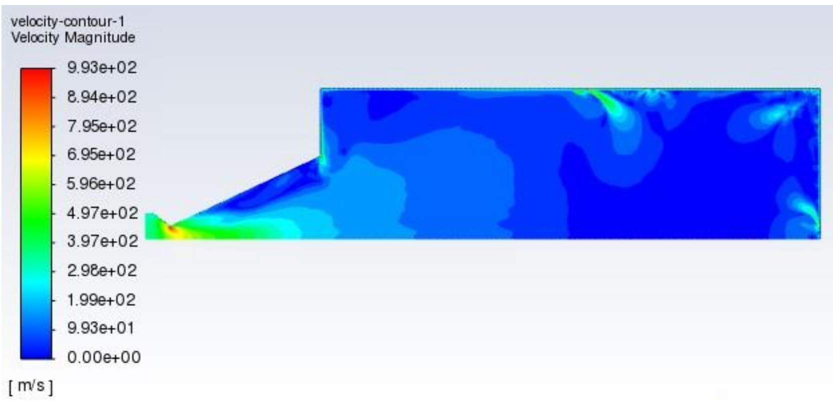

Fig. 16. Velocity magnitude of Mach number 1

Fig. 17. Velocity magnitude of Mach number 2

6. Conclusion

The nozzle design for flow travel in a specific direction and at a specific Mach number (1 and 2) was observed. The results obtained for both bell-shaped and conical-shaped nozzles showed variations in pressure, temperature, velocity, and Mach number along the boundary. Comparing the two Mach numbers, Mach 2 performed better in both nozzle shapes. In the bell-shaped nozzle at Mach 2, the dynamic pressure was measured at 3.47 e+05 Pa, the static temperature at 0.9 e+03, and the velocity magnitude at 0.36 e+03. In the conical-shaped nozzle at Mach 2, the dynamic pressure was measured at 7.93 e+07 Pa, the static temperature at 6.59 e+02 Pa, and the velocity magnitude at 10.4 e+03. Based on the ability to handle higher temperatures, the bell-shaped nozzle outperformed the conical-shaped nozzle.

References

[1] Baidya, Raman, Apostolos Pesyridis, and Maxim Cooper. "Ramjet nozzle analysis for transport aircraft configuration for sustained hypersonic flight." *Applied Sciences* 8, no. 4 (2018): 574. https://doi.org/10.3390/app8040574

[2] Kantu, Bhairab Prasad. "Design and Analysis of a Bell Type Rocket Nozzle." *International Research Journal of Engineering and Technology (IRJET)* 9, no. 3 (2022): 646-649.

[3] Nagineni, Aneesha, and A. Udaya Deepika. "Design and Analysis of Missile Nozzle." In *IOP Conference Series: Materials Science and Engineering, vol. 455, no. 1, p. 012016. IOP Publishing, 2018. https://doi.org/10.1088/1757-899X/455/1/012016*

[4] Toufik, Hamitouche, Sellam Mohamed, Kbab Hakim, Bergheul Saïd, and Lagab Lynda. "Design and performances of the dual-bell nozzle." In *2016 IEEE Aerospace Conference*, pp. 1-7. IEEE, 2016. https://doi.org/10.1109/AERO.2016.7500518

[5] Sreenath, K. R., and A. K. Mubarak. "Design and analysis of contour bell nozzle and comparison with dual bell nozzle." *International Journal of Research and Engineering* 3, no. 6 (2016): 52-56.

[6] Wuye, Dai, Liu Yu, Cheng Xianchen, and Tang Haibin. "Aerospike nozzle performance study and its contour optimization." In *37th Joint Propulsion Conference and Exhibit*, p. 3237. 2001. https://doi.org/10.2514/6.2001-3237

[7] Hagemann, Gerald, Hans Immich, Thong Van Nguyen, and Gennady E. Dumnov. "Advanced rocket nozzles." *Journal of Propulsion and Power* 14, no. 5 (1998): 620-634. https://doi.org/10.2514/2.5354

[8] Davis, Kate, Elizabeth Fortner, Michael Heard, Hannah McCallum, and Hunter Putzke. "Experimental and computational investigation of a dual-bell nozzle." In *53rd AIAA Aerospace Sciences Meeting*, p. 0377. 2015. https://doi.org/10.2514/6.2015-0377

[9] Balaji, Krushna P., P. SrinivasaRao, and B. Balakrishna. "Analysis of dual bell rocket nozzle using computational fluid dynamics." *International Journal of Research in Engineering and Technology* 2, no. 11 (2013): 412-417. https://doi.org/10.15623/ijret.2013.0211060

[10] Shiva, Shankar, Babu Shailesh, Kumar Sai, and Rao TBS. "Thermo-Structure of Dual-Bell Nozzle." *International Journal of Engineering and Innovative Technology (IJEIT)* 4, no. 11 (2015): 292-299.

[11] Humphreys, Robert P., H. Doyle Thompson, and Joe D. Hoffmann. "Design of maximum thrust plug nozzles for fixed inlet geometry." *AIAA Journal* 9, no. 8 (1971): 1581-1587. https://doi.org/10.2514/3.49960

[12] Rommel, Th, G. Hagemann, C-A. Schley, G. Krulle, and D. Manski. "Plug nozzle flowfield analysis." *Journal of Propulsion and Power* 13, no. 5 (1997): 629-634. https://doi.org/10.2514/2.5227

[13] Stark, Ralf, Chloé Génin, Christian Mader, Dietmar Maier, Dirk Schneider, and Michael Wohlhüter. "Design of a film cooled dual-bell nozzle." *Acta Astronautica* 158 (2019): 342-350. https://doi.org/10.1016/j.actaastro.2018.05.056

[14] Pandey, K. M., and A. P. Singh. "CFD analysis of conical nozzle for mach 3 at various angles of divergence with fluent software." *International Journal of Chemical Engineering and Applications* 1, no. 2 (2010): 179. https://doi.org/10.7763/IJCEA.2010.V1.31

[15] Joshi, Prapti, Tarun Gandhi, and Sabiha Parveen. "Critical designing and flow analysis of various nozzles using CFD analysis." *International Journal of Engineering, Research & Technology* 9, no. 02 (2020): 421-424. https://doi.org/10.17577/IJERTV9IS020208

[16] Shakya, Vipul, and Devendra lohia. "CFD Analysis of Convergent Divergent of Supersonic Nozzle." *International Research Journal of Engineering and Technology (IRJET)* 7, no. 5 (2020): 7280-7284.

[17] Nair, Prasanth P., Abhilash Suryan, and Heuy Dong Kim. "Computational study on flow through truncated conical plug nozzle with base bleed." *Propulsion and Power Research* 8, no. 2 (2019): 108-120. https://doi.org/10.1016/j.jppr.2019.02.001

[18] Zhang, Yufei, Haixin Chen, Miao Zhang, Meihong Zhang, Zhao Li, and Song Fu. "Performance prediction of conical nozzle using navier-stokes computation." *Journal of Propulsion and Power* 31, no. 1 (2015): 192-203. https://doi.org/10.2514/1.B35164

[19] Benderradji, Razik, Hamza Gouidmi and Abdelhadi Beghidja. "Effect of the fluidic injection on the flow of a converging-diverging conical nozzle." *International Journal of Energetica (IJECA)* 5, no. 1 (2020): 7-13. https://doi.org/10.47238/ijeca.v5i1.114

[20] Pillai, Sarath, Nandu S. Kumar, V. H. Akshay, Sibin Joju K., and C. Pedda Peeraiah. "Flow Separation Control in Rocket Nozzle." *International Research Journal of Engineering and Technology (IRJET)* 7, no. 8 (2020): 4789-4800.

[21] Khan, Sher Afghan, Abdul Aabid, Fharukh Ahmed Mehaboobali Ghasi, Abdulrahman Abdullah Al-Robaian, and Ali Sulaiman Alsagri. "Analysis of area ratio in a CD nozzle with suddenly expanded duct using CFD method." *CFD Letters* 11, no. 5 (2019): 61-71.

[22] Rosly, Nurhayati, Qamarul Ezani Kamarudin, and Bambang Basuno. "Comparative Study on Various Numerical Schemes for Solving Nozzle Flow Problems." *CFD Letters* 12, no. 6 (2020): 93-106. https://doi.org/10.37934/cfdl.12.6.93106

[23] Pujowidodor, Hariyotejo, Ahmad Indra Siswantara, Budiarso Budiarso, Gun Gun Ramdlan Gunadi, and Candra Damis Widiawaty. "A New Modified k-ε Turbulence Model for Predicting Compressible Flow in Non-Symmetrical Planar-Curvature Converging-Diverging Supersonic Nozzle." *CFD Letters* 12, no. 7 (2020): 57-69. https://doi.org/10.37934/cfdl.12.7.5769