KARYA ILHAM PUBLISHING
CONNECTING MINDS, BRIDGING IDEAS, INSPIRING READERS

Advances in Fluid, Heat and Materials Engineering

Journal homepage:
https://karyailham.com.my/index.php/afhme/index
ISSN: 3083-8134

Analysis of Supersonic Flow in a Converging-Diverging Nozzle with Geometry Variation

Foo Yu Heng[1], Ishkrizat Taib[1,*]

[1] Department of Mechanical Engineering, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia

ARTICLE INFO

Article history:
Received 25 April 2026
Received in revised form 10 May 2026
Accepted 3 June 2026
Available online 30 June 2026

Keywords:
Computational Fluid Dynamics (CFD); converging-diverging nozzle; supersonic flow; grid independence test; Mach number

ABSTRACT

Converging-diverging (C-D) nozzles are widely used in aerospace engineering, propulsion systems, and high-speed fluid transport applications due to their ability to accelerate compressible flow from subsonic to supersonic conditions. The performance of a C-D nozzle is strongly influenced by its geometric configuration, which affects important flow characteristics such as Mach number distribution, velocity, and pressure variation. However, improper nozzle expansion may lead to unstable flow behavior and reduced acceleration efficiency, making geometry optimization an important aspect of nozzle design. This study aims to investigate the effect of nozzle geometry variation on supersonic flow behavior in a converging-diverging nozzle using Computational Fluid Dynamics (CFD). A two-dimensional axisymmetric model was developed and simulated using ANSYS Fluent with a density-based solver under steady-state conditions. Air was treated as a compressible fluid, while the energy equation and Shear Stress Transport (SST) k-ω turbulence model was applied to improve solution accuracy for high-speed flow analysis. A grid independence test was first conducted using six mesh cases with different element sizes to ensure that the numerical results were independent of mesh resolution. Outlet velocity was selected as the primary parameter for convergence assessment. The results showed that the percentage difference in outlet velocity between the finer mesh cases was less than 5%, confirming grid independence. The mesh with an element size of 1.5 mm and approximately 168,000 nodes was selected for further analysis. Three different nozzle geometries were then compared by modifying the curvature of the nozzle profile equation. Among the tested configurations, Geometry 1, defined by $A=0.1+0.5x^2$, produced the highest outlet Mach number of 1.8885 and the highest outlet velocity of 447.86 m/s, indicating the best supersonic performance. In contrast, the steeper expansion profile of Geometry 3 resulted in lower Mach number and weaker acceleration. It can be concluded that nozzle geometry has a significant influence on supersonic flow performance, where a smoother and more gradual diverging section improves pressure distribution and enhances flow acceleration. CFD proved to be an effective tool for evaluating nozzle performance and supporting nozzle design optimization.

* *Corresponding author.*
E-mail address: iszat@uthm.edu.my

https://doi.org/10.37934/afhme.9.1.3544a

1. Introduction

The study of compressible flow is fundamental in many engineering applications, particularly in aerospace, propulsion systems, and high-speed fluid transport [1,2]. One of the most important devices used to control and accelerate compressible flow is the converging-diverging (C-D) nozzle [3,4]. This type of nozzle is designed to accelerate fluid from subsonic to supersonic speeds by utilizing changes in cross-sectional areas, making it essential in applications such as rocket engines and supersonic wind tunnels [5,6].

In a converging-diverging nozzle, the flow first accelerates in the converging section until it reaches sonic conditions at the throat and subsequently expands in the diverging section to achieve supersonic velocities [4]. The performance of such a nozzle is highly dependent on its geometric configuration, which influences parameters such as Mach number distribution, pressure variation, and flow acceleration characteristics [7,8]. Therefore, understanding the effect of nozzle geometry on flow behavior is critical for optimizing design and performance. Fundamental fluid mechanics principles governing nozzle flow are also well established in classical studies of internal compressible flow [9].

With the advancement of computational tools, Computational Fluid Dynamics (CFD) has become a powerful method for analyzing complex flow phenomena without the need for extensive experimental setups [10,11]. CFD allows for detailed visualization and quantitative analysis of flow variables such as velocity, pressure, and Mach number within the nozzle [12,13]. Finite volume methods are commonly applied in CFD simulations because of their strong conservation properties for compressible flows [14].

In this study, a two-dimensional converging-diverging nozzle is modelled and analysed using Computational Fluid Dynamics (CFD). While the fundamental behaviour of supersonic flow in nozzles is well established, there is a need to further investigate on how variations in nozzle geometry influence flow characteristics such as Mach number distribution and pressure variation. Understanding these effects is important for improving nozzle performance and design efficiency in practical engineering applications. Therefore, the objectives of this study are to conduct a grid independence test to ensure solution accuracy, and to evaluate the effect of different nozzle geometries on supersonic flow behaviour by analysing Mach number, velocity, and pressure distributions along the nozzle.

2. Methodology

2.1 Geometry of 2D Converging-Diverging Nozzle

A two-dimensional converging-diverging (C-D) nozzle was developed to simulate compressible flow behaviour. The nozzle geometry was defined using a coordinate-based approach, where the upper wall profile was generated using Eq. (1) [15].

$$A = 0.1 + x^2 \text{ for } -0.5 < x < 0.5 \tag{1}$$

The geometry consists of four boundary segments: the curved upper wall representing the nozzle contour, vertical lines at the inlet and outlet, and a horizontal axis line representing the centreline of the nozzle. These boundaries were combined to form a closed 2D domain as shown in Figure 1 for flow simulation. For the geometry comparison study, variations of the nozzle profile were generated by modifying the curvature of the governing equation. This allows investigation of how different expansion rates affect the flow characteristics within the nozzle.

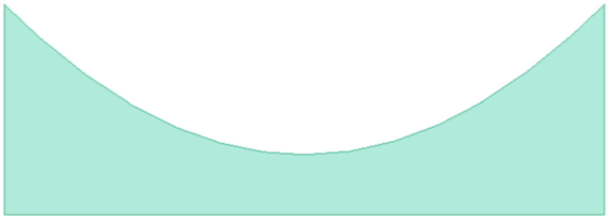


Fig. 1. 2D geometry of the nozzle

2.2 Mesh Generation

The computational domain was discretized using structured quadrilateral elements. A face meshing method was applied to ensure better control over mesh quality and alignment with the flow direction. A grid independence test (GIT) was performed to determine an appropriate mesh size that provides accurate results without excessive computational cost. Multiple mesh cases were generated by varying the global element size, resulting in different node counts. The mesh sizes ranged from coarse to fine, and simulation results were compared to evaluate convergence.

The final mesh selected for further analysis was based on the convergence of velocity results, where the percentage difference between consecutive mesh refinements was found to be less than 5%. This mesh provided a balance between accuracy and computational efficiency (Figure 2).

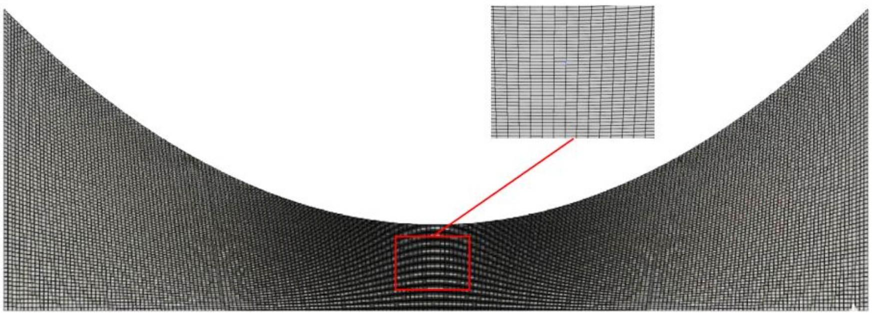


Fig. 2. Meshing

2.3 Boundary Condition and Solver Setup

The simulation was conducted using a density-based solver under steady-state conditions in a two-dimensional axisymmetric domain. This solver is particularly suitable for compressible flow analysis, especially for high-speed and supersonic flow regimes, due to its strong coupling between pressure and density [10]. Air was used as the working fluid and treated as a compressible medium. The energy equation was enabled to account for temperature variations and compressibility effects, which are essential for accurately capturing supersonic flow behavior [11]. Turbulence effects were modeled using the Shear Stress Transport (SST) k-ω model, which provides improved accuracy in predicting boundary layer development and flow behavior in regions with adverse pressure gradients [16,17]. The SST k-ω model is widely used for high-speed internal flow simulations because of its improved near-wall prediction capability and robustness under adverse pressure gradients [18,19].

The boundary conditions were defined to represent the operating conditions of the nozzle. A pressure inlet boundary condition was applied at the inlet to specify the total pressure of the incoming flow, while a pressure outlet boundary condition was imposed at the exit to control the static pressure (Figure 3). The nozzle walls were modeled as stationary with a no-slip condition, ensuring that the fluid velocity at the wall is zero. These boundary conditions allow the simulation to realistically capture the acceleration of flow through the converging-diverging nozzle (Figure 4).

The solution was initialized using the standard initialization method, with initial values computed from the inlet boundary condition to provide a stable starting point for the simulation. A total of 2000 iterations were performed for each case to ensure that the solution reached convergence. Convergence was assessed based on the reduction of residuals and the stabilization of key flow variables such as velocity and Mach number throughout the computational domain.

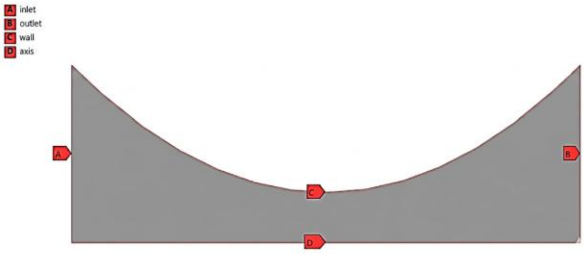


Fig. 3. Name selection label

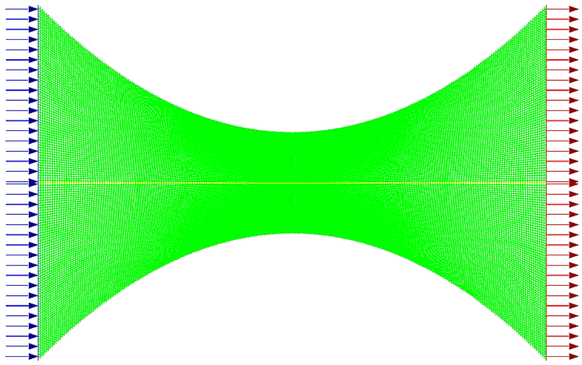


Fig. 4. Inlet and outlet of converging-diverging nozzle

2.4 Grid Independency Test (GIT)

To ensure that the numerical results are independent of mesh size, a grid independence test was conducted. Several mesh configurations were generated by systematically reducing the element size with increasing node counts were simulated, and key output parameters such as outlet velocity were recorded. The percentage difference between consecutive mesh cases was calculated (Eq. (2)). New as the finer meshing; while old as the coarser meshing. A mesh was considered grid-independent when the difference between two successive refinements was less than 5% [13]. Based on this criterion, the selected mesh was used for all subsequent simulations to ensure consistency and accuracy. Similar mesh convergence behaviour has also been reported in previous CFD studies involving converging-diverging nozzles [20].

$$Percentage\ Difference = \frac{|New - Old|}{New} \times 100 \quad (2)$$

2.5 Geometry Comparison Study

To investigate the effect of nozzle geometry on supersonic flow behaviour, multiple nozzle configurations were analysed by modifying the curvature of the nozzle profile equation, resulting in different expansion rates. All simulations were conducted using the same mesh strategy, boundary conditions, and solver settings to ensure a consistent and fair comparison between the different geometries. The flow characteristics were evaluated based on the distribution of Mach number along the nozzle, velocity profiles, and static pressure variation. In addition, centreline plots of Mach number against the axial position were generated to provide a clear comparison of flow acceleration and to illustrate how the different geometrical configurations influence the development of supersonic flow within the nozzle.

3. Results and Discussion

3.1 Grid Independency Test

To ensure that the numerical results obtained from the simulation are independent of the mesh size, a grid independence test was conducted. Several mesh configurations with different element sizes were generated, resulting in varying node counts. For each mesh case, simulations were performed under identical boundary conditions and solver settings, and key output parameters were recorded for comparison.

Table 1 shows the mesh configurations, including element size, number of nodes, and the corresponding outlet velocity and Mach number obtained from each simulation. As the mesh was refined from a coarse to a finer configuration, noticeable variations in the results were observed, indicating that mesh size has a significant influence on solution accuracy. To quantify the variation between successive mesh refinements, the percentage difference in outlet velocity was calculated using Eq. (2). The calculated percentage differences show that the variation in velocity decreases as the mesh becomes finer. In particular, the difference between Case 5 and Case 6 was found to be less than 5%, indicating that the solution is approaching grid independence.

Table 1
Grid independence test result

Case	Element size (mm)	Node count	Output velocity (m/s)	Percentage difference (%)
1	5	15265	665.54	-
2	4	24030	648.69	2.60
3	3	42362	577.58	12.31
4	2	94864	456.11	26.63
5	1.5	168012	423.95	7.59
6	1.3	223290	404.57	4.79

Figure 5 shows the velocity distribution along the nozzle centerline for all mesh cases. It can be observed that the velocity increases gradually in the converging section and rises significantly after the throat region as the flow accelerates into the diverging section. The finer mesh cases show a more stable and smoother velocity profile compared to the coarser meshes, indicating improved numerical accuracy. In particular, the velocity profiles of Case 5 and Case 6 show very close agreement, suggesting that the solution begins to converge at higher mesh densities. This supports the selection of Case 5 as the grid-independent mesh for further analysis.

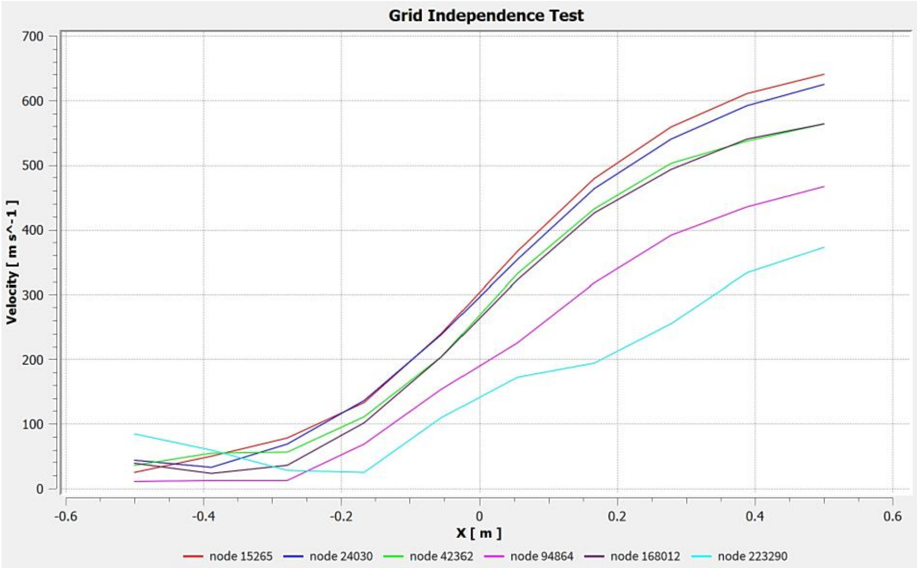


Fig. 5. Velocity distribution along nozzle centerline for grid independence test

3.2 Geometry Comparison Study

To investigate the effect of nozzle geometry on supersonic flow behaviour, three different nozzle profiles were analysed by modifying the curvature of the governing equation. Geometry 1 was defined by Eq. (3), representing a gentler expansion profile, Geometry 2 used the baseline Eq. (1), and Geometry 3 was defined by Eq. (4), representing a steeper expansion profile. All simulations were conducted using the same mesh strategy, boundary conditions, and solver settings to ensure a fair comparison.

$$A = 0.1 + 0.5x^2 \tag{3}$$

$$A = 0.1 + 2x^2 \tag{4}$$

Figure 6 shows the Mach number distribution along the nozzle centerline for the three geometries. It can be observed that all nozzle configurations successfully accelerated the flow from subsonic to supersonic conditions after the throat region. However, significant differences were observed in the rate of acceleration and the final outlet Mach number. Geometry 1 produced the highest outlet Mach number of 1.8885, followed closely by Geometry 2 with 1.8666, while Geometry 3 produced the lowest value of 1.4542.

This indicates that the gentler diverging profile in Geometry 1 provides more effective and stable flow expansion compared to the steeper geometry. Previous studies have shown that nozzle expansion rate strongly influences outlet Mach number and pressure distribution [7]. A smoother area expansion allows the pressure to decrease more gradually, which promotes continuous acceleration of the flow and improves supersonic performance [8]. In contrast, the rapid expansion in Geometry 3 may cause stronger adverse pressure gradients and flow instability, reduce the effectiveness of acceleration and result in a lower Mach number [16]. Numerical investigations of compressible nozzle flow have reported similar trends in Mach number reduction under excessive diverging expansion [15].

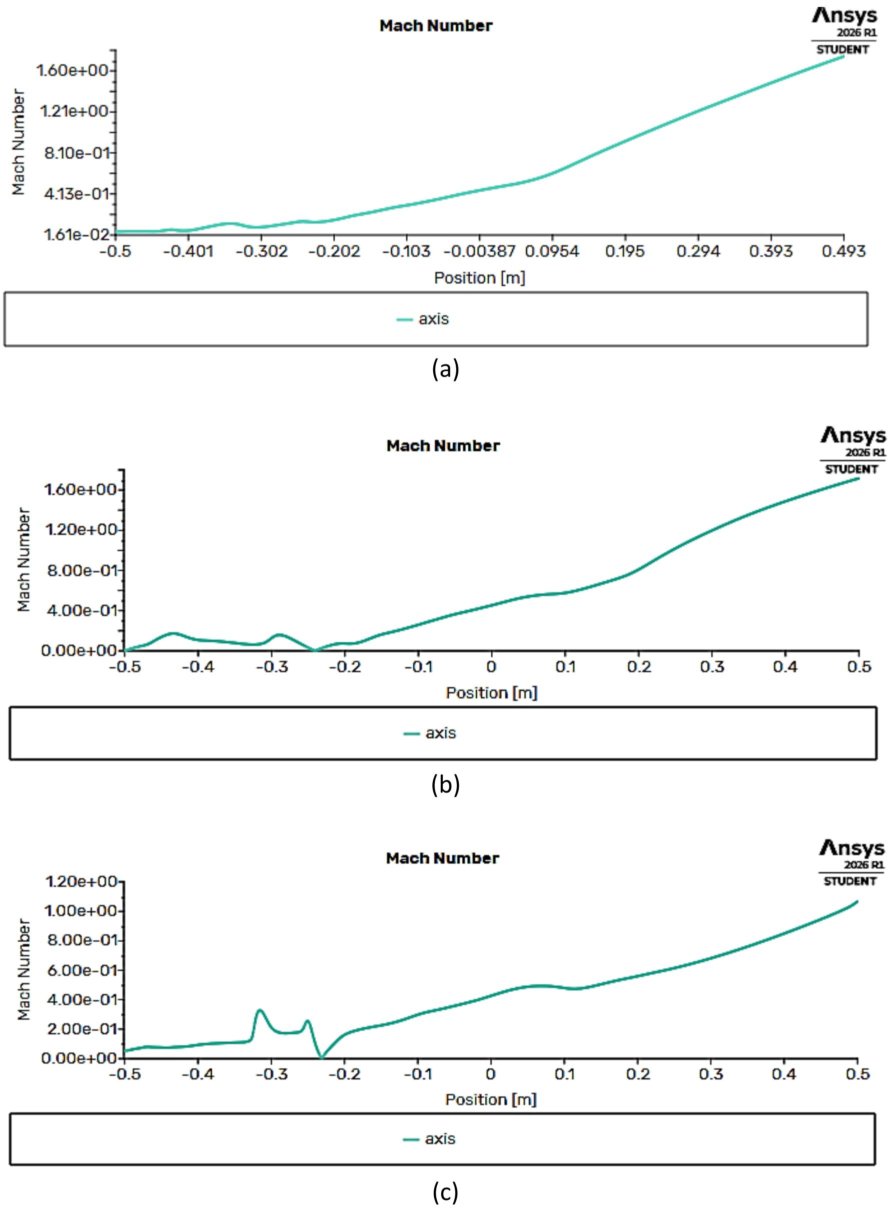
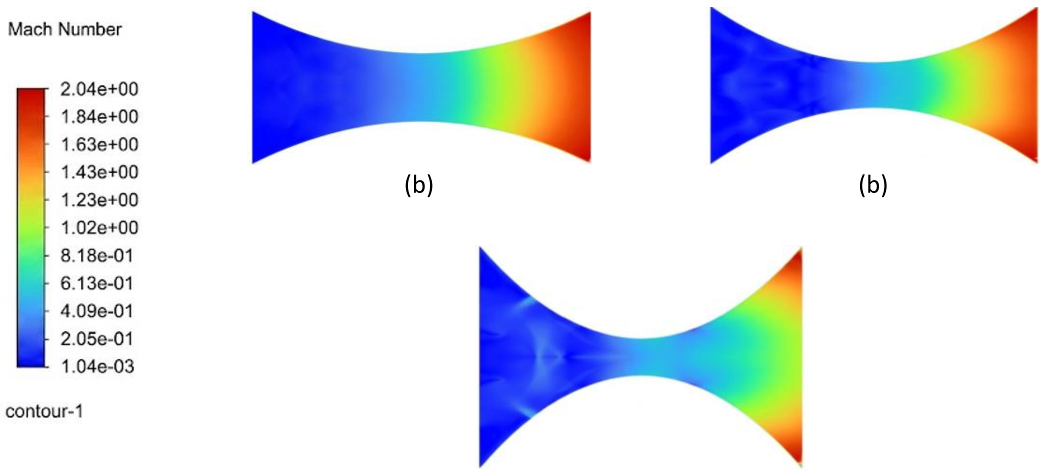


Fig. 6. Chart of Mach number against centerline Axis (a) Geometry 1 (b) Geometry 2 (c) Geometry 3

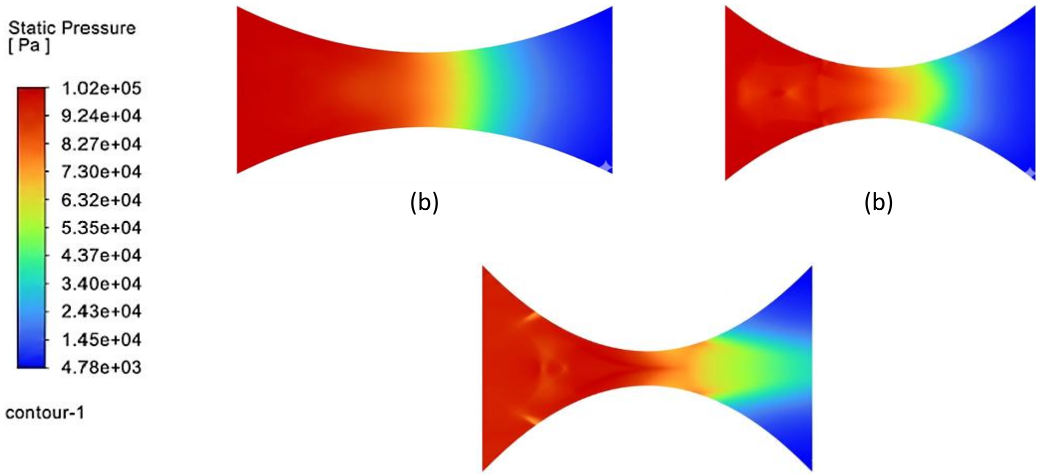
The Mach contour distributions further support this observation. Geometry 1 shows a larger and more uniform supersonic region in the diverging section, indicating better flow acceleration throughout the nozzle. Geometry 2 demonstrates a similar trend but with slightly lower Mach values. Geometry 3 shows a smaller supersonic region and less uniform flow distribution, suggesting that excessive expansion can negatively affect nozzle performance (Figure 7).

Furthermore, the static pressure contours illustrate the relationship between pressure drop and flow acceleration. In all three cases, pressure decreases significantly from the inlet to the outlet as the fluid expands through the nozzle. Geometry 1 exhibits a smoother and more controlled pressure reduction, which contributes to stable acceleration and higher outlet Mach number. On the other hand, Geometry 3 shows a more rapid pressure variation, which may reduce flow stability and weaken the acceleration process (Figure 8).



(c)

Fig. 7. Chart of Mach number against centerline axis (a) Geometry 1 (b) Geometry 2 (c) Geometry 3



(c)

Fig. 8. Static pressure contour (a) Geometry 1 (b) Geometry 2 (c) Geometry 3

The numerical comparison of outlet Mach number and outlet velocity is summarized in Table 2. Geometry 1 achieved the highest outlet velocity of 447.86 m/s, followed by Geometry 2 with 423.95 m/s, while Geometry 3 produced the lowest velocity of 349.66 m/s. The velocity results are consistent with the Mach number analysis, confirming that Geometry 1 provides the best overall performance among the three nozzle designs. From the results, it can be concluded that nozzle geometry has a significant influence on supersonic flow behavior. A gentler diverging section improves flow acceleration by maintaining a more favorable pressure gradient and reducing flow disturbances. Therefore, Geometry 1 demonstrates the most effective design for achieving higher supersonic performance in the present study.

Table 2
Geometry comparison result

Geometry	Outlet Mach number	Outlet velocity (m/s)
1	1.8885	447.86
2	1.8666	423.95
3	1.4542	349.66

4. Conclusions

This study investigated the supersonic flow behavior in a converging-diverging nozzle using Computational Fluid Dynamics (CFD). A two-dimensional axisymmetric model was developed and simulated using a density-based solver under steady-state conditions. The objective of the study was to evaluate the effect of nozzle geometry variation on flow performance and to ensure solution accuracy through a grid independence test.

The grid independence test was conducted using several mesh configurations with different element sizes and node counts. Outlet velocity was selected as the primary parameter for convergence assessment. The results showed that the percentage difference between the finer mesh cases decreased to less than 5%, indicating that the solution had reached grid independence. Based on this criterion, the mesh with an element size of 1.5 mm and approximately 168,000 nodes were selected for further analysis as it provided a good balance between computational efficiency and solution accuracy.

For the geometry comparison study, three nozzle profiles were analyzed by modifying the curvature of the governing equation. The results showed that all geometries were able to accelerate the flow from subsonic to supersonic conditions. However, the nozzle geometry significantly affected the Mach number distribution and outlet velocity. Geometry 1, with the gentler diverging profile $A = 0.1 + 0.5x^2$, produced the highest outlet Mach number of 1.8885 and the highest outlet velocity of 447.86 m/s. In contrast, Geometry 3, with the steeper profile $A = 0.1 + 2x^2$, produced the lowest outlet Mach number and velocity.

The results indicate that a smoother and more gradual area expansion improves pressure distribution and promotes more stable flow acceleration, leading to better supersonic performance. Excessive expansion, on the other hand, may reduce flow stability and weaken acceleration efficiency. Therefore, Geometry 1 was identified as the most effective nozzle design among the three configurations studied.

In conclusion, CFD proved to be an effective tool for analyzing compressible flow behavior and evaluating nozzle performance. The findings of this study demonstrate that nozzle geometry plays a critical role in determining supersonic flow characteristics and should be carefully considered in nozzle design for engineering applications such as propulsion systems and high-speed fluid transport.

References

[1] Anderson, John David. *Modern Compressible Flow: With Historical Perspective. 3rd ed*. New York: McGraw-Hill, 2003.
[2] White, Frank M. Fluid Mechanics Eight Edition. New York, NY, US, Mcgraw-Hill Education (2016): 242.
[3] Liepmann, H. Wolfgang, and A. Roshko. *Elements of Gasdynamics*. New York: Dover Publications, 2001.
[4] Shapiro, Ascher H. *The dynamics and thermodynamics of compressible fluid flow*. New York: Ronald Press (1953).
[5] Cengel, Yunus, and John Cimbala. *Ebook: Fluid Mechanics Fundamentals and Applications (si units)*. McGraw Hill, 2013.
[6] Fox, Robert W., Alan T. McDonald, and John W. Mitchell. *Fox and McDonald's introduction to fluid mechanics*. John Wiley & Sons, 2020.
[7] Mohamed, Souici, Aouissi Mokhtar, and T. Ben Chatti. "Numerical simulation of the compressible flow in convergent-divergent nozzle." *International Journal of Heat and Technology* 35, no. 1 (2017): 673-677. https://doi.org/10.18280/ijht.350328
[8] Khan, Sher Afghan, Omar Mohamed Ibrahim, and Abdul Aabid. "CFD analysis of compressible flows in a convergent-divergent nozzle." *Materials Today: Proceedings* 46 (2021): 2835-2842. https://doi.org/10.1016/j.matpr.2021.03.074
[9] Munson, Bruce Roy. *Wileyplus Blackboard Card for Fundamentals of Fluid Mechanics*. John Wiley & Sons, 2012.
[10] Anderson, John D. *Computational Fluid Dynamics: The Basics with Applications*. New York: McGraw-Hill, 2002.

[11] Versteeg, Henk Kaarle. *An introduction to computational fluid dynamics the finite volume method, 2/E*. Pearson Education India, 2007.
[12] Patankar, Suhas. *Numerical heat transfer and fluid flow*. CRC press, 2018. https://doi.org/10.1201/9781482234213
[13] Ferziger, Joel H., Milovan Perić, and Robert L. Street. *Computational methods for fluid dynamics*. Vol. 3. Berlin: springer, 2002. https://doi.org/10.1007/978-3-642-56026-2
[14] Hirsch, Charles. "Numerical computation of internal and external flows." *Computational methods for inviscid and viscous flows* 2 (1990).
[15] Khalid, Muhammad Waqas, and Muhammad Ahsan. "Computational fluid dynamics analysis of compressible flow through a converging-diverging nozzle using the k-ε turbulence model." *Engineering, Technology & Applied Science Research* 10, no. 1 (2020): 5180-5185. https://doi.org/10.48084/etasr.3140
[16] Menter, Florian R. "Two-equation eddy-viscosity turbulence models for engineering applications." *AIAA journal* 32, no. 8 (1994): 1598-1605. https://doi.org/10.2514/3.12149
[17] Wilcox, David C. *Turbulence modeling for CFD*. Vol. 2. La Canada, CA: DCW industries, 1998.
[18] Pope, Stephen B. "Turbulent flows." *Measurement Science and Technology* 12, no. 11 (2001): 2020-2021. https://doi.org/10.1088/0957-0233/12/11/705
[19] Schlichting, Hermann, Klaus Gersten, and E. KRAUSE. *Boundary Layer Theory. 8th edn*. Springer Berlin, Heidelberg (2000). https://doi.org/10.1007/978-3-642-85829-1
[20] Noh, Mohd Hafiz Mohd. "Numerical investigation of chocked converging-diverging nozzles for thruster application." *IIUM Engineering Journal* 12, no. 3 (2011). https://doi.org/10.31436/iiumej.v12i3.67