Noise Suppression of a C-D Nozzle by Using Chevrons

Achyuth Kumar D[1], Prashanth D[2], Shaurya Rao T[3], Dr. Sravanthi G [4]

[1,2,3]*Student, Dept. of Aeronautical Engineering, IARE, Hyderabad, India*
[4]*Guide, Dept. of Aeronautical Engineering, IARE, Hyderabad, India*

[1]achyuth.2daggumati@gmail.com
[2] prashanthdarsi@gmail.com
[3]shauryatula@gmail.com
[4]g.sravanthi@iare.ac.in

Abstract— **Convergent–divergent (C–D) nozzles play a critical role in high-speed propulsion systems such as rocket engines, gas turbines, and supersonic aircraft. These nozzles are designed to accelerate fluid flow from subsonic to supersonic velocities by utilizing pressure and temperature gradients within the nozzle geometry. However, high-speed jet flows emerging from these nozzles generate significant acoustic noise due to turbulent mixing, shock waves, and pressure fluctuations.**

The present research focuses on the aerodynamic and acoustic analysis of a convergent–divergent nozzle using computational fluid dynamics (CFD) techniques. The geometry of the nozzle was designed using CATIA V5, and numerical simulations were carried out in ANSYS Fluent to evaluate flow characteristics such as velocity distribution, pressure variation, and temperature changes along the nozzle length. Special emphasis is placed on analyzing the effectiveness of chevron structures integrated at the nozzle exit to reduce acoustic emissions.

The CFD analysis investigates compressible flow behavior under different pressure conditions and identifies the optimal working pressure that minimizes acoustic power generation. The results show that the addition of chevrons significantly improves mixing between the jet and ambient air, thereby reducing shock-associated noise and turbulent fluctuations.

The findings of this study contribute to the development of improved nozzle designs capable of achieving both enhanced propulsion efficiency and reduced environmental noise pollution.

***Keywords*—Convergent and Divergent Nozzle, Chevrons, *CFD, Ansys, Catia v5*.**

I. Introduction

High-speed propulsion systems rely heavily on efficient nozzle designs to convert thermal energy into kinetic energy. Among various nozzle types, the convergent–divergent nozzle, commonly known as the De Laval nozzle, is widely used for accelerating gases to supersonic speeds.

The design of a convergent–divergent nozzle allows fluid flow to first accelerate through a converging section until it reaches sonic velocity at the throat. Beyond the throat, the diverging section allows the flow to expand further, resulting in supersonic velocities.

Although these nozzles are highly effective in generating high-speed flows, they also produce intense noise due to:

• Turbulent shear layer interactions
• Shock cell structures
• Pressure mismatch at the nozzle exit
• Mach wave radiation

The noise generated by supersonic jets has significant implications in aerospace engineering. It contributes to environmental noise pollution, affects aircraft design regulations, and may also lead to structural fatigue in nearby components.

To address these challenges, researchers have explored various noise reduction techniques such as:

• Chevron nozzles
• Acoustic liners
• Variable geometry nozzles
• Multi-stream nozzles
• Flow control devices

Among these methods, chevron nozzles have gained considerable attention due to their passive noise suppression capability. Chevron structures consist of triangular serrations at the nozzle exit that enhance mixing between the jet exhaust and surrounding air.

The present research focuses on analyzing the aerodynamic and acoustic behavior of a convergent–divergent nozzle equipped with chevrons. Computational simulations are performed to investigate flow characteristics and identify design parameters that minimize acoustic noise generation.

A. *Background*

A convergent–divergent nozzle operates based on the principles of compressible fluid flow. When a gas flows through a nozzle, its velocity, pressure, and temperature change depending on the cross-sectional area and pressure conditions.

The fundamental working principle of a C–D nozzle can be described as follows:

1. Subsonic flow enters the converging section.
2. Velocity increases as the cross-sectional area decreases.
3. The flow reaches sonic velocity at the throat.
4. Beyond the throat, the diverging section allows further expansion.
5. The flow becomes supersonic in the diverging region.

This process converts pressure energy into kinetic energy, enabling high-speed jet flows.

The Mach number (M), which represents the ratio of flow velocity to the speed of sound, plays a crucial role in analyzing nozzle performance

$$M = V / a$$

Where:

V=Flow velocity
a = Speed of sound

If:

$M < 1 \rightarrow$ Subsonic flow
$M = 1 \rightarrow$ Sonic flow
$M > 1 \rightarrow$ Supersonic flow

In supersonic conditions, pressure and velocity relationships become nonlinear, and shock waves may form due to sudden changes in flow properties

.

B. *Theory*

The operation of a convergent–divergent (C–D) nozzle is based on the principles of compressible fluid flow and conservation laws of mass, momentum, and energy. In a C–D nozzle, fluid initially enters the converging section at subsonic velocity. As the cross-sectional area decreases, the flow accelerates due to the conversion of pressure energy into kinetic energy.

The velocity continues to increase until the flow reaches sonic conditions (Mach number equal to one) at the throat, which is the minimum cross-sectional area of the nozzle. Beyond the throat, the nozzle diverges, allowing the gas to expand further and accelerate to supersonic speeds under appropriate pressure ratios. According to compressible flow theory, the variations in velocity, pressure, and temperature within the nozzle follow the isentropic flow relations, where an increase in velocity corresponds to a decrease in pressure and temperature.

In practical applications, supersonic jets emerging from the nozzle generate acoustic noise due to turbulent mixing and shock wave interactions. To mitigate these effects, modifications such as chevrons are incorporated at the nozzle exit to enhance mixing between the jet flow and the surrounding atmosphere, thereby reducing shock-associated noise and improving acoustic performance

C. *Aim and objectives*

The primary aim of this study is to analyze the aerodynamic and acoustic behavior of a **convergent–divergent (C–D) nozzle** using computational fluid dynamics (CFD) techniques. The research focuses on investigating how variations in pressure influence the flow characteristics inside the nozzle, including velocity, pressure, and temperature distribution. Another important goal of the study is to evaluate the effectiveness of **chevron structures** incorporated at the nozzle exit in reducing acoustic noise generated by high-speed jet flows.

By performing numerical simulations using ANSYS Fluent and modeling the nozzle geometry in CATIA V5, the study aims to identify the optimal operating conditions and design parameters that minimize noise generation while maintaining efficient nozzle performance.

Ultimately, the research seeks to contribute to the development of improved nozzle designs that enhance propulsion efficiency and reduce environmental noise in aerospace and industrial applications.

- To design a convergent–divergent nozzle geometry using CATIA V5 based on standard design parameters such as inlet diameter, throat diameter, and nozzle length.

- To develop a computational model of the nozzle and import the geometry into ANSYS Workbench for performing CFD simulations.
- To generate an appropriate computational mesh for the nozzle model, ensuring sufficient refinement near critical regions such as the throat and nozzle exit to capture accurate flow behavior.
- To define suitable boundary conditions such as inlet pressure, outlet pressure, and wall conditions for simulating compressible flow through the nozzle.
- To analyze the variation of flow parameters including velocity, pressure, and temperature throughout the nozzle using CFD post-processing tools.
- To study the behavior of compressible flow inside the convergent–divergent nozzle and evaluate how the flow accelerates from subsonic to supersonic speeds.
- To investigate the effect of chevron structures at the nozzle exit on the mixing characteristics of the jet flow and their role in reducing acoustic noise.
- To evaluate the acoustic performance of the nozzle by identifying pressure fluctuations and flow instabilities that contribute to jet noise.
- To determine the optimal pressure conditions that result in reduced acoustic power levels while maintaining efficient aerodynamic performance.
- To compare the simulation results with existing literature in order to validate the behavior of the designed nozzle and confirm the effectiveness of noise suppression techniques

II. Literature Review

Olivia Domanski (2023)
Olivia Domanski conducted a study on the optimization of high-speed De Laval nozzles to reduce acoustic energy generated by supersonic exhaust jets.

The research focused on modifying the geometric configuration of the nozzle, particularly the exit region, in order to improve aerodynamic performance while minimizing noise generation. The nozzle model was developed using CATIA V5 and the aerodynamic behavior was analyzed using computational fluid dynamics (CFD) simulations in ANSYS. The study compared conventional nozzle designs with modified exit geometries such as truncated nozzles. Parameters such as velocity distribution, pressure variation, and acoustic energy levels were analyzed.

The results showed that geometric modifications at the nozzle exit can significantly influence the formation of shock structures and turbulent flow patterns, which are major contributors to jet noise. The research concluded that optimized nozzle geometries can reduce acoustic emissions while maintaining efficient flow expansion and aerodynamic performance

N. Shalom, Ron Rabin, and A. J. Abinesh (2023)
N. Shalom, Ron Rabin, and A. J. Abinesh conducted a study on the design and analysis of a convergent–divergent nozzle with the objective of improving its working effectiveness and efficiency.

The researchers investigated the relationship between nozzle geometry and flow characteristics using computational analysis. Important parameters such as Mach number distribution, pressure ratio, and velocity variations were examined to understand how compressible flow behaves inside the nozzle.

The study highlighted that the throat section of the nozzle plays a critical role in determining the transition from subsonic to sonic flow conditions.

It was observed that when the pressure ratio between the inlet and outlet increases, the flow accelerates smoothly through the converging section and reaches sonic velocity at the throat before expanding to supersonic speeds in the diverging section. Although the study primarily focused on improving nozzle performance and reducing operational costs, it provided valuable insights into the fundamental behavior of compressible flow within convergent–divergent nozzles.e the blockchain technology to secure the original component [1].

Hussien W. Mashi (2019)
Hussien W. Mashi investigated the effect of nozzle profile on the performance of convergent–divergent nozzles by analyzing variations in flow parameters such as Mach number, pressure distribution, and shock wave behavior. The study examined how different nozzle contour shapes influence the expansion and acceleration of compressible flow.

The results indicated that the geometry of the nozzle profile plays a significant role in determining the efficiency of the flow expansion process. A well-designed nozzle contour allows smooth acceleration of the gas and minimizes the formation of shock waves and flow separation.

Conversely, poorly designed profiles can lead to unstable shock structures, pressure losses, and reduced nozzle performance. The research also highlighted the formation of shock cell structures in supersonic jets due to pressure mismatches between the nozzle exit and the surrounding atmosphere. The findings emphasize the importance of optimizing nozzle geometry to improve both aerodynamic efficiency and flow stability.

Zhengwu Chen (2024)
Zhengwu Chen conducted an experimental study to investigate noise suppression in supersonic impinging jets using chevron nozzles. The primary objective of the research was to analyze how chevron structures at the nozzle exit influence the acoustic characteristics of high-speed jet flows.

In this study, different nozzle configurations were tested experimentally to evaluate their effect on sound pressure levels and jet flow behavior. The results showed that chevrons significantly reduce the amplitude of acoustic waves and help mitigate discrete tonal noise generated by supersonic jets.

This reduction occurs because chevrons enhance mixing between the high-velocity jet and the surrounding ambient air, which disrupts large turbulent structures responsible for noise generation.

The study also concluded that chevrons can reduce noise without significantly affecting the thrust performance of the nozzle. These findings demonstrate that chevron nozzles are an effective passive noise suppression technique in aerospace propulsion systems.

III. Methodology

The methodology adopted in this study focuses on the design, simulation, and analysis of a convergent–divergent (C–D) nozzle to investigate its aerodynamic and acoustic characteristics.

The overall process involves creating the nozzle geometry, performing computational fluid dynamics (CFD) simulations, and analyzing the resulting flow parameters such as velocity, pressure, and temperature. The simulation process was carried out using **CATIA V5 for modeling and ANSYS Fluent for numerical analysis**.

Geometry Design

The first step in the methodology involved designing the convergent–divergent nozzle geometry using CATIA V5 software. The nozzle consists of three main sections: the converging section, the throat region, and the diverging section. The converging section is designed to gradually reduce the cross-sectional area of the nozzle, which accelerates the fluid flow. The throat is the narrowest section of the nozzle where the flow reaches sonic velocity. Beyond the throat, the diverging section allows the flow to expand and accelerate to supersonic speeds under suitable pressure conditions.

The nozzle also incorporates **chevron structures at the exit**, which are triangular serrations designed to improve mixing between the jet flow and the surrounding air. These chevrons help reduce acoustic noise generated by high-speed exhaust jets.

The geometric specifications of the nozzle are summarized below.

Total nozzle length: 170 mm
Inlet diameter: 60 mm
Convergent section length: 60 mm
Divergent section length: 130 mm
Chevron length: 10 mm
Chevron angle: 60°

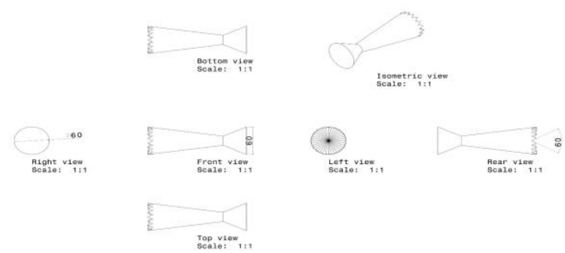

Fig. 3.1(Iso metric view)

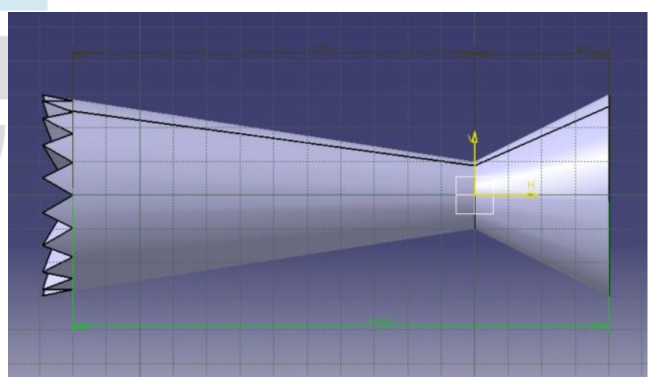

Fig. 3.2 (Side view)

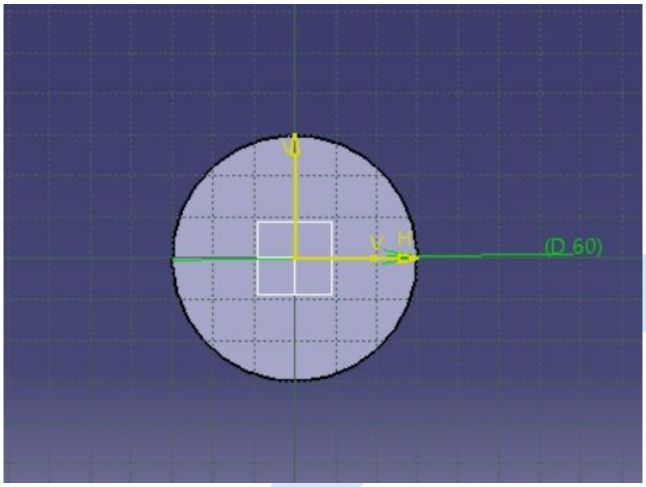

Fig. 3.3 (Front view)

Importing Geometry into ANSYS

After completing the design in CATIA V5, the geometry was exported and imported into ANSYS Workbench for simulation. The geometry was checked for any inconsistencies or errors before proceeding with the simulation process. Proper alignment and scaling were ensured to maintain accurate dimensions during the CFD analysis.

Mesh Generation

Mesh generation is a critical step in CFD analysis because it divides the computational domain into small elements where the governing equations of fluid flow are solved. In this study, a structured mesh was generated for the nozzle geometry. Special attention was given to mesh refinement in regions where high gradients in velocity and pressure were expected, such as the throat region and the nozzle exit.

A finer mesh near the walls and throat region was used to capture boundary layer effects and accurately represent the flow behavior in critical areas. Mesh quality parameters such as skewness and element size were carefully controlled to ensure numerical stability and accurate simulation results.

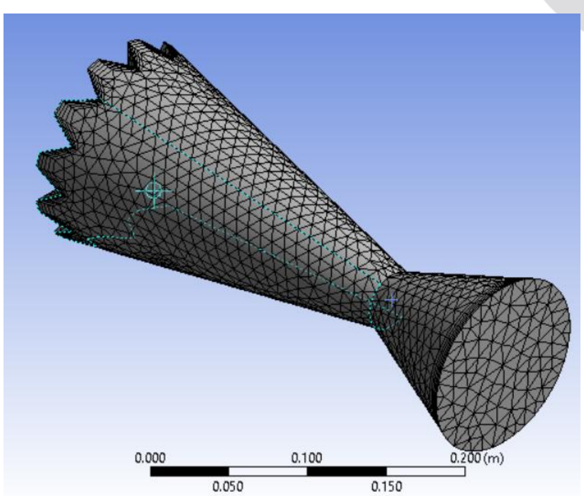

Fig. 3.4 (Mesh)

Boundary Conditions

Appropriate boundary conditions were defined to simulate the compressible flow through the nozzle. The inlet boundary was assigned a high-pressure condition to represent the supply of compressed gas entering the nozzle. The outlet boundary was defined with atmospheric pressure to simulate the expansion of the jet into the surrounding environment.

The nozzle walls were treated as stationary walls with no-slip boundary conditions. This condition assumes that the velocity of the fluid at the wall surface is zero relative to the wall. The flow was considered compressible due to the high velocities involved in supersonic jet flows.

Solver Setup

The CFD simulation was performed using ANSYS Fluent. A pressure-based solver was selected to analyze the compressible flow within the nozzle. The k–epsilon turbulence model was used to account for turbulence effects in the high-speed jet flow.

The simulation parameters included:

•Compressible flow model
•Turbulence modeling using k–ε model
•Energy equation for temperature analysis
• Iterative solution method for solving governing equations

These settings allowed accurate prediction of velocity, pressure, and temperature distributions within the nozzle.

Simulation Process

After defining the mesh and boundary conditions, the simulation was initiated in ANSYS Fluent. The solver iteratively calculated the governing equations for mass, momentum, and energy conservation until the solution reached convergence. During the simulation process, residual values for continuity, momentum, and energy equations were monitored to ensure that the solution was stable and accurate.

The simulation continued until the residual values decreased below the specified convergence criteria, indicating that the solution had stabilized.

IV. RESULTS

The CFD simulation of the convergent–divergent nozzle was carried out using ANSYS Fluent to analyze the behavior of compressible flow through the nozzle. The simulation provided detailed information about the distribution of velocity, pressure, and temperature along the nozzle length. The results were visualized using contour plots and residual graphs, which help in understanding the aerodynamic characteristics and flow behavior inside the nozzle.

Velocity Distribution

The velocity contour obtained from the simulation illustrates the acceleration of fluid through the convergent section of the nozzle. At the inlet region, the fluid enters with relatively low velocity. As the flow moves toward the converging section, the reduction in cross-sectional area causes the velocity to increase gradually. This increase in velocity occurs because the pressure energy of the fluid is converted into kinetic energy.

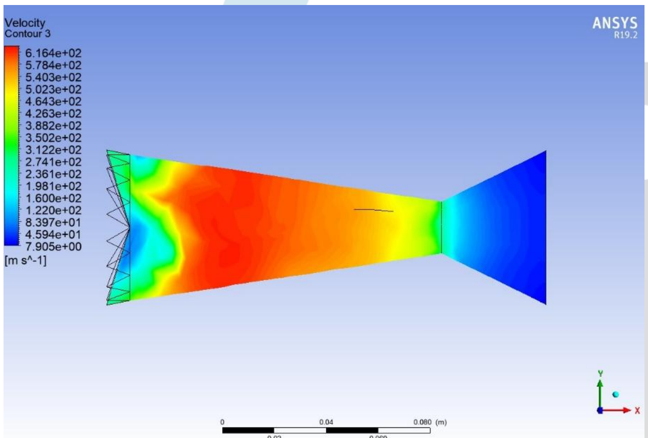

Fig 4.1 (Velocity Distribution)

The maximum velocity occurs at the throat region, which is the narrowest section of the nozzle. In the simulation results, the velocity near the throat reaches approximately **600 m/s**, indicating that the flow approaches sonic conditions. Beyond the throat, the flow enters the diverging section where the fluid expands and continues moving at high velocity.

In the diverging region, the velocity distribution becomes more uniform as the jet expands outward. The presence of chevron structures at the nozzle exit helps enhance mixing between the jet and the surrounding air. This improved mixing reduces the intensity of turbulent structures that contribute to acoustic noise generation.

Pressure Distribution

The pressure contour obtained from the CFD simulation shows a clear variation in pressure along the nozzle length. At the inlet section, the pressure is relatively high because the fluid enters the nozzle under compressed conditions. As the flow moves through the converging section, the pressure gradually decreases due to the increase in fluid velocity.

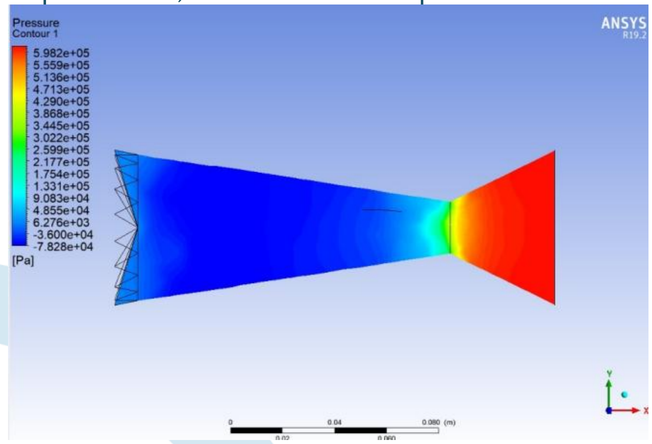

Fig 4.2 (Pressure Distribution)

The lowest pressure is observed at the throat region where the velocity is highest. This behaviour is consistent with the principles of compressible flow, where an increase in velocity results in a decrease in pressure. After passing through the throat, the flow enters the diverging section where the pressure begins to increase again as the fluid expands and decelerates.

The pressure recovery in the diverging section indicates that the nozzle is effectively converting pressure energy into kinetic energy and allowing controlled expansion of the fluid. The smooth pressure variation observed in the simulation also indicates stable flow conditions without significant flow separation.

Temperature Distribution

The temperature contour obtained from the simulation shows the variation of temperature along the nozzle length. At the inlet region, the temperature is relatively high due to the compressed state of the fluid. As the flow accelerates toward the throat, the temperature gradually decreases.

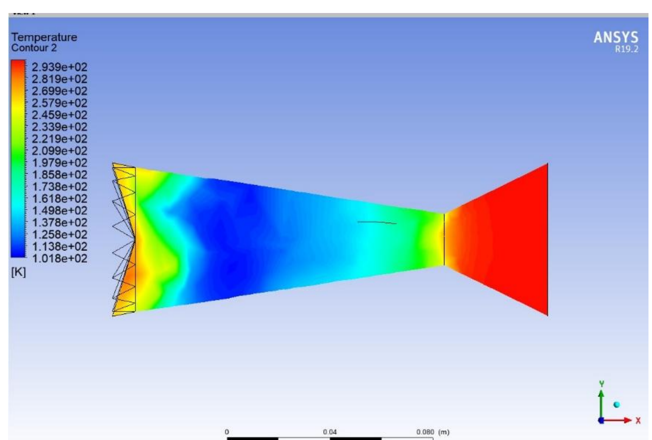

Fig 4.3 (Temperature Distribution)

This decrease in temperature occurs because part of the internal energy of the fluid is converted into kinetic energy as the velocity increases. The lowest temperature is observed at the throat region where the velocity reaches its maximum value.

Beyond the throat, as the flow expands in the diverging section, the temperature begins to increase slightly. This increase occurs due to the deceleration of the fluid and partial recovery of thermal energy. The temperature variation observed in the simulation is consistent with theoretical predictions for compressible flow in convergent–divergent nozzles

V. CONCLUSION

The present study focused on the aerodynamic and acoustic analysis of a **convergent–divergent (C–D) nozzle** using computational fluid dynamics (CFD) simulations. The nozzle geometry was designed using CATIA V5 and analyzed in ANSYS Fluent to investigate the behavior of compressible flow through the nozzle. The simulation results provided detailed insights into the variations of velocity, pressure, and temperature within the nozzle and their influence on overall flow performance.

From the CFD analysis, it was observed that the velocity of the fluid increases significantly as it moves through the converging section of the nozzle due to the reduction in cross-sectional area. The flow reaches its maximum velocity at the throat region, where the Mach number approaches sonic conditions. Beyond the throat, the flow expands in the diverging section, allowing the gas to accelerate further and maintain high velocity while undergoing pressure and temperature changes. The pressure distribution showed a gradual decrease from the inlet toward the throat, followed by partial pressure recovery in the diverging section. Similarly, the temperature distribution indicated a decrease in temperature as the flow accelerated and a slight increase during expansion in the diverging region. These results are consistent with theoretical predictions of compressible flow in convergent–divergent nozzles.

The study also examined the influence of **chevron structures** incorporated at the nozzle exit. The presence of chevrons improved the mixing of the high-velocity jet with the surrounding atmosphere. This enhanced mixing reduced the size and intensity of turbulent structures that are responsible for generating acoustic noise in supersonic jets. As a result, the nozzle design demonstrated improved acoustic performance while maintaining effective aerodynamic characteristics.

The convergence of residual values during the simulation confirmed that the numerical solution was stable and reliable. The contour plots obtained from the analysis clearly illustrated the flow acceleration, pressure variation, and temperature changes along the nozzle length, providing a comprehensive understanding of the internal flow behavior.

Overall, the research demonstrates that **CFD simulation is an effective tool for analyzing and optimizing nozzle performance**. Proper design of the convergent–divergent geometry, combined with the use of chevron structures, can significantly reduce acoustic emissions without compromising propulsion efficiency. The findings of this study contribute to the development of improved nozzle designs for applications in aerospace propulsion systems, gas turbines, and other high-speed jet systems.

Future work may involve experimental validation of the CFD results, optimization of chevron geometry, and investigation of advanced turbulence models to further enhance the accuracy of acoustic predictions and overall nozzle performance

VI. REFERENCE

[1] Olivia, Domanski. (2023). Optimization of the High-Speed De-Laval Nozzle to Reduce the Acoustic Energy by Using the Truncated Nozzle. Springer proceedings in energy, 807-818. doi: 10.1007/978-3-031-30171-1_84Ahmad, R.W., Hasan, H., Yaqoob, I., Salah, K., Jayaraman, R. and Omar, M., 2021. Blockchain for aerospace and defense: Opportunities and open research challenges. *Computers & Industrial Engineering*, 151, p.106982.

[2] N., Shalom., Ron, Rabin., A., J., Abinesh., A, Keo, Sam., Amir, Roshan. (2023). Convergent Divergent Nozzle Design and Analysis to Increase Working Effectiveness. International Journal For Science Technology and Engineering, 11(8), 1303-1309. doi: 10.22214/ijraset.2023.55337

[3] Arkan, Al-Taie., Hussien, W, Mashi., Ali, M, Hadi. (2019). The effect of convergent-divergent nozzle profile on its performance. 19(1), 14-43. doi: 10.32852/IQJFMME.V19I1.262.

[4] Chen, Bao., Weipeng, Li., Fei, Wu., Zhengwu, Chen. (2024). Experimental study of the noise suppression of supersonic impinging jets with chevrons. Applied Acoustics, 221, 109982-109982. doi: 10.1016/j.apacoust.2024.109982.