



e-ISSN: 2319-8753 | p-ISSN: 2347-6710

# IJRSET

International Journal of Innovative Research in  
**SCIENCE | ENGINEERING | TECHNOLOGY**

# INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

Volume 13, Issue 7, July 2024

**ISSN** INTERNATIONAL  
STANDARD  
SERIAL  
NUMBER  
INDIA

Impact Factor: 8.423

 9940 572 462

 6381 907 438

 [ijrset@gmail.com](mailto:ijrset@gmail.com)

 [www.ijrset.com](http://www.ijrset.com)

# CFD Analysis of Compressible Flows in A Convergent-Divergent Nozzle

SATHISH.V, MAKESH.M

ME-Engineering Design, Paavai Engineering College, Namakkal, Tamilnadu, India

Professor of Mechanical Department, Paavai Engineering College, Namakkal, Tamilnadu, India

**ABSTRACT:** A nozzle is a mechanical device that utilizes pressure energy and fluid enthalpy to increase the outflow velocity and control fluid flow direction. To obtain the nozzle duct's shock pattern, the flow inside the nozzle must be supersonic with a Mach number greater than one. Experimentally, the shock pattern is obtained for a nozzle with a Mach number of 2 and a nozzle pressure ratio (NPR) equivalent to 7 and below. For Mach  $M = 2$ , the necessary NPR is equal to 7.82 for correct expansion. When the NPR is greater than 7.82, flow from the nozzles is under-expanded. For NPR less than 7.72, the flow from the nozzle is over-expanded. In this paper, the computational fluid mechanics (CFD) technique was used to simulate the nozzle flow based on one experimental investigation. A two-dimensional transient compressible flow of air through a supersonic nozzle was simulated using ANSYS Fluent software. A time-dependent flow using the density-based implicit solver was employed to analyze the simulation results. The results illustrate that the CFD technique simulates fluid flows and the formation of shock in a duct, providing useful information about fluid dynamics analysis.

## I. INTRODUCTION

Convergent-divergent nozzles are indispensable in numerous engineering applications, particularly in propulsion systems and fluid dynamics control, where they leverage pressure energy and fluid enthalpy to amplify outflow velocity while adeptly managing fluid flow direction. Key to their function is the establishment of a supersonic flow regime within the nozzle, characterized by a Mach number surpassing unity. Attaining desired performance hinges on precise control of parameters like Mach number and nozzle pressure ratio (NPR), with critical thresholds identified through experimental investigations.

Notably, for a Mach number of 2, an NPR of 7.82 ensures optimal expansion, deviations from which result in under-expansion or over-expansion of the flow. In this context, computational fluid dynamics (CFD) emerges as a potent tool, enabling comprehensive simulation and analysis of complex flow phenomena within these nozzles. This paper undertakes a thorough CFD analysis, building upon experimental insights, to elucidate the intricate flow dynamics within convergent-divergent nozzles, with a particular emphasis on the formation and characteristics of shock waves.

Leveraging the capabilities of ANSYS Fluent software, a two-dimensional transient simulation of compressible airflow through a supersonic nozzle is conducted, employing a time-dependent approach utilizing the density-based implicit solver to capture dynamic flow behavior. The results obtained from these CFD simulations are anticipated to furnish valuable insights into nozzle performance, thereby facilitating informed design and optimization strategies across a spectrum of engineering applications.

## II. OBJECTIVE

- Understand the Concept of C-D nozzle.
- About De Laval nozzle.
- Creating the axisymmetric nozzle geometry.
- Study use of face split.
- Creating structural mesh.
- Study the effect of Mach number.
- Solver setup (density based),
- Solve using it Study and analysis Temperature, pressure, Mach number, velocity variation etc., At different divergent angle and inlet pressure.
- CFD processing of setup and solution.

## III. METHODOLOGY

This research utilizes Computational Fluid Dynamics (CFD) to analyze transient compressible flow through a two-dimensional convergent-divergent (C-D) nozzle, focusing on optimizing variables like Mach number, nozzle pressure ratio (NPR), and employing a density-based implicit solver. Using ANSYS 18.1 for modeling and simulations, preliminary results were obtained to guide further investigation and achieve the study's objectives.

### 3.1 C-D Nozzle:

A converging-diverging (CD) nozzle is designed to produce supersonic flow at its exit by ensuring proper rear pressure conditions. Within the nozzle, the flow is isotropic, becoming supersonic as it exits. Combustion chamber exhaust gases pass through the nozzle, converging at the throat—the narrowest part—where flow choking occurs, determining the system's mass flow rate. Unlike convergent nozzles, which can't achieve supersonic speeds (Mach numbers above one), CD nozzles are essential for reaching these high velocities.

### 3.2 Nozzle back pressure:

A converging-diverging (CD) nozzle is designed to produce supersonic flow at the exit, necessitating specific back pressure conditions. Inside the nozzle, the flow is isotropic and transitions to supersonic as it exits. Exhaust gases from the combustion chamber pass through the nozzle, converging at the throat—the narrowest point—where the flow chokes and sets the mass flow rate. Convergent nozzles alone cannot achieve supersonic speeds (Mach numbers above one), making CD nozzles essential for attaining these high-velocity conditions.

**Bernoulli's principle** states that in a streamline flow, as the speed of a fluid increases, its pressure decreases, and vice versa, with the total energy per unit volume remaining constant.

Mathematically, for an incompressible, frictionless fluid, the principle is expressed as

$$P + \frac{1}{2}\rho v^2 + \rho gh = \text{constant}$$

### 3.3 Converging section of nozzle:

As the nozzle narrows from the entrance to the throat region, the flow velocity increases, reaching its maximum at the throat. In a convergent nozzle, as the cross-sectional area decreases, the fluid velocity rises and the pressure drops, illustrating the inverse relationship between pressure and area. To ensure a constant flow rate through the nozzle's narrowest section, the fluid must accelerate. This effect, where velocity increases and static pressure decreases in a constricted area, is known as the Venturi effect. Convergent nozzles are used to accelerate subsonic fluids, and with a sufficiently high nozzle pressure ratio, the flow can reach sonic speeds at the throat. Conversely, divergent nozzles slow down subsonic fluids but can accelerate sonic or supersonic flows.

### 3.4 Compressible fluid flow:

Under standard temperature and pressure conditions, the volume or density of a fluid remains constant. Gases, however, exhibit different behavior, changing their volume and density with even small variations in temperature or pressure. When pressure is applied to a compressible fluid, its density can change significantly. In contrast, an "incompressible fluid" is a substance that maintains a constant volume and does not compress under external pressure.

### 3.5 C-D nozzle application:

Steam turbine and rocket nozzles both use the converging-diverging (C-D) nozzle design. Reaction engine nozzles serve as propelling nozzles in certain steam turbines and jet engines, where they constrict the flow to form an exhaust jet, maximizing the exit velocity of the gases. These nozzles are utilized in various fields such as automobiles, jets, power plants, and rockets to generate the thrust needed for propulsion.

### 3.6 De Laval nozzle:

The De Laval nozzle, also known as the converging-diverging (C-D) nozzle, was pioneered by Swedish engineer Gustaf de Laval in the 19th century. It's a crucial component in various propulsion systems, including steam turbines, rocket engines, and supersonic jet engines. The nozzle operates based on gas behavior at subsonic and supersonic speeds. In subsonic flow, gas velocity increases as the nozzle narrows, reaching sonic velocity (Mach 1.0) at the throat due to choked flow. As the gas transitions to the expanding section, it accelerates to supersonic speeds (Mach > 1.0). The nozzle's ability to smoothly transition between subsonic and supersonic flow is essential for efficient propulsion and has been extensively studied in gas dynamics.

### 3.7 Ideas and assumptions of the appliance for the analysis of gas flow through de Laval nozzles necessitates are:

- - Combustion gas is often idealized as a perfect gas for simplification purposes.
- - Gas flow is assumed to be isentropic, meaning entropy remains constant, frictionless, and adiabatic, with minimal or no heat exchange.
- - The flow progresses linearly from the gas inlet to the exhaust gas exit along the nozzle's axis of symmetry.
- - Due to the high velocity of the flow, the gas behavior is compressible, meaning changes in pressure significantly affect its density and volume
- 

### 3.8 Material used for nozzle manufacturing are: -

Materials used include brass, stainless steel, cast iron, copper, tungsten carbide, nickel, silicon carbide, super alloys, titanium, nickel alloys, and tantalum

## IV. COMPUTATIONAL FLUID DYNAMICS SIMULATIONS

In the realm of fluid mechanics, turbulence presents a formidable challenge for engineers and scientists due to its intricate and unpredictable nature. Computational Fluid Dynamics (CFD) emerges as a potent tool utilized to simulate turbulent airflow around various structures, including vehicles. However, the precision of CFD simulations is constrained by our comprehension of turbulent flows. To tackle these hurdles, the High-Fidelity Turbulence (HiFi TURB) initiative, spearheaded by NUMECA International, endeavors to pioneer innovative solutions for refining



turbulent flow modeling.

Harnessing the capabilities of High-Performance Computing (HPC) and integrating Artificial Intelligence (AI) and Machine Learning (ML) methodologies, this project aims to propel turbulence model advancement forward. Central to the endeavor is the construction of an extensive database comprising high-fidelity, scale-resolving simulations that capture the intricate features of turbulent flows.

Neural Concept, a collaborative partner, contributes expertise in Deep Learning to devise a toolchain for analyzing the substantial volume of data generated by these simulations. Employing Geometry-based Variational Auto-Encoders (VAE), NUMECA gains valuable insights into the interrelations among statistically averaged variables in turbulent flows.

This innovative technique condenses data into meaningful representations, facilitating efficient data management and analysis. Through the convergence of AI, ML, and high-fidelity simulations within the HiFi-TURB project, the prospect of more precise predictions in aeronautical applications emerges. By deepening our comprehension of turbulence, this integrated approach promises to enhance the efficiency and safety of aircraft design and operation.

#### 4.1 Understanding Fluid Dynamics:

Fluid dynamics, a branch of physics, delves into the intricate dynamics and characteristics of fluids, encompassing both gases and liquids. From the aerodynamics of aircraft wings to the dynamics of blood circulation, this field scrutinizes a broad spectrum of phenomena. It unravels the underlying principles governing fluid behavior, including velocity, pressure, and the complex phenomenon of turbulence. Essentially, fluid dynamics is the exploration of how fluids navigate through space, impacting a diverse range of natural and engineered systems.

#### 4.2 The Role of CFD in Engineering:

Computational Fluid Dynamics (CFD) has become indispensable in numerous engineering and scientific fields. Its utilization enables engineers to fine-tune designs, assess performance, and predict fluid system behaviors. Particularly within aerospace, automotive, energy, and environmental sectors, CFD plays a critical role in improving efficiency, strengthening safety measures, and advocating for sustainable practices. Essentially, CFD functions as a flexible instrument for streamlining operations and driving innovation across diverse industries.

#### 4.3 CFD Fundamentals:

CFD breaks down fluid flow equations, like the Navier-Stokes equations, into solvable components on a grid. By iteratively solving these equations, it approximates fluid flow momentum. Incorporating turbulence modeling, CFD accurately simulates turbulent flow and Turbulent Kinetic Energy (TKE) variations, offering insights into fluid behavior in diverse scenarios.

#### 4.4 CFD Workflow and Process:

The process and workflow of CFD involves several meticulous stages, each critical for precise simulation outcomes. It starts with defining the problem and importing system geometry. Next, a computational grid is generated for accuracy, and boundary conditions are chosen to mimic real-world constraints. The solver is then configured to solve governing equations numerically, followed by post-processing to analyze results. Successful execution of each step is essential for reliable and insightful CFD simulations.

#### 4.5 Applications of CFD:

CFD serves as a versatile asset utilized across diverse industries for multifaceted purposes. In aerospace, it refines aircraft designs, evaluates aerodynamic performance, and examines airflow impacts on structural integrity. Automotive engineering benefits from CFD by optimizing vehicle aerodynamics, reducing drag, and improving fuel efficiency. Moreover, CFD plays a critical role in environmental modeling, HVAC system design, biomedical engineering, and other fields where fluid flow dynamics are crucial. Overall, CFD acts as a foundational tool for

enhancing efficiency, performance, and comprehension across various disciplines.

#### 4.6 Advancements and Future Perspectives:

CFD continues to evolve, propelled by advancements in turbulence models, numerical algorithms, and computational power. High-performance computing and cloud-based simulations enhance the speed and detail of analyzing complex flow phenomena. Integration with optimization algorithms and machine learning further refines design processes and decision-making. This synergy between advanced computational methods and emerging technologies opens new frontiers for CFD, enabling more efficient problem-solving and innovation across industries.

#### 4.7 Summary of CFD Simulation:

CFD, a branch of fluid dynamics, focuses on simulating and assessing fluid flow in engineering scenarios. It blends principles from fluid mechanics, mathematics, and computer science to solve the equations governing fluid motion. This method enables detailed examination of complex fluid flow phenomena that may be difficult to study experimentally. Essentially, CFD offers a robust framework for comprehensively analyzing fluid behavior in various engineering applications, providing insights for the experimental methods alone may be challenging to attain.

#### 4.8 GOVERNING EQUATION IN COMPUTATIONAL FLUID DYNAMICS:

The examination of thermo-fluids revolves around fundamental governing equations rooted in the conservation laws of fluid properties. These equations are anchored in three essential principles:

- Conservation of Mass, encapsulated in the Continuity Equation,
- Conservation of Momentum, expressed through Newton's Second Law,
- Conservation of Energy, embodied by the First Law of Thermodynamics or the Energy Equation.

These laws assert that within a closed system, mass, momentum, and energy remain constant. In essence, they underscore the necessity for conservation in all processes.

The investigation of fluid flow with thermal changes relies on certain physical properties. The three unknowns which must be obtained simultaneously from these three basic conservation equations are the velocity  $v \rightarrow$ , pressure  $p$  and temperature  $T$ . Yet  $p$  and  $T$  are considered the two required independent thermodynamic variables.

The final of the conservation equations also contains four other thermodynamic variables; density  $\rho$ , enthalpy  $h$ , viscosity  $\mu$ , and thermal conductivity  $k$ ; the last two of which are also transport properties. These four properties are uniquely determined by the value of  $p$  and  $T$ .

Fluid flow should be analyzed to know  $v$ ,  $e$ ,  $c$ ,  $v$ ,  $p$  and  $T$  throughout every point of the flow regime. Furthermore, the method of fluid flow observation based on kinematic properties is a fundamental issue. The movement of fluid can be investigated with Lagrangian or Eulerian methods.

Lagrangian description of fluid motion is based on the theory of following a fluid particle that is large enough to detect properties. Initial coordinates at time  $t_0$  and coordinates of the same particle at time  $t_1$  must be examined. To follow millions of separate particles through the path is almost impossible.

In the Eulerian method, instead of following any specific particle across the path, the velocity field is examined as a function of space and time.

This missile example precisely explains the two methods.

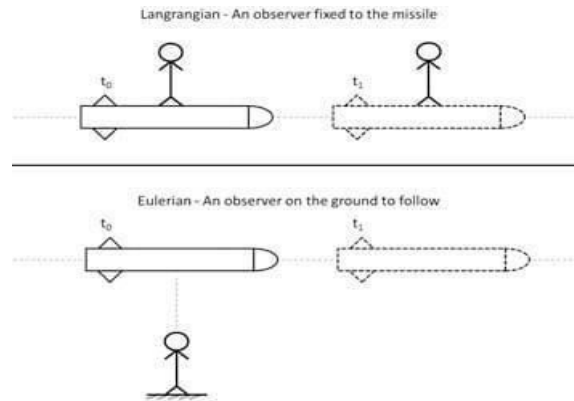


Figure.4.1 Lagrangian and Eulerian methods

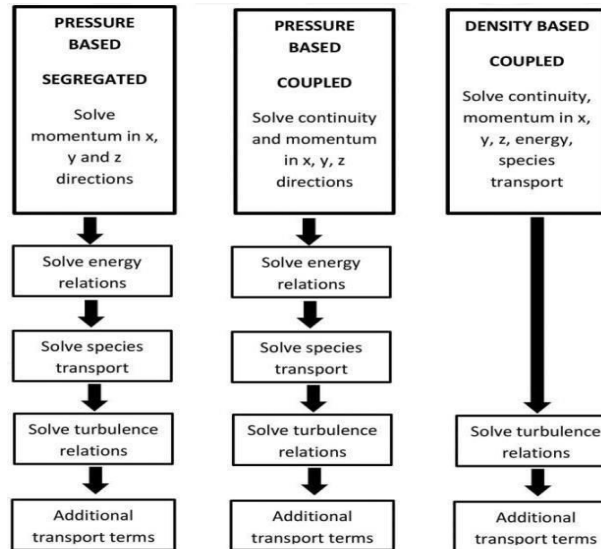
#### 4.9 THE ANSYS FLUENT SOFTWARE:

ANSYS FLUENT stands out as a premier tool for discretely solving the governing equations in fluid dynamics. Renowned for its versatility, it excels in reproducing the behavior and providing insights across a broad spectrum of fluid flows, whether compressible or incompressible, laminar or turbulent, and steady or unsteady. Its extensive range of mathematical models caters to diverse transport phenomena, including heat transfer and chemical reactions, applicable to both simple and intricate geometries. FLUENT finds applications in various domains, encompassing devices like heat exchangers, nozzles, complex-shaped ducts, turbo machinery, and components of heat engines. It is equally adept at analyzing external flows, flow through compressors, pumps, fans, and multiphase flows, among other scenarios.

Table 4.1 Equivalencies associated with the variable.

Equation	Variable ( $\phi$ )
Continuity	1
X momentum	U
Y momentum	V
Z momentum	W
Transport of turbulent kinetic energy	K
Dissipation of turbulent kinetic energy	$\epsilon'$

**4.9.1 ANSYS FLUENT SOLVER PROCESS:**



**Figure.4.2 ANSYS fluent solver performances**

FLUENT solvers offer two distinct methods for solving discretized equations: segregated and coupled. The segregated mode is best for flow problems with pronounced velocity gradients, like those in boundary layer formations, and minimal compressibility effects ( $Mach < 0.3$ ), making it suitable for vehicular aerodynamics and internal flows. Conversely, the coupled solver is ideal for scenarios with significant compressibility effects, such as supersonic flows and combustion chambers, where shock waves from substantial pressure and density changes are common.

In numerical terms, the segregated solver mode addresses the flow governing equations by solving velocity and pressure equations sequentially, step by step. In contrast, the coupled solver mode simultaneously tackles continuity, momentum, and species exchange equations on a global scale.

**4.10 MAIN PARAMETERS AND BOUNDARY CONDITIONS:**

ANSYS FLUENT, user-friendly yet requiring parameter adjustments for optimal usage, allows users to specify dimensionality (2D or 3D) before importing meshes. Opting for 2D analysis is suitable for studying uniform cross-sections in infinitely long bodies with minimal depth gradients. Additionally, it offers double precision, beneficial for scenarios with significant flow variable gradients or when enhanced convergence is required.

ANSYS FLUENT allows users to adjust the number of core processors for internal parallel processing, particularly useful for handling non-stationary models, multiphase and reactive problems, large grid sizes, and complex geometries like aircraft and vehicles. Furthermore, it enables scaling grid dimensions using both English and international systems, accommodating scenarios ranging from flows between distant building zones to microfluidic problems.

It also facilitates the selection of constant values to compute non-dimensional parameters such as friction coefficient and lift and drag coefficients. Accurate representation of boundary conditions is crucial in any computational fluid dynamics (CFD) model, ensuring fidelity to real-world conditions tailored to the mathematical properties of the differential equations.

**4.10.1 Dirichlet boundary condition:**

Flow variable values are prescribed at the boundary, commonly associated with inlet flow and isothermal walls.



#### 4.10.2 Neumann boundary condition:

Gradient-normal flow variable values are specified for this boundary condition, often found at symmetric and adiabatic boundaries.

#### 4.10.3 Mixed boundary conditions:

Boundary conditions are categorized into physical (e.g., solid walls) and artificial (e.g., outflows), with the latter serving as mathematical approximations to real flow behavior, focusing on relevant regions and reducing computational costs while maintaining accuracy.

ANSYS FLUENT provides a range of boundary condition types for simulating diverse flow problems, offering flexibility and accuracy in hydrodynamic studies.

#### 4.11 MACH NUMBER:

The Mach number in fluid dynamics measures the ratio of flow velocity to the local speed of sound, offering insights into compressibility characteristics.

$$M = u/c \text{ where}$$

M is Mach number,

u is the local flow velocity with respect to the boundaries,

c is the speed of sound in the medium

#### 4.12 SIMULATION USING CFD FLUENT:

It is an engineering tool that helps us to conduct Experimentation, to perform the geometry or simulation of C-D nozzle and to study the parameter (density based) such as velocity, temperature, pressure, Mach number.

For the simulation we must consider: -

The Air is flowing.

- A) It is a pressure inlet.
- B) At the outlet pressure is lower than at the inlet.
- C) At the throat of C-D nozzle velocity and pressure decreased.

### V.MODELING

The C-D nozzle was modeled in SolidWorks with dimensions: total length 190 mm, inlet diameter 96 mm, throat diameter 40 mm, outlet diameter 136 mm, convergent angle 39°, divergent angles 19° and 29°, using stainless steel.

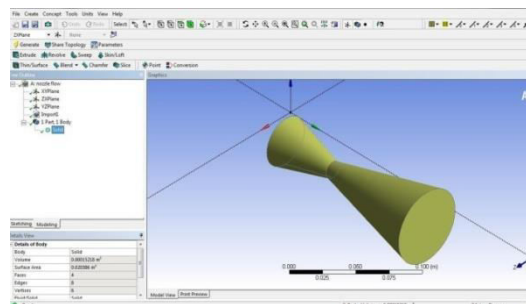


Figure.5.1 ANSYS Modelling

## 5.1 GEOMETRY:

In ANSYS Workbench, double-clicking the geometry cell opens the ANSYS Design Modeler. We create an axisymmetric part of the nozzle size with hard flow behavior and set the size function to curvature for the curved section at 19°. The figure shows the C-D nozzle geometry with a 29° divergent angle.

### 5.1.1 DRAFTING:

In ANSYS, the C-D nozzle was drafted with an axisymmetric design, applying hard flow behavior and curvature size function for the 19° section, resulting in a final geometry with a 29° divergent angle.

## 5.2 MESHING

The geometric body was represented as finite elements in ANSYS CFD. After modeling the nozzle, a mesh was generated with face meshing applied for structural mapping at a 19° divergent angle. Figure 3c shows the 2D geometry with a 29° divergent angle.

## 5.3 BOUNDARY CONDITIONS (BC):

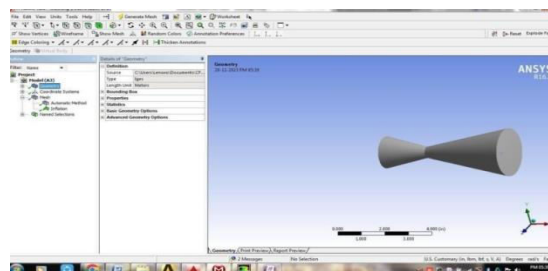
Boundary conditions were set in ANSYS with inlet pressure at 101,235 Pa and temperature at 300 K for the first test, and for the third test, pressure at 102,325 Pa, velocity at 62.8 m/s, and temperature at 300 K.

## 5.4 SOLVER SETUP, CFD POST PROCESSING:

The density-based solver was used for accurate compressible flow analysis, with 2D steady-state conditions and ideal gas density. Air was selected as the material, with output parameters set to static pressure, total pressure, Mach number, and axial velocity. Reference value was set to the inlet, and the number of iterations was 1400.

## VI. DRAFTING AND CALCULATING

### 6.1 NOZZLE SOLID MODEL:



Nozzle diameter =10mm, convergent diameter=30mm, Divergent diameter=60mm

**Figure.6.1 Nozzle solid model view**

6.2 NOZZLE MESH:

Element size = 0.025mm

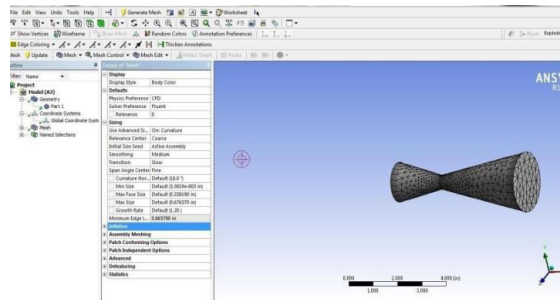


Figure.6.2 Mesh control view

6.3 CALCULATING RESIDUAL:

Residual time: 0.25s, Residual count: 500 times



Figure.6.3 Residual calculation view

6.4 RESIDUAL STREAM I:

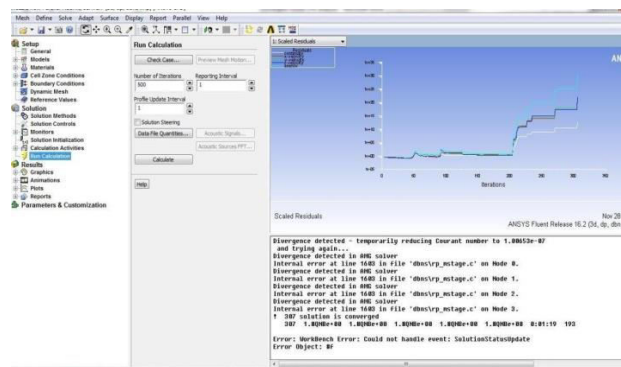


Figure.6.4 Residual stream view

## VII. SOLUTION AND RESULT

### 7.1 PRESSURE FLOW

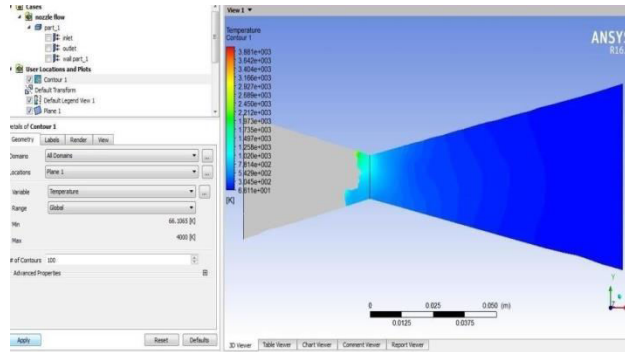


Figure.7.1 Pressure flow view

### 7.2 TEMPERATURE FLOW

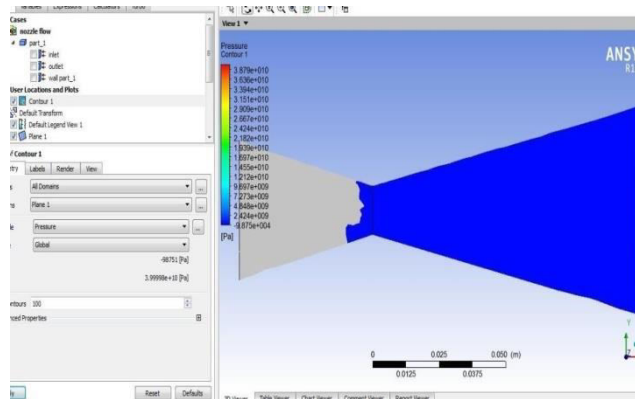


Figure.7.2 Temperature flow view

### 7.3 VELOCITY FLOW

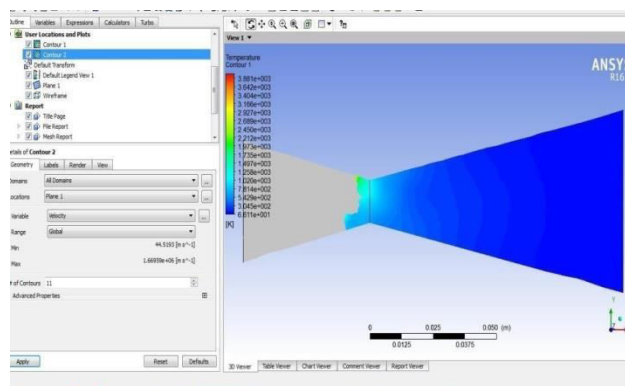


Figure.7.3 Velocity flow view



### 7.4 VELOCITY STEAMLINE FLOW

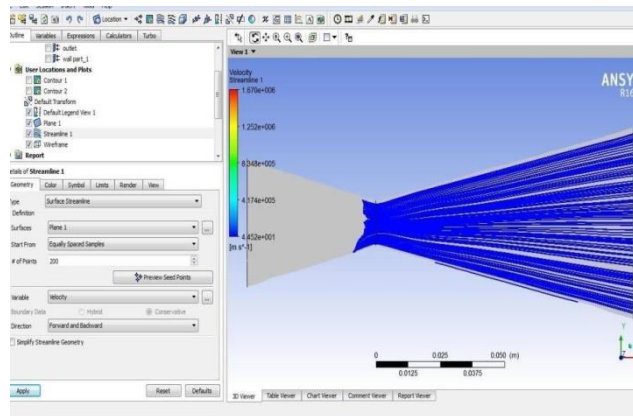


Figure.7.4 Velocity streamline view

### 7.5 CHARTING PRESSURE

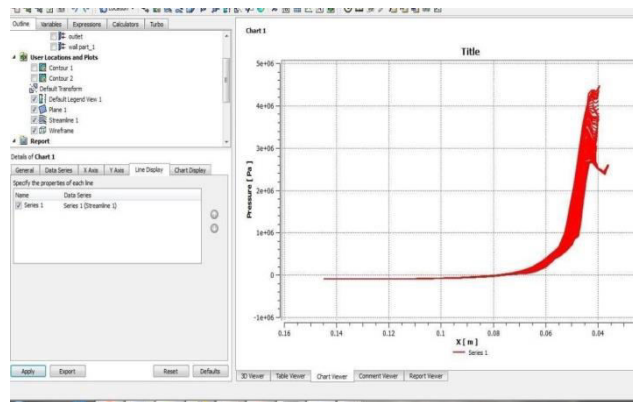


Figure.7.5 Charting pressure view

### 7.6 CHARTING VELOCITY

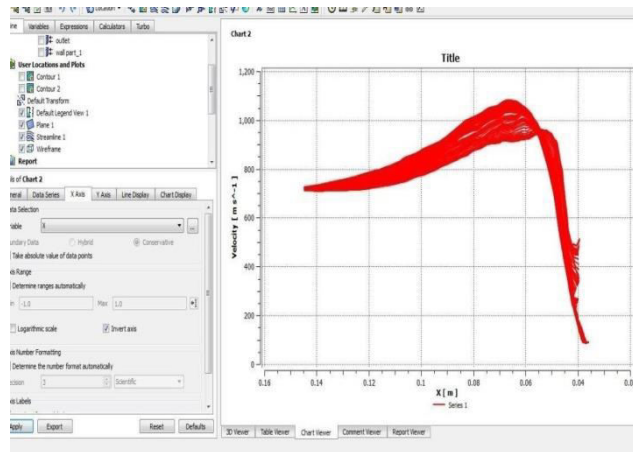


Figure.7.6 Charting velocity view

## VIII. RESULT

### 7.7 NOZZLEFLOW:

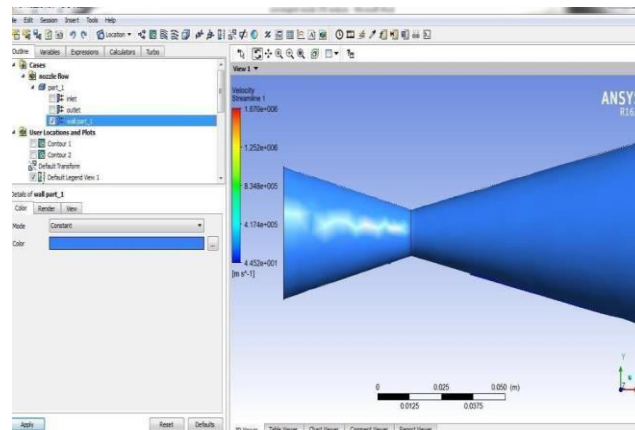


Figure.7.7 Nozzle flow view

## 7.8 LOCATION AND PLOTS:

### 7.8.1 CONTOUR I:

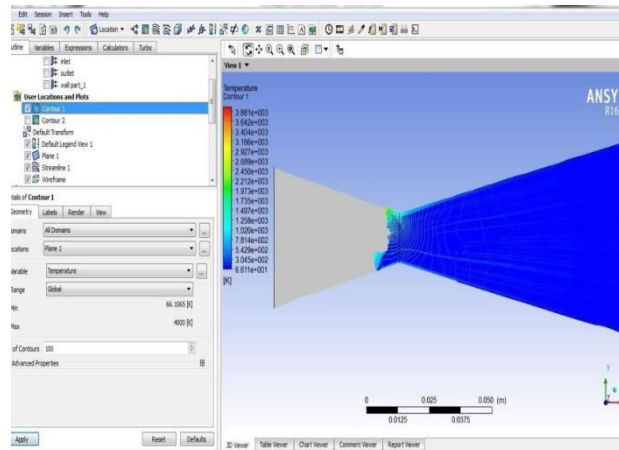


Figure.7.8 Nozzle contour plot I view

### 7.8.2 CONTOUR II:

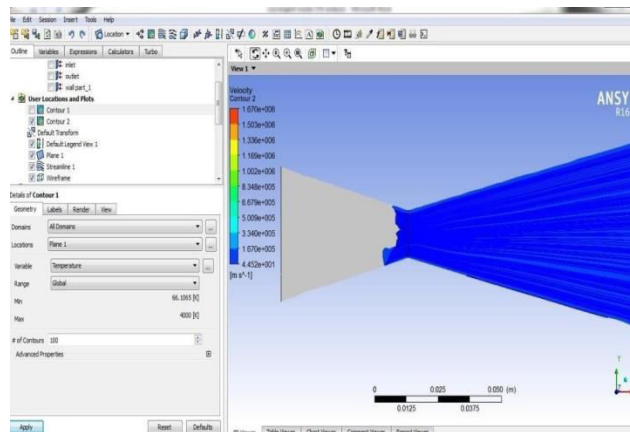


Figure.7.9 Nozzle contour plot II view

## VIII. CONCLUSION

In this project, a convergent-divergent (C-D) nozzle with the following specifications was analyzed:

- Total length: 190 mm
- Inlet diameter: 96 mm
- Throat diameter: 40 mm
- Outlet diameter: 136 mm
- Convergent angle: 39°
- Divergent angles: 19° and 29°

The analysis was conducted using SolidWorks for design and ANSYS Fluent for flow field analysis. Key conditions and parameters for the simulation were:

- 2D steady-state conditions
- Ideal gas density
- Compressible flow
- Inlet pressure: 102,325 Pa
- Inlet velocity: 62.8 m/s
- Inlet temperature: 300 K
- Number of iterations: 1400

The study aimed to produce supersonic flow near the exit plane and increase the outlet velocity. The simulation outputs included various flow properties such as pressure, temperature, velocity, and density, demonstrating the nozzle's capability to control fluid characteristics and achieve the desired flow acceleration.

#### REFERENCES

1. International Journal of Engineering Technology Science and Research IJETSRS [www.ijetsrs.com](http://www.ijetsrs.com) ISSN2394–3386 Volume5, Issue 4April 2018. Ali, A., Neely, A., Young, J. Blake, B., & Lim, J.Y. (2010). Numerical Simulation of Fluidic Modulation of Nozzle Thrust. In 17<sup>th</sup> Australasian Fluid Mechanics Conference (pp.5–8).
2. G. Satyanarayana, Ch. Varun, S.S Naidu, “CFD analysis of convergent-divergent nozzle” in an International Journal of Scientific and Research.
3. Gustaf De-Laval, “Concepts and CFD analysis of De-Laval nozzle” in International Journal of Mechanical Engineering and Technology (IJMET)Volume7, Issue5, September1991.
4. Trong Bui, Bogdan-Alexandru Belega, “CFD analysis of flow in Convergent-divergent nozzle”, international conference of scientific paper AFASES2015 Brasov,28-30May2015.
5. Sutton book, Sutton, George P, Rocket Propulsion Elements, An Introduction to the Engineering of Rockets 6th Edition pages636 publisher, Wiley-Inter science, year1992, id0471529389.
6. Internationalconferenceofscientificpaperafases2015brasov,28-30.
7. A. Khan, Shembhakar,2008,"ViscousFlow International Conference on Aerospace Science and Technology, Bangalore.
8. R. Boyanapalli et al., 2013, "Analysis of Composite De-Laval Nozzle Suitable for rocket applications", International Journal of Innovative Technology and Exploring Engineering, 2, pp.336-344.
9. V. Lijo, H.D. Kim, T. Setoguchi, S. Matsuo, 2010, "Numerical simulation of transient flows in a rocket propulsion nozzle", International Journal of Heat and Fluid Flow,31, pp.409-417.
10. Q. Xiao, H.M. Tsai, D. Papa moschou, 2007, "Numerical investigation of supersonic flow separation", AIA Journal, 45, pp. 532-541.
11. In Convergent -Divergent Rocket Engine Nozzle Using Computational Fluid Dynamics, AFASES,2015.
12. International Journal of Innovative Technology and Exploring Engineering (IJITEE) ISSN:2278-3075, Volume-9 Issue-6, April 2020.
13. Sudhakar BVVN, Sekhar BPC, Mohan PN, Md Ahmad T (2016) Modeling simulation of convergent-divergent nozzle using computational fluid dynamics. Int Res J Eng Technol (IRJET) 03:2395– 0072. 15.Biju Kuttan P, M Sajesh - Optimization of Divergent Angle of a Rocket.





INTERNATIONAL  
STANDARD  
SERIAL  
NUMBER  
INDIA



# INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH

IN SCIENCE | ENGINEERING | TECHNOLOGY

 9940 572 462  6381 907 438  [ijirset@gmail.com](mailto:ijirset@gmail.com)



[www.ijirset.com](http://www.ijirset.com)

Scan to save the contact details