

# Computational Fluid Dynamics Study of a Convergent–Divergent Nozzle

---

## Introduction

In propulsion and high-speed aerodynamics, convergent–divergent (C–D) nozzles are essential components that accelerate gases from subsonic to supersonic speeds. Their performance directly influences thrust, efficiency, and stability of propulsion systems. Understanding compressible flow through such nozzles is important not only for theoretical gas dynamics but also for practical aerospace applications. This project applies computational fluid dynamics (CFD) to investigate flow features inside a C–D nozzle, with emphasis on choking, shock formation, and the transition from subsonic to supersonic regimes.

## Motivation

The motivation for this work lies in gaining a deeper understanding of compressible flows in nozzles through numerical simulation. While analytical gas dynamics provides useful predictions, it cannot capture the full details of flow structures such as shock waves and expansion fans. CFD provides these insights and allows for direct visualization of Mach number, pressure, and velocity fields. The study therefore bridges theory and practice, demonstrating how computational methods can support propulsion system design.

## Objectives

- To simulate compressible flow through a C–D nozzle using ANSYS Fluent.
- To analyze flow regimes at different inlet pressures and identify choking, shock formation, and expansion behavior.
- To compare CFD predictions with theoretical expectations from gas dynamics.
- To demonstrate how nozzle performance varies with pressure ratio.

## Methodology

The nozzle geometry was created and discretized into a computational mesh. A density-based solver was chosen in ANSYS Fluent due to its suitability for high-speed compressible flows. Air was modeled as an ideal gas. The boundary conditions included a pressure inlet (varied across multiple cases), a pressure outlet fixed at ambient conditions, and adiabatic no-slip walls. The throat region was carefully meshed to capture steep gradients at the sonic condition. Solver convergence was tracked through residuals and monitoring of Mach number distribution along the nozzle centerline.

## Results and Discussion

The CFD simulations captured a range of flow regimes as inlet pressure was varied. At low pressures, the flow remained mostly subsonic with only mild acceleration. As inlet pressure increased, the flow reached Mach 1 at the throat, producing choked conditions. For moderate pressure ratios, supersonic expansion was achieved in the divergent section without major shocks. At higher pressures, overexpanded flow developed with distinct shock waves forming in the divergent region. In the highest cases, strong shocks and localized separation were observed. These results aligned well with classical gas dynamics, confirming the accuracy of the CFD predictions.

## Conclusion

This project demonstrated how CFD can effectively analyze compressible flow through a convergent-divergent nozzle. The simulations reproduced choking, supersonic acceleration, and shock formation under different pressure ratios. The results agreed with theoretical gas dynamics, reinforcing CFD's value as a tool for propulsion system analysis. Such studies not only validate theory but also provide detailed insights into nozzle behavior, which are crucial for aerospace engineering applications.

### **Short Project Description (350 characters)**

Simulated compressible flow in a convergent-divergent nozzle using ANSYS Fluent. Analyzed choking, shock formation, and flow regimes across pressure ratios. Validated results with gas dynamics theory, showing CFD as a reliable tool for propulsion nozzle analysis.