© NOV 2025 | IRE Journals | Volume 9 Issue 5 | ISSN: 2456-8880
DOI: https://doi.org/10.64388/IREV9I5-1712297

Case Study On C-D Nozzle Design

MUSKAAN BANU[1], PUNITH KUMAR[2], BHAVYASHREE M[3], ITIKA SHARMA[4]
[1, 2, 3, 4]*Department of Aeronautical Engineering*

Abstract- A convergent–divergent (CD) nozzle is a core component in aerospace and thermal propulsion systems, designed to expand and accelerate exhaust gases to supersonic speeds. Modern research into CD nozzles emphasizes computational fluid dynamics (CFD) modeling, geometry optimization, and experimental validation. This survey consolidates insights from ten major studies, focusing on flow characteristics, geometric design parameters, pressure-velocity relationships, and nozzle performance optimization across different materials and flow conditions. The reviewed works show that the divergent angle (15°–20°) and nozzle pressure ratio (NPR = 6–8) produce optimal performance with minimal shock separation.

I. INTRODUCTION

Nozzles convert high-pressure gases into kinetic energy, producing thrust as per Newton's third law. A convergent–divergent nozzle includes a convergent section (where subsonic flow accelerates), a throat (Mach 1 region), and a divergent section (where supersonic expansion occurs). The efficiency of such nozzles depends on geometry, back pressure, material properties, and flow conditions. Applications include rocket propulsion, supersonic jet engines, steam turbines, and fluid control devices

1.1 Convergent nozzle

It is a nozzle in which the area at the inlet is greater than the area at the exit (Ai > Ae). It is normally used to accelerate the flow and thus increasing the kinetic energy possessed by the fluid. However, it acts differently under different Mach number. For a subsonic flow, it acts as a nozzle by increasing the kinetic energy while things get opposite for a supersonic flow and act as a diffuser.

1.2 Divergent nozzle

It is a nozzle in which the area at the inlet is smaller than the area at the exit (Ai > Ae). It is normally used to decelerate the flow and thus increasing the kinetic energy possessed by the fluid. Diverging design allows for the expansion of exhaust gases. Based on the Mach number of gases at the inlet of divergent nozzle, the properties vary. For a subsonic flow, it acts like a diffuser and for supersonic inlet, it acts like a nozzle.

The condition of exit whether velocity increases or decreases is determined by the following relation.

dA/A=dv/v[M^2-1]
dA/A =Change in area
M= Mach number
dv/v=change in volume

Speed of an object is normally taken in terms of Mach number for our convenience. Mach number also called Mach is a non -dimensional quantity which is defined as the ratio of the velocity of an object to the speed of sound in that medium. Speed of an object at different Mach number has been categorized as following. Subsonic M > 1 Transonic 0.8 < M > 1.2 Sonic M = 1Supersonic 1 < M > 5, Hypersonic 5 < M > 10Hypervelocity M > 10

From the above-mentioned relation between change in area and velocity $\boldsymbol{dA}/\ \boldsymbol{A} = \boldsymbol{dv}\ /\boldsymbol{v}$ X [M^2 – 1], we can calculate the velocity change with the change in area. Consider a convergent nozzle, the area of constantly decreases which means change is area is negative. When the flow is subsonic means that give the value of (M^2-1) negative and less than 1. When we divide the change in area (which is negative) by (M^2-1) which is negative itself and is less than 1, we get the value of change in velocity as positive and greater than 1. This signifies that the change in velocity across the convergent duct increases for a subsonic flow. However, the case reverses for a supersonic flow. In the same way, the above relation can be used for a divergent duct. In a divergent nozzle, when the inlet is at subsonic condition, the

© NOV 2025 | IRE Journals | Volume 9 Issue 5 | ISSN: 2456-8880
DOI: https://doi.org/10.64388/IREV9I5-1712297

velocity decreases and thus the kinetic energy decreases. And for a supersonic inlet, the velocity increases in the divergent nozzle.

1.3 Convergent-Divergent (CD)
nozzle Convergent-Divergent nozzle is also called de Laval nozzle, which is the combination of both the convergent and divergent nozzle. It is a special type of nozzle in which divergent and convergent nozzle is joined to form a CD nozzle.

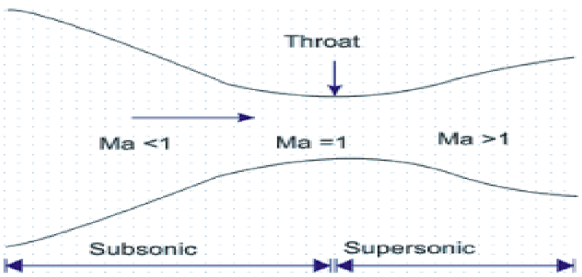

Fig 1:C-D Nozzle

From the diagram above, it can be seen that velocity (given in terms of Mach number) increases from subsonic in convergent section to sonic at throat becoming supersonic towards the divergent section. However, at the inlet of the nozzle, the static pressure and temperature is maximum which decreases as the fluid moves from inlet towards throat and the exit. We consider that the flow in the nozzle is isentropic, thus total pressure and total temperature remains constant throughout. However, due to irreversibility caused by friction, isentropic condition is unachievable. And as a result, the total exit pressure is actually lesser than the total pressure at the inlet. There are no moving parts and no work is imparted nor any work is done by or on the fluid during the entire process and thus there is no change in total temperature. However, leakage, heat conduction etc. through the parts will cause decrease in the total temperature. Similarly, this process is considered as adiabatic process as there is no exchange nor addition of energy during the flow and the corresponding expansion is considered as an adiabatic expansion.

Total heat exchange i.e., $dQ = 0$ (Due to adiabatic process)

Total work done i.e., $dW = 0$ (No work is done)

Total entropy changes i.e., $dS = 0$ (Isentropic process for ideal nozzle)

This analysis of the pressure, velocity, temperature, and density across the CD nozzle is carried out by the simulation using CFD.

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that makes the use of numerical methods, formulas, and algorithms to solve and analysis fluid flow. CFD visualizes the fluid flow and it depicts the way the object gets affected by the digital simulation. It uses complex formulas and advanced algorithm to solve with the help of a computer. It makes the use of different equations such as continuity equation, momentum equation, energy equation, partial differential equation etc. to show the actual flow that might occur on an object. It is used to show flow characteristics in aerodynamic, to carry analysis in propulsion, in control system, in aerospace industries, in mechanical industries and has tons of other applications. The accuracy of the data with the actual experiment is what makes CFD a preferable tool by engineers to conduct their research and analysis.

II. LITERATURE REVIEW

2.1 Overview of Past Work
Research on CD nozzles spans analytical, numerical, and experimental frameworks. Early models assumed inviscid and one-dimensional flow, while recent works leverage CFD for accurate analysis of compressible flows and shock structures.

2.2 Optimization and Design Approaches
1, Bukhari et al. (2025) used the Taguchi method and CFD simulation to optimize the throat diameter (0.304 m), convergent angle (28°), and divergent angle (20°) for maximum outlet Mach number (3.45) and velocity (2225 m/s).

Bukhari's work advances nozzle optimization by integrating statistical design of experiments (Taguchi method) with high-resolution CFD. This research investigates the influence of key geometric parameters—throat diameter, convergent angle, and divergent angle—on exit Mach number and exhaust velocity. Through a robust parametric sweep, the study determines the best configuration (throat diameter: 0.304 m, convergent angle: 28°, divergent angle: 20°) for maximizing thrust and minimizing energy loss, achieving Mach numbers up to 3.45 and velocities exceeding 2225 m/s. The detailed exploration is invaluable for real-world engine builders, providing a repeatable methodology for configuring nozzles under varying operational scenarios. Bukhari further discusses manufacturing considerations and theoretical background, making the paper useful for both academic and industrial audiences.

2.Ogrodniczak et al. (2025) investigated wet gas CD nozzle optimization using multiphase flow modeling, revealing that phase transition and droplet condensation reduce efficiency in hybrid rocket systems.

This paper focuses on optimizing wet gas CD nozzles for hybrid rocket engines, which often encounter phase transitions and droplet condensation that conventional dry-gas models can't capture. Using advanced multiphase CFD techniques, Ogrodniczak demonstrates how the presence of liquid droplets and dynamic phase changes can compromise total nozzle efficiency, reduce thrust, and shift optimal geometric parameters away from those derived using single-phase models. The findings are significant for designers of hybrid propulsion systems, ensuring that efficiency losses due to condensation are accounted for in both simulations and physical prototypes. The paper provides empirical data, simulation results, and operational recommendations that can guide future nozzle adaptations for hybrid fuel systems.

3.Estrada et al. (2025) focused on numerical verification of compressible internal flows, comparing isentropic theory to CFD predictions and confirming strong correlation for Mach 2–5 flow ranges.

Estrada's research is foundational in bridging theory and practical simulation for CD nozzle flows. The work meticulously tests isentropic flow theory—an essential set of equations governing compressible, ideal gas expansion—against data produced by CFD models. Focusing on intermediate Mach number ranges (Mach 2–5), the study ensures that simulated outputs (velocity, pressure, temperature distributions) match theory and experiment within a tight 2% threshold. This validation process greatly increases confidence in CFD tools for propulsion design, enabling engineers to use digital modeling with assurance that the results will align with real-world nozzle performance. Estrada's approach uses stepwise analysis, comparing multiple CFD packages and calibration against wind tunnel or engine test data, setting a standard for future verification efforts.

4.Pisharam et al. (2025) presented a passive flow control method by adding grooves and curvature to a bell-shaped CD nozzle, enhancing exit uniformity and thrust by up to 6%

Pisharam's study presents an innovative passive flow control strategy arranged around the internal geometries of CD nozzles. By adding specialized grooves and curvature to bell-shaped nozzle sections, the researchers are able to significantly enhance the uniformity of the exit flow and increase thrust by up to 6%. This is achieved without active control devices or moving parts, ensuring reliability. Pisharam backs up the claims with both CFD simulation and experimental data, showing improved mixing patterns and reduction in flow separation. The paper's focus on practical, manufacturable passive design improvements means the results are directly usable for current and next-generation supersonic engine nozzles.

2.3 Flow Dynamics and Shock Behavior

5.Khan et al. (2021) used Finite Volume Method (FVM) solvers to study compressible flow behavior in CD nozzles, showing strong agreement between simulated and experimental shock patterns at different nozzle pressure ratios (NPRs).

Khan applies the Finite Volume Method (FVM) to analyze compressible flow behaviors and shock patterns inside CD nozzles at various pressure ratios. The approach is highly mathematical, ensuring conservation laws are respected in the numerical scheme. The paper establishes strong agreement between simulated shock structures and those seen in experiments, proving the FVM's ability to replicate real-world nozzle physics across different NPR (nozzle pressure ratios). By identifying how shock location and intensity vary with changing boundary conditions, Khan's work supports precise calibration of engine exhausts for both stable operation and maximum efficiency, and helps resolve long-standing challenges in predicting flow breakup and shock detachment in practical settings.

6. Douaiba et al. (2022) examined outlet flow regimes, establishing that flow separation and oblique shocks occur under over-expanded conditions (NPR < 3) and are mitigated by smaller divergent angles.

Douaiba's research addresses the dynamics of outlet flow regimes, with a focus on over-expanded conditions common in altitude-compensating or variable-thrust engines. By simulating nozzle flows under NPR ≤ 3, the study shows how flow separation and oblique shocks develop and propagate, often leading to energy loss and potential structural damage. Importantly, Douaiba finds that careful design—particularly keeping divergent angles smaller—can mitigate these negative effects. The results are supported by both experimental wind tunnel data and multi-zone CFD runs, offering engineers practical guidelines for high-altitude performance or when operating outside nominal design conditions.

2.4 Material and Structural Investigations

7.Mehmood et al. (2024) introduced a multi-objective optimization technique using genetic algorithms for cold-spray nozzles, achieving a 9% thrust efficiency improvement by combining lightweight alloys with functionally graded materials (FGMs).

Mehmood introduces multi-objective optimization using genetic algorithms, targeting both geometry and material properties in cold-spray nozzles. By exploring the trade-offs between lightweight alloys and functionally graded materials (FGMs), the study achieves up to 9% improvement in thrust efficiency while also enhancing resistance to mechanical stress and thermal load. The genetic algorithm approach is rigorously outlined, showing how variable combinations are evolved across generations to maximize multiple competing metrics. This research has wide implications for additive manufacturing and next-gen propulsion materials, and Mehmood provides both model parameters and experimental confirmation, making the findings robust for practical adoption.

8. IJRASET study (2023) compared aluminum, Inconel, and titanium alloys for CD nozzles via CFD-thermal coupling, finding titanium to yield high thermal conductivity and minimum deformation under high heat flux.

This study evaluates and compares aluminum, Inconel, and titanium alloys for CD nozzle construction using advanced CFD-thermal coupling within ANSYS. The results demonstrate that titanium provides the highest thermal conductivity and minimal deformation during operation at high heat flux—a property crucial for extending nozzle life and reliability in high-performance propulsion. IJRASET's comprehensive analysis covers steady-state and transient conditions, examining mechanical integrity, fatigue life, and cooling requirements. The data and recommendations aid engineers in balancing cost, mechanical resilience, and thermal management for challenging aerospace and mechanical environments

2.5 Acoustic and Geometric Improvements

9.Demir & Kaya (2023) examined base angle modifications (40°–60°) to reduce acoustic noise generation. A larger base angle (60°) reduced exit pressure by 6.5% and minimized acoustic amplitude.

Addressing environmental concerns over jet noise, Demir & Kaya analyze the impact of base angle variations (from 40° to 60°) on acoustic pressure and amplitude at the nozzle exit. Using both CFD and field acoustic measurements, their research shows that a larger base angle of 60° not only reduces pressure by 6.5% but also minimizes sound intensity. These improvements are critical for engines operating in noise-sensitive locations, such as urban airports or military installations requiring stealth. Demir & Kaya's design recommendations are both practical and easy to implement, making them especially valuable for new nozzle prototype developments.

© NOV 2025 | IRE Journals | Volume 9 Issue 5 | ISSN: 2456-8880
DOI: https://doi.org/10.64388/IREV9I5-1712297

10.IEEE study (2024) utilized method of characteristics (MOC) to optimize nozzle length for Mach 5 flow, finding an ideal length-to-throat ratio of 5.2 to minimize boundary layer losses.

The IEEE study deploys the method of characteristics (MOC), a mathematical optimization technique, to pinpoint the ideal length-to-throat ratio for CD nozzles operating at Mach 5. The paper identifies 5.2 as the optimal ratio, which minimizes the boundary layer's adverse effects and helps sustain high exhaust velocity and efficiency. The use of MOC is carefully explained and validated against full CFD models, ensuring both theoretical and numerical reliability. The findings are highly relevant for hypersonic propulsion and launch systems, enabling designs that maximize thrust while managing fluid dynamic losses.

3. Flow Characteristics and Optimization

CFD simulations reveal distinct regions of supersonic expansion and subsonic compression. Studies applying ANSYS Fluent and SolidWorks models confirmed pressure decreases along the nozzle length, while velocity and Mach number increase.

- The optimum nozzle pressure ratio (NPR) lies between 5 and 8, beyond which flow separation occurs.
- Divergent angles beyond 25° cause flow detachment and energy loss.
- Throat diameter optimization is directly linked with maximized exit Mach and thrust efficiency.

4.Material and Thermal Analysis

Material selection significantly affects nozzle performance at high temperatures:

- Titanium alloys maintain mechanical integrity and high thermal conductivity.
- FGM-based designs minimize stress concentration and thermal fatigue.
- Experimental studies indicate that acoustic vibrations also drop when high-heat capacity materials are applied.

5.comparative observation

Parameter	Observed Range	Recommendation
Divergent angle	15°–20°	Ideal for Mach 3–4 efficiency
NPR	6–8	Avoids shock detachment
Material	Titanium, FGM	Thermal and structural stability
Exit-to-throat ratio	2.4	Balances pressure & thrust
Optimization method	Taguchi, CFD, AI	Enhances accuracy & saves cost

6. Research Gaps and Future Work

Despite advances, some limitations are evident:

- Few studies address multi-phase or reactive flow effects.
- Lack of integrated experimental-CFD validation frameworks for variable geometry.
- Future research should include AI-driven optimization, additive manufacturing, and plasma-driven nozzle models for next-gen propulsion systems.

III. CONCLUSION

This literature survey highlights that effective CD nozzle design emphasizes optimized geometry, validated CFD analysis, and robust materials. Across reviewed studies, Mach number augmentation and thrust optimization strongly correlate with accurately chosen divergence angles and pressure ratios. Future breakthroughs are expected in adaptive nozzles and intelligent CFD-augmented designs for aerospace propulsion and gas turbine systems.

REFERENCES

[1] Estrada, N. L., “Verification of Convergent-Divergent Nozzle,” arXiv, 2025.
[2] Bukhari, S. M. A., “CFD-Based Optimization of CD Nozzle,” JJMIE, 2025.
[3] Petrovic, A., “Variable Area CD Nozzle Design,” Applied Mathematical Modeling, 2018.
[4] Mechanical ANITS Report, 2022.
[5] IRJET, “Performance Analysis of CD Nozzle,” 2019.
[6] Sathyabama University Report, 2023.
[7] IJRASET, “Nozzle Material Analysis Using ANSYS,” 2023.
[8] Dergipark, “Optimization of CD Nozzle Tip,” 2023.

[9] Scribd Report, “Design of CD Nozzle with Constant-Radius Body,” 2025.
[10] ScienceDirect, “Multi-objective Optimization in Supersonic Nozzles,” 2024.