Numerical study of shock cell structure and spread characteristics of non-circular jets

Abhinay raj S[1], Parthipan S[2], Joshua Messu S[3], Dr. Sandeep J [4]

[1,2,3]Student, Dept. of Aeronautical Engineering, IARE, Hyderabad, India [4]Guide,
Dept. of Aeronautical Engineering, IARE, Hyderabad, India
[1]abhinayrajsadala@gmail.com [2]parthipansiddamalla@gmail.com [3]joshuamessu2004@gmail.com
[4] j.sandeep@iare.ac.in

Abstract—The flow characteristics of supersonic jets issuing from non-circular nozzles are of significant importancein propulsion systems, high-speed exhaust configurations, and aeroacoustic control applications. Unlike conventional circular jets, geometrically asymmetric nozzles generate complex flow features that substantially alter mixing behaviour, shock-cell patterns, and noise generation mechanisms. In this study, a computational investigation is conducted to examine the influence of nozzle cross-sectional geometry—namely circular, square, and triangular shapes—on shock-cell development and jet spreading at an exit Mach number of 3.0.

Three-dimensional, steady-state Reynolds-Averaged Navier–Stokes (RANS) equations are solved using the k–ω SST turbulence model within ANSYS Fluent 2023 R1. To isolate the effect of geometry, all nozzle configurations are designed with an identical exit area of 29.93 mm². The results indicate that both nozzle shape and the corresponding hydraulic diameter play a critical role in determining jet development characteristics. Configurations with smaller hydraulic diameters exhibit reduced potential core lengths and enhanced velocity decay rates.

Furthermore, non-circular jets demonstrate increased lateral spreading compared to circular jets, primarily due to the formation of strong corner-induced vortical structures that enhance entrainment. The presence of axis-switching phenomena in square and triangular jets further intensifies mixing and modifies downstream flow evolution. These findings provide valuable insights for the design and optimization of supersonic nozzles in aerospace propulsion and high-speed flow applications.

Keywords — Non-circular jets; supersonic jet flow; shock-cell structure; jet mixing; hydraulic diameter; axis switching; k–ω SST model; CFD; vortex dynamics; nozzle geometry.

I, Introduction

Jet flows represent a fundamental class of fluid dynamic phenomena and are integral to a wide range of engineering applications, including aerospace propulsion, industrial jet systems, and high-speed exhaust configurations. In supersonic regimes, jets exhibit complex flow features such as shock-cell structures, shear layer instabilities, and turbulent core development, all of which significantly influence mixing efficiency, thrust characteristics, and acoustic emissions. While circular nozzles have been extensively studied due to their geometric simplicity, they do not always provide optimal performance. In contrast, non-circular nozzles introduce geometric asymmetry that can enhance mixing, modify noise characteristics, and enable better control over jet development when properly understood.

In compressible supersonic flows, shock-cell patterns arise due to a mismatch between the nozzle exit pressure and the ambient pressure. This mismatch generates alternating expansion and compression regions, forming a quasi-periodic structure along the jet centerline. For circular nozzles, these shock cells are axisymmetric and relatively predictable. However, for non-circular geometries such as square or triangular nozzles, the inherent asymmetry distorts the shock-cell arrangement into a three-dimensional structure, making both experimental characterization and numerical prediction more challenging.

A key geometric parameter influencing jet behaviour is the hydraulic diameter, D_h , defined as four times the flow area divided by the wetted perimeter. This parameter serves as an effective length scale to represent wall–fluid interaction effects across different nozzle shapes. Jets issuing from nozzles with smaller hydraulic diameters generally exhibit enhanced mixing rates and reduced potential core lengths, although the implications of these effects depend on the specific application.

With advances in computational capability, Computational Fluid Dynamics (CFD) has emerged as a powerful approach for analysing supersonic jet flows. By numerically solving the governing conservation equations on discretized domains, CFD provides detailed insight into spatial variations of key flow variables such as Mach number, pressure, temperature, and turbulence quantities. This level of detail is often difficult to obtain experimentally, especially for fully three-dimensional supersonic flows.

The present study performs a comparative numerical investigation of circular, square, and equilateral triangular nozzle geometries operating at a design exit Mach number of 3.0. Simulations are conducted under perfectly expanded, under-expanded, and over-expanded conditions to evaluate the influence of nozzle shape on shock-cell structure, jet spreading characteristics, potential core length, and turbulent mixing behaviour. The objective is to isolate geometric effects and provide insights relevant to the design of high-speed nozzle systems.

Significance of Non-Circular Jets

Non-circular nozzles fundamentally alter the exit flow distribution, leading to distinct advantages over conventional circular configurations. The asymmetric geometry promotes the formation of additional vortical structures within the shear layer, which enhances entrainment of surrounding fluid and accelerates mixing. Furthermore, modifications in turbulence structure can reduce dominant noise-producing mechanisms, particularly in subsonic and low-supersonic regimes.

From a propulsion perspective, nozzle geometry influences shock-cell spacing and strength, enabling potential optimization of thrust characteristics for specific operating conditions. In thermal and industrial applications, non-circular jets also provide improved control over heat transfer distribution, allowing either broader coverage or localized enhancement depending on design requirements.

Shock-Cell Structure

Shock-cell formation is a defining feature of imperfectly expanded supersonic jets. In under-expanded conditions, where exit pressure exceeds ambient pressure, the jet undergoes expansion through oblique waves followed by recompression, producing the characteristic diamond-shaped pattern observed in flow visualization techniques such as Schlieren imaging. The spacing of these shock cells depends on parameters such as nozzle diameter and pressure ratio.

For non-circular jets, the shock-cell structure becomes inherently three-dimensional. The lack of geometric symmetry leads to variations in cell spacing along different directions, while interactions between shock waves and corner-induced shear layers introduce additional flow complexity. These effects cannot be accurately captured using simplified two-dimensional assumptions, necessitating full three-dimensional analysis.

Jet Spreading Behaviour

The rate at which a jet expands downstream—referred to as the spreading rate—is a critical factor governing entrainment, mixing efficiency, and the persistence of supersonic flow regions. Circular jets exhibit well-established spreading characteristics; however, non-circular jets display anisotropic spreading behaviour. For example, jets issuing from rectangular or triangular nozzles may expand more rapidly along one axis than another.

A notable phenomenon in non-circular jets is axis switching, where the major and minor axes interchange as the jet develops downstream. This behaviour arises due to azimuthal pressure gradients and the redistribution of momentum caused by the non-uniform geometry. Axis switching significantly enhances mixing and contributes to the faster breakdown of the jet core compared to circular configurations.

Numerical Considerations

Accurate numerical prediction of supersonic jet flows requires careful treatment of several key aspects. First, the turbulence model must effectively capture both free shear layer development and shock–boundary layer interactions. Second, the computational grid must be sufficiently refined to resolve shock structures without excessive computational cost. Third, boundary conditions must accurately represent the intended nozzle pressure ratios. Finally, validation against established experimental or numerical data is essential to ensure the reliability of simulation results. These considerations form the basis of the methodology adopted in this work.

II. Literature Review

Literature Review

The study of supersonic jets issuing from non-circular nozzles has been an active area of research for several decades, encompassing experimental investigations, analytical formulations, and computational simulations. Prior work has consistently demonstrated that deviations from circular geometry introduce significant modifications in jet structure, mixing characteristics, and acoustic behaviour. The following studies represent key contributions that inform the present investigation.

S. Bogadi and B. T. N. Sridhar (2019) explored the influence of diagonal expansion ramps integrated at the exit of rectangular nozzles. Their analysis revealed that such geometric modifications induce strong streamwise vortical structures within the jet plume. These vortices enhance turbulence intensity, thereby accelerating entrainment of ambient fluid and promoting rapid mixing. As a consequence, the potential core length is reduced and the supersonic region decays more quickly. Their findings highlight the effectiveness of passive geometric modifications in altering jet behaviour without requiring external energy input, making them particularly attractive for practical propulsion systems.

Prasanta Kumar Mohanta and co-authors (2016) conducted a comparative study on jets emerging from square and hexagonal nozzles while maintaining identical throat-to-exit area ratios. Despite this constraint, noticeable differences in jet development were observed, indicating a strong dependence on nozzle geometry. Configurations with larger wetted perimeters relative to flow area exhibited enhanced mixing rates due to increased interaction between the jet and surrounding fluid. The study also demonstrated close agreement between Schlieren flow visualisation and numerical simulations, reinforcing the reliability of CFD techniques for analysing complex jet flows.

The work of D. M. Mitchell et al. (2013) focused on under-expanded elliptic jets, with particular emphasis on near-field flow behaviour. A key observation was the occurrence of axis-switching, where the major and minor axes of the jet interchange as it propagates downstream. This phenomenon results in intensified turbulence and a more intricate shock-cell configuration compared to circular jets. Additionally, the enhanced mixing leads to a shorter potential core, suggesting that elliptic nozzles can achieve improved mixing efficiency within a limited spatial extent.

K. Lee and collaborators (2012) investigated the impact of chevron geometries on jet mixing and noise reduction in subsonic conditions. Chevron nozzles introduce controlled asymmetry through serrated edges at the nozzle exit, generating streamwise vortices that disrupt large-scale coherent structures in the shear layer. Their results indicated that optimized chevron configurations can effectively reduce noise levels while maintaining acceptable thrust performance. This study demonstrates that even small geometric alterations at the nozzle exit can significantly influence jet acoustics and flow structure.

Earlier work by M. Kaushik (2006) examined the effect of introducing notches at the exit of circular sonic nozzles. These notches act as passive flow control elements, generating additional vortical structures that enhance mixing within the shear layer. The modification was found to disrupt the regular shock-cell pattern, shorten the potential core, and increase jet spreading rates. The results further support the concept that passive geometric modifications are an effective means of controlling supersonic jet behaviour.

🔍 Research Gap (Important for your paper)

Although previous studies have established that geometric modifications significantly influence jet development, most investigations have focused on specific configurations such as elliptic, rectangular, or modified circular nozzles. A systematic comparison between fundamentally distinct geometries—such as circular, square, and triangular nozzles—under identical operating conditions remains limited.

III. Methodology

The approach taken in this study combines three-dimensional CAD modelling, structured and unstructured meshing, and density-based CFD simulation to produce a consistent set of flow-field data for each nozzle configuration. Each stage is described in turn below.

3.1 Problem Description

Three nozzle geometries are considered: circular (used as the reference case), square, and equilateral triangular. All three share the same convergent-divergent internal profile and the same exit area (29.93 mm²), so differences in the computed flow field can be attributed directly to exit shape rather than to differences in mass flow or throat area. The design exit Mach number is 3.0, and three nozzle pressure ratio (NPR) conditions are simulated: perfect expansion, over-expansion, and under-expansion. Simulations are carried out in ANSYS Fluent 2023 R1 using steady-state, three-dimensional RANS.

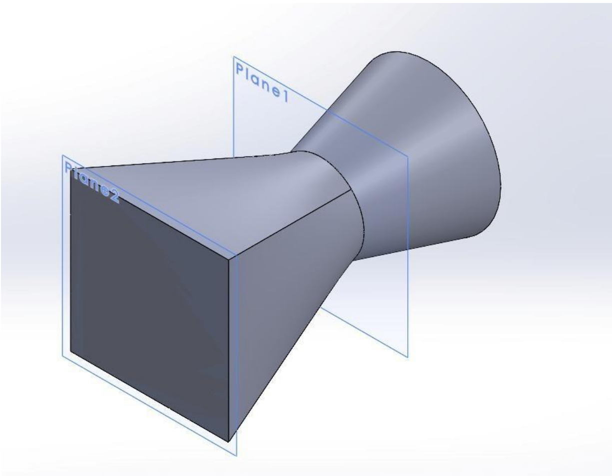
Parameter	Circular Nozzle	Square Nozzle	Triangular Nozzle
Exit Area (mm²)	29.93	29.93	29.93
Exit Dimension	d = 6.17 mm	L = 5.47 mm	L = 8.30 mm
Inlet Diameter (mm)	5.0	5.0	5.0
Throat Diameter (mm)	3.0	3.0	3.0
Convergent Half-Angle	15°	15°	15°
Divergent Half-Angle	15°	15°	15°
Design Mach Number	3.0	3.0	3.0

3.2 Nozzle Geometry and Design

All nozzles share a 5.0 mm inlet diameter and a 3.0 mm throat diameter, with 15° half-angles on both the converging and diverging sections. The diverging section transitions from the circular throat to the exit cross-section at the exit plane. Exit dimensions were derived from the equal-area constraint:

- Circular nozzle: exit diameter d = 6.17 mm
- Square nozzle: side length L = 5.47 mm (area = L^2 = 29.93 mm²)
- Equilateral triangular nozzle: side length L = 8.30 mm (area = $(\sqrt{3}/4)L^2$ = 29.93 mm²)

Three-dimensional solid models were created in SolidWorks 2022. The computational domain was extended approximately 20 hydraulic diameters downstream of each exit plane to capture the complete shock-cell structure and allow the jet to develop without undue influence from the outlet boundary.



3.3 Computational Domain and Meshing

Meshes were generated in ANSYS Meshing. An unstructured tetrahedral topology was adopted for all three configurations because the non-circular exit geometries make structured meshing impractical. Local refinements were applied at the nozzle walls, throat edges, and the downstream plume region where shock gradients are steepest. Inflation layers were added at all solid boundaries to resolve the near-wall region adequately.

The final mesh for each geometry contains approximately 3.0 million elements and 558,000 nodes. Mesh quality was assessed using the Report Quality function in ANSYS Fluent; after applying the built-in improve-quality command, the minimum orthogonal quality reached 0.177 (the accepted minimum for high-speed CFD is 0.1) and the maximum aspect ratio was 19.0, which is acceptable for unstructured meshes in this class of flow.

Flow Condition	Inlet Total Pressure (Pa)	Ambient Back Pressure (Pa)
Perfect Expansion	3,721,327	101,325
Overexpansion	2,533,125	101,325
Underexpansion	6,079,500	101,325

3.4 Governing Equations

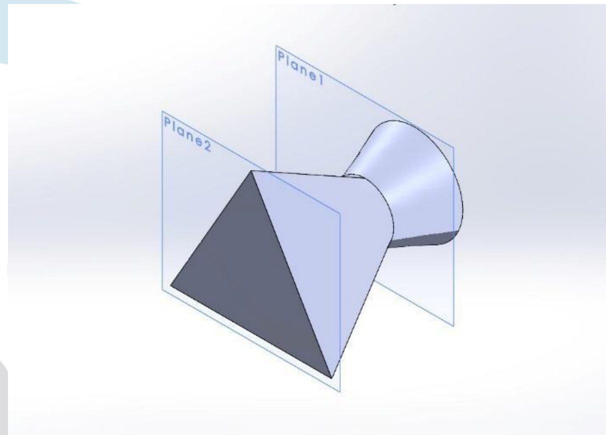
The flow is described by the compressible, steady-state RANS equations expressing conservation of mass, momentum, and energy:

$$\partial\rho/\partial t + \nabla\cdot(\rho U) = 0$$

$$\partial(\rho U)/\partial t + \nabla\cdot(\rho UU) = -\nabla p + \nabla\cdot\tau$$

$$\partial(\rho E)/\partial t + \nabla\cdot(\rho UH) = \nabla\cdot(\tau\cdot U) + \nabla\cdot(k\nabla T)$$

Here ρ is density, U is the velocity vector, p is static pressure, τ is the viscous stress tensor, E is total energy, H is total enthalpy, k is thermal conductivity, and T is temperature.



3.5 Turbulence Modelling

Turbulence closure is provided by the k-ω SST model. This two-equation model blends the k-ε formulation in the free-stream, where it performs well, with the k-ω formulation near walls, where adverse pressure gradients and potential separation require better near-wall accuracy. For supersonic jet flows, which combine intense free-shear mixing with shock-boundary layer interactions at the nozzle walls, this blended approach is well-suited and is widely used in the literature for comparable problems.

3.6 Fluid Properties and Thermodynamic Model

Air is treated as a compressible ideal gas. Density is computed from the ideal gas law $\rho = p/(RT)$, where R = 287 J/(kg·K). Dynamic viscosity is temperature-dependent and computed using Sutherland's law:

$$\mu = \mu_0 (T/T_0)^{(3/2)} \times (T_0 + S)/(T + S)$$

where $\mu_0 = 1.716 \times 10^{-5}$ kg/(m·s) at $T_0 = 273.15$ K, and S = 110.4 K. The specific heat ratio is $\gamma = 1.4$ and Cp = 1006.43 J/(kg·K).

3.7 Boundary Conditions

Identical boundary conditions are applied to all three geometries across all three NPR cases:

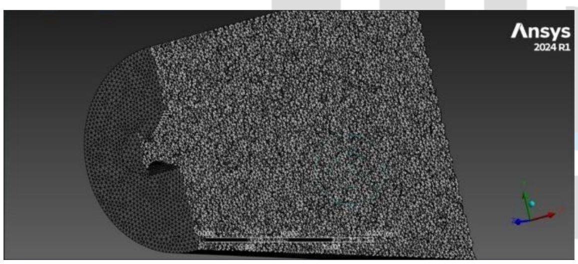
- Inlet (Pressure Inlet): total pressure set per the NPR condition; total temperature = 300 K.
- Outlet (Pressure Outlet): ambient static pressure = 101,325 Pa for all cases.

- Nozzle Walls: no-slip, adiabatic condition at all solid surfaces.
- Far-field Boundaries: pressure outlet at ambient conditions to simulate free exhaust into still air.

3.8 Solver Settings

A density-based coupled solver was chosen because it is designed for flows where pressure and density are tightly coupled, as they are in high-speed compressible jets. The principal settings were:

- Solver type: Density-Based, Steady-State
- Flux discretisation: Roe-FDS (Flux Difference Splitting)
- Spatial discretisation: second-order upwind for all variables
- Gradient computation: Least Squares Cell Based
- CFL number: started at 0.5, raised gradually to 2.0 as the solution stabilised
- Convergence criterion: residuals $< 1 \times 10^{-5}$ for all equations
- Parallelisation: 8-core parallel processing.



3.9 Post-processing

All results were post-processed in ANSYS CFD-Post. The following quantities were extracted for each geometry and NPR condition:

- Mach number contours on the nozzle centreline plane to reveal shock-cell spacing and Mach disk position.
- Static pressure along the jet axis to quantify the periodicity of shock cells.
- Velocity vector fields to track jet spreading and axis-switching in the non-circular cases.
- Turbulent kinetic energy contours to compare mixing enhancement across nozzle types.
- Three-dimensional Mach number isosurfaces to characterise the full shock structure.

The local Mach number was derived in CFD-Post from $M = |V| / \sqrt{\gamma RT}$, where $|V|$ is the velocity magnitude from the u, v, w components, $\gamma = 1.4$, $R = 287$ J/(kg·K), and T is the local static temperature.

3.10 Grid-Independence Study

To confirm that results are not unduly sensitive to mesh resolution, a grid-independence study was carried out for the circular nozzle under perfect expansion. Three refinement levels were tested: coarse (~1.5 million elements), medium (~3.0 million, the adopted mesh), and fine (~5.0 million). Comparing the axial Mach number distribution across the three levels showed that the medium and fine meshes agree to within 1%, while the coarse mesh introduces noticeable discrepancies near the first shock cell. The medium mesh was therefore selected as offering the best balance between accuracy and computational cost.

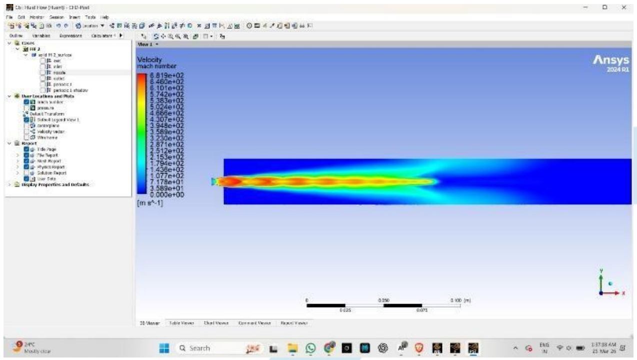
IV. Results

The simulations were run to convergence with residuals below 1 $\times 10^{-5}$ in all cases. The inlet total pressure of 3,721,327 Pa against an ambient back pressure of 101,325 Pa produced a strongly under-expanded jet at the design Mach number of approximately 3.0. The following subsections describe the key features of the flow field for each nozzle geometry.

4.1 Mach Number Distribution and Shock Cell Formation

In all three cases the flow accelerates through the convergent-divergent passage and reaches supersonic conditions at the exit plane. Immediately downstream, the Mach number distribution shows alternating high and low regions along the jet axis—the signature of a periodic shock-cell train.

The circular nozzle produces a well-ordered, axisymmetric shock-cell pattern with consistent spacing. The cells remain distinguishable over several diameters, reflecting the relatively slow mixing of a round jet.



The square nozzle shows noticeably distorted cells. Uneven pressure gradients at the flat faces and corners generate asymmetric expansion waves; as a result the cells become less regular and their apparent spacing shortens with distance downstream.

The triangular nozzle produces the most irregular shock pattern of the three. The sharp corners create concentrated vortices that interact strongly with the shock structure from early in the near-field, causing rapid degradation of the periodic cell train. This indicates that the triangular jet is exchanging momentum with the surrounding fluid far earlier than either of the other two.

4.2 Potential Core Length and Velocity Decay

The potential core—the region where centreline Mach number stays close to the exit value—provides a convenient measure of how far the jet retains its initial momentum.

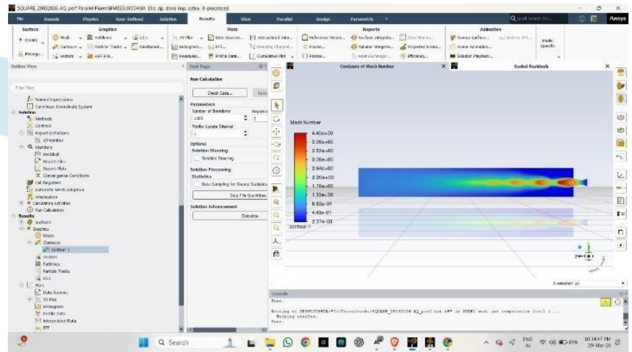
The circular jet maintains the longest potential core. The absence of geometric features that promote vortex shedding means the shear layer grows slowly, and the supersonic core persists well beyond the exit plane.

The square jet's core is shorter. Corner vortices continuously feed turbulence into the shear layer, drawing ambient fluid inward and eroding the core from outside.

The triangular jet has the shortest core by a clear margin. Its three sharp corners generate strong, persistent vortices that interact intensely with the shear layer from the very start of the plume, producing rapid centreline velocity decay and efficient momentum exchange with the surroundings.

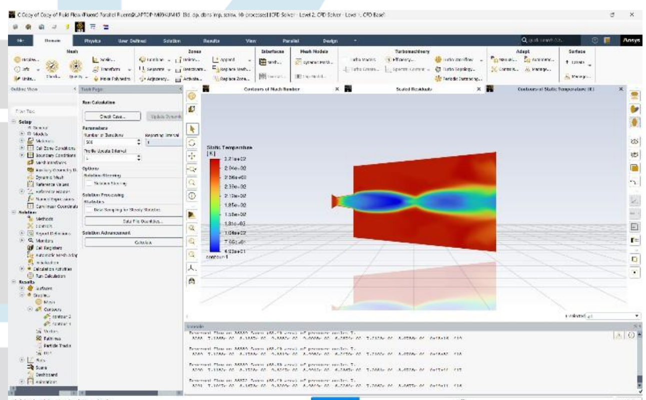
4.3 Jet Spreading Characteristics

Lateral spreading was assessed by tracking the transverse growth of the plume. The circular jet spreads slowly and remains concentrated near the axis. The square jet spreads noticeably faster, driven by the streamwise vortices shed from its corners. The triangular jet spreads fastest of all: the three corners each contribute a strong vortex, and their combined effect produces aggressive lateral growth and high entrainment. This rapid spreading is consistent with the short potential core and the irregular shock pattern described above.



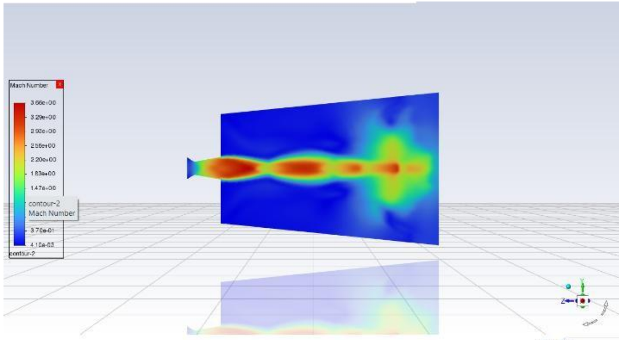
4.4 Axis-Switching Phenomenon

Both non-circular jets display axis switching, though to different degrees. In the square case, the redistribution of momentum is moderate: opposite corners interact symmetrically, and the axis switch is gradual. In the triangular case the situation is more dramatic. The three-fold asymmetry means the velocity distribution is strongly non-uniform across the cross-section from early in the plume, and the axes interchange over a shorter distance. This behaviour accelerates the breakdown of the jet core and contributes to the high overall mixing rates seen in that geometry.



4.5 Pressure and Temperature Distribution

The centreline pressure trace mirrors the shock-cell pattern: high-pressure compression zones alternate with low-pressure expansion regions, with amplitude decreasing downstream as mixing proceeds. The triangular jet shows the fastest amplitude decay, consistent with its rapid mixing; the circular jet sustains pressure oscillations over a longer distance.



Temperature follows a complementary trend. The flow cools as it accelerates to high Mach number, then gradually warms as it mixes with the ambient. Non-circular jets recover to near-ambient temperature more quickly than the circular jet, reflecting their higher turbulence levels and greater entrainment.

4.6 Influence of Hydraulic Diameter

All three nozzles share the same exit area, but their hydraulic diameters differ. The circular nozzle has the largest D_h (equal to its geometric diameter), while the triangular nozzle has the smallest, because its relatively long perimeter reduces D_h for a given area. The results confirm that smaller hydraulic diameter correlates with faster decay, shorter core, and higher turbulence—consistent with the increased wall–fluid interaction .

4.7 Turbulence and Vortex Dynamics

Turbulent kinetic energy contours reveal that the circular jet develops turbulence gradually along its shear layer, with the highest values concentrated near the edge of the jet. The non-circular jets show elevated turbulence much earlier and at higher levels overall, with the triangular nozzle producing the strongest vortical structures of the three. These corner vortices pair with and intensify the natural shear-layer instability, promoting rapid entrainment and explaining the short core lengths and high spreading rates observed.

4.8 Comparative Summary

Taken together, the results establish a clear ranking: the circular nozzle provides the most stable and predictable flow, with well-defined shock cells and slow mixing; the square nozzle offers a middle ground, with measurably better mixing than the circle but less complexity than the triangle; and the triangular nozzle delivers the most aggressive mixing, fastest spreading, and most disrupted shock structure. Whether this ranking translates directly into preferred nozzle choice depends on the application—there are situations where a long, stable supersonic core is desirable—but for applications where rapid mixing or wide coverage is needed, the triangular nozzle is clearly advantageous.

V. Conclusion

The present investigation analysed the influence of nozzle exit geometry on the shock-cell structure and spreading characteristics of supersonic jets operating at a design Mach number of 3.0. Three-dimensional steady-state Reynolds-Averaged Navier–Stokes (RANS) simulations, coupled with the k–ω SST turbulence model and a density-based solver in ANSYS Fluent, were performed for circular, square, and equilateral triangular nozzle configurations with identical exit areas to isolate geometric effects.

The results demonstrate that nozzle geometry significantly governs the evolution of shock-cell structures. The circular nozzle exhibited a highly symmetric and periodic shock pattern, whereas the square configuration introduced moderate asymmetry. In contrast, the triangular nozzle generated strongly distorted and rapidly dissipating shock structures, primarily due to intensified corner-induced vortical interactions.

A clear relationship was observed between geometric asymmetry and potential core length. Increased asymmetry led to a reduction in core length and a more rapid decay of centreline velocity. Among the configurations studied, the triangular jet showed the shortest potential core, while the circular jet sustained supersonic flow over a longer downstream distance. This behaviour is directly linked to enhanced turbulence generation within the shear layers of non-circular geometries.

The hydraulic diameter emerged as a critical scaling parameter, even under constant exit area conditions. Nozzles with smaller hydraulic diameters, corresponding to higher perimeter-to-area ratios, exhibited accelerated mixing and faster jet decay. This highlights the importance of considering hydraulic diameter alongside geometric shape during nozzle design.

Axis-switching behaviour was identified in both square and triangular jets, leading to redistribution of momentum across the jet cross-section and further enhancement of mixing. The effect was more pronounced in the triangular configuration, reflecting its higher degree of geometric non-uniformity.

The predicted pressure and temperature fields were consistent with established compressible flow theory, supporting the validity of the numerical methodology and solver setup adopted in this study.

From an engineering standpoint, the findings confirm that non-circular nozzles provide a viable passive strategy for enhancing jet mixing, increasing spreading rates, and potentially reducing noise levels without the need for active flow control mechanisms. The triangular nozzle demonstrated superior mixing performance, whereas the circular configuration remains advantageous for applications requiring a stable and extended supersonic core.

Future work may extend this analysis to additional nozzle geometries, such as hexagonal or multi-lobed configurations, and to a broader range of nozzle pressure ratios. Such studies would contribute to the development of a comprehensive design framework for high-speed nozzle applications in aerospace propulsion and related engineering systems.

VI References

[1] S. Bogadi, B.T.N. Sridhar. (2019) Decay of supersonic rectangular jet issuing from a nozzle with diagonal expansion ramps. Thermal Science, 23(6B), 3929–3940.

[2] P. K. MOHANTA, "Study of Decay Characteristics of Hexagonal and Square Supersonic Jet," DE GRUYTER, 2016.

[3] D.M. Mitchell, D.R. Honnery, J. Soria. (2013). Title of the article. Experiments in Fluids, 54(4), Article number or page range.

[4] K. Lee, J. Bae, B. Kong, S. Lee. (2014). Effect of chevron tip count and shape on the subsonic jet mixing noise emitted from hot air lance Journal of Mechanical Science and Technology, 28(11), 4573-4581.

[5] M. Kaushik, P.S. Thakur, E. Rathakrishnan. (2006). Studies on the effect of notches on circular sonic jet mixing Journal of Propulsion and Power, 21(1), 211-214.