

CFD Based Geometry Re-Design & Performance Analysis of a Scramjet Intake

Dev Sharma

devsharma4122003@gmail.com

Chandigarh University

prof. Uma Shanker

Chandigarh University

Research Article

Keywords: Scramjet intake, Intake redesign, Computational fluid dynamics (CFD), Aerodynamic performance, Shock-wave interaction

Posted Date: October 26th, 2026

DOI: <https://doi.org/10.21203/rs.3.rs-11260066/v1>

License:   This work is licensed under a Creative Commons Attribution 4.0 International License.

[Read Full License](#)

Additional Declarations: No competing interests reported.

CFD Based Geometry Re-Design & Performance Analysis of a Scramjet Intake

Abstract: - The aerodynamic performance of a scramjet intake depends on efficient supersonic compression, reduced total-pressure losses, and suitable flow distribution at the combustor interface. The present study investigates multiple redesigned scramjet intake configurations using 2-dimensional computational fluid dynamics (CFD) simulations. Systematic geometric modifications are introduced to the baseline configuration to examine their influence on internal shock-wave interactions, compression characteristics, and downstream flow development. Numerical simulations are performed using a density-based, steady-state solver coupled with the SST (k- ω) turbulence model. The redesigned configurations are evaluated at an inlet static pressure of 52 kPa over a Mach-number of 4. A structured computational grid is employed, and grid-independence assessment is performed to minimize spatial discretization effects. The flow field is evaluated using Mach number, static pressure, static temperature, velocity distribution, shock-wave structure, pressure recovery, and total-pressure losses. Particular attention is given to oblique shock-wave interactions with the intake walls and their influence on compression and exit-flow characteristics. The comparative analysis provides insight into the aerodynamic response of modified intake geometries and their suitability for effective supersonic compression with controlled aerodynamic losses.

Keywords: - Scramjet intake; Intake redesign; Computational fluid dynamics (CFD); Aerodynamic performance; Shock-wave interaction.

1. Introduction: -

Scramjet (Supersonic Combustion Ramjet) engines are airbreathing propulsion systems designed for hypersonic flight, eliminating mechanical compressors by utilizing strategically contoured *inlet-isolator geometries*. Incoming freestream air undergoes compression through a series of oblique shock waves generated by the vehicle's forebody and compression ramps. This process rapidly decelerates and compresses the supersonic airflow into a high-pressure, high-velocity stream ready for stable, continuous supersonic combustion within the downstream combustor.

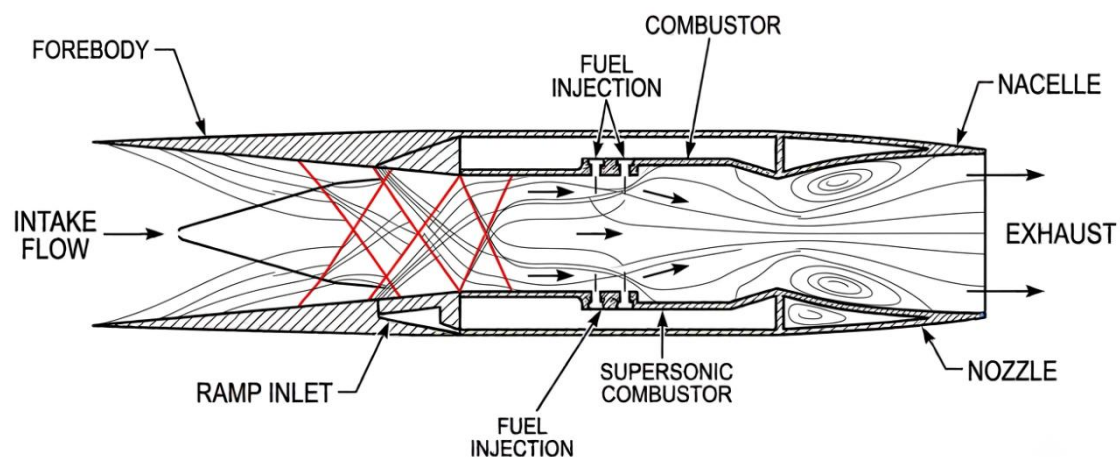


Fig no: -1 A schematic depicting the structural architecture of a scramjet engine and delineating its operational principles.

A critical operational challenge involves internal aerodynamic phenomena like shock wave-boundary layer interactions (SBLIs), boundary layer separation, and shock-shock interactions. These adverse

viscous-inviscid interactions generate low-momentum separation bubbles and severe flow distortion. Ultimately, this structural degradation reduces the total pressure recovery ratio, compromises overall compression efficiency, and risks triggering catastrophic engine unstart across lower speed ranges and varied off-design flight envelopes.

To mitigate these severe flow losses, modern researchers utilize diverse passive and active flow-control methodologies. While some designs implement upward-protruding ramp grooves to generate counter-rotating vortex pairs, alternative configurations deploy porous wall ventilation and passive boundary layer bleed gaps to absorb localized boundary layer detachment. Additionally, automated genetic-algorithm-based optimization procedures geometrically modify compression ramps for simultaneous aerodynamic drag minimization and pressure recovery maximization.

Complementing intake aerodynamics, complex downstream transient flow fields govern combustion stabilization within the engine. High-speed diagnostic methodologies, including colour Schlieren visualization and non-intrusive luminescent pressure-sensitive paint, confirm that rising backpressure forces fuel-induced bow shocks upstream. This induced boundary layer separation paradoxically benefits performance by augmenting fuel plume penetration depth, dispersion, and macromixing to achieve stable, self-reinforcing flame-holding within dual recessed cavity flameholder.

2. Literature Review and objective: -

Implement a passive geometric flow control method to mitigate shock-boundary layer interactions (SBLI) in a Mach 4 mixed-compression supersonic intake. High-resolution numerical simulations utilizing a specialized turbulence model analyse downward, mid-surface, and upward-protruding ramp grooves. Upward-protruding configurations generate stable sub-boundary layer counter-rotating vortex pairs. These vortices re-energize the boundary layer, collapsing the low-momentum throat separation bubble by 83% and lowering downstream flow distortion by 30% without substantial aerodynamic drag penalties.[1]

Examine transient flow fields and flame stabilization within a supersonic scramjet combustor utilizing dual fuel-injector arrays and opposing cavity flameholders. Transient numerical simulations track the evolution of oblique shock fronts, expansion fans, and shear layers. Downstream backpressure drives fuel-induced bow shocks upstream, increasing cavity pressure and inducing localized wall boundary layer separation. This detachment enhances fuel plume penetration, macromixing, and supersonic stream blending, establishing a self-sustaining wave-chemistry feedback loop for stable combustion anchoring.[2]

Investigate time-dependent flow fields and combustion stabilization within a scramjet assembly featuring face-to-face recessed flame-anchoring cavities and twin fuel injection manifolds. Transient computational fluid dynamics chart the internal flow structures during initial fuelling phases toward equilibrium. Downstream pressure rises force shock fronts upstream, elevating pocket pressure and inducing boundary layer separation. This localized detachment benefits performance by augmenting the penetration depth, dispersion, and mixing of primary and secondary fuel plumes via highly coupled gas-dynamic wave oscillations.[3]

Present a multi-objective aerodynamic optimization of a Mach 5 hypersonic intake ramp. Utilizing a Reynolds-Averaged Navier-Stokes (RANS) framework, the computational model evaluates various turbulence models, validating the setup with a fine mesh and the (k)-(omega) SST model. Parameterizing the ramp geometry via Bézier splines allows automated shape generation. Coupled with a genetic algorithm, the optimization evaluates generated designs to minimize the aerodynamic drag coefficient while maximizing the static and total pressure recovery ratios.[4]

Experimentally evaluate wall ventilation techniques to control shock-wave/boundary-layer interactions (SBLI) within a Mach 2.2 mixed-compression supersonic intake. Integrating a shallow cavity beneath a porous surface near the cowl-shock impingement zone aims to mitigate shock-induced flow separation and stagnation pressure losses. Testing variations in contraction ratios and wall porosities via static pressure measurements and Schlieren flow visualization demonstrates that an 18.8% porous surface successfully minimizes separation bubble length, shock wave intensity, and wall static pressure.[5]

Conduct a numerical investigation of a two-dimensional Mach 5 scramjet inlet-isolator to enhance total pressure recovery. Utilizing a density-based solver with the SST (k)-(omega) turbulence model, the authors evaluate shock wave-boundary layer interactions, boundary layer separation, and lip-shock-induced detachment. The study validates the solver against experimental benchmarks before independently assessing two discrete flow-control strategies: passive boundary layer bleed gaps and varying cowl deflection angles, ultimately combining them into an optimized integrated configuration.[6]

Develop a non-intrusive luminescent measurement system utilizing pressure-sensitive paint (PSP) to characterize a generic Mach 5 scramjet inlet-isolator. Oxygen-quenched luminophores generate comprehensive global surface pressure maps across the compression ramp and isolator sidewalls, validated via discrete transducer readings. Supplemented by colour Schlieren visualization and (k)-(omega) SST CFD simulations across multiple angles of attack (AoA), the methodology captures complex three-dimensional shock-shock interactions and separation bubbles to accurately calculate downstream compression and kinetic energy efficiencies.[7]

3. Methodology: -

This research transitions into numerical modelling using ANSYS Fluent R1 2021 to investigate hypersonic and high-supersonic regimes at Mach 4 and above. The working fluid is modelled as an ideal gas with specified thermophysical properties. To replicate actual conditions within a high-speed wind tunnel, the freestream parameters are configured with a static pressure of 52000 Pascal and a static temperature of 300 Kelvin.

The governing equations implemented in these simulations are the Reynolds-Averaged Navier–Stokes (RANS) equations, which encompass the conservation of mass (continuity), momentum (Navier-Stokes), and energy. To resolve turbulent behaviours accurately, the Shear Stress Transport (SST) k- ω model is chosen. As a robust two-equation closure model, it is highly reliable for predicting turbulence across diverse flow fields, particularly in high-velocity aerodynamics.

This framework operates via a dual-transport mechanism solving for both turbulent kinetic energy (k) and the specific dissipation rate (omega). This specific model is preferred because of its superior near-wall treatment, a critical requirement when evaluating boundary layer phenomena in supersonic and hypersonic flows.

A density-based solver is utilized to effectively resolve the highly compressible flow characteristics inherent to high Mach numbers and Sutherland viscosity. Furthermore, spatial discretization of the system is executed utilizing a second-order upwind scheme to ensure high-fidelity resolution of shock waves and gradient changes.

Sutherland's three-coefficient method determines dynamic viscosity, coupling it to temperature variations for realistic ideal gas modelling. Additionally, a rigorous grid independence study and solver validation were conducted to confirm the accuracy and reliability of ANSYS Fluent for these high-speed simulations.

3.1 Solver validation

The following subsections detail the inlet geometry and boundary conditions utilized to validate the numerical solver, followed by the grid independence analysis.

3.2 Base line Geometry and other Novel Design Geometry

To validate the numerical framework, the 2D scramjet inlet-isolator configuration from [8] was adopted as the baseline model for all subsequent simulations.

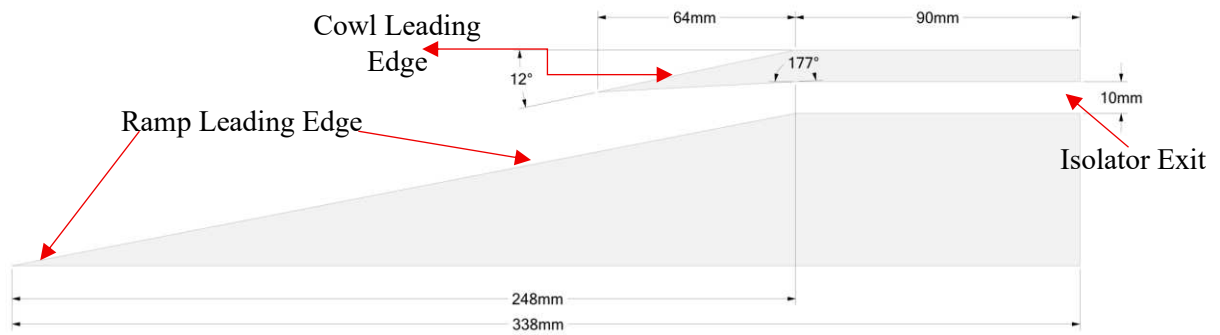
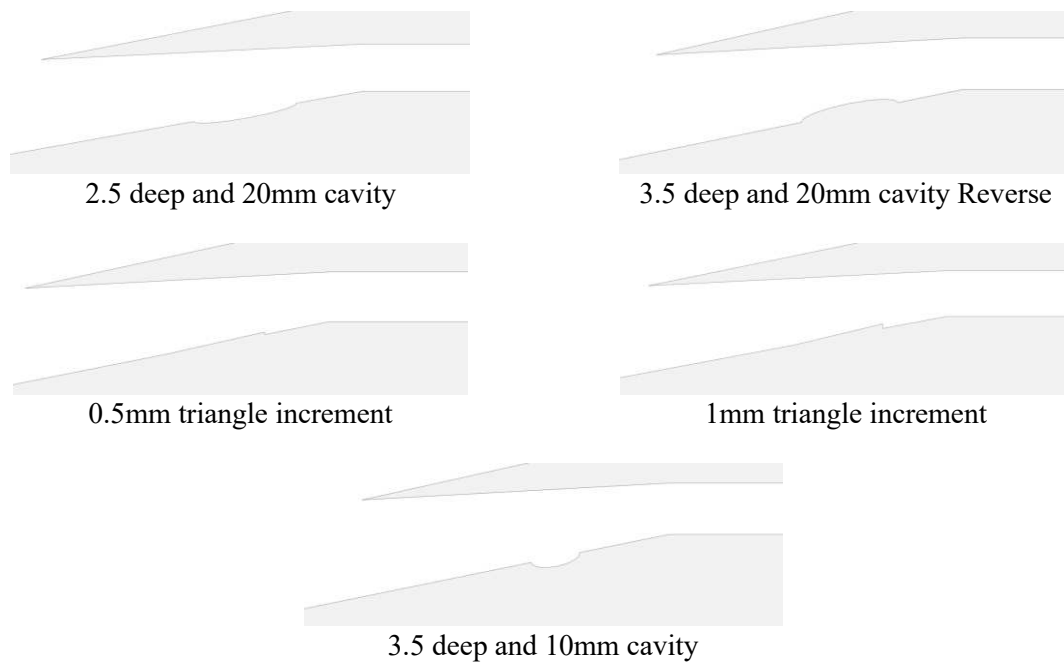


Fig no: - 2 Base line Geometry



3.3 Typical Mesh and Boundary Condition

The computational domain was discretized using a structured mesh with a refined face mesh applied to the scramjet inlet-isolator region. As shown in the fig 3, the mesh is highly dense around the ramp, cowl, and internal flow passage to accurately resolve shock waves, boundary-layer development, and strong pressure and temperature gradients. Away from the inlet-isolator, the mesh becomes comparatively coarser, reducing computational cost. This meshing strategy is suitable for Mach 4 to flow conditions with 52000 Pa pressure and 300 K temperature. The mesh density an efficient balance between computational numerical accuracy. This refined mesh helps obtain reliable distributions of pressure, temperature, velocity, and Mach number throughout the computational domain.

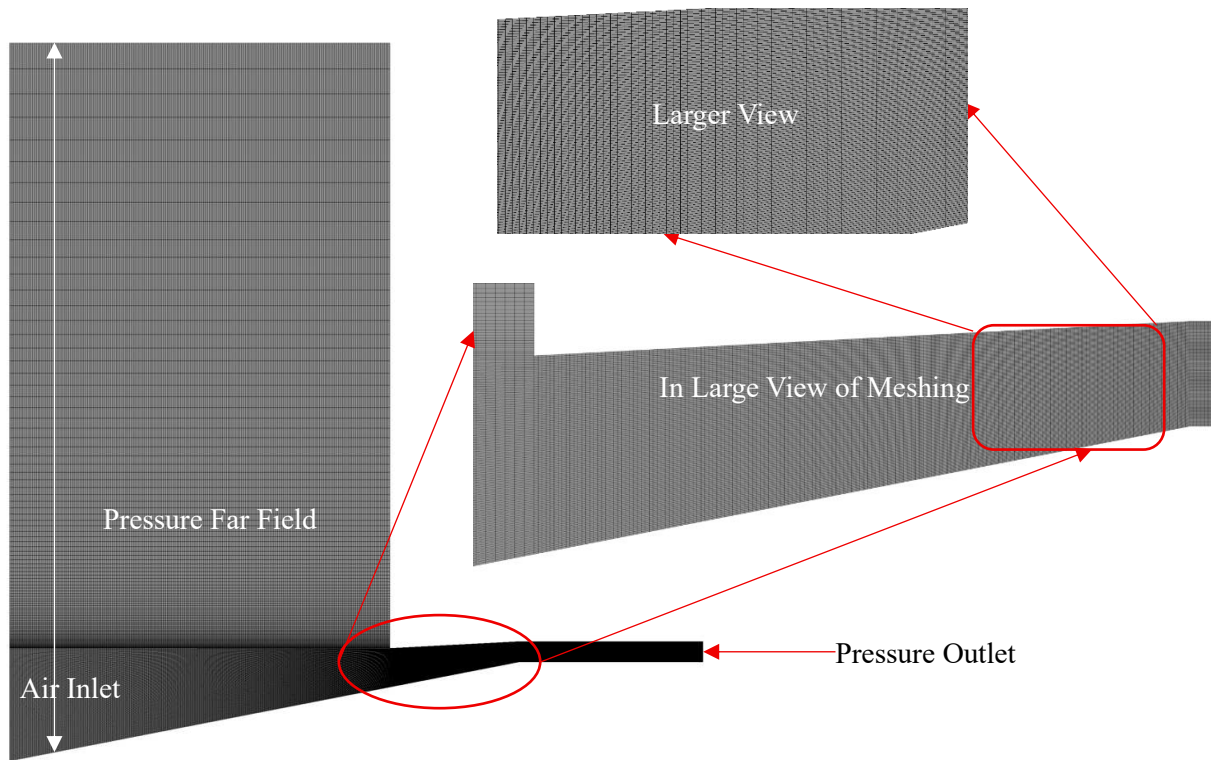


Fig: - 3 Detailed mesh and boundary types

3.4 Grid Independence Test

The grid independence test (GIT) was conducted using varying numbers of elements: coarse (50 thousand elements), medium (80 thousand elements), fine (1.2 lakh elements), and fine (1.5 lakh elements) and more. The configuration remained constant while simulations varied the number of mesh elements, as shown in Fig. 4. Results indicated that the fine mesh closely matches with the computational and experimental data, ensuring accuracy with a wall y^+ value below 1, also facilitating the fair comparison with paper.

Table no: -1

S. no.	Mesh Type	Number of Element
1.	Very Coarse	50,000
2.	Coarse	80,000
3.	Medim	1,20,000
4.	Fine	1,50,000
5.	Very Fine	2,00,000

The resulting table data revealed that the Fine mesh configuration provides the most optimal balance, as its outputs closely match both the established computational models and the available experimental validation data. Furthermore, this specific fine grid density successfully ensures high-fidelity accuracy, which is essential for capturing precise boundary layer flow physics. Consequently, selecting this grid resolution facilitates a rigorous, fair comparison with reference papers while optimizing computational efficiency during the entire research project. This systematic validation process ultimately guarantees that all future simulation runs remain dependable, robust, and highly accurate.

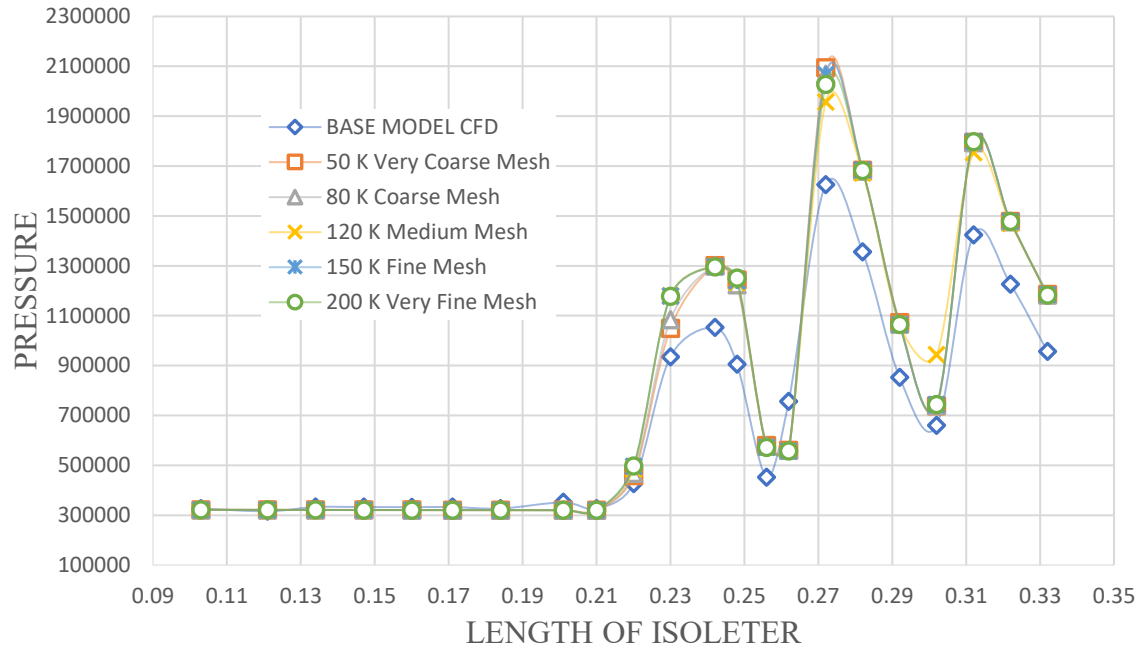


Fig: - 4 GIT comparison graph of static pressure distribution on the ramp surface

4 Results and Discussion

The present investigation focuses on improving the performance of the inlet isolator by incorporating a passive bleed system and evaluating the influence of different cowl deflection angles on its aerodynamic characteristics. In addition, an integrated configuration is developed by combining the most effective designs identified from the passive bleed and cowl deflection angle investigations into a single geometry. The performance of this combined configuration is then numerically assessed to determine its overall effectiveness. This integrated design approach is intended to achieve an improved inlet configuration capable of enhancing engine performance while maintaining better pressure recovery.

4.1 The effect of cavity and increment design on scramjet intake

The effect of cavity and incremental ramp configurations on the aerodynamic performance of a scramjet intake ramp is investigated through CFD analysis. The cavity modifies the local flow field by inducing recirculation and shear-layer interactions, while the incremental ramp geometry provides controlled supersonic compression through successive changes in ramp angle. Their influence on shock-wave structure, pressure distribution, Mach-number variation, boundary-layer behaviour, and pressure recovery can be evaluated to determine the effectiveness of the modified intake configuration.

4.2 Effect of flow characteristics on Pressure; Mach contours

In high-speed aerodynamic applications such as scramjet inlets, the integration of passive bleed systems can substantially modify the flow field and improve overall aerodynamic behaviour. A passive bleed arrangement is mainly employed to regulate boundary-layer development and control shock-wave interactions within the inlet. Through strategically positioned bleed holes, high-pressure air is allowed to discharge from the flow passage, thereby decreasing the size and intensity of separation regions generated by lip-induced shocks along the ramp. The bleed system also helps alleviate the effects of strong shock waves within the isolator by releasing excess pressure and weakening shock-induced flow separation. The resulting changes can be observed through Mach number contours and pressure distribution profiles, which generally indicate a more uniform flow structure and improved pressure recovery toward the isolator exit.

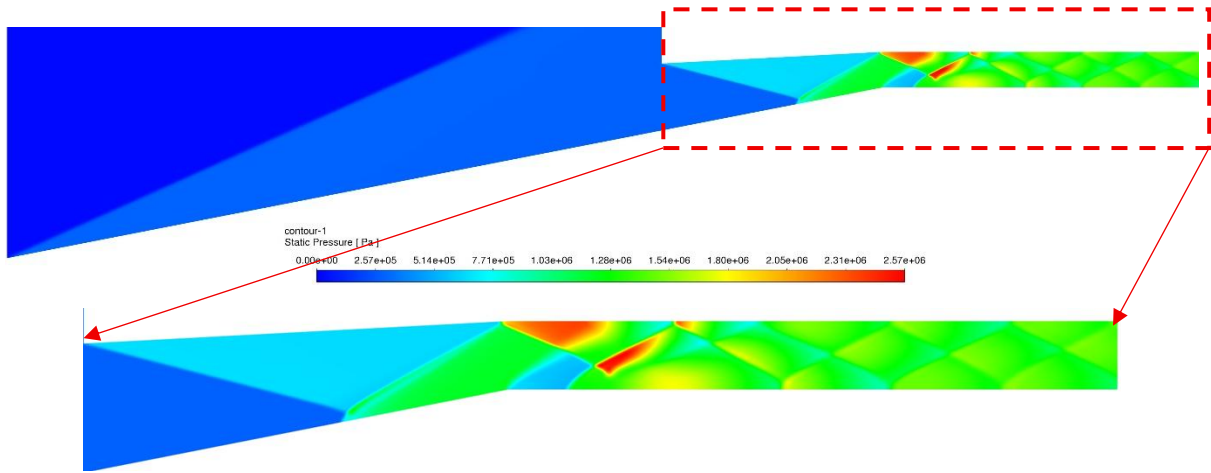


Fig no: - 5 Base model pressure contour at 4 Mach

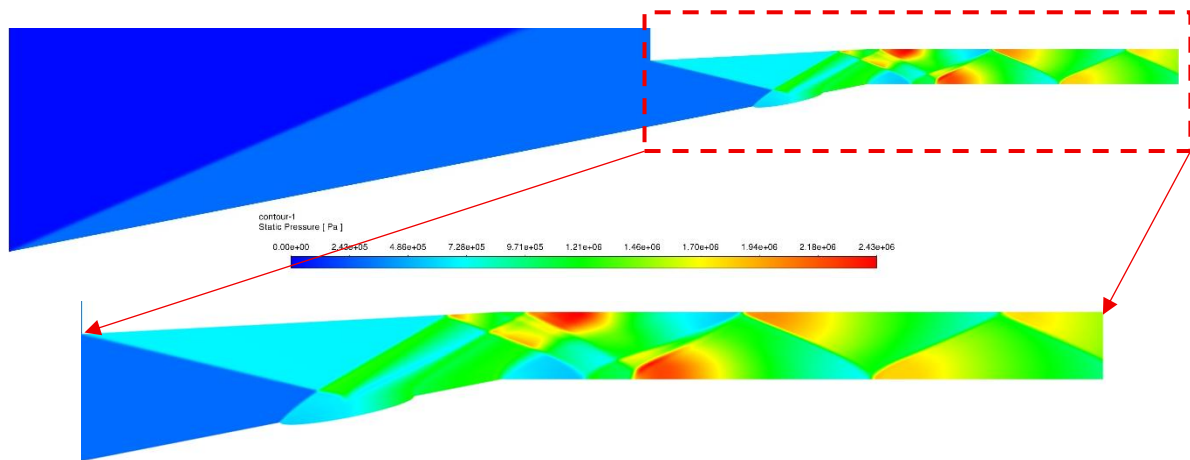


Fig no: - 6 2.5 deep and 20mm cavity model pressure contour at 4 Mach

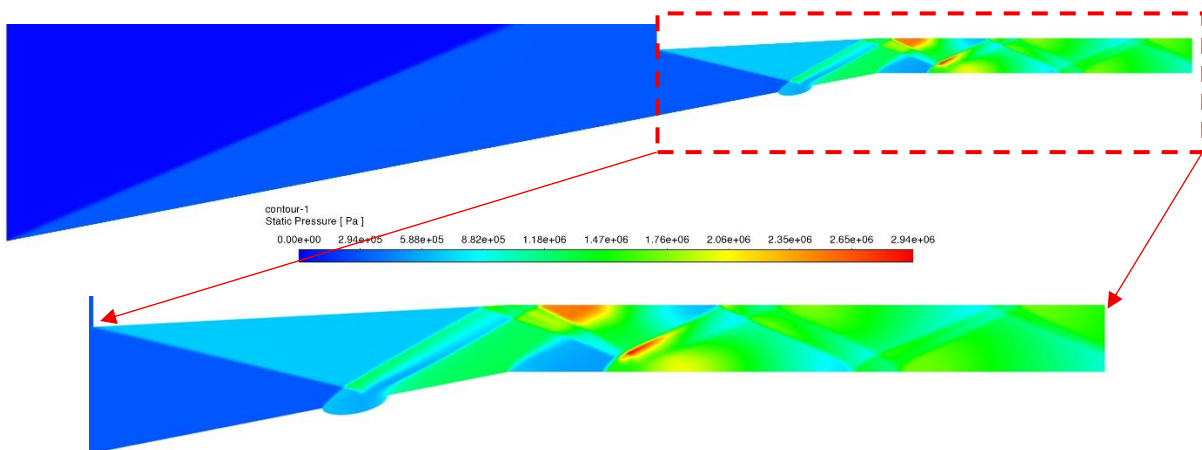


Fig no: - 7 3.5 deep and 10mm cavity model pressure contour at 4 Mach

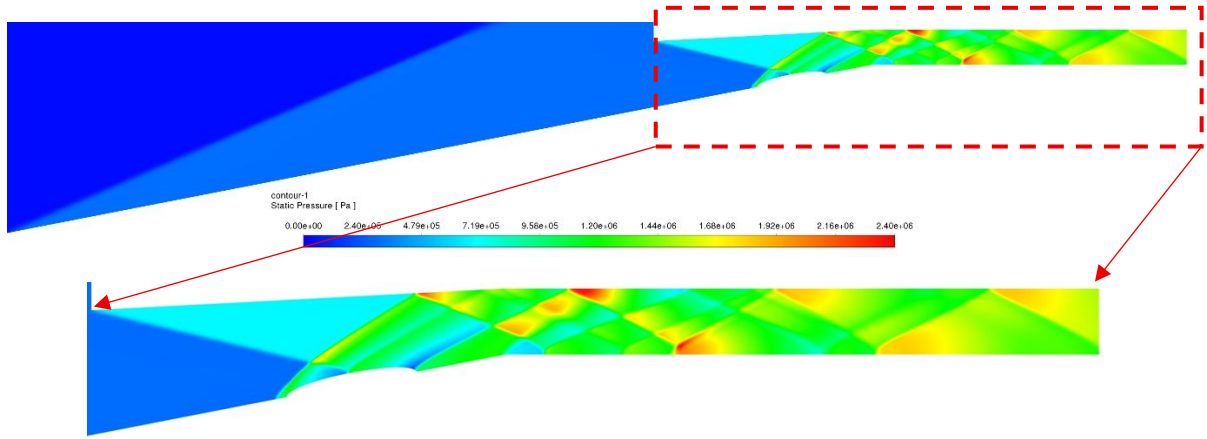


Fig no: - 8 3.5 deep and 20mm cavity Reverse model pressure contour at 4 Mach

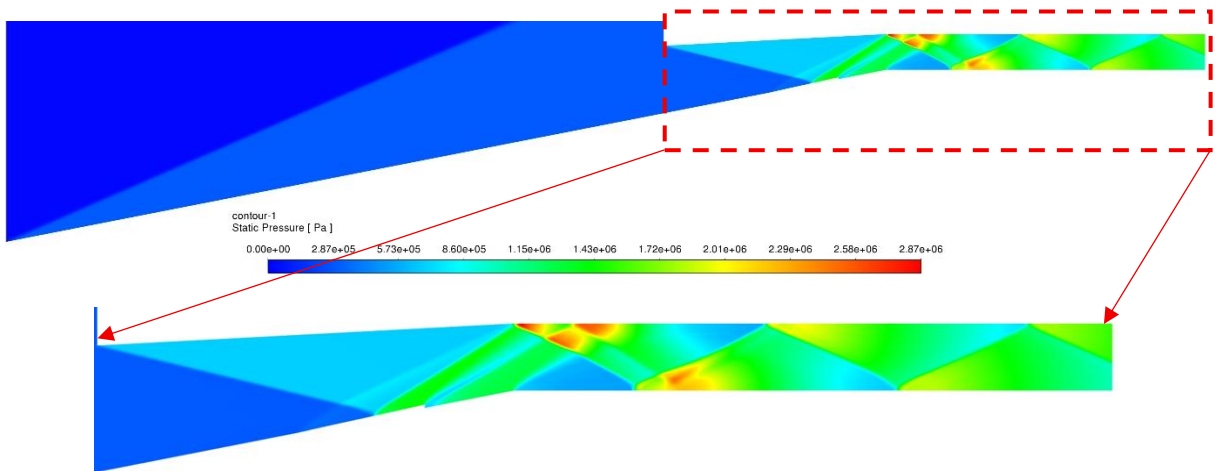


Fig no: - 9 0.5mm triangle increment model pressure contour at 4 Mach

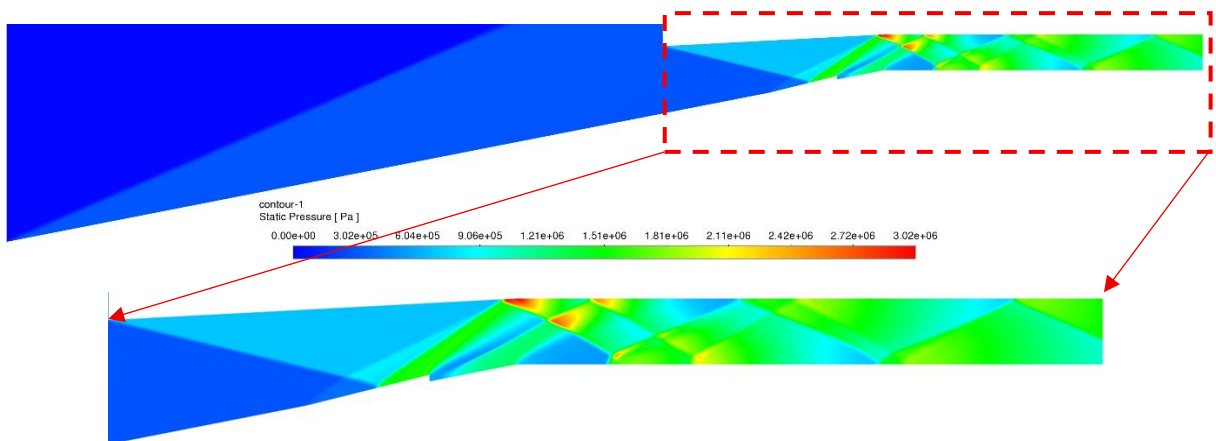


Fig no: - 10 1mm triangle increment model pressure contour at 4 Mach

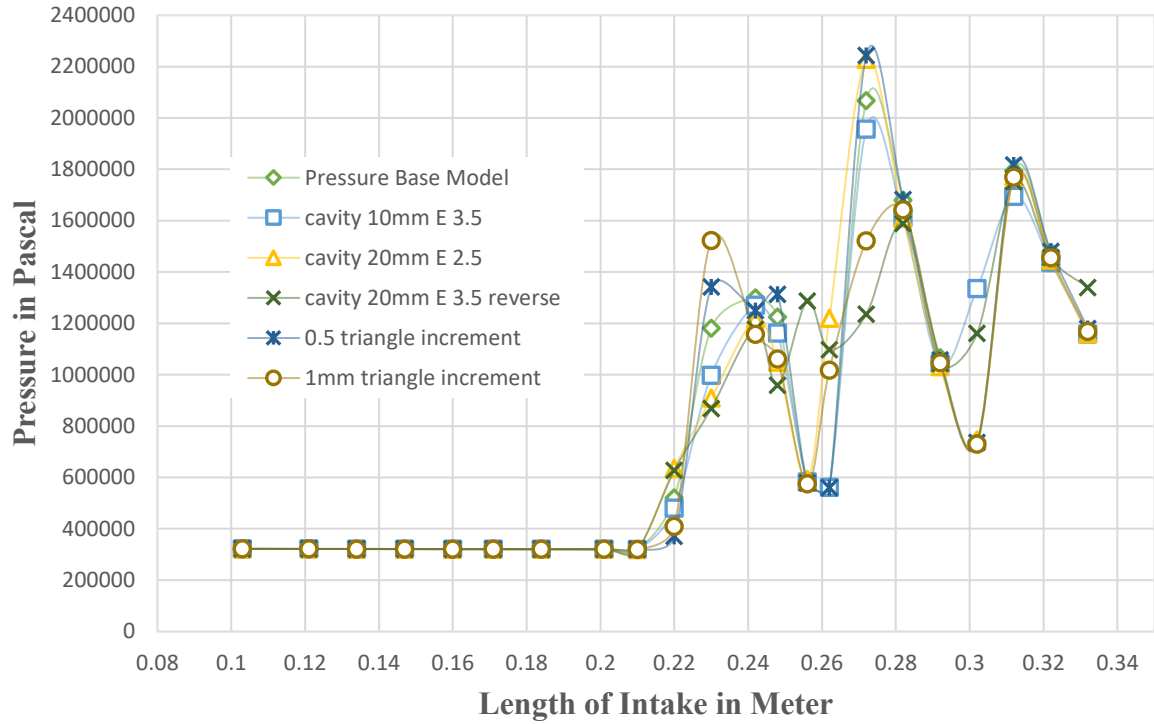


Fig no: - 11 Comparative Analysis of Intake Pressure for Cavity and Triangle Modifications

This graph represents a Computational Fluid Dynamics (CFD) analysis evaluating the internal flow field and static pressure distribution of a supersonic/hypersonic intake operating at Mach 4. The aerodynamic performance of a baseline model (BM) is systematically compared against multiple passive flow control configurations. These include open cavities (10mm and 20mm depths with expansion ratios $E=2.5$ and 3.5 , plus a reverse cavity setup) and micro-triangular increments (0.5mm and 1mm). The flow visualization contours resolve complex shock-wave/boundary-layer interactions (SWBLI) and the subsequent formation of a shock train within the isolator section. Upstream of the normalized intake length ($x < 0.20$), the flow field remains unperturbed, exhibiting a uniform static pressure across all configurations. Downstream ($x > 0.20$), the onset of oblique shock reflections induces a severe adverse pressure gradient. This triggers highly oscillatory pressure profiles with pronounced peaks localized between $x = 0.27$ and $x = 0.32$. The variations in peak magnitudes indicate that integrating cavities and micro-geometry increments effectively modulates shock topology and boundary layer separation. Consequently, these structural modifications offer a viable mechanism for optimizing pressure recovery and mitigating flow instability in high-speed propulsion systems.

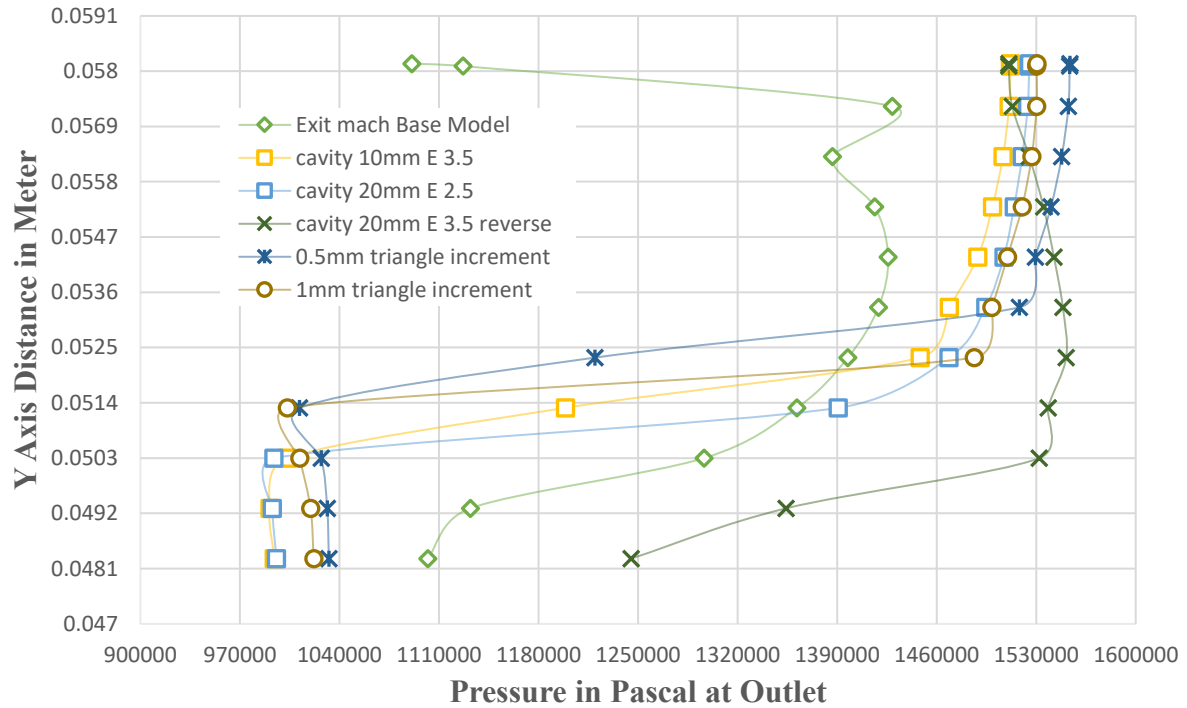


Fig no: - 12 Outlet pressure Comparative Analysis at Exit for Cavity and Triangle Modifications

The Exit outlet pressure reveals distinct behaviours along the intake height from 0.0483 m to 0.0582 m, demonstrating a clear trade-off between the geometric modifications and pressure recovery. In the upper core region ($Y > 0.053$ m), the baseline model exhibits the lowest exit static pressure, fluctuating around 1.38 to 1.43 MPa. In stark contrast, all modified configurations substantially elevate the core pressure, creating a highly uniform plateau around 1.50 to 1.55 MPa. The lower wall region ($Y < 0.052$ m) introduces a drastic inversion in performance metrics. While most cavity configurations and triangle increments see their pressures drop sharply to a minimum baseline plateau around 1.00 MPa, the "cavity 20mm E 3.5 reverse" model acts as a major outlier. It maintains an exceptionally high exit pressure profile throughout the lower shear layer, staying above 1.25 MPa and peaking near 1.54 MPa at $Y = 0.0505$ m. This highly uneven pressure profile at the lower bounds indicates that while the reverse cavity modification achieves superior overall pressure magnitude at the exit plane, it introduces greater flow non-uniformity across the vertical height compared to the highly stabilized micro-triangular configurations.

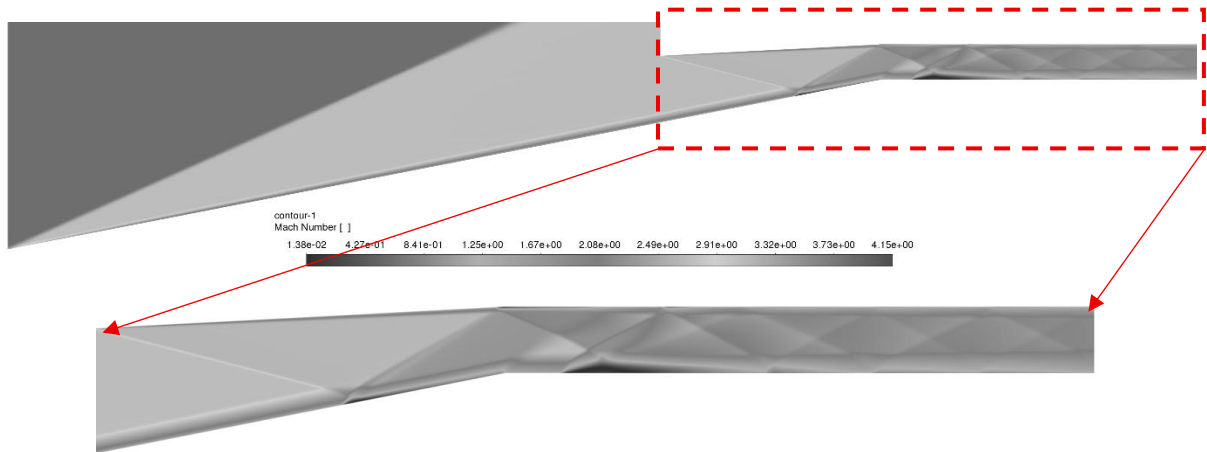


Fig no: - 13 Base model Mach contour at 4 Mach

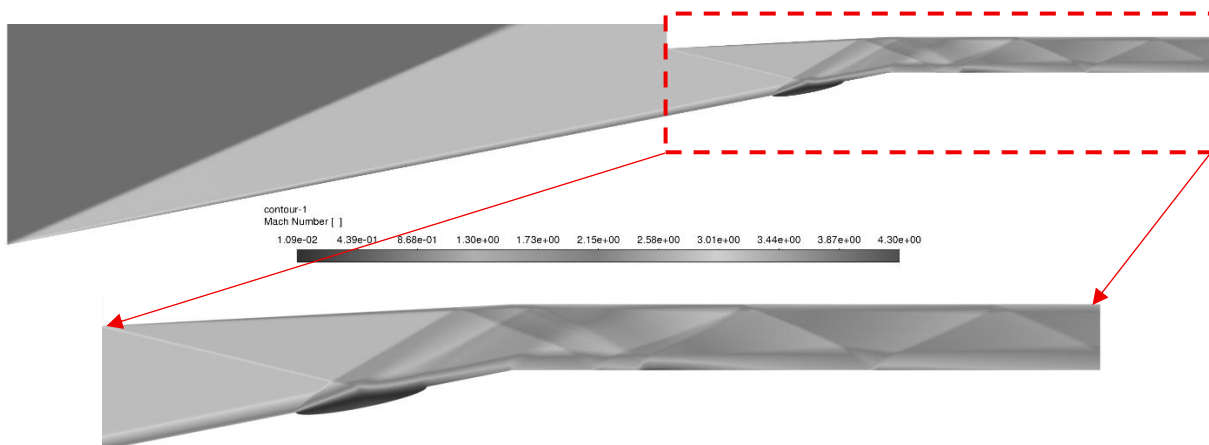


Fig no: - 14 2.5 deep and 20mm cavity model Mach contour at 4 Mach

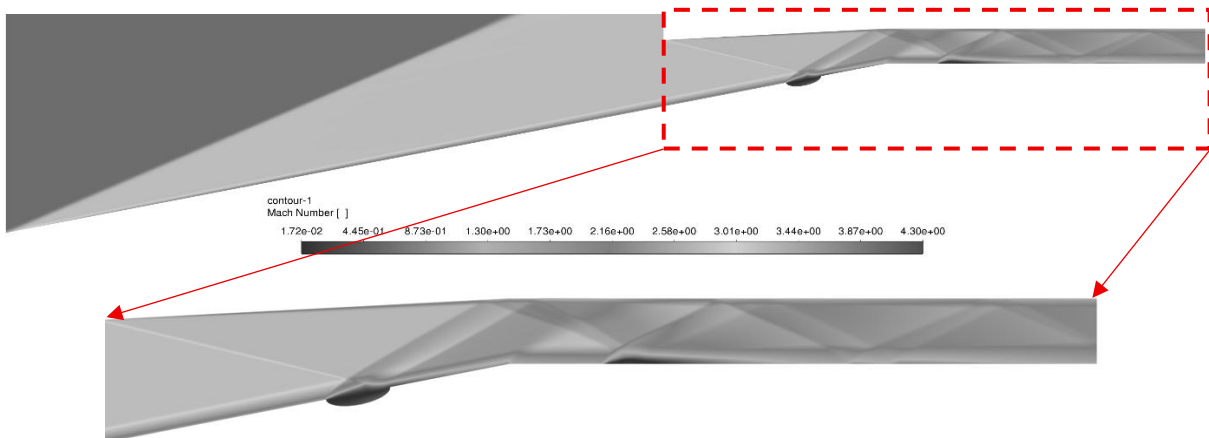


Fig no: - 15 3.5 deep and 10mm cavity model Mach contour at 4 Mach

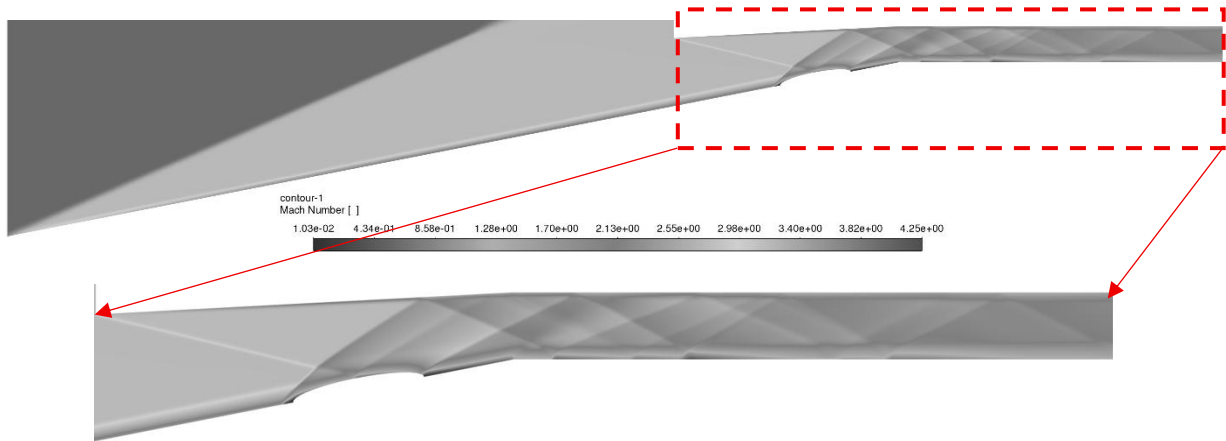


Fig no: - 16 3.5 deep and 20mm cavity Reverse model Mach contour at 4 Mach

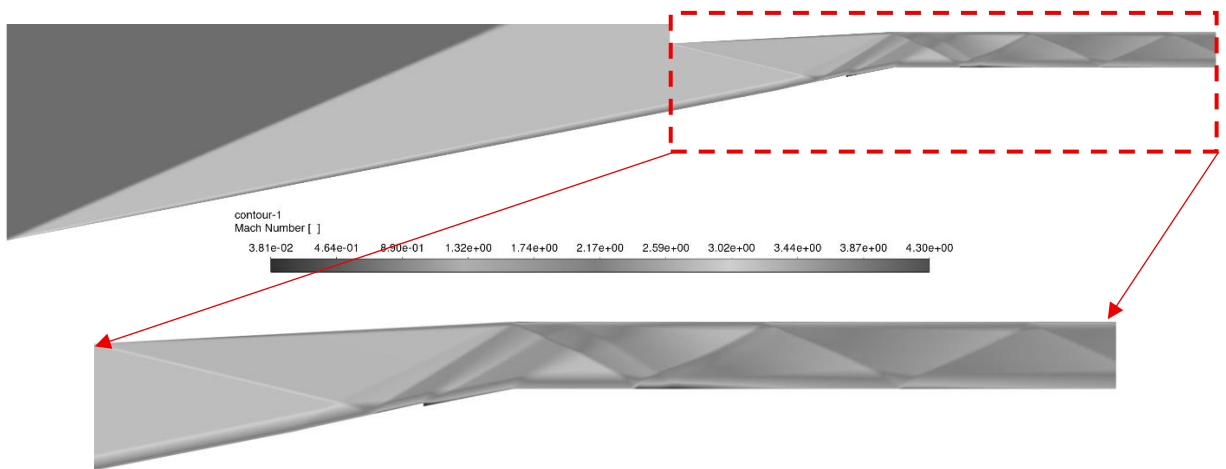


Fig no: - 17 0.5mm triangle increment model Mach contour at 4 Mach

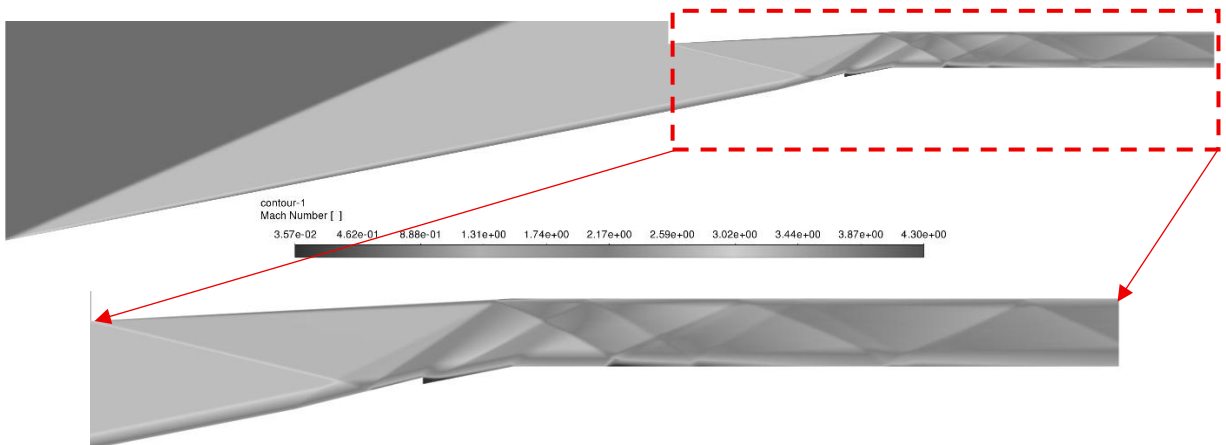


Fig no: - 18 1mm triangle increment model Mach contour at 4 Mach

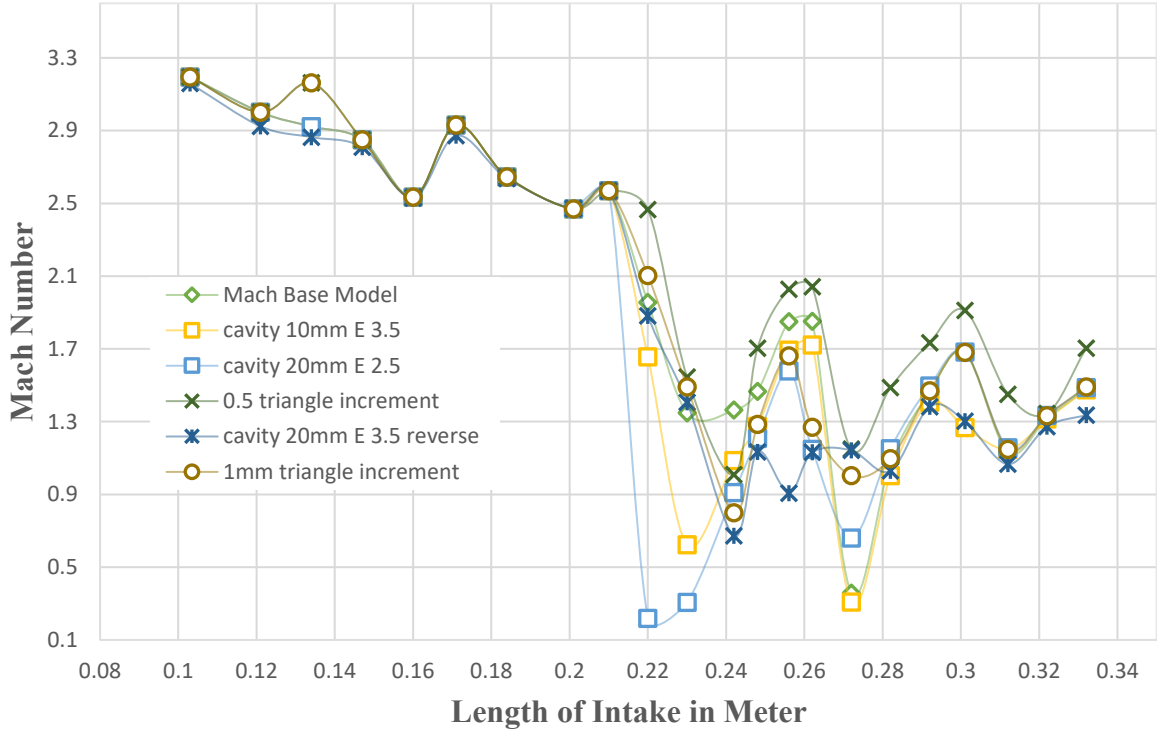


Fig no: - 19 Comparative Analysis of Intake Mach Number for Cavity and Triangle Modifications

The Mach contour plots resolve the development of a complex oblique shock train that drives flow deceleration through successive shock reflections. Upstream of the normalized intake length ($x < 0.20$), the freestream flow undergoes initial compression, causing the Mach number to transition smoothly from approximately 3.1 down to 2.4 across all models. Downstream ($x > 0.20$), strong shock-wave/boundary-layer interactions (SWBLI) trigger severe flow expansion and re-compression zones. This behaviour is evidenced by a sharp drop in the core flow velocity, with local Mach numbers plunging to minimum values between 0.6 and 1.2 within the $x = 0.22$ to 0.28 zone. The highly oscillatory velocity recovery beyond $x = 0.28$ highlights how geometric modifications alter shock reflection angles and boundary layer growth, serving as an effective means to manage flow kinetic energy and prevent intake unstart.

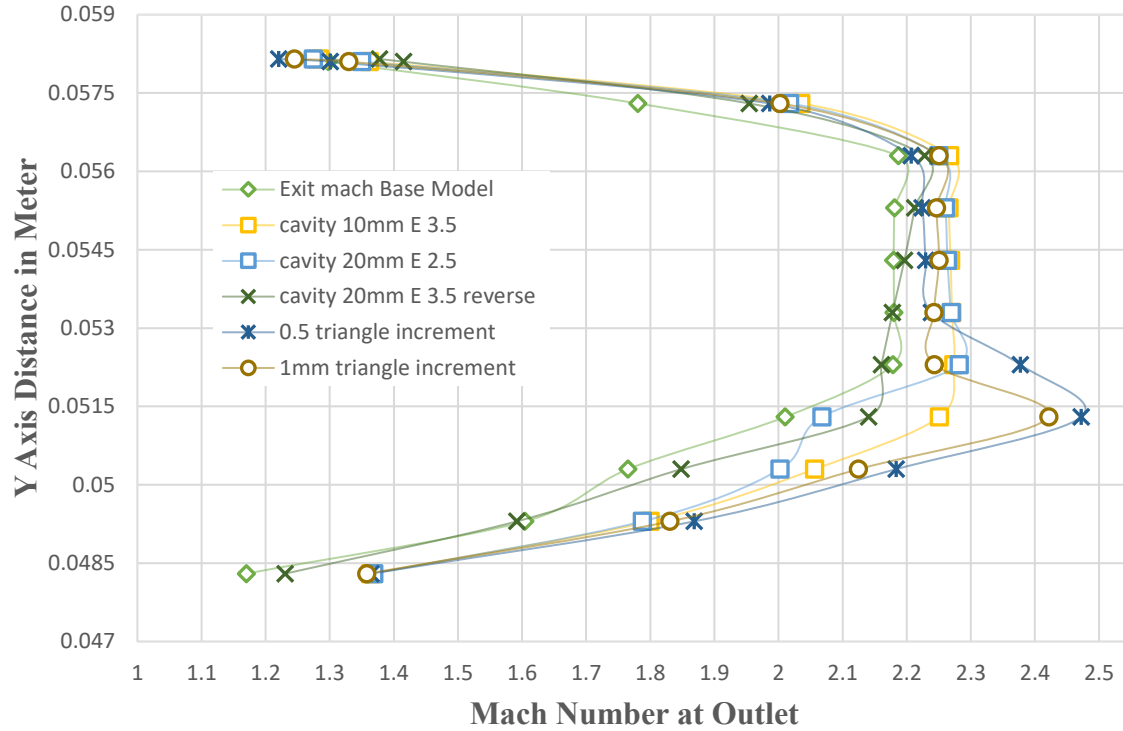


Fig no: - 20 Exit Mach Number Comparative Analysis for Cavity and Triangle Modifications

The plotted profiles illustrate the exit flow field characteristics and boundary layer development across the intake height, spanning a Y-axis range from 0.0483 m to 0.0582 m. In the core flow region spanning approximately between $Y = 0.0525$ m and $Y = 0.0565$ m the Mach number exhibits a highly stabilized plateau across all tested models, maintaining steady velocities between 2.18 and 2.28. However, severe flow gradients develop near the upper and lower wall boundaries due to strong viscous interactions. Near the lower wall region ($Y < 0.0520$ m), the models incorporating passive controls specifically the 0.5mm and 1mm micro-triangular increments demonstrate an aggressive local acceleration, tracking peak localized Mach numbers up to 2.48 before sharply dropping near the surface. Conversely, the "cavity 20mm E 3.5 reverse" and the "exit Mach base model" yield significantly lower Mach values throughout the lower shear layer, plunging down to 1.16 - 1.23 near the bottom wall. This variation highlights how localized passive geometric modifications can heavily re-energize the boundary layer, altering velocity gradients and uniformity at the intake exit.

5. Conclusion

Based on the CFD Based Geometry Re-Design & Performance Analysis of a Scramjet Intake file, the comparison should focus on pressure recovery, pressure uniformity, shock structure, Mach-number distribution, and exit-flow uniformity. The study compares the baseline with 10/20 mm cavity configurations, reverse cavity, and 0.5 & 1 mm triangular increments.

- The modified configurations increase the core exit pressure from the baseline's approximately 1.38 - 1.43 MPa to about 1.50 - 1.55 MPa.
- The reverse 20 mm cavity, $E = 3.5$ produces high pressure near the lower wall, reaching approximately 1.54 MPa, but the file specifically notes that it produces greater pressure non-uniformity.
- At the outlet, the core-flow Mach number is relatively stable for the tested configurations at approximately 2.18 - 2.28.
- The 0.5 mm and 1 mm triangular designs produce local acceleration up to approximately Mach 2.48, whereas the reverse cavity and baseline fall to about Mach 1.16 - 1.23 near the lower wall.

5.1 Pressure Distribution Detail

The pressure distribution inside the scramjet intake is strongly influenced by the cavity geometry and triangular increment configurations because these modifications change the development of shock waves and their interaction with the boundary layer. In the baseline configuration, the pressure remains relatively low in the upstream region and increases downstream as the flow passes through successive compression and shock-wave interactions. The modified configurations produce a higher-pressure level in the core region, with the outlet pressure generally reaching approximately 1.50 - 1.55 MPa, compared with about 1.38 - 1.43 MPa for the baseline case. The pressure variation becomes more significant after approximately 0.20 m along the intake, where oblique shock reflections generate a stronger adverse pressure gradient. The 20 mm reverse cavity with an expansion ratio of 3.5 produces a high local pressure, reaching approximately 1.54 MPa near the lower wall. However, its pressure distribution is less uniform across the outlet height. In comparison, the triangular increment configurations provide a more controlled pressure distribution while maintaining the increased core pressure. These results indicate that geometric modifications can effectively alter shock strength, reduce undesirable flow separation effects, and improve the compression process. Therefore, pressure recovery should be evaluated not only from the maximum pressure obtained but also from the uniformity of the pressure distribution at the intake exit.

5.2 Mach Number Distribution Detail

The Mach-number distribution provides an important indication of how effectively the modified intake controls the supersonic flow and manages shock-wave interactions. At the upstream section of the intake, all configurations show a similar reduction in Mach number as the incoming flow undergoes initial compression. The Mach number decreases smoothly from approximately 3.1 to 2.4 before the flow reaches the stronger shock-interaction region. Beyond approximately 0.20 m, the flow experiences repeated shock reflections and shock-boundary-layer interactions, producing significant variations in the local Mach number. In the region between approximately 0.22 and 0.28 m, the local Mach number decreases to values between about 0.6 and 1.2, indicating strong flow deceleration caused by the shock system and boundary-layer interaction. At the intake exit, the core-flow region becomes considerably more stable, with Mach numbers generally remaining between approximately 2.18 and 2.28. The 0.5 mm and 1 mm triangular increment configurations produce local acceleration near the lower wall, with peak Mach numbers reaching approximately 2.48. In contrast, the reverse 20 mm cavity and baseline configuration show lower Mach numbers near the lower wall. These results demonstrate that passive

geometric modifications can control shock reflections, influence boundary-layer development, and redistribute flow kinetic energy. The triangular configurations therefore provide a controlled mechanism for modifying the exit-flow characteristics.

Table no: - 1

S. no.	Design	Pressure behaviour	Mach behaviour	Flow uniformity
1.	Baseline	Lower core pressure	Strong low-Mach region near wall	Reference
2.	10 mm cavity, E=3.5	Higher exit pressure	Modified shock structure	Improved pressure recovery
3.	20 mm cavity, E=2.5	Higher exit pressure	Modified shock structure	Improved compression
4.	20 mm cavity, E=3.5 reverse	Very high local pressure	Low Mach near lower wall	High pressure, less uniform flow
5.	0.5 mm triangle increment	Higher pressure	Local acceleration up to -2.48	Strong near-wall acceleration
6.	1 mm triangle increment	Higher pressure	Local acceleration up to -2.48	Strong near-wall acceleration

Among the investigated passive geometric modifications, the micro-triangular increment configurations demonstrated favourable pressure recovery and controlled Mach-number development at the intake exit. Compared with the cavity configurations, the triangular modifications produced a relatively stabilized core flow while simultaneously re-energizing the near-wall region. The 0.5 mm and 1 mm configurations therefore provide promising candidates for further optimization.

6. Reference

1. Y. A. Gujar *et al.*, "Passive Flow Control of Mach 4 Mixed-Compression Intake Using Chevron Grooves," *FME Transactions*, vol. 54, no. 1, pp. 53–69, 2026. https://www.mas.bg.ac.rs/_media/istrazivanje/fme/vol54/1/5_n.k._gahlot_et_al.pdf
2. J. C. Small, L. Zhang, B. G. Crawford, and V. Viti, "Flow and Flame Stabilization in Scramjet Engine Combustor with Two Opposing Cavity Flameholders," *Aerospace*, vol. 12, no. 8, p. 723, Aug. 2025. <https://doi.org/10.3390/aerospace12080723>
3. S. Emami, C. A. Trexler, A. H. Auslender, and J. P. Weidner, "Experimental Investigation of Inlet-Combustor Isolators for a Dual-Mode Scramjet at a Mach Number of 4," NASA Langley Research Centre, Hampton, VA, USA, NASA Tech. Paper 3502, May 1995. <https://ntrs.nasa.gov/api/citations/19950021922/downloads/19950021922.pdf>
4. F. De Vanna, D. Bof, and E. Benini, "Multi-Objective RANS Aerodynamic Optimization of a Hypersonic Intake Ramp at Mach 5," *Energies*, vol. 15, no. 8, p. 2811, Apr. 2022. <https://doi.org/10.3390/en15082811>

5. H. Gunasekaran, T. Thangaraj, T. Jana, and M. Kaushik, "Effects of Wall Ventilation on the Shock-wave/Viscous-Layer Interactions in a Mach 2.2 Intake," *Processes*, vol. 8, no. 2, p. 208, Feb. 2020. <https://doi.org/10.3390/pr8020208>
6. D. K. Bajaj, S. T. Kansara, N. K. Nath, A. Vashishtha, and D. Sahoo, "Enhancement of scramjet inlet efficiency by geometrical modifications," *Advances in Aerodynamics*, vol. 7, no. 1, Art. no. 6, Apr. 2025. <https://doi.org/10.1186/s42774-025-00206-7>
7. C. Idris, M. R. Saad, H. Zare-Behtash, and K. Kontis, "Luminescent Measurement Systems for the Investigation of a Scramjet Inlet-Isolator," *Sensors*, vol. 14, no. 4, pp. 6606–6632, Apr. 2014. <https://doi.org/10.3390/s140406606>