



BASE FLOW CONTROL THROUGH A TRIANGULAR CAVITY WITH SUDDEN EXPANSION AT SUPERSONIC MACH NUMBER FOR LOWER AREA RATIO

**Shamitha Shetty¹, Sher Afghan Khan², Ambareen Khan³, Mohammad
Nishat Akhtar⁴, Elmi Abu Bakar⁵, Khizar Ahmed Pathan⁶**

¹ Department of Mathematics, Nitte (Deemed to be University), Nitte
Meenakshi Institute of Technology (NMIT), Department of Mathematics,
Bengaluru-560064, Karnataka, India

² Department of Mechanical & Aerospace Engineering, Faculty of
Engineering, IIUM, Kuala Lumpur, Malaysia

^{3,4,5} School of Aerospace Engineering, Universiti Sains Malaysia, Tuanku Syed
Sirajuddin Engineering Campus, 14300 Nibong Tebal, Penang, Malaysia,

⁶ CSMSS Chh. Shahu College of Engineering, Chhatrapati Sambhajinagar
(Aurangabad) 431011, Maharashtra, India,

Email: ¹shamithashetty092@gmail.com, ²sakhan@iium.edu.my,
³khanambareen5@gmail.com, ⁴nishat@usm.my, ⁵meelmi@usm.my,
⁶kapathan@csmsseengg.org

Corresponding Author: **Mohammad Nishat Akhtar**

<https://doi.org/10.26782/jmcms.2026.10.00007>

(Received: July 16, 2026; Revised: September 24, 2026; Accepted: October 15, 2026)

Abstract

This investigation examines passive control tactics for administering base pressure at Mach $M = 1.8$. Because the base pressure in the separated section is reduced and below the back pressure, it contributes a sizable portion of the total drag. Hence, we must restrain or regulate this parameter. Consequently, we use base drag to extend the range of aerospace vehicles and save energy. We select expansion levels to expose the nozzle to both adverse and favorable pressure conditions, allowing us to evaluate the control mechanism's efficacy. We select three cavity geometries to evaluate the flow-regulation mechanism by placing the cavities at various positions across a range of duct sizes. The simulated results show that the cavity is effective only when the viscous layer exiting the nozzle enters the cavity; otherwise, the control is ineffective, regardless of whether the reattachment point occurs before or after the cavity. The present investigation shows that the cavity is most effective when located 8.5 mm from the base; however, at other cavity locations, its effectiveness is negligible

Shamitha Shetty et al

except at NPRs 3 and 5. At these two NPRs, the exiting flow is associated with oblique shock waves and an adverse pressure gradient, which reduces base pressure.

Keywords: Triangular Cavity, Base Flow, Area Ratio, Supersonic Mach Number, Duct Size

I. Introduction and Background

Sudden expansion or an abrupt area increase is common in automotive and aerospace applications. This investigation focuses on manipulating base pressure and, consequently, the base drag to reduce fossil fuel consumption and mitigate global warming. This research paper examines control approaches to achieve base pressure at high speed, specifically at Mach 1.8, a screech-prone Mach number. As propulsion advances, missiles and aircraft continually seek to enhance performance and reduce drag to boost aerodynamic efficiency; understanding and controlling base pressure is critical. The abrupt increase in axisymmetric flow is a complex process distinguished by stream partitioning, recirculation, and reconnection. It has two primary regions: the primary flow near the separated stream. The point where the splitting streamline meets the wall is called the reattachment position. A large body of literature addresses rapid-expansion challenges. However, these studies focus on specific flow and geometric features.

Fig. 1 displays the C-D nozzle with a tube. In this investigation, we will explore compressible streams in a 2-D axisymmetric CD nozzle across various duct segments, expansion levels, and cavity locations.

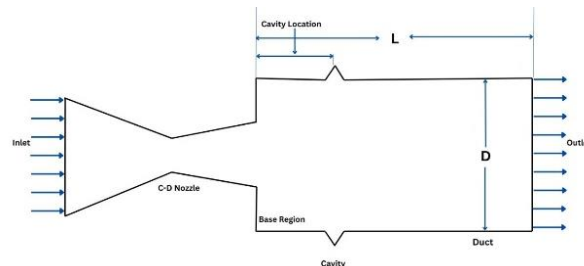


Fig. 1. Nozzle and duct with cavities

An unexpected increase in the flow's cross-sectional area leads to a low-pressure separated region. Research on diffusers and expanding ducts consistently shows that geometry, enlargement ratio, and turbulence properties strongly affect separation and reattachment [XXII, VII, XXVIII, VIII]. Multiphase studies revealed that phase interactions can significantly alter pressure recovery and vortex development. These insights have enabled more reliable models for intermittent and two-phase flows [V]. Further work on gas-solid and multiphase configurations confirms that sudden expansion plays a crucial part in pressure fluctuations and energy dissipation [IV, XX, IX].

In engineering applications such as nozzles, propulsion systems, and internal flows, heat transfer, secondary vortex formation, and inlet conditions strongly affect performance [XXVII]. Meanwhile, pressure recovery and heat transfer remain largely governed by diffuser design and upstream parameters [XXIII]. At high speeds,

Shamitha Shetty et al

expansion levels, gas temperature, and interactions within the viscous layer directly affect base drag [XIX].

Reviews of flow-management schemes assess the effectiveness of static and dynamic methods, such as ribs, cavities, and flow injection, in stabilizing viscous layers and reducing drag [XXI]. Computational approaches such as machine learning and hybrid CFD-ANN models have emerged as practical tools for predicting base pressure. These methods effectively correlate outcomes with inertia levels, expansion levels, flow relief, and tube segments [XXV]. ANN-assisted studies further validate these trends for different aerodynamic configurations [XI]. One review summarizes the influence of ribs, cavities, steps, and tabs, showing that optimal control occurs when devices are located near the natural reattachment point [XII]. By examining the combined influence of NPR, L/D ratio, and rib geometry, this study extends prior findings on passive control [X]. It offers new insights into base-pressure management in high-speed flow.

Baig et al. [VI] and Rehman et al. [XXVI] experimentally controlled base pressure using microjets at various expansion levels and area ratios at high Mach numbers. The findings revealed that the controller remains useful only if the nozzle is under-expanded. The smallest pipe segment required for the flow to be associated with the tube is approximately 1D or 2D for lower area ratios. It is 3D for higher area ratios and higher Mach numbers. Khan et al. [XVI] conducted experiments to regulate the flow field at various Mach numbers, relief, and NPR. Base pressure is strongly affected by expansion and inertia levels, as well as the area ratio. Khan et al. [XV] used passive control techniques to reduce drag by applying dimples to a suddenly expanded duct. In view of the above analysis, it may be concluded that to study base-pressure phenomena at supersonic Mach numbers, where screech tones are possible,

Pandey and Rathakrishnan [XXIV] experimentally investigated base-pressure control at Mach $M = 1.74$ using cavities with dimensions 3:3 and 6:3. They found that cavity depth does not affect base-pressure regulation. In contrast, cavity width plays an influential role in regulating base pressure. They also concluded that if the cavity is placed near the reattachment location and the shear layer enters the cavity, only the cavity becomes effective and regulates the base pressure.

Khan Ambareen et al. [XIV,-XIII] numerically studied the effect of ribs on base pressure and flow development in a suddenly expanded flow at sonic Mach numbers, considering various rib locations and geometries. Akhtar et al. [III] numerically studied a converging-diverging nozzle and the effect of duct length on the flow field and flow development at supersonic Mach numbers. Akhtar et al. [II] studied the impact of microjets on the wall pressure distribution in a suddenly expanded duct at high Mach numbers.

Khan et al. [XVIII] conducted an experimental study using multiple cavities at subsonic Mach numbers. Abid et al. [I] numerically studied sudden expansion at Mach 2.2. Khan et al. [XVII] experimentally studied the impact of expansion levels at various Mach numbers and area ratios.

The current study assesses the effectiveness of different cavity geometries in regulating base pressure and, consequently, base drag, which can be substantial at transonic inertia levels. This study uses three cavity geometries with aspect ratios of 3:3, 3:6, and 6:3.

Shamitha Shetty et al

To the author's knowledge, no one has studied how cavities of different dimensions can manipulate base pressure.

II. Computational Fluid Dynamics

Governing Equations

For this manuscript, we performed the CFD analysis in ANSYS Fluent using the finite volume method (FVM) to simulate turbulent flow through a converging–diverging nozzle with a suddenly expanded duct and corrugated ribs.

The fundamental equations solved in ANSYS Fluent are the governing equations of fluid flow, namely:

Continuity Equation (Mass Conservation)

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \quad (1)$$

where:

- ρ = fluid density
- $\vec{V}$ = velocity vector

This equation guarantees conservation of mass throughout the nozzle and duct domain.

Momentum Equation (Navier–Stokes Equation)

$$\rho \left(\frac{\partial \vec{V}}{\partial t} + \vec{V} \cdot \nabla \vec{V} \right) = -\nabla p + \nabla \cdot \tau + \rho \vec{g} \quad (2)$$

where:

- p = static pressure
- τ = viscous stress tensor
- $\vec{g}$ = gravitational acceleration

This equation governs acceleration, shock formation, recirculation zones, shear-layer interaction, and vortex generation due to the corrugated ribs within the suddenly expanded duct.

Energy Equation

$$\frac{\partial}{\partial t} (\rho E) + \nabla \cdot [\vec{V} (\rho E + p)] = \nabla \cdot (k_{eff} \nabla T + \tau_{eff} \cdot \vec{V}) \quad (3)$$

where:

- E = total energy
- T = temperature
- k_{eff} = effective thermal conductivity

This equation captures compressibility effects, shock-wave heating, expansion waves, and energy exchange inside the nozzle–duct configuration.

Turbulence Model Equations (Standard k–ε Model)

The current work states that the standard k-ε turbulence model was used. Therefore, Fluent additionally solved two transport equations:

Turbulent Kinetic Energy Equation

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot (\rho k \vec{V}) = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + G_k - \rho \varepsilon \quad (4)$$

Turbulent Dissipation Equation

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \nabla \cdot (\rho \varepsilon \vec{V}) = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right] + C_{1\varepsilon} \frac{\varepsilon}{k} G_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (5)$$

Modeling

Table 1 lists the constraints and the geometry size studied. The nozzle dimensions are estimated for M = 1.8. The geometry and shape are built on the sizes in Table 1. The diameter ratio is 1.7.

Table 1: Constraints of the existing investigation

Constraint	Length
Convergent angle	15°
Divergent angle	5°
Inlet radius	10.85 mm
Exit radius	5 mm
Critical radius	4.17 mm
Convergent size	25.0 mm
Divergent length	9.51 mm
Tube diameter	17 mm
Tube length	L/D ratio = 1 to 6

Cavity Dimensions and Locations

The cavity dimensions considered are 3:3, 3:6, and 6:3. Table 2 illustrates the cavity sizes for these w:h fractions. The cavity placement locations are L/D = 0.5-3.

Table 2: The cavity property is based on the aspect ratio

Cavity Geometry	Breadth (w) (mm)	Height (h) (mm)
1	3	3
2	3	6
3	6	3

Meshing and Conditions at the Boundary

A key step in the CFD process is the mesh. When selecting the free-face mesh class, the 2D type representation is configured accordingly. Elements were sized for each line segment (edge) after using the organized mesh option. The line segments determined the element size, and face meshing produced similarly shaped elements. The mesh neutrality check was then performed. Fig. 2 demonstrates the element type and size applied throughout this verification. The mesh-independent status for the 17 mm tube without ribs (L/D = 6) was achieved with different element sizes, all built on the same geometric model.

Shamitha Shetty et al

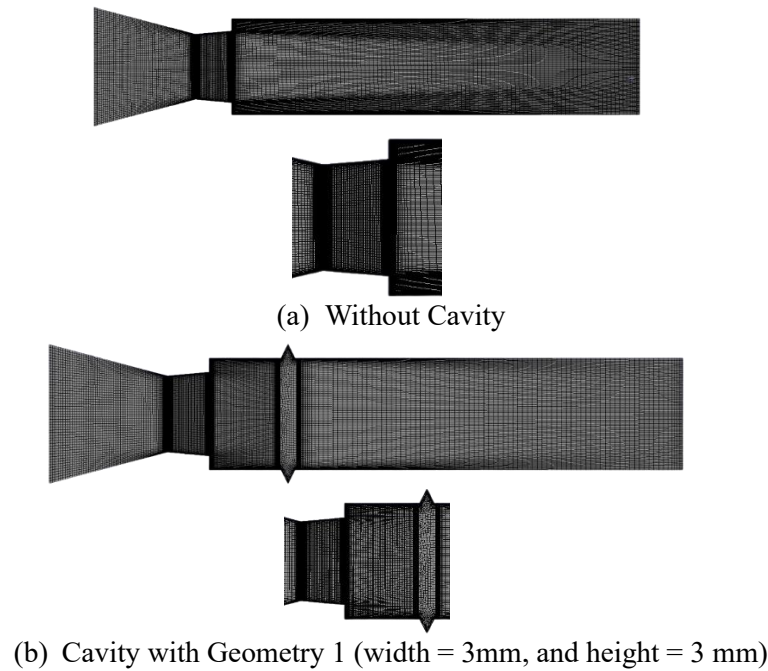


Fig. 2: Nozzle and duct with cavity geometries

Validation of Test Results

The ANSYS Workbench software used Fluent-based CFD methods throughout the fluid flow analysis. The model was built using DesignModeler. Fig. 4 shows a nozzle that sharply broadens into a tube with a cavity. In the test by Pandey and Rathakrishnan [XXIV], the measurements of the CD nozzle with an abrupt tube are detailed below.

Mesh Independence Test

Table 3 presents mesh-neutrality findings, a key step in ensuring reliable simulation results across different mesh refinements. Element dimensions vary from coarse to fine, while the number of nodes and elements remains consistent within each mesh. When the mesh happens to be finer, both nodes and elements increase meaningfully—from 2,472 nodes and 2,301 elements in the coarsest mesh to 1,474,416 nodes and 1,471,157 elements in the finest. The finest mesh offers the greatest detail but demands substantial computational capability. Intermediary meshes strike a balance between accuracy and efficiency.

Table 3: Mesh Independence Test

Element dimensions	Coarsest (5mm)	Coarse (4 mm)	Medium 1	Medium 2	Fine	Finer	Finest
Nodes	2472	3241	9469	17144	104060	297586	1474416
Elements	2301	3040	9163	16754	103168	296093	1471157

Based on trends in node and element counts, the highest-resolution mesh likely yields excellent, precise results (see Fig. 3a). Nonetheless, refining the mesh beyond a certain level may yield diminishing returns in precision. However, it would drastically increase

Shamitha Shetty et al

computational time. A careful analysis of the table indicates that the "Fine" or "Finer" mesh options might offer the optimal compromise between correctness and efficiency. These settings notably increase the number of nodes and elements assessed for medium meshes but avoid the high computational costs of the finest mesh. If the results are similar between the fine and finest meshes, further refinement isn't necessary, as it would only consume additional computational resources without improving the results. Therefore, the finer meshes are the most promising choices for future simulations. Figure 3(c) shows the average value of Y^+ on the duct wall. The Y^+ value ranges from 05 to 65 across different NPRs. These Y^+ values are acceptable for the selected turbulent model and near-wall function. Figures 3(d) to (e) show the Y^+ contours on the wall and at the duct base corner, respectively.

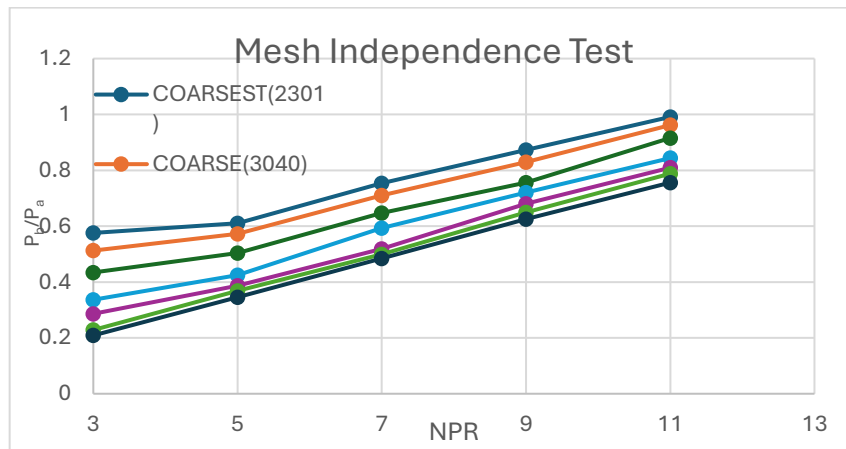


Fig. 3(a). Mesh Independence Test

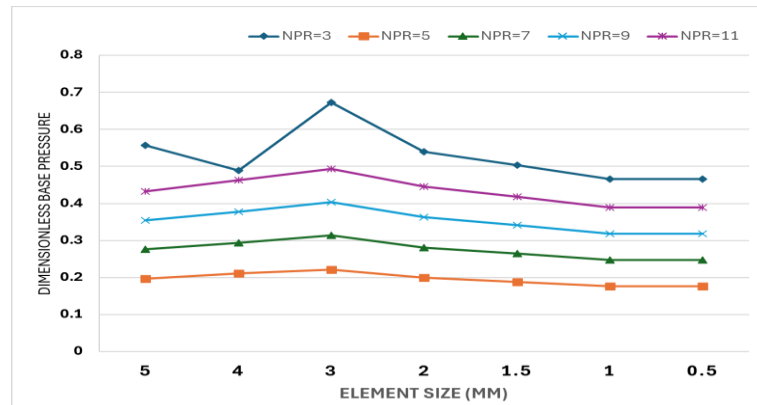


Fig. 3(b). Base pressure variations with mesh element size

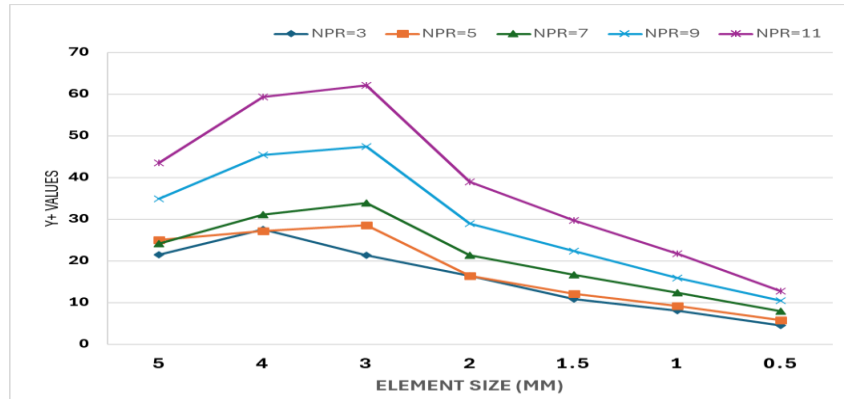


Fig. 3(c). Average wall Y+ variations with mesh element size

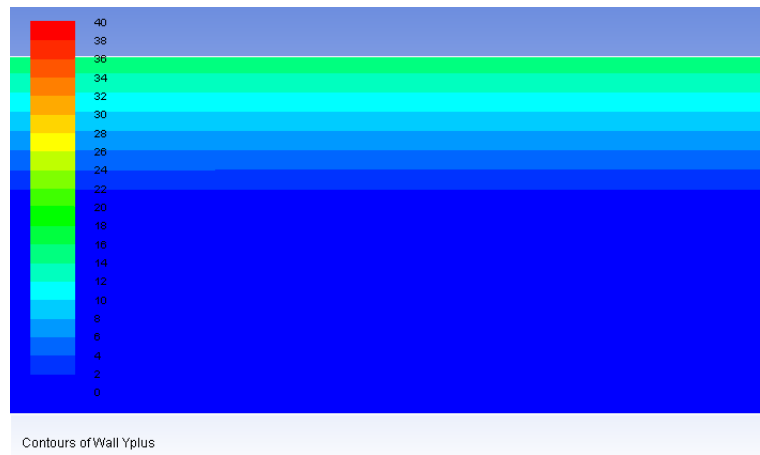


Fig. 3(d). Contour of Y+ at the mid-duct wall



Fig. 3(e). Contour of Y+ on the base corner of the duct

Fig. 3. Grid Independence Test and Y+ values

Validation of CFD Results

The initial section confirms the C-D nozzle in a circular pipe, as reported in the earlier control study by Pandey & Rathakrishnan [XXIV]. Tests were carried out at Mach 1.74 and NPR 2.65 (see Fig. 4). Validated geometries include L/D ratios of 2-10. We then simulate the same model with modified dimensions to emulate conditions at Mach 1.8 & numerous NPRs. The chosen NPRs remain 3, 4, 5, 7, and 11, while the L/D ratios extend from 1 to 6.

Geometry Validation

To validate the geometry against prior experimental work (Pandey & Rathakrishnan, XXIV), we replicate the flow factors using 2D CFD (ANSYS Fluent) for a cavity aspect ratio of 1 and subsequently compare the results. The focus is on significant flow parameters such as pressure and velocity. The geometry employed for validation is derived from Fig. 3.2 and Table 1 as detailed in Chapter 3. The tube size varies with L/D, stretching from 2 to 10.

Convergent-Divergent Axisymmetric Nozzle

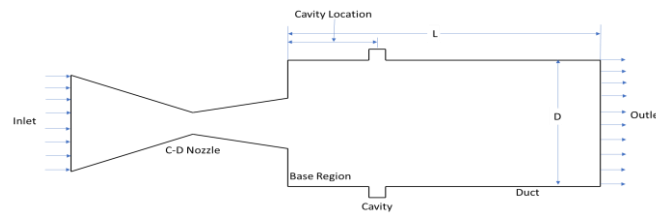


Fig. 4. Schematic of the experimental model.

Fig. 4 depicts an experimental setup with a CD axisymmetric nozzle that suddenly widens into a larger cross-section within a circular duct containing an annular cavity. This configuration, used to control base flow, is based on earlier work by Pandey and Rathakrishnan [XXIV]. The setup dimensions, shown in Fig. 1, match those listed in Table 1. All measurements are given in millimeters, and the Mach number is 1.74.

The cavity aspect ratio is defined as the width-to-depth ratio; in the experimental model, it is 3:3, with width and depth set to 3 mm. The study also examines a cavity aspect ratio of 2 (6:3). The configurations of the axisymmetric convergent-divergent nozzle model with a cavity, simulated using ANSYS Workbench 2023 and DesignModeler, are shown below for L/D ratios ranging from 2 to 10.



Fig. 5: $M = 1.74$, $L/D = 2$



Fig. 6: $M = 1.74$, $L/D = 4$



Fig. 7: $M = 1.74$, $L/D = 6$



Fig. 8: $M = 1.74$, $L/D = 8$



Fig. 9: $M = 1.74$, $L/D = 10$

Meshing

The Automatic Method is used on all quads, as shown in Fig. 10, to start the meshing process. The element size of 0.001 m is consistently applied across all C-D nozzle dimensions.

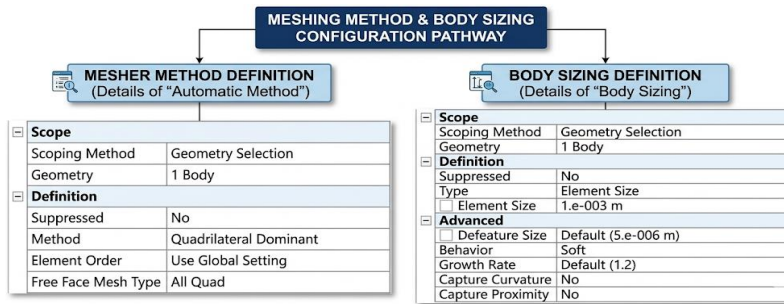


Fig. 10. The meshing panel includes the Automatic Method and Element Size.

The meshing procedure for $L/D = 2$ is illustrated in Fig. 11 and applies to all L/D values with the same mesh density.

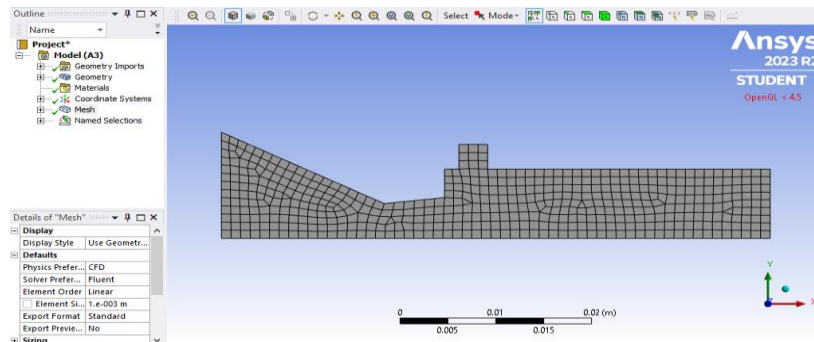


Fig. 11. Mesh for element magnitude of 0.001 m for $L/D = 2$.

CFD Simulation Setup

General

Before starting the CFD simulation, certain setup steps are necessary. These involve arranging general settings, models, materials, boundary conditions, initialization, and calculations. The solver is density-based. For 2D space, choose the axisymmetric preference to ensure the analysis focuses on a 2D axisymmetric problem. The complete solver setup is shown in Fig. 12.

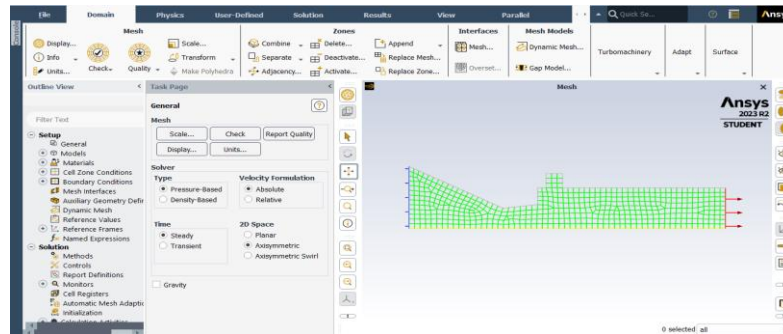


Fig. 12. General solver setup.

Models

The model enables the energy equation, as shown in Fig. 13. Under the viscous model settings, the default is the realizable k-epsilon turbulent model with a scalable wall function. The standard wall functions for near-wall treatment are also selected.

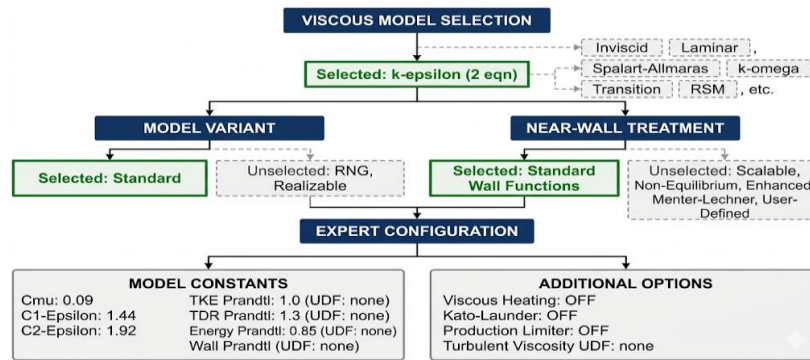


Fig. 13. Geometry model's panel.

Materials

In the material selection, air is chosen as the fluid, with density set to that of an ideal gas and viscosity specified by the Sutherland Law, as shown in Fig. 14. Other properties, such as specific heat, thermal conductivity, and molecular weight, are set to constant values.

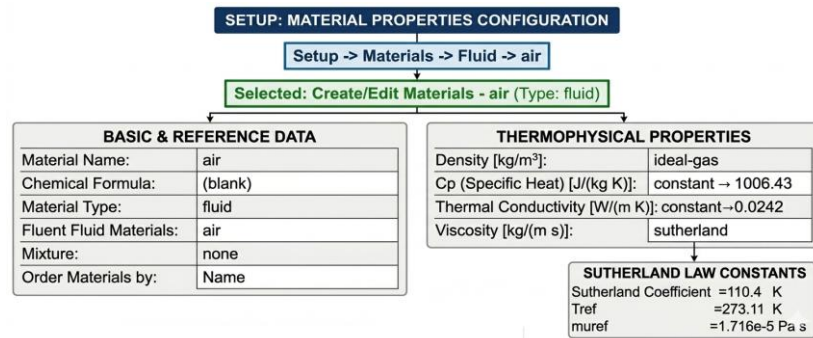


Fig. 14. Fluid material selection.

Boundary Conditions

Fig. 15 illustrates the boundary conditions at the nozzle inlet, and the Gauge Total Pressure (GTP) is calculated using the NPR formula provided below. For all other zones, the GTP must be set to 0. The computation of NPR is shown in the equation below:

$$NPR = \frac{GTP + P_{atm}}{P_{atm}} \quad (7)$$

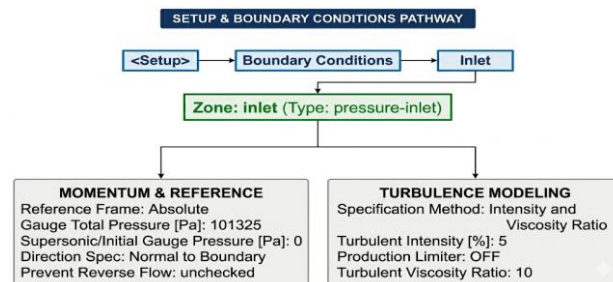


Fig. 15. Boundary conditions for pressure inlet.

Initialization and Calculation

Next, we set the solution initialization to hybrid initialization and the number of iterations to 5000, as shown in Fig. 16.

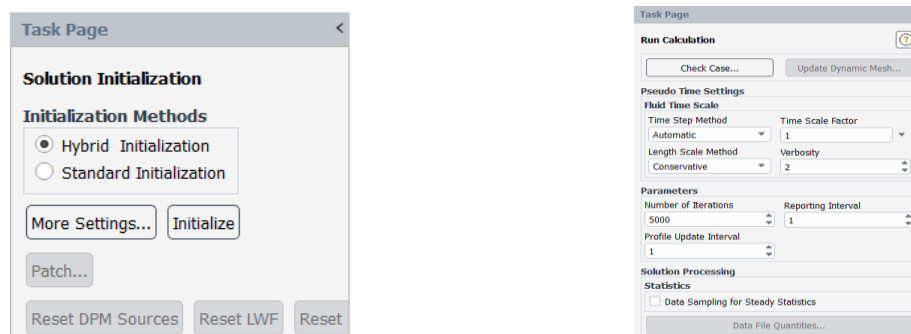


Fig. 16. Solution initialization and calculation panel.

Validation Result

Pandey & Rathakrishnan [XXIV] conducted tests at a Mach number of 1.74, with NPR values of 2.10-3.48, area ratios of 10-2.89, and L/D ratios of 1-10. In this work, we compare our results with those presented in Fig. (c) of Pandey & Rathakrishnan's [XXIV] earlier research, specifically at an NPR of 2.65 (P01/Pa). Table 5 shows that Pandey & Rathakrishnan [XXIV] demonstrated the effect of base pressure across different L/D ratios under constant Mach number, NPR, and area ratio conditions. To validate our results, we tested L/D ratios from 2 to 10.

Table 5: Comparison of the present study with earlier work and the error.

L/D	Pandey & Rathakrishnan [XXIV]	Present work (Ansys Fluent)		Percentage Error (%)
	Base Pressure Ratio	Pressure (Pa)	Base Pressure Ratio	
2	0.727	-23753.8	0.766	5.31
4	0.538	-46178.9	0.544	1.16
6	0.523	-51986.1	0.487	6.90
8	0.539	-51143.5	0.495	8.12
10	0.558	-50234.0	0.504	9.64

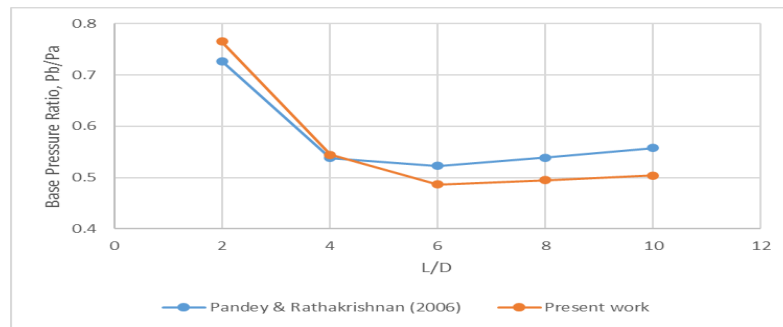


Fig. 17(a). Comparison of Experimental and CFD Results.

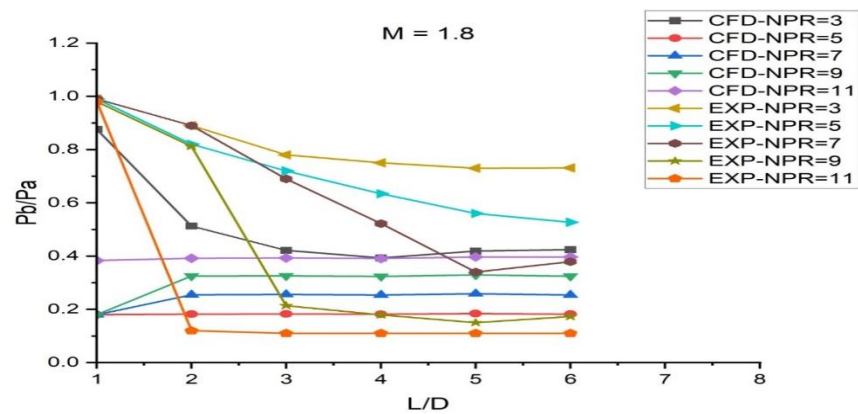


Fig. 17(b): Comparison of Experimental and CFD Results.

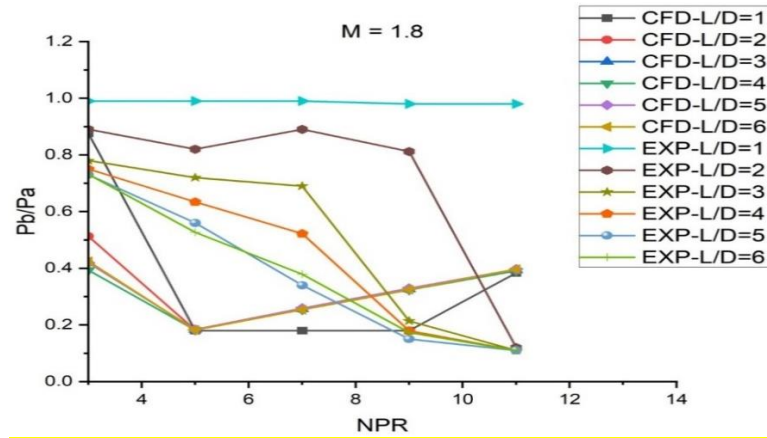


Fig. 17(c). Comparison of Experimental and CFD Results.

Figs. 17(b) to (c) compare experimental and CFD results at Mach 1.8 for the same area ratio but without control, with Khan et al. [XVII]. The figures show large discrepancies between the CFD and experimental results for the following reasons.

III. Results and Discussion:

In under-expanded conditions, the flow accelerates, diverges, and experiences increased rotation and expansion-induced spin, which can lead to early reattachment and a longer reattachment segment. An oblique shock redirects the flow toward the duct centerline, reducing this reattachment length. Base pressure and flow fluctuations are associated with the principal vortex, and wave strength is notably influenced.

These findings show passive control of the base flow through a cavity with Mach $M = 1.8$ and a tube radius of 8.5 mm, resulting in a diameter ratio of 1.7. Using isentropic flow relations and since entropy does not change, the NPR required for ideal expansion at $M = 1.8$ is 5.8, which was established as the model's design condition. For overexpansion scenarios, the chosen NPR values (3, 5) are below the NPRs necessary for ideal expansion. Conversely, for under-expanded conditions, we select 7, 9, and 11, corresponding to 21%, 60%, and 90% above the essential magnitude for proper expansion.

Literature supports the numerical assessment results. K. M. Pandey and E. Rathakrishnan [XXIV] conducted experimental studies on the effect of a cavity on base pressure, confirming that their findings align with the current results. As illustrated in Fig. 3, the CFD analysis matches the experimental data.

When the cavity is 5.5 mm from the base

Fig. 18 shows the effects of the present analysis for a cavity located 8.5 mm from the base, for three cavity aspect ratios: 3:3, 3:6, and 6:3. As discussed in the initial introduction, unlike other passive control methods, the flow control using a cavity is only effective when the flow falls into the cavity; otherwise, the presence of the cavity will not be felt, depending on the area ratio, inertia levels, tube size ratio, and NPR.

Shamitha Shetty et al

The duct-to-nozzle diameter ratio primarily determines the reattachment. However, in this case, the diameter ratio is 1.7, so the reattachment segment is projected to be lower than $L/D = 1$. This may explain why the cavity is effective when placed at 8.5 mm. Further, it is observed that once the tube size reaches 17 mm, the base pressure magnitude is slightly higher than in larger duct segments ($L/D = 1-3$). Across the three cavity geometries, increasing the cavity depth from 3 mm to 6 mm does not significantly alter the base pressure. These discoveries reveal that the parameters controlling base pressure are passive, namely width, not cavity depth. The cavity forms whenever it is long enough to accommodate the viscous layer as it falls inside (Fig. 18(a)- (e)).

When we observe the findings of this investigation for duct sizes, namely 85 mm & 102 mm, it is found that the magnitude of the base pressure has marginally changed; this may be due to the larger tube segments, and the atmospheric pressure is insufficient to affect the flow of the tube; otherwise, the outcomes of the study are in the same order of magnitude.

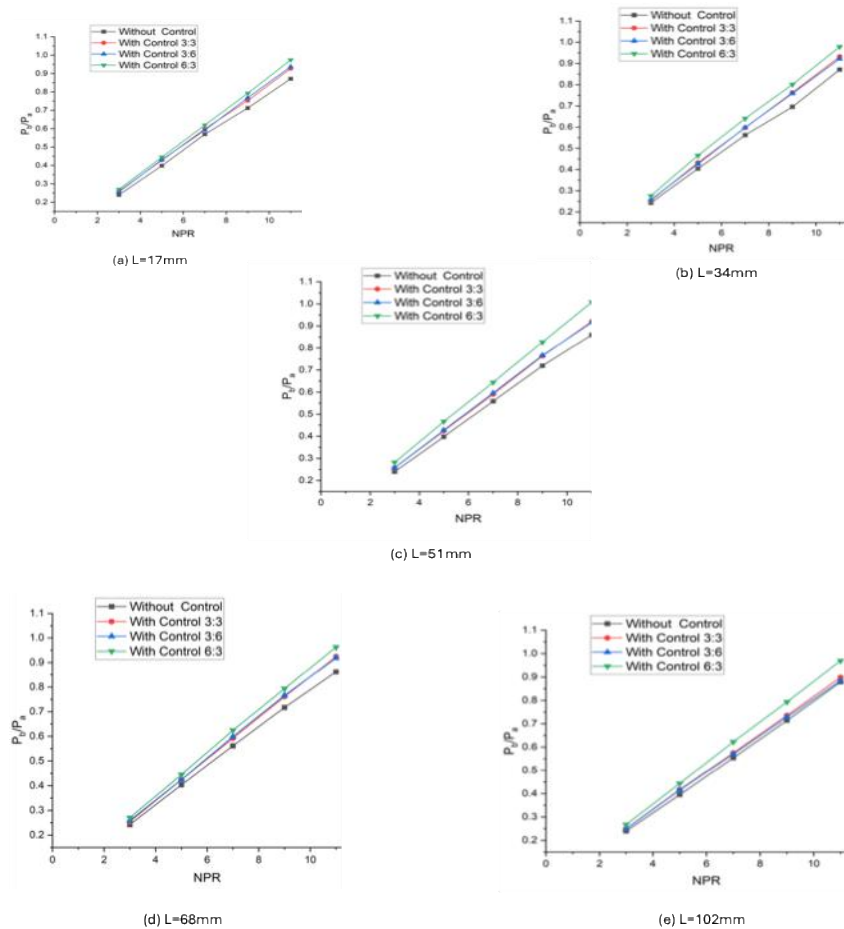


Fig. 18. P_b/P_a Vs. NPR for several cavities

When the cavity is at 17 mm from the base

At 17 mm from the base, the results indicate it is ineffective; once the nozzles are properly expanded, base pressure increases linearly, with or without supervision. Again, control efficacy becomes significant once the nozzle flow passes through an under-expanded stream. The cavity is also not helpful, as its position prevents the shear layer from interacting with it; the flow reconnected to the tube wall before reaching the cavity, as shown in Fig. 19(a)-(e).

The findings of this investigation appear identical for the remaining duct segments, namely, 34 to 102 mm. The effect of geometry has also vanished.

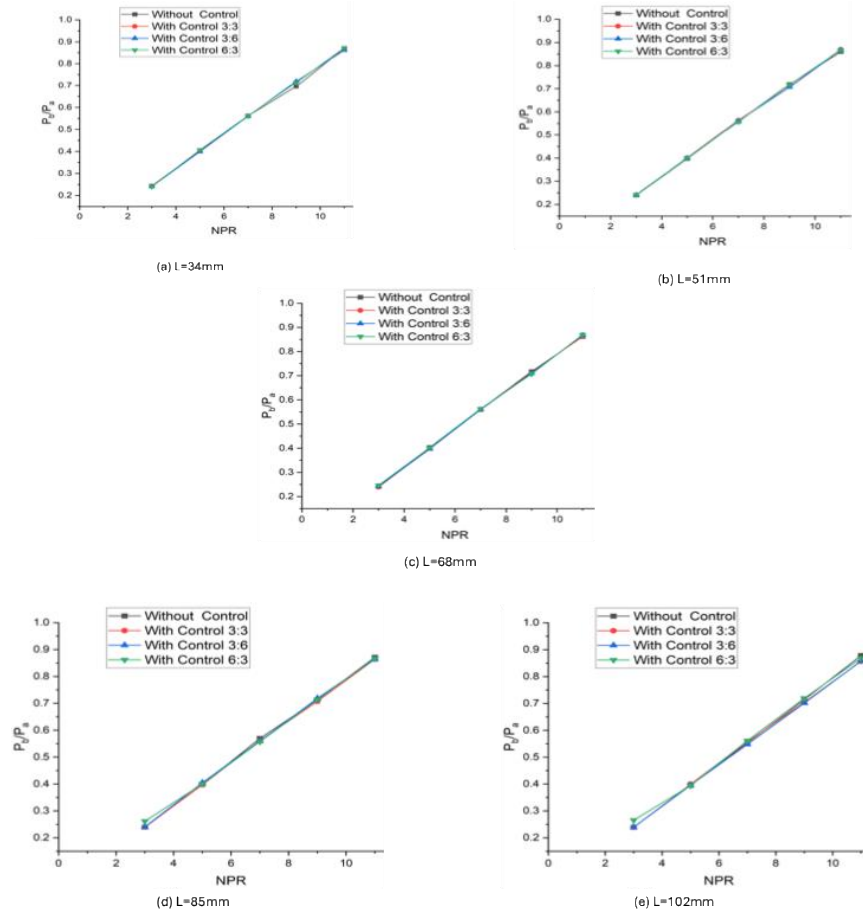


Fig. 19. P_b/P_a Vs. NPR for several cavities

When the cavity is located at different locations, namely $L/D = 1.5, 2.0, 3.0$, and 4.0 , the results were unchanged from the cases without ribs. Hence, results for these cavity locations are not shown to avoid a monotonous pattern. This is because the shear layer does not enter the cavity, so the base pressure values do not change.

Contours at various NPRs for Duct size $L/D = 6$

This section shows velocity, pressure, and streamline contours at various NPRs.

Shamitha Shetty et al

Velocity, Pressure, and Streamline at NPR = 3

Figs. 20(a) to (c) at NPR = 3, the nozzles are facing an adverse pressure gradient of 0.52. As the nozzle is over-expanded, oblique shock waves augment the flow leaving the nozzle, and a recirculation region forms at the corner. After passing through the cavity, the flow accelerates further, as shown in Fig. 20(a). Similarly, the pressure contour is shown in Fig. 20(b), where the pressure is low but recovers beyond the cavity. Likewise, Fig. 20(c) shows the streamline contour. Two recirculation zones are visible: the first at the base of the cavity and the second near the cavity.

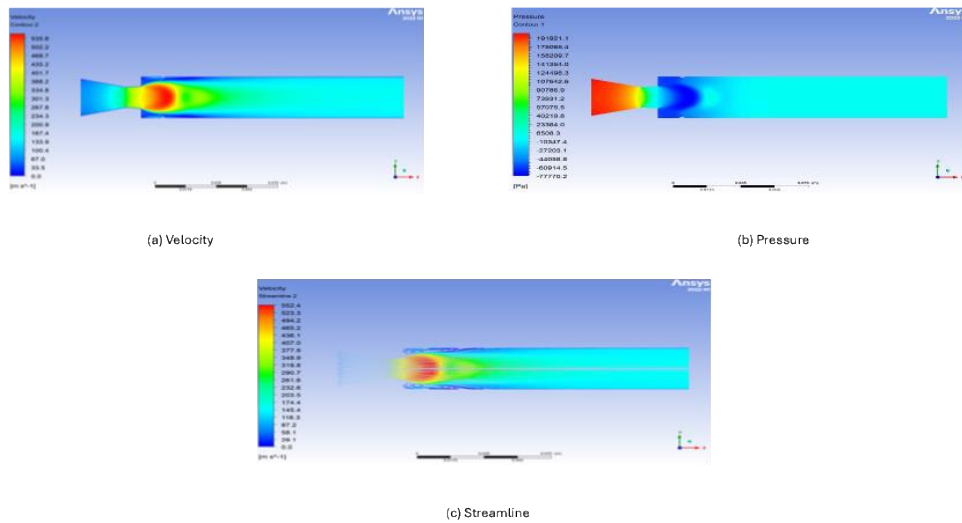


Fig. 20. Various Contours at a nozzle pressure ratio of 3

Pressure, Velocity, and Streamline at NPR = 7

Fig. 21 shows the velocity, pressure, and streamline contours once the NPR is further increased to 7. For this NPR, the nozzle is under-expanded, with an under-expansion level of 1.2. Under these conditions, as the viscous layer exits the nozzle, an expansion fan accompanies the flow, increasing the reattachment length, as shown in Figs. 25 (a) to (c).

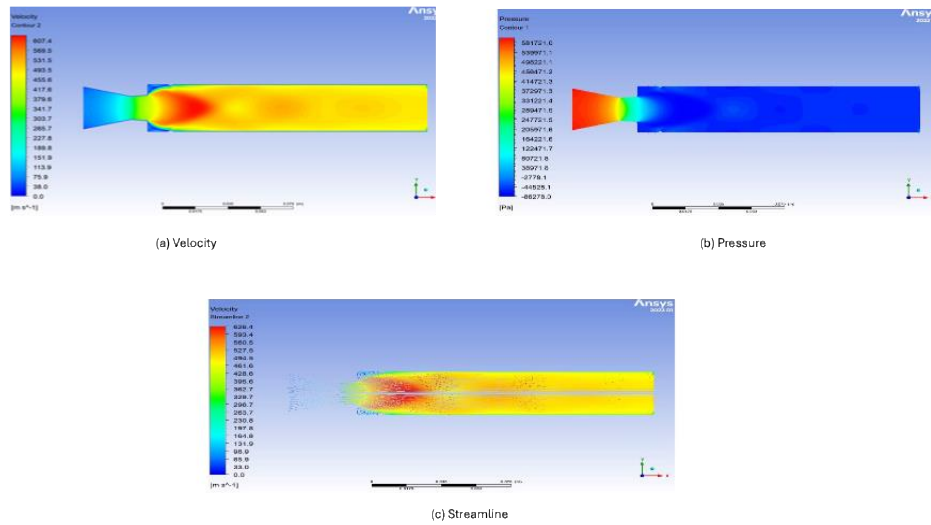
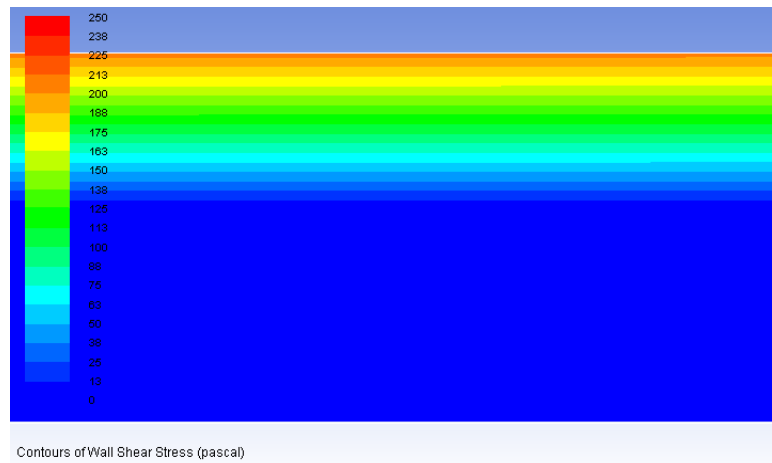


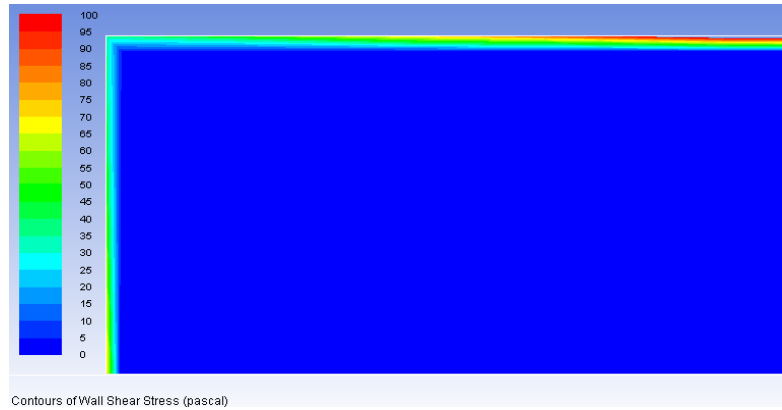
Fig. 21. Various Contours at nozzle pressure ratio = 7

Wall Shear Stress

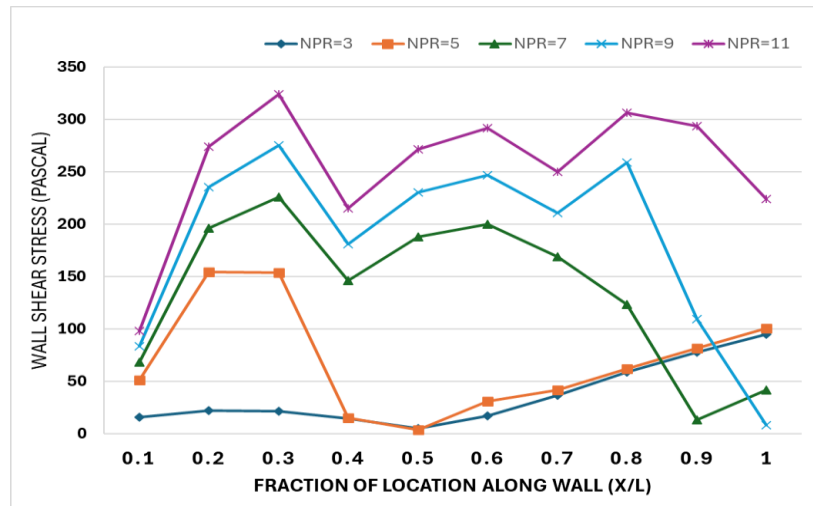
Fig. 22 shows the wall shear stress along the duct wall for $L/D = 6$. Figures 22(a & (b) show the wall shear stress contours in the middle portion of the duct and at the base corner of the duct wall, respectively, at NPR = 7. In Figure 22c, x/L is the fraction of the total wall length, where L is the total wall length. Figure 22c shows that wall shear stress depends on the reattachment point location and the nozzle pressure ratio. The sudden rise in shear stress indicates the reattachment point. The reattachment point is around $x/L=0.2$ to 0.3 for NPRs 5, 7, 9, and 11.



a) Contour of Wall Shear Stress at middle portion of duct at NPR = 7



b) Contour of wall shear stress at the base corner portion of the duct at NPR = 7



c) Wall shear stress distribution plots along the wall

Fig. 22. Wall shear stress distribution along the wall for $L/D=6$

IV. Conclusions

The base pressure depends on inertia, tube size, cavity location, and expansion. The cavity differs fundamentally from other passive control mechanisms, such as ribs, boattail, step body, and vortex-locking methods. In most passive control techniques, the location of the flow-control mechanism is not critical, except for a cavity. Specifically, when a cavity serves as the control mechanism, it is effective only if the flow exiting the nozzle enters the cavity. If it reattaches before or after the cavity, the cavity is not felt and cannot affect the flow confined to the cavity or the enlarged duct. The base-pressure results clearly demonstrate this trend. The cavity is successful only when fixed at $L/D = 0.5$ (i.e., 8.5 mm) from the base; at other cavity locations, the control technique is ineffective. However, at other cavity locations ($L/D = 1.0$ to 3.0), the control still has an impact, provided the nozzle operates under adverse conditions (e.g., nozzle pressure ratios of 3 and 5) and the flow is over-expanded. These results

Shamitha Shetty et al

reiterate that, whenever the flow control mechanism is employed, whether active or passive, it is effective under favorable pressure conditions. Base pressure results show that at lower NPRs, namely 3 and 5, the control decreases base pressure when the nozzles operate in an over-expanded condition. Hence, the cavity's downstream locations are ineffective when the nozzle is ideally expanded or under-expanded.

Conflict of Interest:

There was no relevant conflict of interest regarding this paper.

References

- I. Aabid, Abdul, et al. "Numerical Simulation of Suddenly Expanded Flow at Mach 2.2." *International Journal of Engineering and Advanced Technology*, vol. 8, no. 3, 2019, pp. 457–62, https://www.researchgate.net/publication/331466434_Numerical_Simulation_of_Suddenly_Expanded_Flow_at_Mach_2.2.
- II. Akhtar, Mohammad Nishat, et al. "Effects of Micro Jets on the Flow Field of the Duct with Sudden Expansion." *International Journal of Innovative Technology and Exploring Engineering*, vol. 8, no. 9S2, 2019, pp. 636–40, 10.35940/ijitee.i1129.0789s219.
- III. Akhtar, Mohammad Nishat, et al. "Numerical Simulations of a CD Nozzle and the Influence of the Duct Length." *International Journal of Innovative Technology and Exploring Engineering*, vol. 8, no. 9S2, 2019, pp. 622–30, 10.35940/ijitee.i1127.0789s219.
- IV. Arabi, Abderraouf, et al. "Pressure Recovery Model for Gas–liquid Two-Phase Flow Across Sudden Expansions." *Experimental and Computational Multiphase Flow*, vol. 6, no. 1, 2024, pp. 14–27, 10.1007/s42757-023-0160-3.
- V. Arabi, Abderraouf, et al. "Identifying the Intermittent Flow Sub-Regimes Using Pressure Drop Time Series Fluctuations." *Experimental and Computational Multiphase Flow*, vol. 6, no. 1, 2024, pp. 28–40, 10.1007/s42757-022-0153-7.
- VI. Baig, Maughal Ahmed Ali, et al. "Control of Base Flows with Micro Jets." *International Journal of Turbo and Jet Engines*, vol. 28, no. 1, 2011, pp. 59–69, 10.1515/tjj.2011.009.
- VII. Davletshin, I. A., et al. "Heat Transfer and Flow Structure in a Plane Diverging Channel." *International Journal of Heat and Mass Transfer*, vol. 189, 2022, article 122744. 10.1016/j.ijheatmasstransfer.2022.122744.
- VIII. Davletshin, I. A., et al. "Secondary Flows in a Flat Diffuser and Their Significance for the Formation of a Turbulence Field." *Thermophysics and Aeromechanics*, vol. 30, no. 6, 2024, pp. 995–1011, 10.1134/S0869864323060033.

- IX. Golubkina, I. V., and A. N. Osipstov. "Compressible Gas-Droplet Flow and Heat Transfer Behind a Condensation Shock in an Expanding Channel." *International Journal of Thermal Sciences*, vol. 179, 2022, article 107576, 10.1016/j.ijthermalsci.2022.107576.
- X. Khan, Ambareen, et al. "Comprehensive CFD Analysis of Base Pressure Control Using Quarter Ribs in Sudden Expansion Duct at Sonic Mach Numbers." *International Journal of Thermofluids*, vol. 24, 2024, article 100908. 10.1016/j.ijft.2024.100908.
- XI. Khan, Ambareen, et al. "Passive Control of Base Pressure: A Review." *Applied Sciences*, vol. 11, no. 3, 2021, article 1334. 10.3390/app11031334.
- XII. Khan, Ambareen, et al. "Experimental Investigation of Suddenly Expanded Flow at Sonic and Supersonic Mach Numbers Using Semi-Circular Ribs: A Comparative Study Between Experimental, Single Layer, Deep Neural Network (SLNN and DNN) Models." *The European Physical Journal Plus*, vol. 138, no. 4, 2023, article 314. 10.1140/epjp/s13360-023-03853-1.
- XIII. Khan, Ambareen, et al. "Numerical Simulation of Suddenly Expanded Flow from Converging Nozzle at Sonic Mach Number." *Proceedings of International Conference of Aerospace and Mechanical Engineering 2019*, Springer, 2020, pp. 349–59. 10.1007/978-981-15-4756-0_29.
- XIV. Khan, Ambareen, et al. "Effect of Ribs in a Suddenly Expanded Flow at Sonic Mach Number." *Heliyon*, vol. 10, no. 9, 2024, article e30313. 10.1016/j.heliyon.2024.e30313.
- XV. Khan, S. A., et al. "Passive Control of Base Drag Employing Dimple in Subsonic Suddenly Expanded Flow." *International Journal of Mechanical and Mechatronics Engineering*, vol. 18, no. 3, 2018, pp. 69–74, https://www.researchgate.net/publication/326684003_Passive_control_of_base_drag_employing_dimple_in_subsonic_suddenly_expanded_flow.
- XVI. Khan, Sher Afghan, et al. "CFD Analysis of Human Powered Submarine to Minimize Drag." *International Journal of Mechanical and Production Engineering Research and Development*, vol. 8, no. 3, 2018, pp. 1057–66, 10.24247/ijmperdjun2018111.
- XVII. Khan, Sher Afghan, and E. Rathakrishnan. "Nozzle Expansion Level Effect on Suddenly Expanded Flow." *International Journal of Turbo and Jet Engines*, vol. 23, no. 4, 2006, pp. 233–58, 10.1515/tjj.2006.23.4.233.
- XVIII. Khan, Sher Afghan, et al. "Passive Control of Base Drag in Compressible Subsonic Flow Using Multiple Cavity." *International Journal of Mechanical and Production Engineering Research and Development*, vol. 8, no. 4, 2018, pp. 39–44. 10.24247/ijmperdaug20185.
- XIX. Kim, Yongchan, et al. "Base Flow and Drag Characteristics of a Supersonic Vehicle with Cold and Hot Jet Flows of Nozzles." *Aerospace*, vol. 10, no. 10, 2023, article 836. 10.3390/aerospace10100836.

- XX. Kopparthy, Saketh, et al. “Numerical Investigations of Turbulent Single-Phase and Two-Phase Flows in a Diffuser.” *International Journal of Multiphase Flow*, vol. 130, 2020, article 103333. 10.1016/j.ijmultiphaseflow.2020.103333.
- XXI. Lee, Shibo, et al. “A Review of Flow Control Strategies for Supersonic/Hypersonic Fluid Dynamics.” *Aerospace Research Communications*, vol. 2, 2024, article 13149. 10.3389/arc.2024.13149.
- XXII. Pakhomov, M. A., and V. I. Terekhov. “Gas-Droplet Flow Structure and Heat Transfer in an Axisymmetric Diffuser with a Sudden Expansion.” *Journal of Applied Mechanics and Technical Physics*, vol. 61, no. 5, 2020, pp. 787–97. 10.1134/S0021894420050132.
- XXIII. Pakhomov, Maksim A., and Viktor I. Terekhov. “Modeling of Turbulent Heat-Transfer Augmentation in Gas-Droplet Non-Boiling Flow in Diverging and Converging Axisymmetric Ducts with Sudden Expansion.” *Energies*, vol. 15, no. 16, 2022, article 5861. 10.3390/en15165861.
- XXIV. Pandey, K. M., and E. Rathakrishnan. “Annular Cavities for Base Flow Control.” *International Journal of Turbo and Jet Engines*, vol. 23, no. 2, 2006, pp. 113–27. 10.1515/TJJ.2006.23.2.113.
- XXV. Rashid, Shagufta, et al. “Review of Wave Drag Reduction Techniques: Advances in Active, Passive, and Hybrid Flow Control.” *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, vol. 236, no. 14, 2022, pp. 2851–84. 10.1177/09544100211069796.
- XXVI. Rehman, Shafiqur, and S. A. Khan. “Control of Base Pressure with Micro-Jets: Part I.” *Aircraft Engineering and Aerospace Technology*, vol. 80, no. 2, 2008, pp. 158–64. 10.1108/00022660810859373.
- XXVII. Salem, Khaled M., et al. “Modeling Two-Phase Gas-Solid Flow in Axisymmetric Diffusers Using Cut Cell Technique: An Eulerian-Eulerian Approach.” *Boundary Value Problems*, vol. 2024, no. 1, 2024, article 150. 10.1186/s13661-024-01955-y.
- XXVIII. Shakirov, R. R., et al. “Kinematic Structure of Flow and the Heat Transfer in Flat Diffuser and Confuser Channels.” *Thermophysics and Aeromechanics*, vol. 29, no. 5, 2023, pp. 759–64. 10.1134/S08698643220500146.