



ARTICLE

Influence of Cavity Position on Flow Control at Large Mach Numbers: A CFD Analysis

Sher Afghan Khan¹, Ridwan¹, Abdul Aabid^{2,*}, Khizar Ahmed Pathan³ and Muneer Baig²

¹Affiliation Mechanical and Aerospace Engineering Department, Faculty of Engineering, International Islamic University, Kuala Lumpur, Malaysia

²Department of Engineering Management, College of Engineering, Prince Sultan University, Riyadh, Saudi Arabia

³Department of Mechanical Engineering, CSMSS Chh. Shahu College of Engineering, Aurangabad, Maharashtra, India

*Corresponding Author: Abdul Aabid. Email: aaabid@psu.edu.sa

Received: 03 May 2026; Accepted: 28 August 2026; Published: 04 September 2026

ABSTRACT: This study investigates the influence of a passive cavity flow-control strategy on base pressure in suddenly expanded supersonic flows through computational fluid dynamics (CFD) simulations. The flow configuration consists of a convergent-divergent (CD) nozzle discharging into an enlarged duct with an area ratio of 2.89. Simulations were performed for Mach numbers of 1.2, 1.4, 1.6, and 1.8, while the cavity was positioned upstream of the expansion at cavity locations corresponding to length-to-diameter ratios (L/D) ranging from 0.5 to 2.0 in increments of 0.5. The effects of nozzle pressure ratio (NPR), Mach number, duct length-to-diameter ratio, and cavity placement on base pressure were systematically examined. The results demonstrate that the cavity significantly enhances base pressure under the investigated operating conditions without adversely affecting the flow field. Base pressure is strongly influenced by Mach number, NPR, duct geometry, and cavity location. Among the configurations considered, the cavity positioned at 0.5D from the nozzle exit consistently provides the most effective pressure recovery across the range of Mach numbers, NPRs, and L/D ratios investigated. Furthermore, increasing NPR promotes stronger under-expanded jet conditions, with highly under-expanded flows observed at NPR = 6 and 8, leading to a corresponding increase in base pressure. These findings highlight the potential of cavity-based passive control for regulating base pressure in suddenly expanded supersonic flows.

KEYWORDS: CFD; supersonic flow; passive control; Mach number; CD nozzle

1 Introduction

Research on turbulent flow remains important due to its applications in the aerospace and automotive industries. Researchers continue to explore the flow field at the blunt region of fuselages, missiles, and shells because it depends on the base pressure in the recirculation region. The literature indicates that the base pressure in the recirculation zone is lower than atmospheric pressure, which is directly related to base drag. The total drag acting on any aerospace vehicle consists of viscous drag, drag due to shock waves at sonic and supersonic Mach numbers, and base drag. Because the base pressure is lower than the ambient pressure, the resulting base drag can be significant, reaching up to 70% of the total drag. Two methods are prevalent for regulating base pressure: passive control and active control. Active control requires an external energy source to regulate base pressure, whereas passive control requires no external energy source. Instead, passive control requires changes in the flow-field geometry using passive methods such as

various rib shapes, cavities, boattails, step bodies, and vortex-locked mechanisms. The major advantage of active control is that it can be switched on or off as needed. In contrast, passive control remains with the system throughout the aerospace vehicle's flight, adding extra system weight.

Pandey et al. [1] demonstrated the effectiveness of cavities in reducing recirculation zones in sudden expansion ducts. Other studies have used dynamic control methods [2] and passive [3] Methods in CD nozzles. Computational Fluid Dynamics simulations have been instrumental in analyzing CD nozzles and in understanding gas-flow dynamics across various nozzles [4–7].

Research across various nozzle configurations has emphasized the importance of modelling accuracy and geometric control. Rao et al. [8] and Najjar et al. [9] both focused on CD nozzle flow, with the former highlighting the necessity of precise boundary conditions for reliable predictions. At the same time, the latter found that the $k-\epsilon$ turbulence model provides better assessments of flow performance than the Spalart–Allmaras model. Similarly, Salvador et al. [10] utilized a homogeneous equilibrium approach to show that accurate cavitation modelling is vital for optimizing diesel injector design. Beyond traditional nozzle flow, Pushpa et al. [11] found that incorporating copper-water nanofluids in baffled annular cavities significantly enhances heat transfer functioning, and Koomullil et al. [12] introduced a versatile mesh generation framework to improve simulation accuracy across complex geometries.

The CFD approach is used to evaluate different turbulence models & to analyze flow behavior in both rough and smooth channel conditions [13]. Recent developments in fluid dynamics have integrated machine learning and advanced computational techniques to study complex phenomena, including shock wave behavior, turbulence prediction, and the influence of structural and thermal effects on flow characteristics [14]. In addition, surrogate-based optimization methods, including the grey wolf optimizer, have been effectively used to enhance the design performance of scramjet nozzles and heat exchangers [15,16]. Khan et al. [17] conducted experimental investigations of flow conditions at Mach numbers of 1 or higher, and the results were subsequently interpreted using machine learning techniques. Aabid et al. [18] investigated high-speed flow control from a CD nozzle using design of experiments and CFD methods. The authors developed this all-Mach THINC-TDU numerical scheme for compressible multi-component flows, including surface tension, cavitation, turbulence modelling, and interface sharpening. It was shown to improve accuracy and exhibit a more robust behavior in simulating intricate multiphase flow events, while also keeping numerical diffusion low and preventing those spurious oscillations, you know, the annoying kind [19].

In this investigation, rectangular cavities of different geometries are used at various locations to control the base pressure. One major issue with cavities is that they are effective only when the dividing streamline reattaches within the cavity; otherwise, they become blind when the flow does not span the cavity's width. Unlike the ribs, where it does not matter whether the rib location is near or far from the reattachment point, in any case, it will impact the flow field in the recirculation zone; however, the effectiveness of the ribs in regulating the flow will be at its best when the rib is placed very close to the reattachment point. Hence, controlling the base pressure at low and high speeds is of interest to researchers in transonic and supersonic flow due to the advent of high-speed fighter planes and missiles. Even a slight increase in base pressure will significantly reduce base drag and, hence, the range of aerospace vehicles, resulting in considerable savings in fossil fuel use and reduced global warming.

The current study focuses on controlling the base flow within a cavity in a duct. The research investigates the effectiveness of a cavity in controlling base pressure by positioning it at multiple locations and at several supersonic Mach numbers, while varying the inlet pressure. A 17 mm-diameter tube is analysed using a 3 mm × 3 mm cavity. This work models a 3-D CD nozzle to control the base pressure and, eventually, the base drag, and to enhance duct flow rates through precise control of pressure parameters.

Unlike some earlier studies that mostly examined passive control devices or cavity configurations under limited operating conditions, this work evaluates cavity placement effects across a wider range of Mach numbers, NPR values, and duct length-to-diameter ratios for an area ratio of 2.89. The results indicate that the 0.5D cavity configuration is the most effective for enhancing base pressure while maintaining a reasonably intact overall flow structure.

2 Problem Definition

This study focuses on passive flow-regulation methods and uses a cavity at the duct's reattachment point to influence base pressure. In Fig. 1, the Convergent-Divergent (CD) nozzle is connected to the tube. When the fluid exits the nozzle into a larger duct section, a low-pressure zone forms at the base, leading to base drag. The cavity is strategically positioned within the duct to enhance base pressure by inhibiting the backflow in the base zone. The CD nozzle dimensions used in this study are the convergent nozzle length is 25 mm, and the diverging nozzle length is 9.51 mm. In contrast, the inlet diameter is 21.7 mm (Theta (C) converging angle = 15 degrees), the throat diameter is 8.34 mm, and the exit diameter from the divergent duct is 10 mm (Theta (C) converging angle = 15 degrees). The length and diameter of the sudden-expansion duct vary depending on the current investigation and are shown in Table 1. Similarly, the cavity location is also clearly defined, as shown in Table 1.

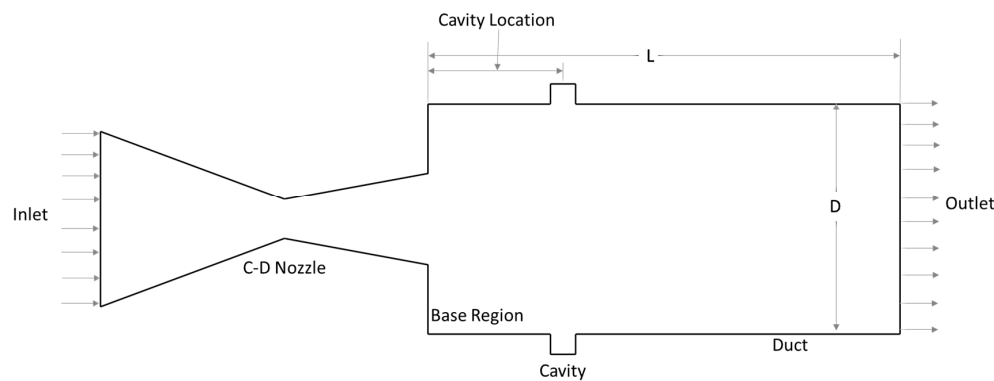


Figure 1: The configuration of the nozzle and duct with cavity.

Table 1: The constraints and levels.

Parameters	Level 1	Level 2	Level 3	Level 4
Mach No.	1.2	1.4	1.6	1.8
L/D (Length/Diameter) Ratio	3	4	5	6
NPR (Inlet pr/Atm. pr)	2	4	6	8
Position of Cavity	0.5D = 8.5 mm	1D = 17 mm	1.5D = 25.5 mm	2D = 34 mm

3 CFD Analysis

3.1 Governing Equations

Computational Fluid Dynamics is a numerical technique for simulating flow and related physical processes. It predicts fluid behavior within a defined domain by solving the governing equations under specified boundary conditions. In CFD, fluid behavior is described by the basic equations of continuity, momentum, and energy.

1. Conservation of Mass Equation

This equation ensures that mass is conserved throughout the flow field:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad (1)$$

2. Momentum Equation

This equation signifies the balance of forces acting on the fluid:

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \quad (2)$$

3. Energy Equation

This equation accounts for the conservation of energy in the system:

$$\frac{\partial(\rho e)}{\partial t} + \frac{\partial(\rho u_j e)}{\partial x_j} = -P \frac{\partial u_j}{\partial x_j} + \frac{\partial q_j}{\partial x_j} + \tau_{ij} \frac{\partial u_i}{\partial x_j} \quad (3)$$

where, ρ is air density, u is speed, P is pressure, τ_{ij} is the viscous stress tensor, q_j is heat flux, and e is energy/unit mass.

3.2 Models and Parameters

The current study investigates the impact of inertia levels (M), tube length-to-diameter ratio (L), cavity position (C), and expansion levels. The tube diameter is 17 mm, and the nozzle exit diameter is 10 mm. Table 1 provides the constraints and their levels. Using ANSYS Workbench, the geometries are meticulously modelled for all parameter combinations, and the models are analysed using Fluent.

Due to the duct's axisymmetric nature, computation time is significantly reduced. The quarter 3D model is chosen for in-depth assessment to ensure optimal outcomes within a reasonable time frame. Fig. 2 illustrates the quarter 3D model and its corresponding boundary conditions. The inlet pressure designates the entry point, and the outlet pressure denotes the duct exit. Inlet pressure is calculated based on the NPR and applied accordingly. At the tube's exit plane, a zero-gauge pressure condition is imposed.

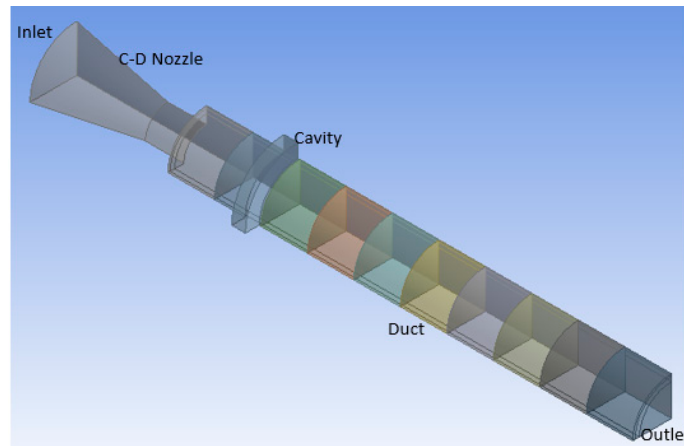


Figure 2: Quarter 3D model and corresponding boundary conditions.

3.3 Meshing and Boundary Conditions

Fig. 3 illustrates the 3D meshed model prepared for the CFD study. The meshing process in ANSYS Workbench uses hexahedral elements. To create these elements, the entire model is split into multiple sub-volumes, each of which is meshed individually. To improve resolution near the wall regions, where strong gradients are expected, edge biasing was applied with a bias factor of 10. In addition, curvature-based sizing with a fine relevance setting was used to ensure the mesh properly captured geometric details and key flow features. After mesh generation, the grid quality was assessed using standard mesh metrics, including skewness and element quality.

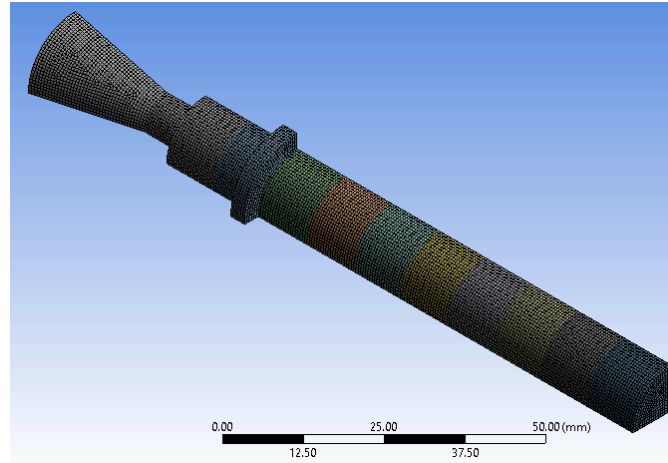


Figure 3: 3D meshed model.

Once the mesh was finalized, the flow analysis was carried out using a steady-state density-based solver for the three-dimensional quarter-symmetric model. A CFD study is conducted employing ANSYS Fluent for all combinations of variables based on a complete factorial design. Based on the governing equations and turbulence formulation, the nozzle flow computations were performed using the Reynolds-averaged Navier–Stokes equations together with the $k-\varepsilon$ turbulence model. The computation employs the K-epsilon turbulent model, chosen for its accuracy and efficiency [17]. The working medium for this study is treated as an ideal gas, specifically air. The variation in viscosity with temperature was modeled using Sutherland’s law, as presented in previous work [17]. The inlet boundary condition was specified in terms of gauge pressure; the outlet pressure was fixed at back pressure. The operating pressure was taken as zero gauge at the duct exit.

For the solution methodology, we used an implicit solver for the steady-flow case with the Roe-FDS flux scheme. To improve numerical accuracy, we adopted second-order upwind discretization for the stream variables, turbulent kinetic energy, and dissipation rate. We obtained the cell-face values using a multidimensional linear reconstruction method. In this method, a higher-order approximation at the cell face is obtained by expanding the cell-centered solution about the cell centroid using a Taylor series. Accordingly, the face value is written as

$$\phi_{f,SOU} = \phi + \nabla\phi \cdot \vec{r} \quad (4)$$

where ϕ is the cell-centered value, $\nabla\phi$ is the gradient in the upstream cell, and $\vec{r}$ is the transposition vector from the centroid of the upstream cell to the face centroid. This formulation improves the spatial accuracy of the numerical prediction by accounting for local variations in the variable within each cell.

Overall, the adopted mesh strategy and numerical settings provided a reliable basis for simulating the compressible flow through the nozzle and enlarged duct. The refinement near critical regions and the use of an appropriate solver and discretization scheme enabled accurate prediction of the pressure field, velocity distribution, and flow development in the base region.

3.4 Grid Convergence Index (GCI) Study

The grid element size plays an important role in CFD investigations. The element size should be adjusted to achieve correct findings with the least calculation time. The grid independence test is performed to optimize the grid size for grid element sizes ranging from 0.1 mm to 5 mm. Table 2 presents the test results for various grid sections and dimensions. Established on the outcomes, the results are constant at a grid element size of 1 mm. The grid element size of 0.5 mm is selected for additional CFD assessment. Table 1 Grid independence test: Number of mesh elements with various element sizes.

Table 2: Mesh convergence tests.

Mesh Element Size in mm	Mesh Nodes	Mesh Elements	Base Pressure
5	2804	652	0.610057
4	2786	632	0.635878
3	3055	720	0.612463
2	6241	1398	0.477677
1	34,164	7884	0.33163
0.5	228,791	55,650	0.332748
0.25	1,719,427	442,519	0.333865
0.1	25,345,765	6,587,205	0.334983

4 Results and Discussion

Pressure at the base is achieved from Fluent software, initially recorded as relative pressure, later converted to dimensionless pressure, and further normalized to atmospheric pressure for dimensionless representation. This transformation enhances the visualization and comprehension of the outcomes. Furthermore, pressure plots are generated in Fluent for detailed analysis.

Pressure plots are illustrated for an L/D ratio of 6, covering nozzle pressure ratios from 2 to 8 and Mach numbers $M = 1.2$ to 1.8 . Figs. 4–23 show the contours of total pressure for different configurations.

4.1 Pressure Contours: $L/D = 6$ Mach (M) = 1.4, and NPR = 2

Figs. 4–8 exhibit pressure contours corresponding to a Mach number of 1.4, an L/D ratio of 6, and an NPR of 2. These illustrations show pressure plots for situations with no cavity and for cavities at various positions along the tube size, from $L/D = 0.5$ to 2 from the nozzle exit. Based on Figs. 4–8, it is evident that the flow from the CD nozzle undergoes overexpansion, and reattachment occurs at a considerable distance downstream in the tube.

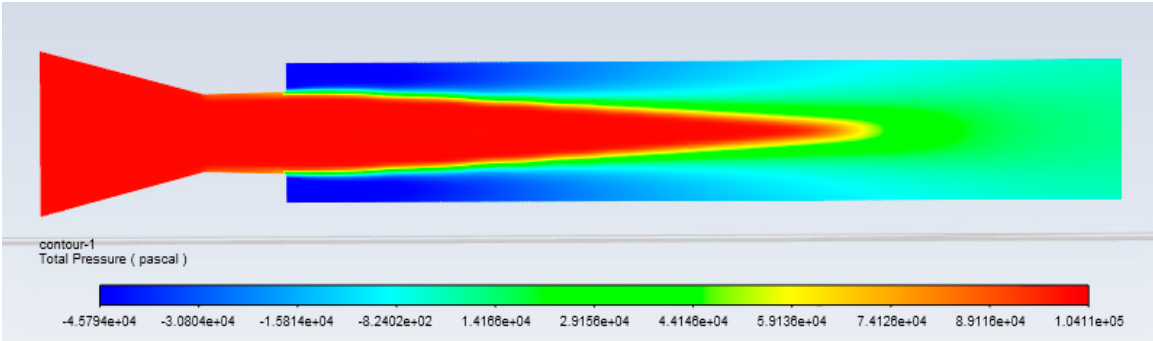


Figure 4: Pressure Contours: Duct without cavity.

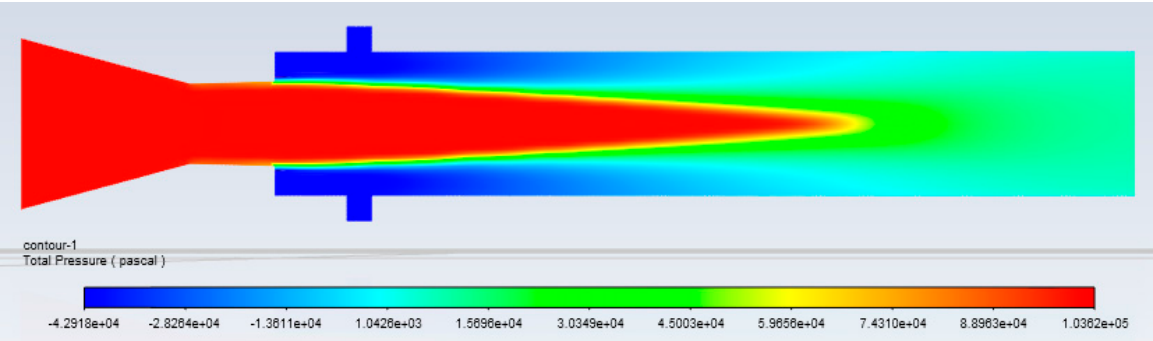


Figure 5: Pressure Contours: Duct with a cavity placed at $L/D = 0.5$.

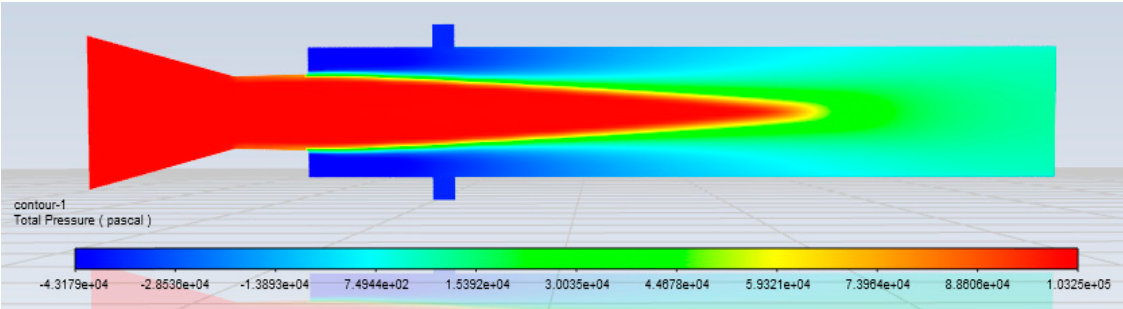


Figure 6: Pressure Contours: Duct with a cavity placed at $L/D = 1.0$.

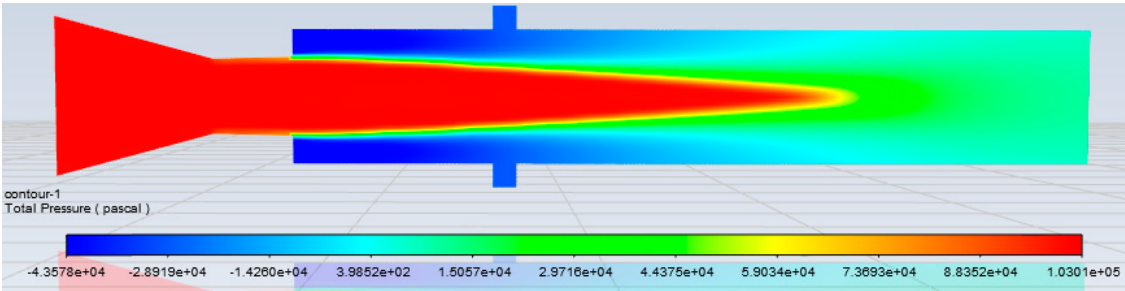


Figure 7: Pressure Contours: Duct with a cavity placed at $L/D = 1.5$.

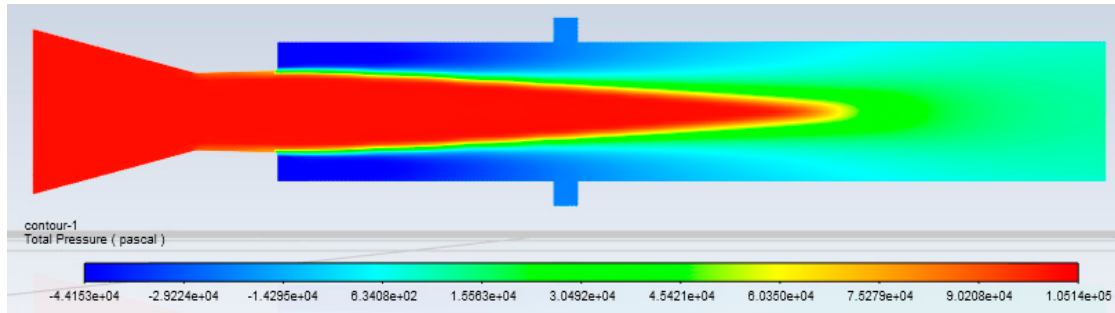


Figure 8: Pressure Contours: Duct with a cavity placed at $L/D = 2.0$.

4.2 Pressure Plots: $L/D = 6$ Mach (M) = 1.4, $NPR = 4$

Figs. 9–13 exhibit pressure contours corresponding to a Mach number $M = 1.4$, an L/D ratio of 6, and an NPR of 4. These illustrations show pressure distributions for situations with no cavity and with a cavity at various positions, over the range $L/D = 0.5$ – 2.0 . Based on Figs. 9–13, it is apparent that the stream from the nozzle undergoes expansion fan and reattaches to the duct near the nozzle exit.

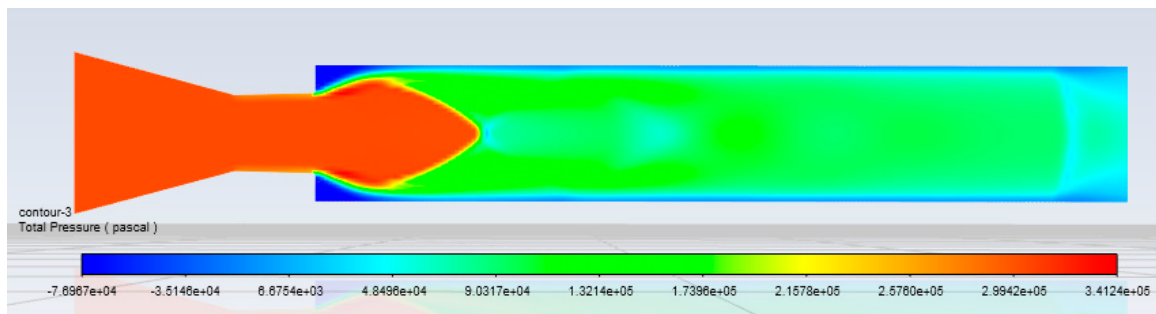


Figure 9: Pressure Contours: Duct without cavity.

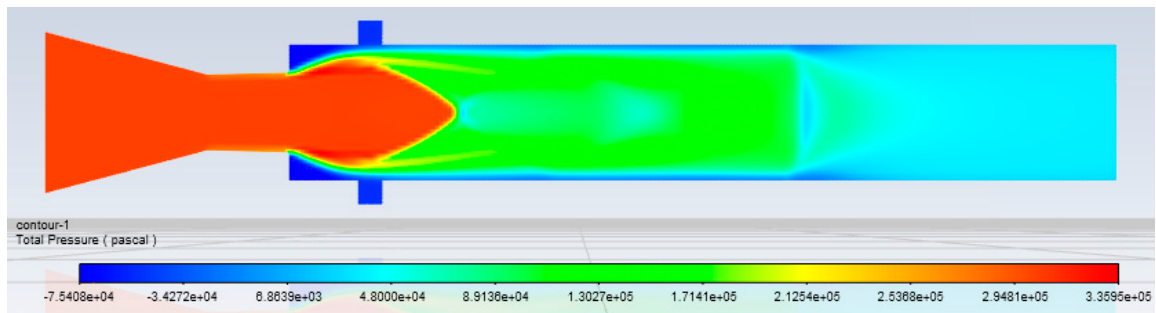


Figure 10: Pressure Contours: Duct with a cavity placed at $L/D = 0.5$.

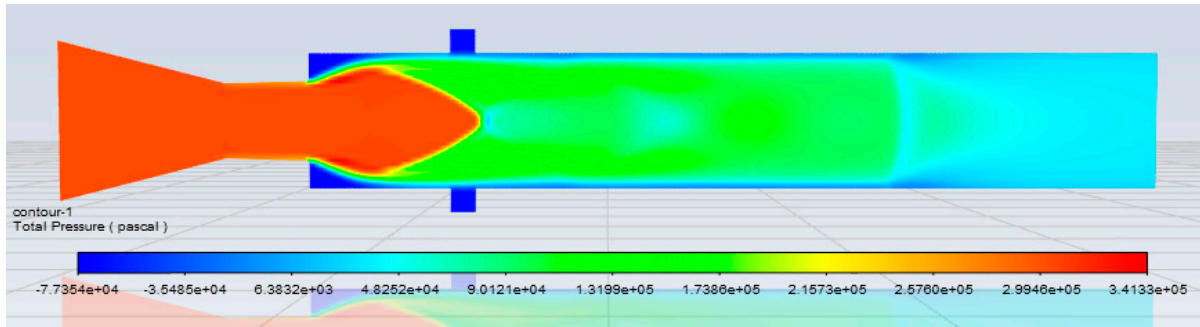


Figure 11: Pressure Contours: Duct with a cavity placed at $L/D = 1.0$.

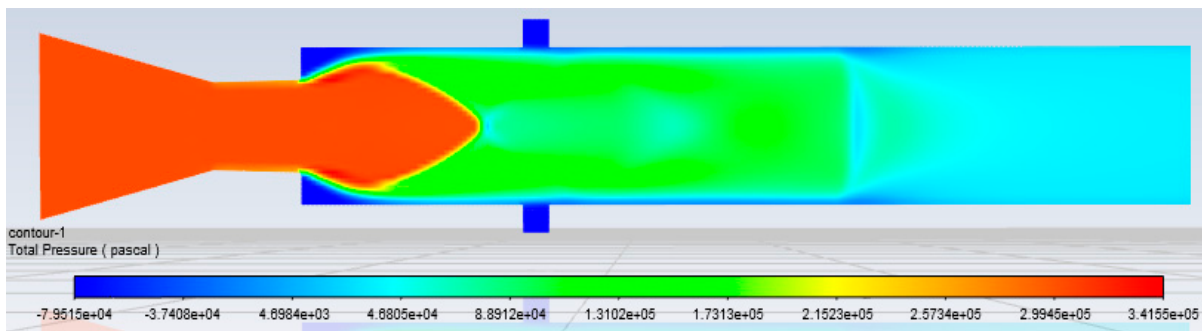


Figure 12: Pressure Contours: Duct with a cavity placed at $L/D = 1.5$.

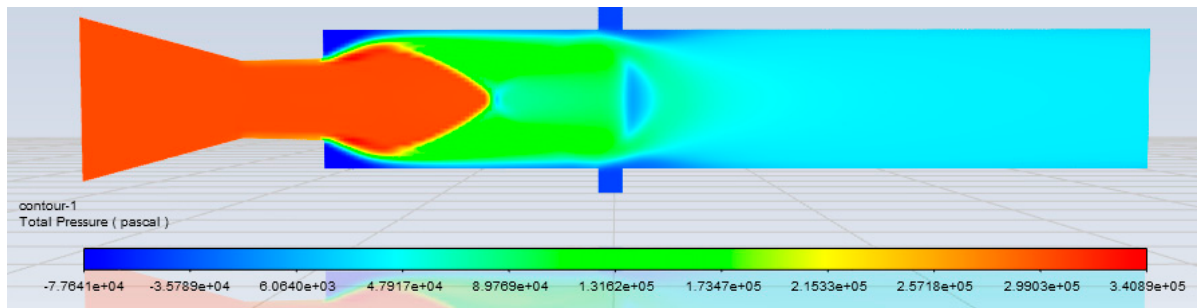


Figure 13: Pressure Contours: Duct with a cavity placed at $L/D = 2.0$.

4.3 Pressure Contours: $L/D = 6$, Mach (M) = 1.4, and NPR = 6

Figs. 14–18 exhibit pressure contours corresponding to a Mach number of 1.4, an L/D ratio of 6, and an NPR of 6. These illustrations depict pressure distributions for conditions with no cavity and with a cavity at various locations, $L/D = 0.5$ to 2. Based on Figs. 14–18, it is evident that the nozzle flow undergoes significant expansion and reattaches to the tube very close to the nozzle exit.

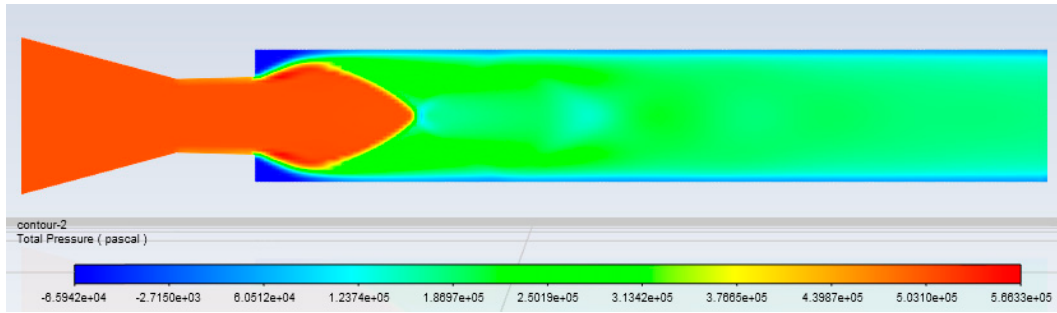


Figure 14: Pressure Contours: Duct without cavity.

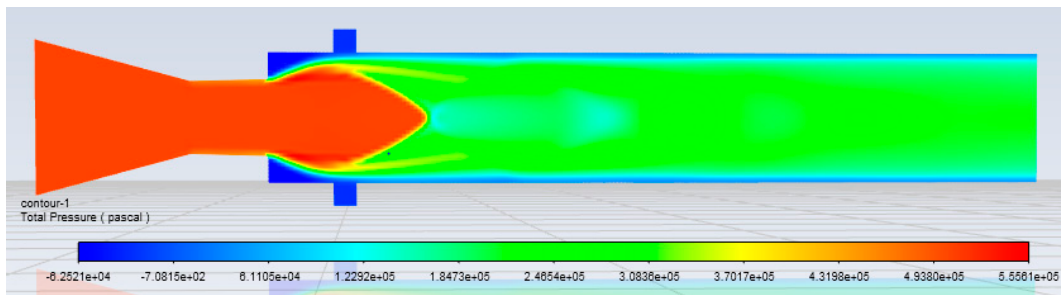


Figure 15: Pressure Contours: Duct with a cavity placed at $L/D = 0.5$.

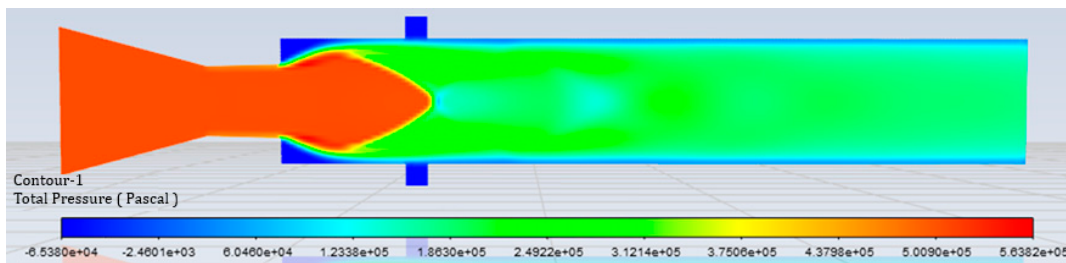


Figure 16: Pressure Contours: Duct with a cavity placed at $L/D = 1.0$.

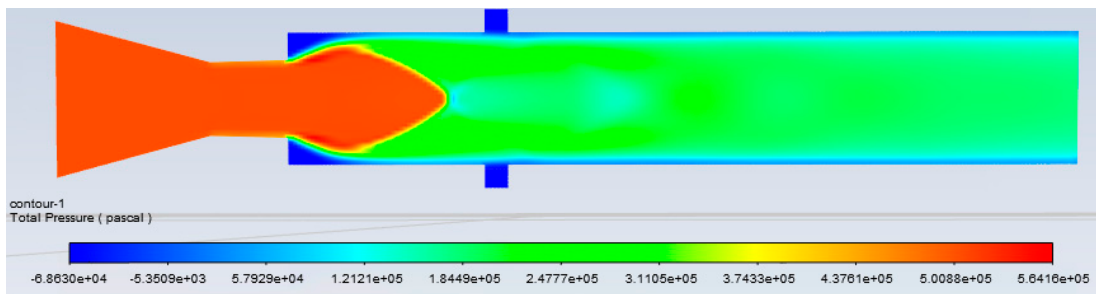


Figure 17: Pressure Contours: Duct with a cavity placed at $L/D = 1.5$.

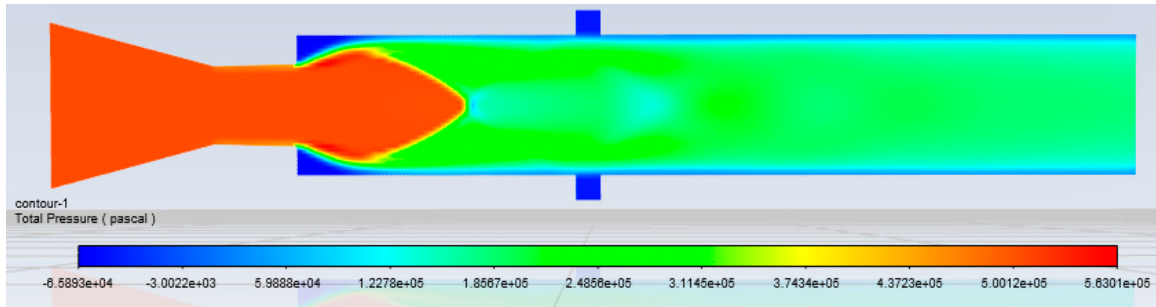


Figure 18: Pressure Contours: Duct with a cavity placed at $L/D = 2.0$.

4.4 Pressure Contours: $L/D = 6$, $Mach(M) = 1.4$, and $NPR = 8$

Figs. 19–23 exhibit pressure contours corresponding to a Mach number of 1.4, an L/D ratio of 6, and an NPR of 8. These illustrations depict pressure spreading for scenarios with no cavity and with a cavity at various locations. Based on Figs. 19–23, it is evident that the stream from the nozzle undergoes substantial under-expansion, and the flow reattaches to the tube very close to the nozzle exit.

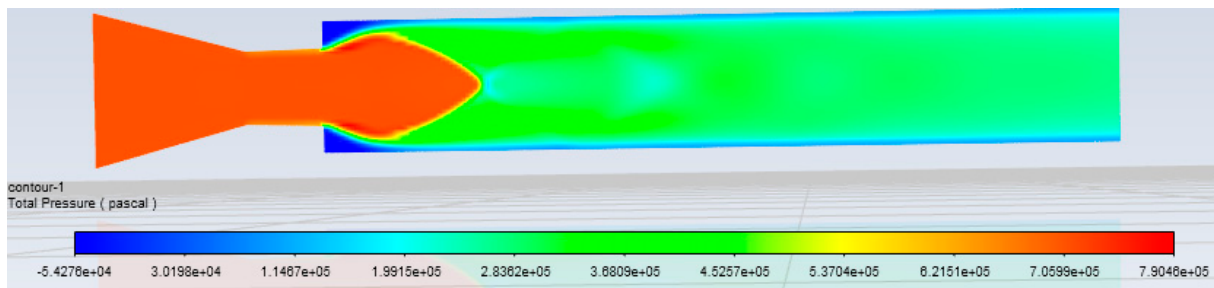


Figure 19: Pressure Contours: Duct without cavity.

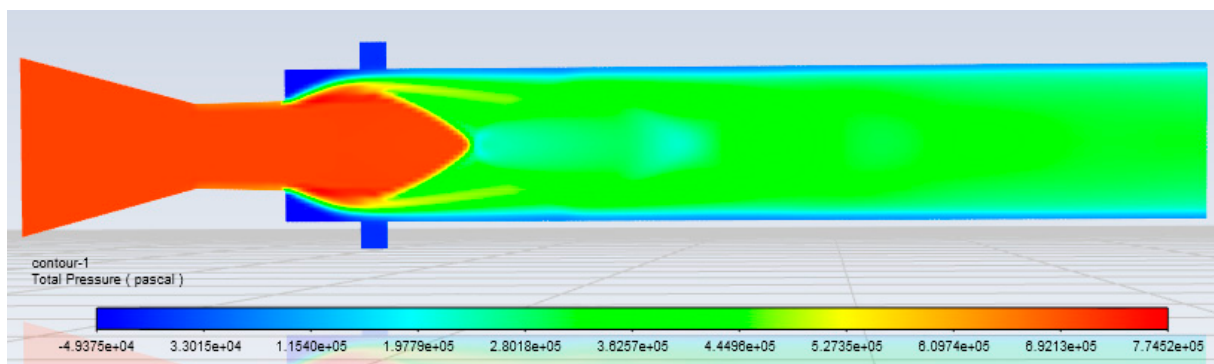


Figure 20: Pressure Contours: Duct with a cavity placed at $L/D = 0.5$.

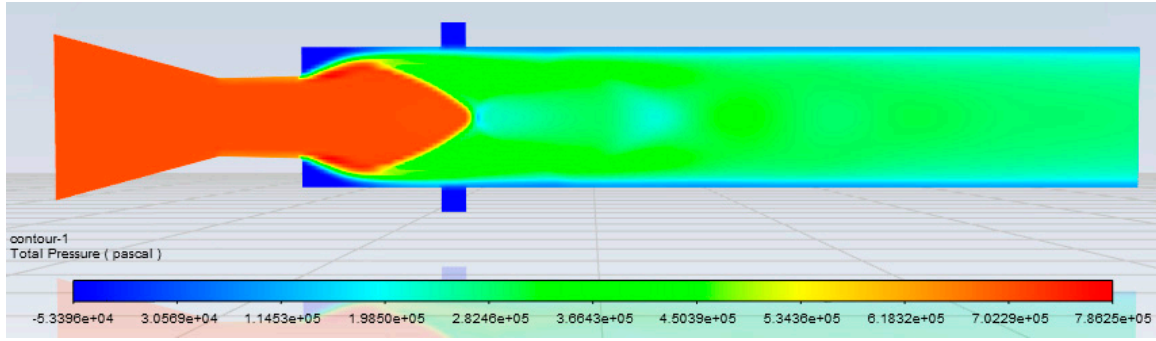


Figure 21: Pressure Contours: Duct with a cavity placed at $L/D = 1.0$.

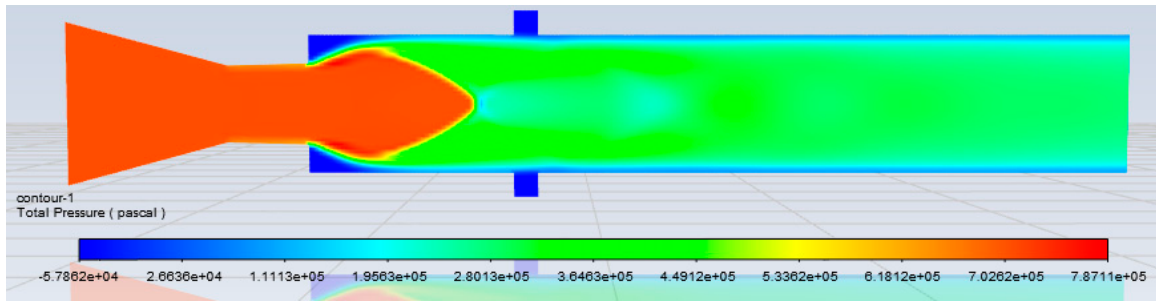


Figure 22: Pressure Contours: Duct with a cavity placed at $L/D = 1.5$.

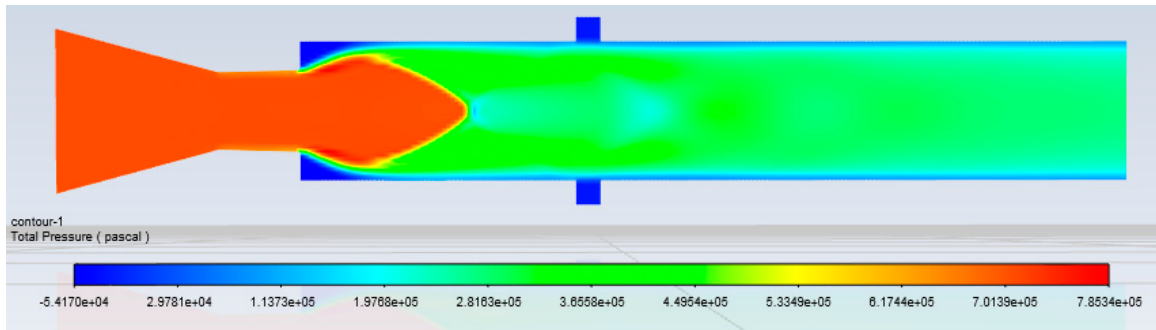


Figure 23: Pressure Contours: Duct with a cavity placed at $L/D = 2.0$.

It is evident that the expansion levels significantly influence the reattachment point in the expanded duct, as indicated by the pressure contours. Lower nozzle pressure ratios result in a longer reattachment length, whereas higher under-expansion levels reduce it. The efficacy of the cavity depends on the reattachment point and the cavity's specific location. Due to the cyclic nature of vortex shedding, the overall flow pattern within the expanded duct may exhibit oscillations.

The fluctuations can happen, particularly pronounced under specific geometric and inertia parameter configurations. The strength of the main vortex at the base is primarily affected by inertia levels, expansion levels, reattachment size, and area ratio. A comprehensive understanding of the changes in base pressure is presented in the subsequent section.

4.5 CFD Analysis Findings for Mach (M) = 1.2

When a flow-regulation mechanism is applied to govern the stream in a tube with an abrupt rise in area, it is imperative to scrutinize the effects of these strategies on the stream pattern in the tube. When applying passive control methods, it is mandatory to ensure that the passive control cavity does not adversely affect the flow area in the enlarged tube. At Mach $M = 1.2$, the NPRs examined are such that at these NPRs, the flow from the nozzle is over- or under-expanded. The expansion levels at these NPRs are $P_e/P_a = 0.83, 1.65, 2.5$, and 3.31 . From these values, it is evident that at $NPR = 2$, only the nozzle is over-expanded, whereas at the remaining NPRs, the nozzle exit flow is under-expanded.

Fig. 24a–d illustrates the base pressure changes, considering the NPR with and without cavity, and different cavity placements when Mach $M = 1.2$. The findings indicate that the cavity is useful only at $L/D = 0.5$, yielding a maximum increase in base pressure. However, for the other cavity locations, namely from $1D$ to $2D$, the influence of the cavity presence is minimal. When we investigate the outcome, keeping the flow physics in mind, we find that when the stream leaving the nozzle is over-expanded, the dividing streamline deflects toward the base, leading to a short reattachment length and an increased base pressure. It is also detected that this decreasing trend continues up to $NPR = 4$, whereas the NPR needed for the ideal expansion is 2.42 . That shows that another factor is influencing the flow, in addition to NPR. We know that the reattachment length rests on the Mach number, NPR, L/D ratio, and the diameter ratio. In this case, the area ratio plays an important role in determining the reattachment length.

On the other hand, when the nozzle is under-expanded, the nozzle exit pressure exceeds the ambient pressure; hence, the flow expands until the flow stream pressure equals the atmospheric pressure. While the nozzle flow undergoes stream expansion, the stream deflects away from the base, increasing the reattachment length and the base pressure.

There is another factor that is responsible for this trend in the base pressure, which is the cavity location, unlike other passive control techniques, where, in any case, the flow stream will experience the presence of the control mechanism, irrespective of the location of the control, and will influence the flow stream. As far as the cavity is concerned, its impact will be felt when the shear layer passes through it. Otherwise, if the flow reconnects to the tube side before or after the cavity position, the cavity location will not affect it.

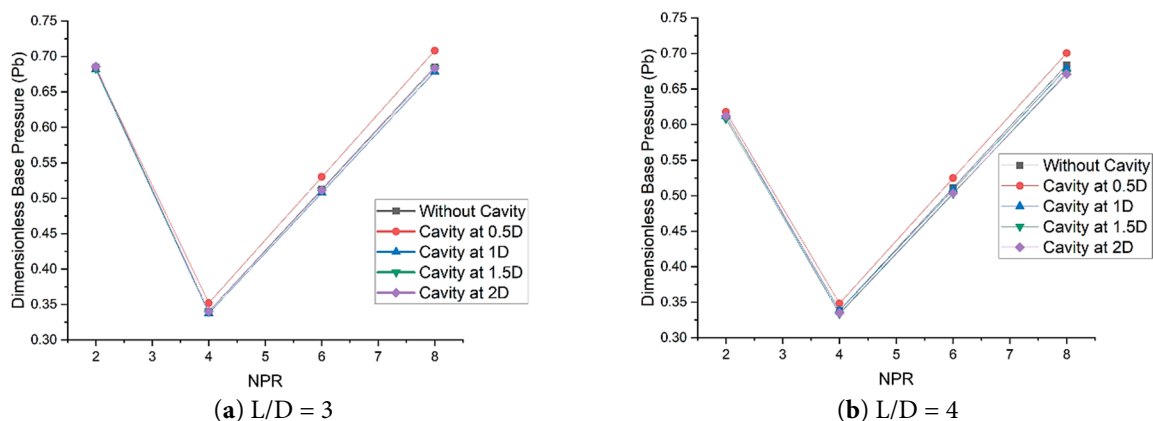


Figure 24: Cont.

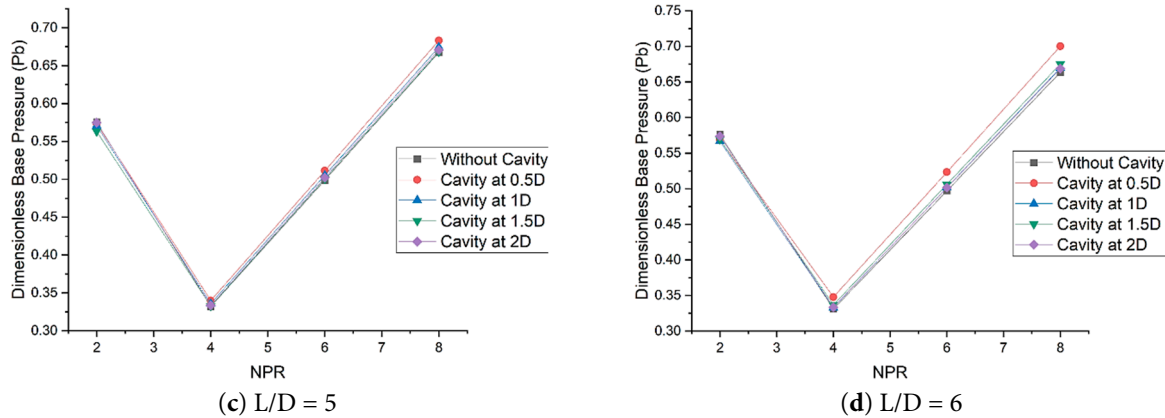


Figure 24: Base pressure vs. NPR, at Mach number = 1.2 and various L/D and cavity locations. (a) $L/D = 3$, (b) $L/D = 4$, (c) $L/D = 5$, (d) $L/D = 6$.

4.6 CFD Analysis Findings for Mach $M = 1.4$

Fig. 25a–d illustrates the base pressure alterations, considering the NPRs with no cavity and with a cavity, and different cavity positions when Mach 1.4. The NPR required for ideal expansion is 3.2. The findings indicate that the cavity is successful only at a location 0.5D from the cavity. Meanwhile, at other cavity locations, the alteration in base pressure is minor. As discussed earlier, the reattachment point is primarily determined by the diameter ratio; the area ratio is 2.89, implying a diameter ratio of 1.7. For this diameter ratio, the reattachment is expected to occur at a distance less than $L/D = 1$. Hence, the effectiveness remains at $L/D = 0.5$, even though the Mach number increases from $M = 1.2$ to 1.4. Since the NPRs of the simulations remained in the range 2 to 8, owing to a rise in the Mach number, the level of expansion changed, with $P_e/P_a = 0.625, 1.25, 1.88$, and 2.5. Compared to the previous Mach number, there is a decrease in the under-expansion level and an increase in the over-expansion levels. Due to changes in the expansion levels, the base pressure changes marginally in magnitude. Also, there is a marginal change in the control mechanism's effectiveness due to the increased inertia levels.

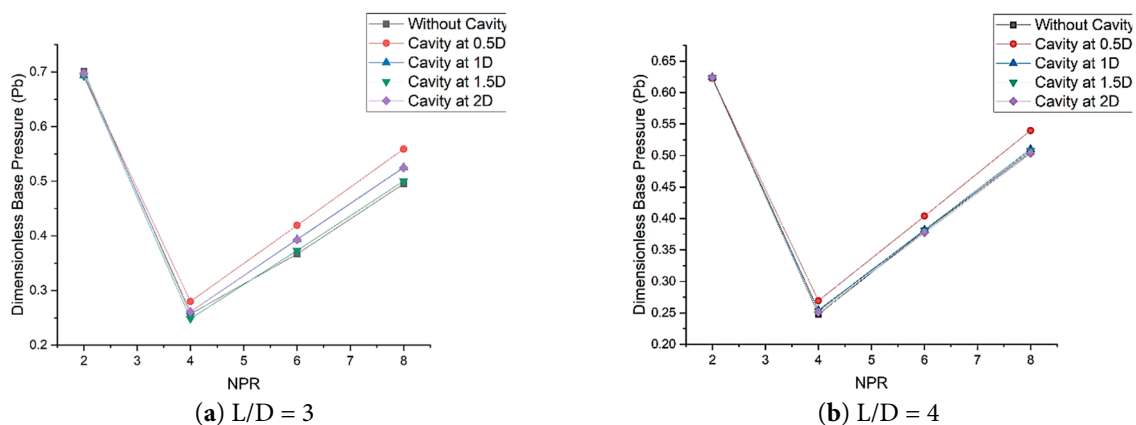


Figure 25: Cont.

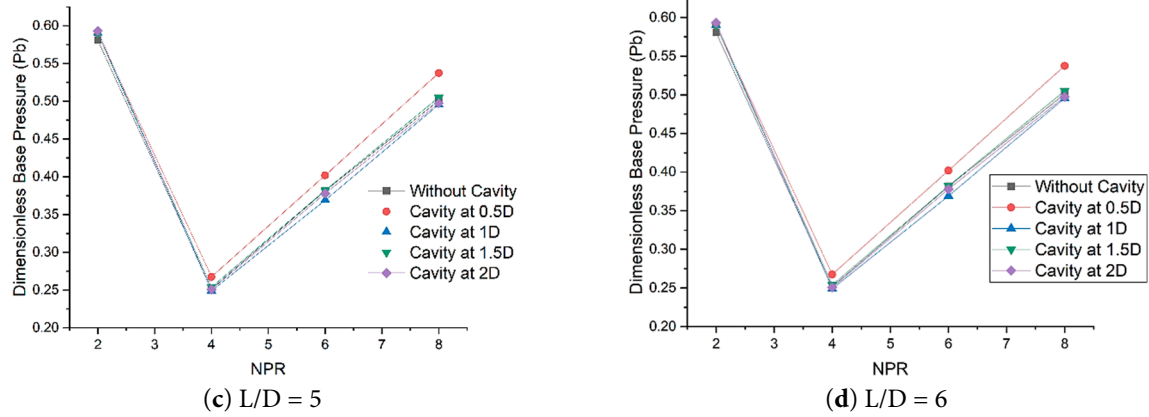


Figure 25: Base pressure vs. NPR, at Mach number = 1.4 and various L/D and cavity locations. (a) $L/D = 3$, (b) $L/D = 4$, (c) $L/D = 5$, (d) $L/D = 6$.

4.7 CFD Analysis Findings for Mach $M = 1.6$

Fig. 26a–d illustrates the base pressure variants, considering the level of expansion with and without a cavity, and different cavity positions, when Mach $M = 1.6$. The NPR necessary for the correct expansion is 4.3. Since the NPRs of the simulations remained the same, owing to a rise in the Mach number, the level of expansion changed, with values of $Pe/Pa = 0.47, 0.76, 1.4$, and 1.86 . Due to higher Mach numbers and changes in expansion level, there is greater variation in base pressure values. The findings indicate that the cavity is useful at 0.5D as the duct diameter remained fixed at 17 mm. Meanwhile, at other cavity locations, the alteration in base pressure is minor.

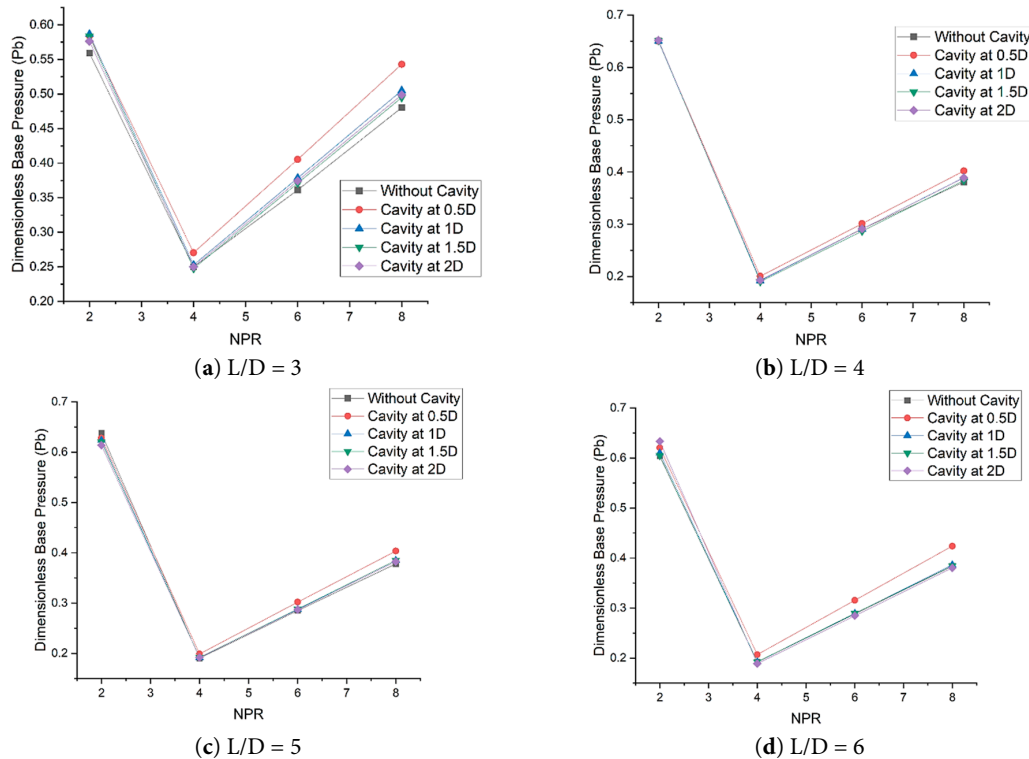


Figure 26: Base pressure vs. NPR, at Mach number = 1.6 and various L/D and cavity locations. (a) $L/D = 3$, (b) $L/D = 4$, (c) $L/D = 5$, (d) $L/D = 6$.

4.8 CFD Analysis Findings for Mach $M = 1.8$

Fig. 27a–d depicts the variations in base pressure concerning the level of expansion, without a cavity and with a cavity, and numerous cavity positions when Mach $M = 1.8$. At Mach $M = 1.8$, the NPR required for perfect expansion is 5.8, significantly higher than at the lowest Mach number, $M = 1.2$. Since the NPRs of the simulations remained in the range 2 to 8, owing to a rise in the Mach number, the level of expansion changed, with $Pe/Pa = 0.344, 0.69, 1.04$, and 1.38 . From Fig. 27, it is evident that the base pressure pattern at Mach $M = 1.8$ is totally altered compared to the previous three Mach numbers. This trend can be attributed to an increase in the Mach number, which, in turn, leads to a longer reattachment length. As inertia and expansion levels rise, the shock wave strength changes, altering the base pressure. For the duct length of 51 mm, the influence of atmospheric pressure is clearly seen in Fig. 27a, where the base pressure values differ from those in the larger duct segment; the effect of the ambient pressure is almost negligible.

Notably, the cavity proves successful only at $NPR = 4$, enhancing base pressure at specific locations. However, at NPR values of 2, 6, & 8, the disparity in base pressure utilizing the cavity is irrelevant. Based on the above results, the cavity is most effective at 0.5D. The detailed results for the base pressure changes due to a cavity at a 0.5D location are discussed in the following section.

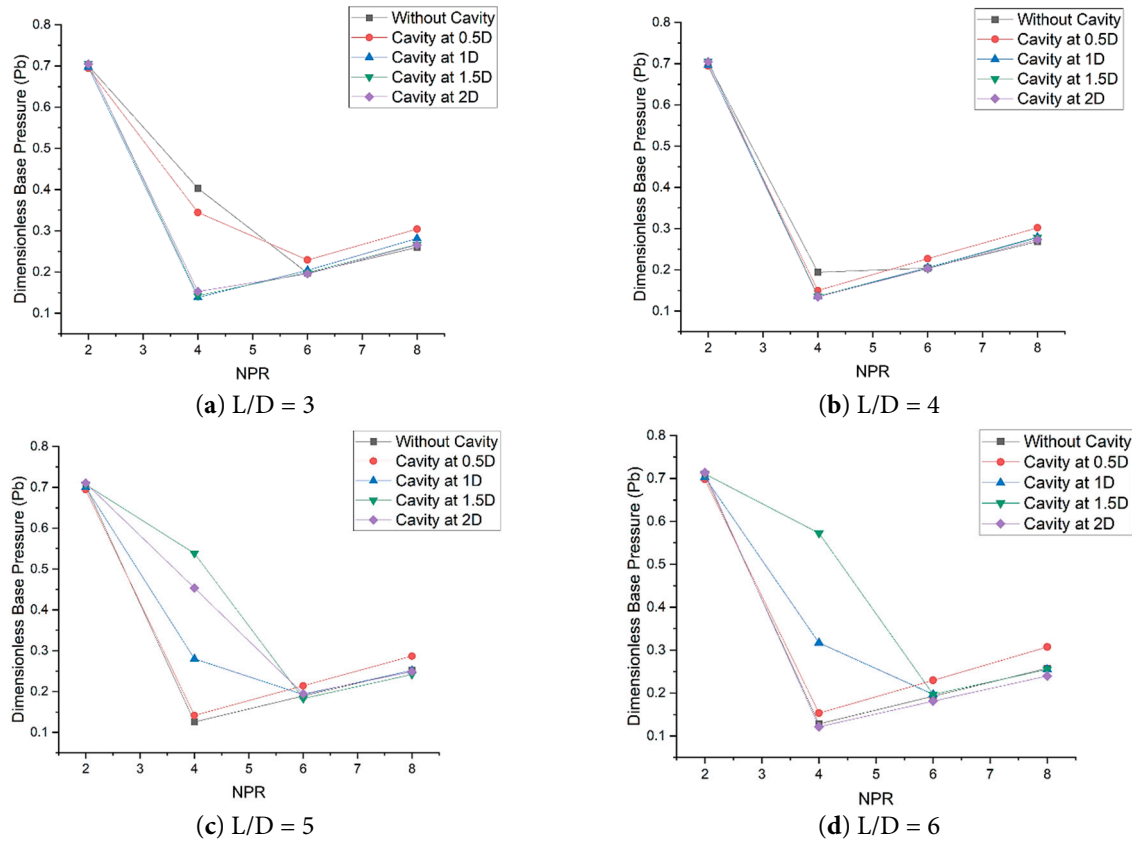


Figure 27: Base pressure vs. NPR, at Mach number = 1.8 and various L/D and cavity locations. (a) $L/D = 3$, (b) $L/D = 4$, (c) $L/D = 5$, (d) $L/D = 6$.

4.9 Effectiveness of Cavity at 0.5D Location

Every time a control mechanism is introduced to regulate the base pressure in a tube with an abrupt increase in duct area, it is crucial to assess the influence of these flow-control techniques on the duct's stream pattern. It must be ensured that the implementation of passive control does not adversely affect

the flow dynamics within the duct. Fig. 28a–d shows detailed results of base pressure variations at a 0.5D location with a cavity, compared with those without a cavity. Based on the results, the cavity is efficient in enhancing the base pressure across all parameter combinations at the 0.5D location.

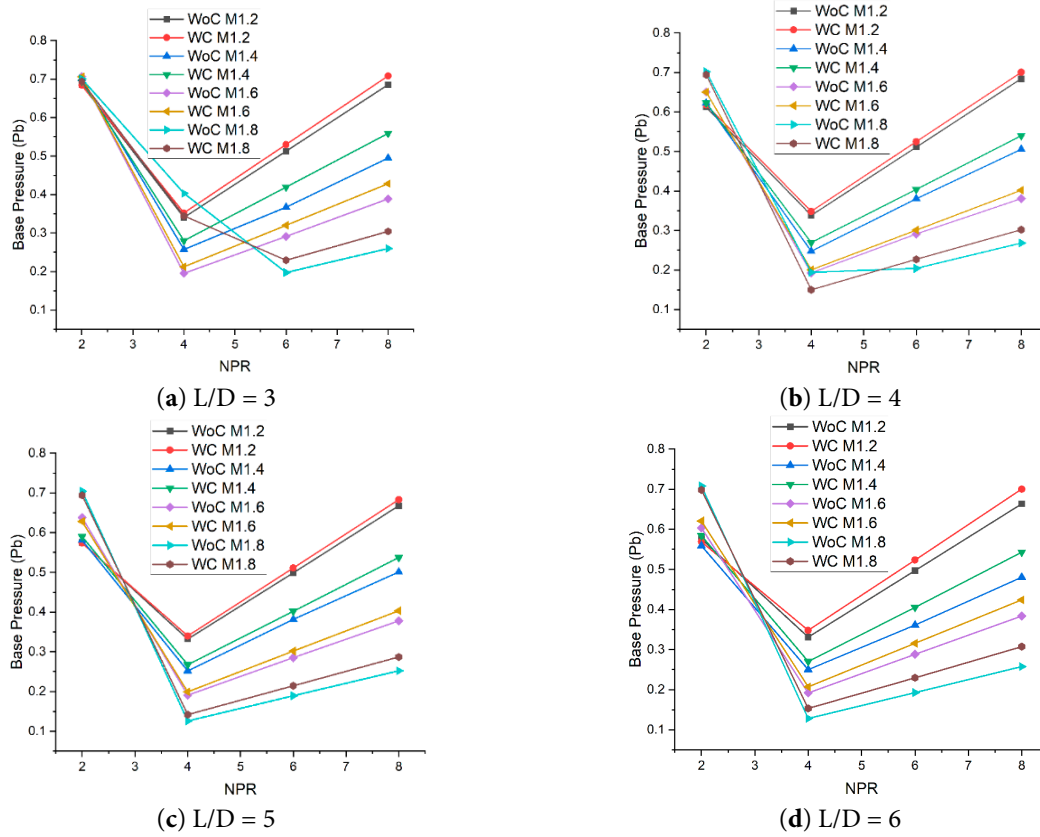


Figure 28: Base pressure vs. NPR, at various Mach numbers & L/D ratios at cavity location = 0.5D. (a) L/D = 3, (b) L/D = 4, (c) L/D = 5, (d) L/D = 6.

4.10 Effectiveness of Cavity at 1D Location

Fig. 29a–d exhibits the detailed results of base pressure variations with a cavity at a 1D location, and these results are compared with the base pressure results without a cavity. Based on the results, the cavity is efficient in improving the base pressure across various parameter combinations at the 1D location.

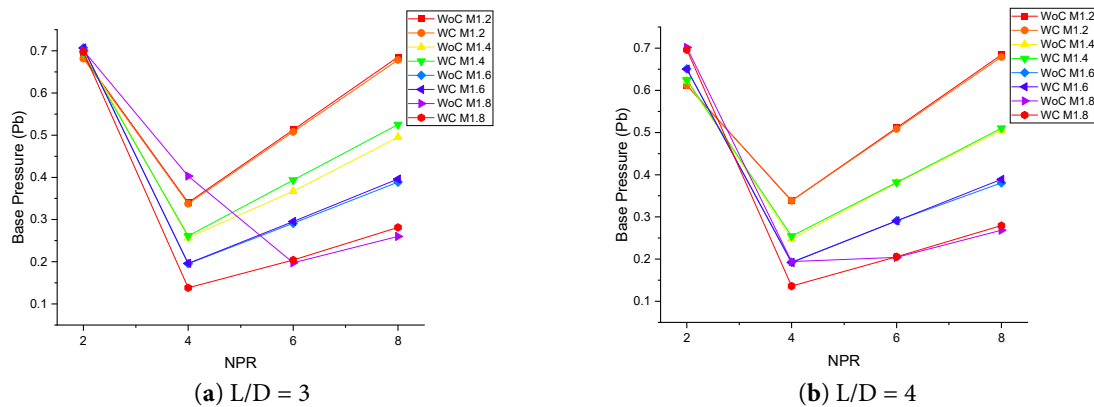


Figure 29: Cont.

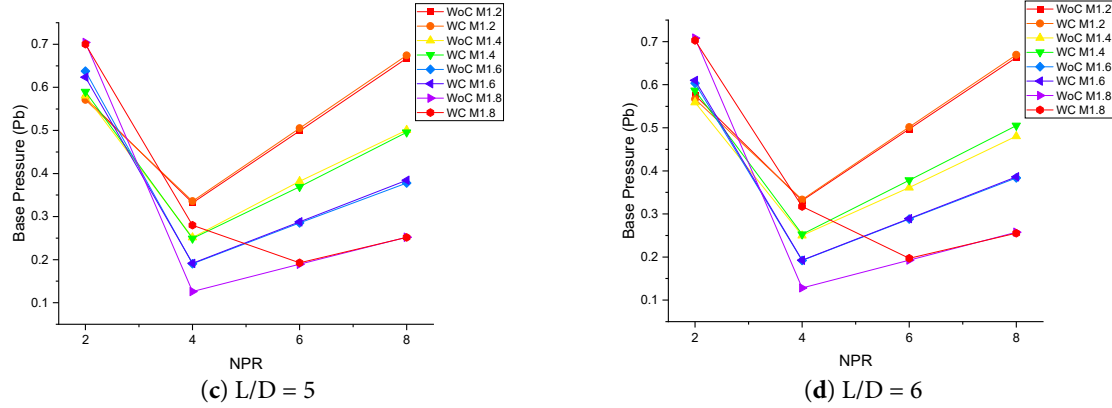


Figure 29: Base pressure vs. NPR, for various Mach numbers & L/D ratios at cavity location = 1D. (a) $L/D = 3$, (b) $L/D = 4$, (c) $L/D = 5$, (d) $L/D = 6$.

4.11 Effectiveness of Cavity when Placed at $L/D = 1.5$

Fig. 30a–d shows the detailed outcomes of base pressure variations with a cavity at a 1.5D location, and these outcomes are matched with those without a cavity. Based on the results, the cavity successfully enhances the base pressure for some parameter combinations at the 1.5D location.

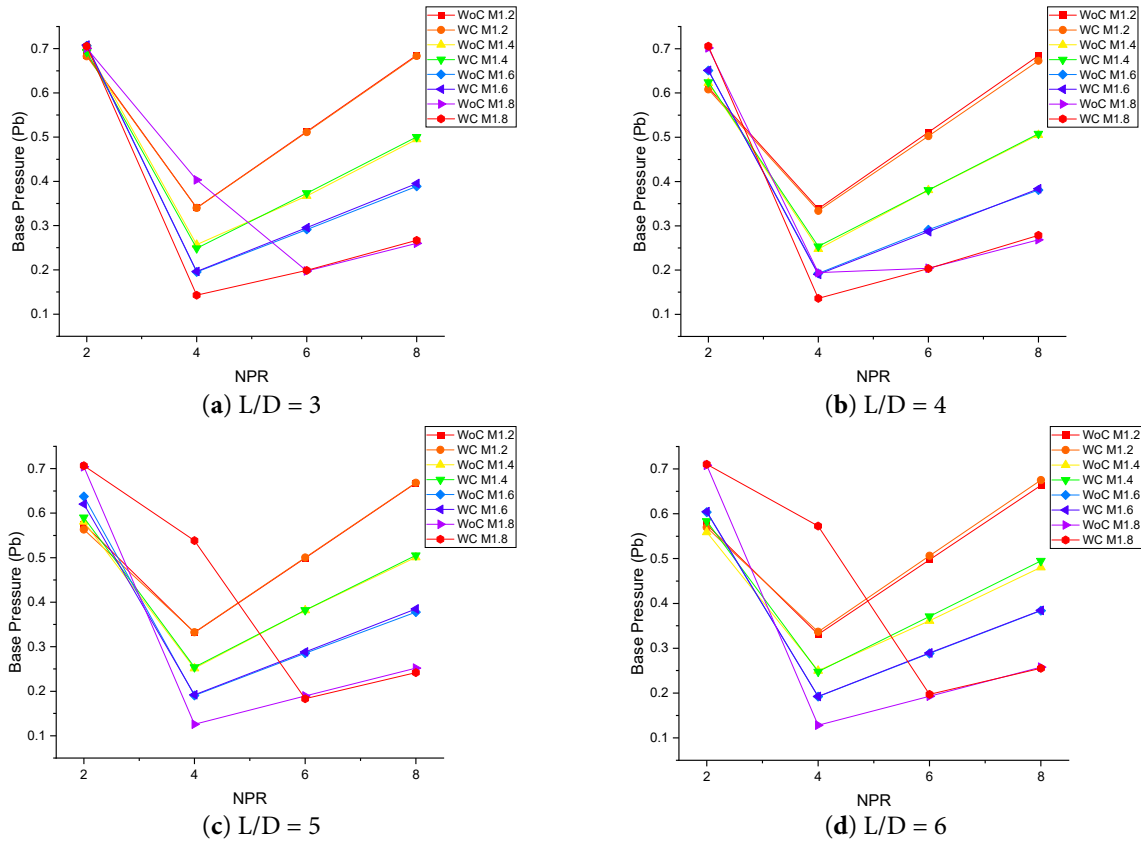


Figure 30: Base pressure vs. NPR, at various Mach numbers & L/D ratios at cavity location = 1.5D. (a) $L/D = 3$, (b) $L/D = 4$, (c) $L/D = 5$, (d) $L/D = 6$.

4.12 Effectiveness of Cavity at 2D Location

Fig. 31a–d shows the detailed results of base pressure variations with a cavity at a 2D location, and these results are compared with the base pressure results without a cavity. Based on the results, the cavity enhances base pressure in some parameter combinations at the 2D location.

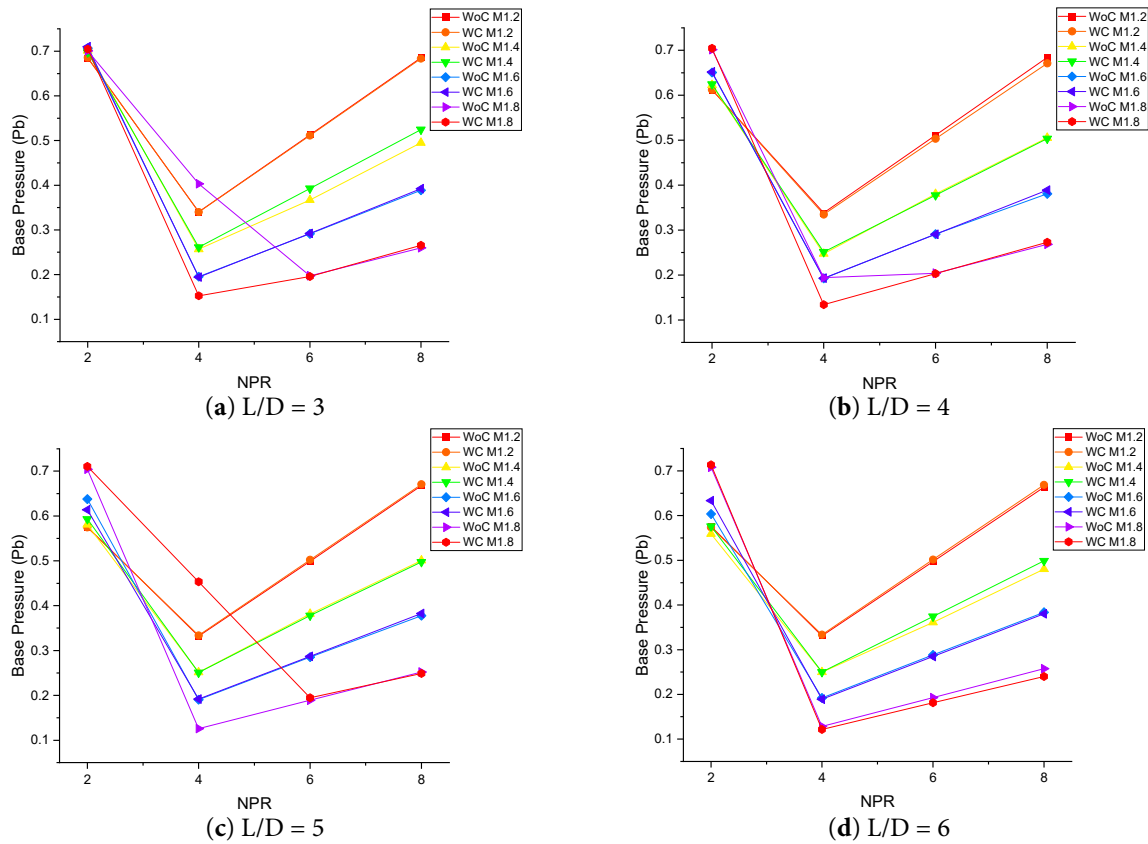


Figure 31: Base pressure vs. NPR, for various Mach numbers & L/D ratios at cavity location = 2D. (a) L/D = 3, (b) L/D = 4, (c) L/D = 5, (d) L/D = 6.

4.13 Turbulence Intensity Contours for L/D = 6, M = 1.4, & Cavity Position L/D = 0.5

The Turbulence Intensity Contours in Fig. 32a–d show the study results. The results for L/D = 6, Mach = 1.4, and NPR = 2 show the cavity at 0.5D. The flow conditions approach over-expansion at NPR = 2, as the flow reaches its over-expansion limit. The turbulence-intensity contours show that the boundary layer forms at the nozzle exit, creating a primary recirculation zone in the base region. The 0.5D cavity location establishes a shear-layer interaction that enables controlled mixing while reducing base-area turbulence fluctuations. The base pressure rises owing to this interaction, confirming that the cavity acts as an effective control point. The reattachment length is moderate. The nozzle begins to transition to under expanded flow conditions at NPR = 4. The higher NPR condition shows that the base pressure drops due to under-expansion, leading to organized turbulence patterns that form a complex structure.

The nozzle operates under expanded at NPR = 6. The turbulence intensity contours show that turbulence intensity increases due to stronger expansion waves and increased downstream turbulence. The cavity reduces turbulence by directly contacting the separated flow, thereby reducing disruptions in the base area. The controlled interaction of flow with higher NPR values leads to increased base pressure, as the

conclusion shows that cavity effectiveness persists at those elevated NPR levels. The flow condition at $\text{NPR} = 8$ becomes highly under expanded, leading to strong expansion waves and increased turbulence levels in the combining layer. The viscous layer contains most of the turbulence, but cavity effects create stable recirculation behavior in the base area.

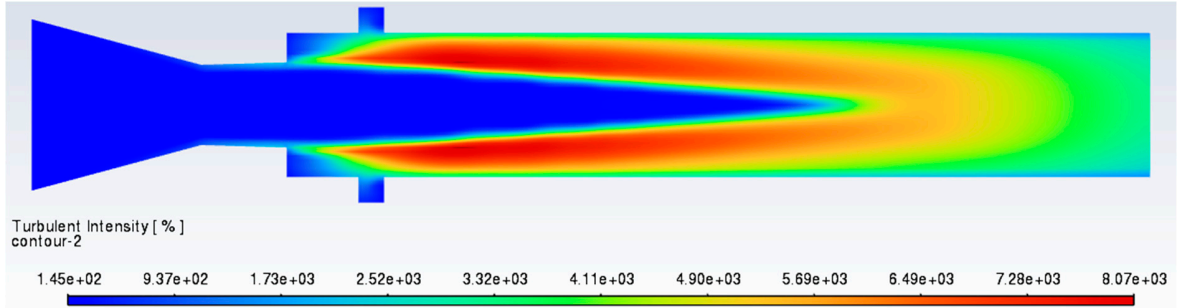
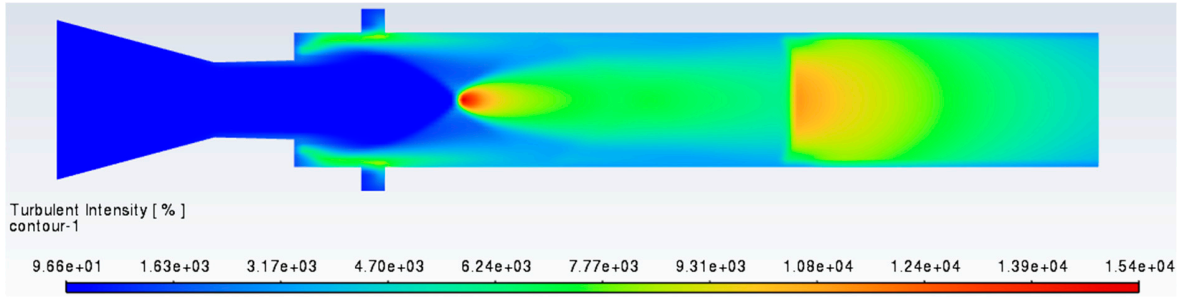
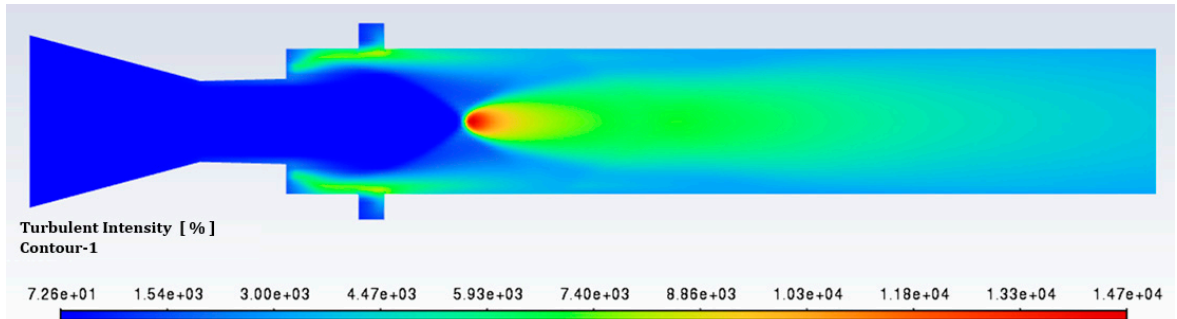
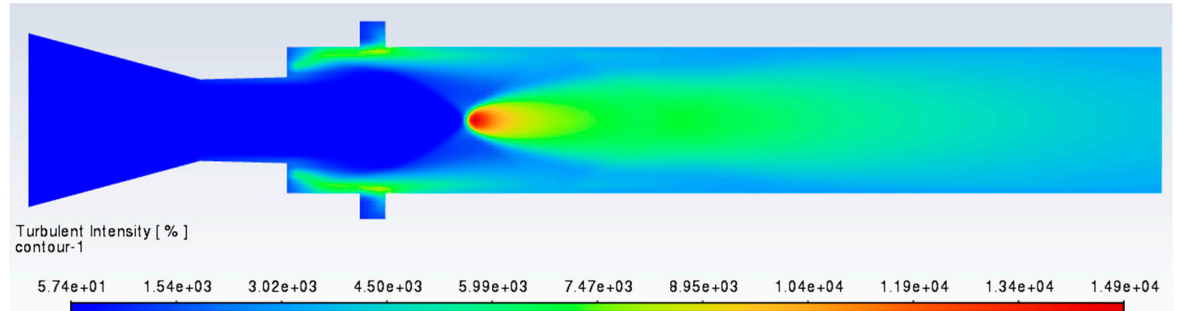
(a) $\text{NPR} = 2$ (b) $\text{NPR} = 4$ (c) $\text{NPR} = 6$ (d) $\text{NPR} = 8$

Figure 32: Turbulence Intensity Contours for $L/D = 6$, $\text{Mach} = 1.4$ and cavity position = $0.5D$. (a) $\text{NPR} = 2$, (b) $\text{NPR} = 4$, (c) $\text{NPR} = 6$, (d) $\text{NPR} = 8$.

4.14 Particle Pathlines for $L/D = 6$, $Mach = 1.4$ and Cavity Location $L/D = 0.5$

Fig. 33a–d displays the particle path lines. The particle path lines establish a recirculation zone that appears behind the sudden expansion. The cavity generates secondary vortical structures that interact with the primary shear-layer flow. The effective reattachment length decreases during this interaction, thereby improving base-mixing performance and enhancing base-pressure regulation. The path lines at NPR = 4 show that flow acceleration increases and the recirculation region lengthens due to under-expansion. The cavity maintains shear-layer control by preventing the reattachment point from moving downstream. The path lines at NPR = 6 show that expanding jets become stronger. The recirculation bubble operates at higher energy levels as its size decreases. The cavity disturbs large vortices, creating more compact and stable base-flow patterns that increase base pressure. The path lines at NPR = 8 show that energetic expansion leads to increased mixing behaviors. The base recirculation area maintains its control through the cavity system, which establishes structured vortex interaction while stronger flow acceleration occurs.

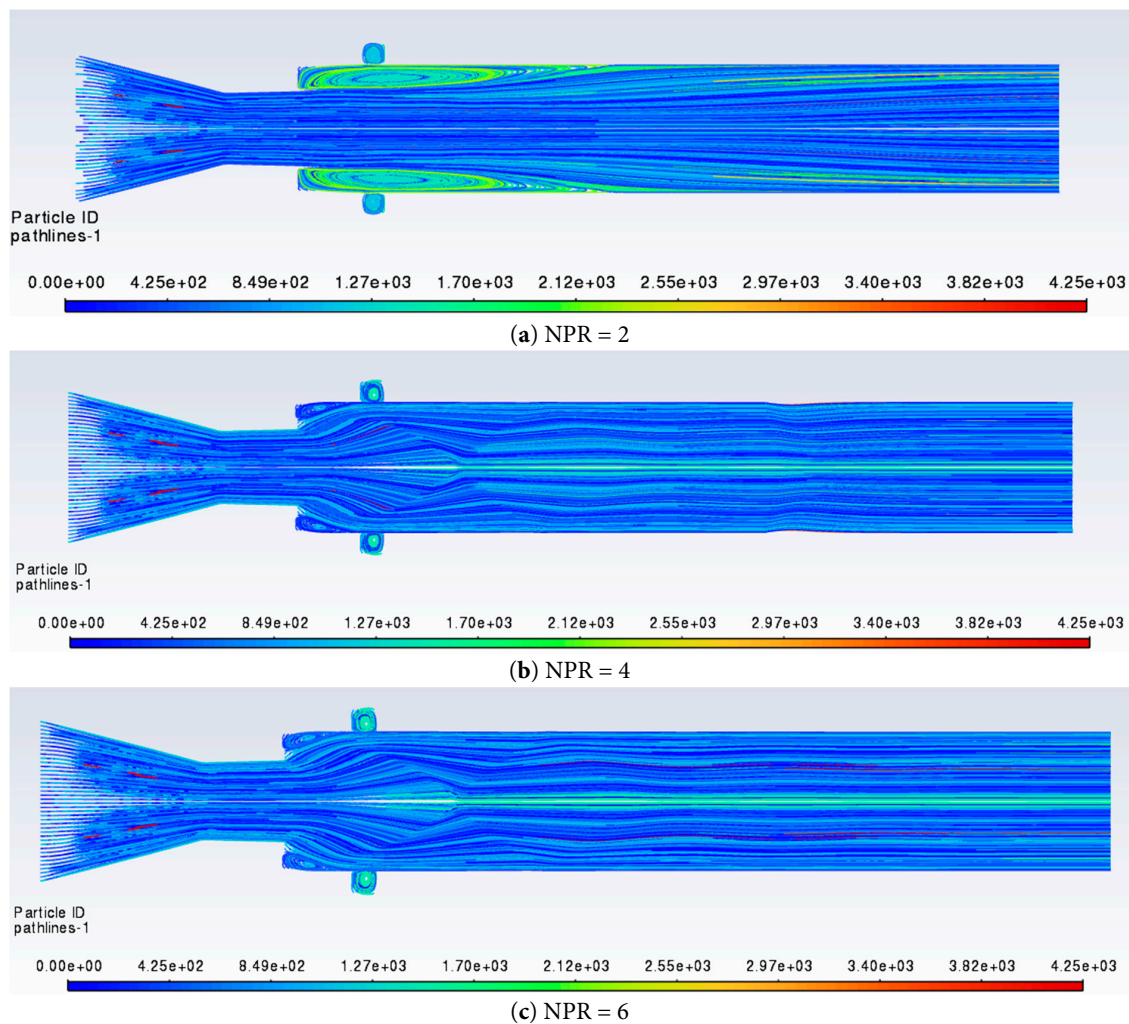


Figure 33: Cont.

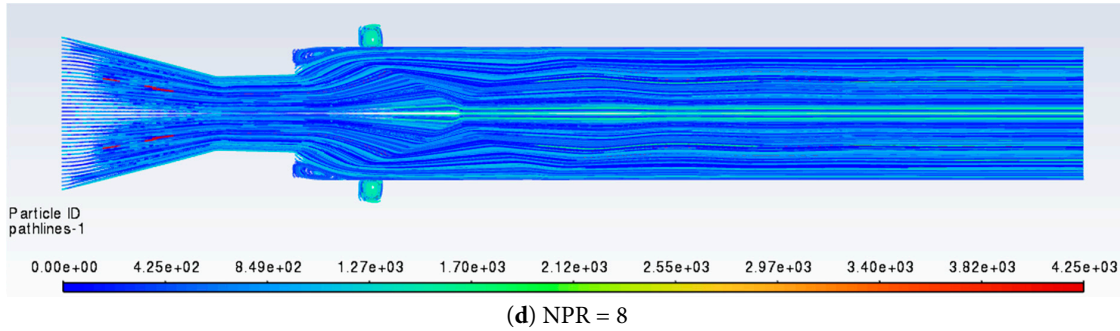


Figure 33: Particle Pathlines for $L/D = 6$, $Mach = 1.4$ and cavity location $= 0.5D$. (a) NPR = 2, (b) NPR = 4, (c) NPR = 6, (d) NPR = 8.

4.15 Static Temperature Contours: $L/D = 6$, $M = 1.4$, & Cavity Position $L/D = 0.5$

Fig. 34a–d shows static temperature contours. At NPR = 2, the base region exhibits higher temperatures because its flow pattern induces recirculation. The cavity improves thermal energy distribution throughout the area by activating mixing, which keeps the thermal field near the base at a constant temperature. The temperature contours at NPR = 4 show that the temperature gradients increase between the jet core and the recirculation zone. The base zone at NPR = 6 and 8 becomes more thermally uniform as the cavity enhances mixing among the high-velocity cold-core flow and the warmer recirculating flow. The improved thermal uniformity near the base aligns with the observed growth in base pressure under highly under-expanded conditions.

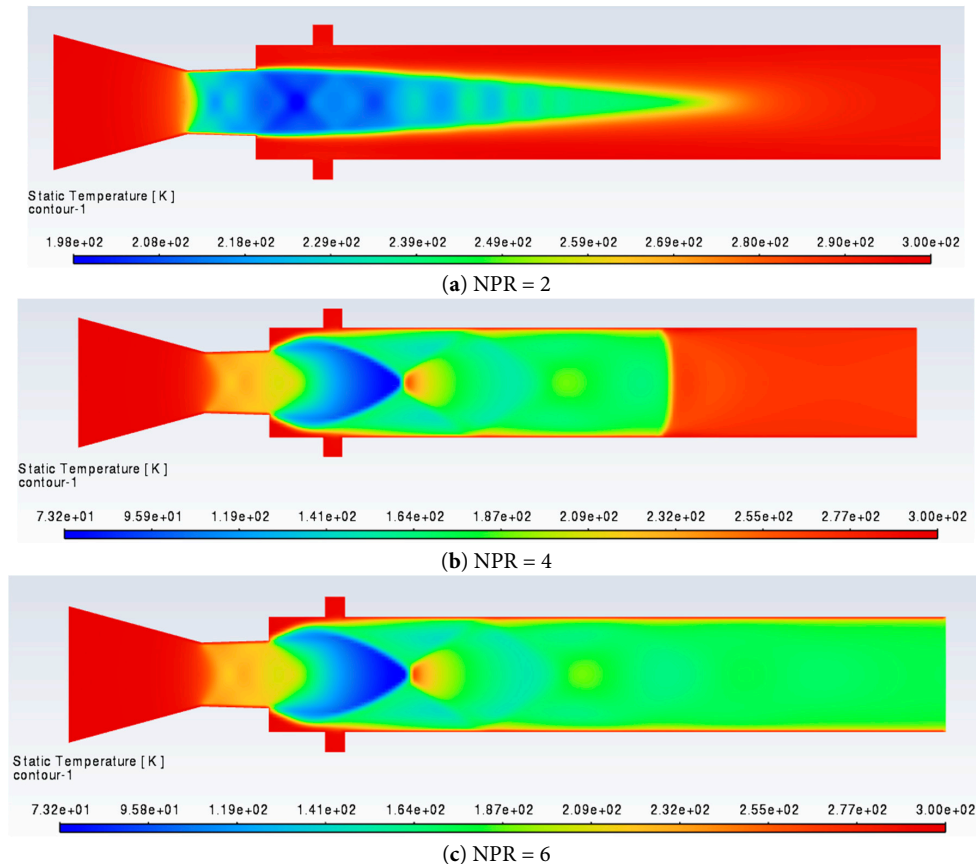


Figure 34: Cont.

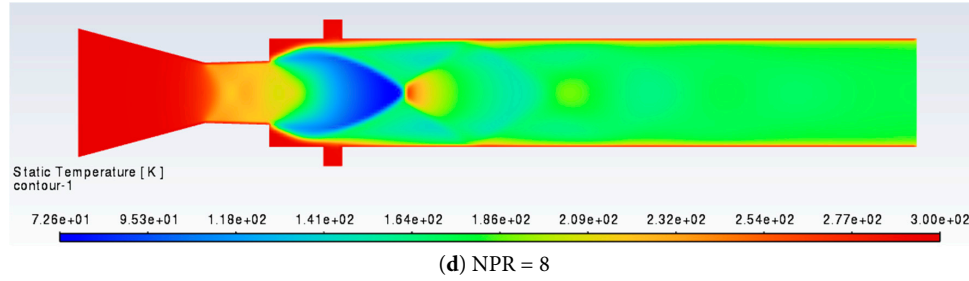


Figure 34: Static Temperature Contours: $L/D = 6$, $M = 1.4$ and cavity position $L/D = 0.5$. (a) NPR = 2, (b) NPR = 4, (c) NPR = 6, (d) NPR = 8.

4.16 Validation of CFD Analysis Results

The current CFD analysis results appear to match those reported in the literature by Pandey and Rathakrishnan [1]. In this study, both configurations, with and without a cavity, are analyzed using CFD. The CFD results appear quite consistent with the experimental results. Fig. 35 presents the CFD and experimental outcomes for different length-to-diameter ratios, and overall, the simulation results align very closely with the experimental data reported in the literature.

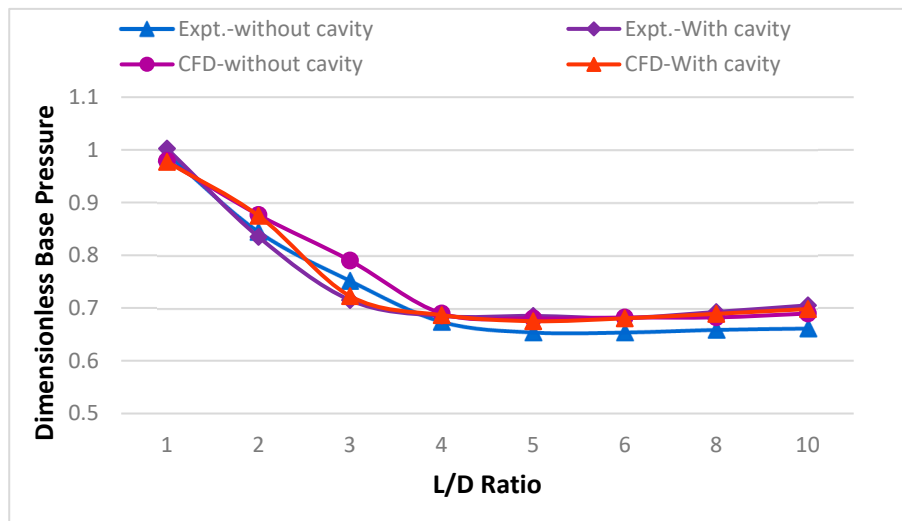


Figure 35: Validation of CFD results with experimental results.

The current CFD analysis results also match those reported in the literature by Pathan et al. [19]. To validate the numerical model (Table 3), the average base pressure was extracted from the duct's base face using the ANSYS Fluent post-processor. The raw gauge pressure data were changed to pressure (Pabs) by adding the back pressure (Pa). This was then normalized to a dimensionless form (P_b/P_a) for comparison against experimental data. Validation was performed across a range of expansion levels (NPR) from 2 to 8 at a constant $M = 1.5$.

The CFD simulations consistently underpredict the base pressure across the entire NPR range, suggesting a systemic bias in the numerical setup. This discrepancy likely stems from the turbulence model's limitations in accurately resolving the high-energy recirculation zones and complex vortex shedding at the duct's base.

Table 3: Validation of current work.

NPR	Without Control (Exp) [19]	Without Control (CFD)	PCD Control (Exp-Avg) [19]	PCD Control (CFD-Avg)
2.0	0.76	0.75	0.88	0.85
5.0	0.35	0.32	0.68	0.64
8.0	0.32	0.29	0.91	0.85

5 Conclusion

Based on the outcomes, it can be inferred that the base flow is effectively regulated by the cavity, especially at a cavity location of 0.5D for an area ratio of 2.89. However, the other cavity locations are ineffective in most parameter combinations, and the alteration in base pressure is insignificant. This lack of effectiveness is attributed to nozzle over-expansion, which reduces the reattachment length and prevents the flow from turning toward the base, rendering the cavity interaction ineffective. The findings indicate that inertia levels, duct length-to-diameter ratio, expansion levels, and cavity location play important roles in controlling base pressure. The cavity demonstrates maximum effectiveness in regulating base pressure when positioned at 0.5D for all NPR values, Mach numbers, and L/D ratios. Since the reattachment length is mainly dependent on the area or diameter ratio, which is 1.7 in this case, the cavity location cannot make a significant contribution to base pressure control, as the viscous layer after exiting from the nozzle does not interfere with the cavity placed at other numerous locations except at L/D = 0.5. As expansion levels rise, the nozzle undergoes under-expansion, reducing base pressure at NPR = 4. Beyond NPR = 4, further increases lead to highly under-expanded conditions, raising base pressure at NPR = 6 and 8. The flow visualization results from all displayed figures confirm that the system exhibits moderate effectiveness at lower NPR due to its over-expansion characteristics. The transition to under-expansion occurs at NPR = 4, which results in lower base pressure levels. The base pressure increases under extremely under-expanded conditions that occur at NPR = 6 and 8. The system uses the cavity to maintain recirculation stability, keeping the flow system intact and preventing any changes to its flow pattern.

Acknowledgement: This research is supported by the Structures and Materials (S&M) Research Lab of Prince Sultan University.

Funding Statement: The authors acknowledge Prince Sultan University's support in paying the article processing charges (APC) for this publication.

Author Contributions: The authors confirm contribution to the paper as follows: writing—original draft preparation, Sher Afghan Khan, Ridwan, Abdul Aabid; conceptualization, Khizar Ahmed Pathan, Sher Afghan Khan; validation, Abdul Aabid, Khizar Ahmed Pathan; resources, Muneer Baig; review and editing, Sher Afghan Khan, Muneer Baig; evaluation, Muneer Baig. All authors reviewed and approved the final version of the manuscript.

Availability of Data and Materials: The data that supports the findings of this study are available from the corresponding author upon reasonable request.

Ethics Approval: Not applicable.

Conflicts of Interest: Sher Afghan Khan, Abdul Aabid and Muneer Baig had no involvement in the peer review of this article and had no access to information regarding its peer review. Full responsibility for the editorial process for this article was delegated to another journal editor. The authors declare no other conflicts of interest.

References

1. Pandey KM, Rathakrishnan E. Annular cavities for base flow control. *Int J Turbo Jet Engines*. 2006;23(2):113–28. [[CrossRef](#)].
2. Khan A, Aabid A, Razak NA, Khan SA, Baig M. Numerical analysis of changes in the flow nature in a sonic flow with corrugated ribs. *Int J Thermofluids*. 2025;28:101307. [[CrossRef](#)].
3. Khan SA, Fatepurwala MA, Pathan KN, Dabeer PS, Baig MAA. CFD analysis of human powered submarine to minimize drag. *Int J Mech Prod Eng Res Dev*. 2018;8(3):1057–66. [[CrossRef](#)].
4. Li H, Choi J, Li B, Kim I, Heo J. Numerical analysis on the gas flow dynamics from a rectangular slot-nozzle for pulse cleaning of filter unit. *Powder Technol*. 2016;297:330–9. [[CrossRef](#)].
5. Yu N, Jourdain R, Gourma M, Shore P. Analysis of De-Laval nozzle designs employed for plasma figuring of surfaces. *Int J Adv Manuf Technol*. 2016;87(1):735–45. [[CrossRef](#)].
6. Stewart K, Rob I, Faik H. CFD analysis of C-D nozzle compared with theoretical & experimental data. *Incas Bull*. 2018;10(2):53–64. [[CrossRef](#)].
7. Wang YS, Xu JL, Huang S, Lin YC, Jiang JJ. Experimental and numerical investigation of an axisymmetric divergent dual throat nozzle. *Proc Inst Mech Eng Part G J Aeronaut Eng*. 2020;234(3):563–72. [[CrossRef](#)].
8. Rao GR, Ramakanth US, Lakshman A. Flow analysis in a convergent-divergent nozzle using CFD. *Int J Res Mech Eng*. 2013;1(1):136–44.
9. Najar NA, Dandotiya DS, Najar FA. Comparative analysis of $k-\epsilon$ and Spalart-Allmaras turbulence models for compressible flow through a convergent-divergent nozzle. *Int J Eng Sci*. 2013;2(8):8–17.
10. Salvador FJ, Jaramillo D, Romero JV, Roselló MD. Using a homogeneous equilibrium model for the study of the inner nozzle flow and cavitation pattern in convergent-divergent nozzles of diesel injectors. *J Comput Appl Math*. 2017;309:630–41. [[CrossRef](#)].
11. Pushpa BV, Sankar M, Mebarek-Oudina F. Buoyant convective flow and heat dissipation of Cu-H₂O nanoliquids in an annulus through a thin baffle. *J Nanofluids*. 2021;10(2):292–304. [[CrossRef](#)].
12. Koomullil R, Soni B, Singh R. A comprehensive generalized mesh system for CFD applications. *Math Comput Simul*. 2008;78(5–6):605–17. [[CrossRef](#)].
13. Kadivar M, Tormey D, McGranaghan G. A comparison of RANS models used for CFD prediction of turbulent flow and heat transfer in rough and smooth channels. *Int J Thermofluids*. 2023;20:100399. [[CrossRef](#)].
14. Mirjalily SAA. A computational investigation of shock wave train in diverse duct geometries using machine learning-adopted $k-\omega$ turbulence model. *Acta Astronaut*. 2024;217:27–36. [[CrossRef](#)].
15. Tong S, Guo M, Tian Y, Le J, Zhang D, Zhang H. Multi-objective optimization design of scramjet nozzle based on grey wolf optimization algorithm and kernel extreme learning machine surrogate model. *Phys Fluids*. 2024;36(2):025104. [[CrossRef](#)].
16. de Paola E, Camussi R, Gasparetti F, Di Marco A, Stoica LG, Capobianchi G, et al. Predicting wall pressure fluctuations on aerospace launchers through machine learning approaches. *Aerospace*. 2024;11(12):972. [[CrossRef](#)].
17. Khan A, Rajendran P, Sidhu JSS, Sharifpur M. 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. *Eur Phys J Plus*. 2023;138(4):314. [[CrossRef](#)].
18. Aabid A, Khan SA. Investigation of high-speed flow control from CD nozzle using design of experiments and CFD methods. *Arab J Sci Eng*. 2021;46(3):2201–30. [[CrossRef](#)].
19. Pathan KA, Dabeer PS, Khan SA. Effect of nozzle pressure ratio and control jets location to control base pressure in suddenly expanded flows. *J Appl Fluid Mech*. 2019;12(4):1127–35. [[CrossRef](#)].