MedCrave
Step into the World of Research
Aeronautics and Aerospace Open access Journal

Research Article
Open Access
CrossMark

Passive control of base pressure at sonic and supersonic Mach numbers

Abstract

This study presents a comprehensive review of the study entitled *Passive Control of Base Flow at Sonic and Supersonic Mach Numbers*. The research investigates passive flow control techniques to improve base pressure and reduce aerodynamic drag in suddenly expanded flows. Base drag significantly contributes to the overall drag experienced by high-speed aerospace vehicles such as rockets, missiles, and aircraft. The study employs Computational Fluid Dynamics (CFD) to analyze the effects of cavity-based passive control devices under various nozzle pressure ratios and flow conditions. The results show that introducing cavities can modify the recirculation region behind the nozzle exit, increasing base pressure and potentially reducing drag. This review examines the study's research objectives, methodology, findings, and engineering significance. It also critically evaluates the research's strengths and limitations. Overall, the study contributes valuable insights into passive flow control techniques and provides a foundation for future investigations into drag reduction technologies for aerospace applications.

Keywords: aerospace, base flow, mach number, fluid dynamics, nozzle pressure

Volume 10 Issue 3 - 2026

Ambareen Khan,[1] Mohammad Nishat Akhtar,[2] Sher Afghan Khan[3]

[1]School of Aerospace Engineering, Universiti Sains Malaysia, Tuanku Syed Sirajuddin Engineering Campus, Nibong Tebal, Malaysia
[2]School of Aerospace Engineering, Universiti Sains Malaysia, Tuanku Syed Sirajuddin Engineering Campus, Nibong Tebal, Penang, Malaysia
[3]Department of Mechanical & Aerospace Engineering, Faculty of Engineering, IIUM, Kuala Lumpur, Malaysia

Correspondence: Sher Afghan Khan, Department of Mechanical & Aerospace Engineering, Faculty of Engineering, International Islamic University, Kuala Lumpur, Malaysia

Received: September 10, 2026 | **Published:** September 23, 2026

Introduction

The performance of high-speed aerospace vehicles is strongly influenced by aerodynamic drag. One major source of drag in supersonic and transonic flight is base drag, which originates from the low-pressure region formed behind a vehicle or nozzle. When a high-speed jet exits a nozzle into a suddenly expanded duct, flow separation occurs, creating a low-pressure recirculation zone. This phenomenon reduces propulsion efficiency and increases fuel consumption.

To address this issue, researchers have explored various flow control techniques to increase base pressure and reduce drag. These techniques generally fall into active and passive flow control methods. Active methods require external energy input, whereas passive methods rely solely on geometric modifications and flow interactions. Due to their simplicity, reliability, and lower operating costs, passive flow control techniques are often preferred in aerospace applications.

The reviewed study investigates the effectiveness of cavity-based passive control in controlling base flow at sonic and supersonic Mach numbers. Through numerical simulations, the authors analyze how cavity geometry influences the flow field, pressure distribution, and recirculation region. The findings provide important information for designing efficient propulsion systems and aerodynamic structures.

Research objectives

a) Investigate the effect of passive flow control on base pressure.

b) Analyze the behavior of flow in a suddenly expanded duct.

c) Examine the influence of nozzle pressure ratio (NPR) on flow characteristics.

d) Evaluate the effectiveness of cavity geometries in controlling the recirculation region.

e) Identify configurations capable of improving aerodynamic performance and reducing base drag.

Literature review

Base flow and base drag

Base drag is a major aerodynamic loss mechanism in high-speed vehicles. It occurs when flow separates from the body surface, creating a wake region with significantly lower pressure than the surrounding flow. The pressure difference between the front and rear surfaces generates drag, reducing overall vehicle efficiency.

For aerospace vehicles operating at supersonic speeds, base drag can contribute a substantial percentage of total aerodynamic drag. Consequently, increasing base pressure has become an important area of aerodynamic design research.

Passive flow control techniques

Researchers have developed several passive flow control techniques to improve pressure recovery and reduce drag.

Cavities are recessed geometric features introduced into the wall surface. They generate vortices that interact with the separated flow and recirculation zone. These vortices enhance momentum transfer and can increase base pressure.

Ribs are protrusions placed along the duct wall. They disturb the boundary layer and modify flow structures, helping to improve pressure recovery in certain flow conditions.

Dimples create localized vortical structures that delay separation and improve flow mixing. They are widely used in aerodynamic applications that require drag reduction.

Spikes are commonly installed on the noses of missiles and projectiles. They alter shock-wave structures and reduce aerodynamic drag at supersonic speeds.

Previous studies

Previous investigations have demonstrated that passive control methods can significantly affect the recirculation region and base pressure. Many researchers reported improved pressure recovery through cavities, ribs, and other geometric modifications. However, the effectiveness of these methods depends on factors such as Mach number, nozzle pressure ratio, area ratio, and geometric dimensions.

Several studies focused on either sonic or supersonic flow conditions, but limited research has examined the combined influence of cavity geometry and operating parameters for specific area ratios. This research gap motivated the authors to conduct a detailed CFD investigation of cavity-based passive flow control.

Research gap

Based on the reviewed literature, the following gaps were identified:

A. Limited studies on cavity effectiveness at specific supersonic Mach numbers.

B. Insufficient understanding of cavity ratio effects.

C. Lack of detailed analysis for area ratio 4 configurations.

D. Need for further numerical investigation of recirculation zone behavior under different nozzle pressure ratios.

Methodology analysis

Research method

The study employed a **Computational Fluid Dynamics (CFD)** approach to investigate the effect of cavity-based passive flow control on base pressure in a suddenly expanded duct. CFD was selected because it allows detailed visualization and analysis of complex flow phenomena such as shock waves, flow separation, recirculation zones, and pressure distributions under supersonic conditions. The numerical approach provides an economical and efficient alternative to experimental testing while maintaining acceptable accuracy.

Geometrical configuration

The flow domain consisted of a **convergent-divergent (C-D) nozzle** connected to a suddenly expanded duct with an **area ratio (AR) of 4**. A cavity was introduced into the enlarged duct to act as a passive control device to manipulate the recirculation region and improve base pressure. The cavity was positioned **1D from the nozzle exit**, where *D* represents the diameter of the enlarged duct.

The flow domain consisted of a **convergent-divergent (C-D) nozzle** connected to a suddenly expanded duct with an **area ratio (AR) of 4**. A cavity was introduced into the enlarged duct to act as a passive control device to manipulate the recirculation region and improve base pressure. The cavity was positioned **1D from the nozzle exit**, where *D* represents the diameter of the enlarged duct (Table 1).

Table 1 Four cavity aspect ratios were investigated:

Cavity aspect ratio
03:03
03:06
06:03
06:06

Numerical approach

The numerical investigation used a Computational Fluid Dynamics (CFD) approach to analyze flow physics and base pressure characteristics under sonic and supersonic conditions. Commercial CFD software was used to model the flow domain, employing the standard realizable k-epsilon turbulence model and a wall-function approach to resolve the complex interactions within the suddenly expanded duct.

Analysis procedure

Flow physics of suddenly expanded flow

When a supersonic jet exits a convergent-divergent (C-D) nozzle into an enlarged duct, the abrupt increase in cross-sectional area causes the flow to expand rapidly.[1] This abrupt enlargement causes boundary-layer separation at the base corner, generating a separated shear layer and a low-pressure recirculation region adjacent to the base.[2] The pressure within this recirculation zone typically drops significantly below the free-stream ambient pressure, creating substantial base drag.[3] In high-speed internal and external flows, such as those in supersonic combustors, missiles, and rockets, this base drag can account for 30% to 60% of the total aerodynamic drag.[4]

Role of passive cavity control

Introducing geometric modifications such as an annular cavity in the path of the suddenly expanded flow serves as an effective passive suppression mechanism.[3] When the separated shear layer passes over the cavity, it spans the cavity volume before reattaching to the shroud or duct wall downstream.[1] The cavity disrupts the large-scale, periodic vortex shedding in the corner recirculation zone, breaking it down into smaller, highly energetic.[1] These smaller turbulent structures promote enhanced mixing and increase momentum transfer between the main supersonic core flow and the recirculating fluid.[4] Consequently, mass entrainment and pressure recovery within the base region occur, raising the base pressure toward atmospheric levels and directly minimizing base drag.[3]

Computational setup and parameters

To investigate the effectiveness of cavities via Computational Fluid Dynamics (CFD), the numerical model is configured using the specific flow and geometric parameters summarized below:

a) Mach Number: 1.4

b) Area Ratio (AR): 4

c) Nozzle Pressure Ratio (NPR): 2 to 10

d) Flow type: Supersonic

e) Cavity position: 1D (one duct diameter) downstream of the nozzle exit

f) Cavity geometries investigated: Four configurations defined by Width (W) and Depth (D): 3 x 3, 3 x 6, 6 x 3, 6 x 6

The computational domain is solved using commercial CFD software applying the Reynolds-Averaged Navier-Stokes (RANS) equations. Turbulence closure is achieved via the standard k - ε turbulence model paired with a standard wall-function approach to resolve near-wall gradients and complex shear-layer interactions.

Step-by-step computational analysis protocol

Step 1: Pre-processing and geometry generation

a) Model the axisymmetric C-D nozzle and the suddenly expanded duct geometry according to the specified area ratio (AR = 4).

b) Integrate the four distinct annular cavity configurations (3 x 3, 3 x 6, 6 x 3, 6 x 6) into the enlarged duct section at a fixed location of 1D from the nozzle exit plane.

Citation: Khan A, Akhtar MN, Khan SA. Passive control of base pressure at sonic and supersonic Mach numbers. *Aeron Aero Open Access J.* 2026;10(3):180–184. DOI: 10.15406/aaoaj.2026.10.00266

c) Generate a high-quality structured or unstructured mesh across the domain. Ensure sufficient grid density and inflation layers along the duct walls, the sudden expansion step, and inside the cavity regions to capture shear-layer separation, the reattachment point, and vortex dynamics.

Step 2: Boundary condition assignment

a) **Inlet:** Define the nozzle inlet using stagnation pressure and temperature conditions corresponding to the tested Nozzle Pressure Ratio (NPR = 2-10).

b) **Outlet:** Set the duct exit boundary as a pressure outlet or supersonic outflow boundary condition.

c) **Walls:** Apply a no-slip, adiabatic wall boundary condition along the nozzle, cavity, and enlarged duct surfaces.

d) **Axis:** Define the centerline of the axisymmetric domain with an axis boundary condition.

Step 3: Solver settings and iteration

a) Initialize the flow field using the inlet boundary values.

b) Select a density-based or pressure-based coupled solver suitable for high-speed compressible flows.

c) Enable the standard k - ε turbulence model with standard wall functions.

d) Set spatial discretization schemes to second-order upwind for flow, turbulent kinetic energy, and turbulent dissipation rate equations to ensure solution accuracy.

e) Run the simulation iteratively until the residuals drop by at least 4 to 6 orders of magnitude and key monitoring parameters such as base pressure and mass flow rate reach a steady state.

Step 4: Flow field post-processing and extraction

A. Extract the static pressure distribution along the base region, cavity walls, and enlarged duct centerline.

B. Calculate the nondimensional base pressure coefficient or base pressure ratio $\left(\frac{P_b}{P_a}\right)$ for each cavity configuration across the range of NPRs.

C. Visualize the flow field using contours of Mach number, static temperature, and static pressure.

D. Plot streamlines and vorticity magnitude contours to confirm the 3 (Power & Motion Conversion/Vortex generation) dynamics, highlighting the primary recirculation zone, secondary vortices inside the cavity, shear layer growth, and reattachment locations along the shroud wall.

Step 5: Performance evaluation and optimization

A. Compare the base pressure values of cavity configurations against the baseline case (sudden expansion duct without a cavity).

B. Assess the influence of cavity width (W) and depth (D) on the momentum transfer coefficient and base pressure enhancement.

C. Identify the optimum cavity geometry that effectively neutralizes the low-pressure suction vacuum, stabilizes the flow, and minimizes base drag to maximize overall system efficiency.

Results

This section reviews past studies of passive flow control, organizing recent experimental and computational advancements across both subsonic and supersonic speed regimes. Numerical and experimental data across both regimes show that inserting unpowered micro-geometric devices alters the base flow topology to counteract severe suction penalties. The quantitative performance of these techniques remains heavily dependent on the controller geometry, its spatial tracking coordinates, and the overarching pressure boundaries.

Sonic regime (M = 1.0) Annular quarter-circular ribs via CFD

Numerical fluid simulations utilizing computational fluid dynamics (CFD) were performed by Mahaboobali et al.,[5] to map internal expanded flows.

CFD Setup and Parameter Specifications: The numerical setup used a structured grid configuration to model a sudden axisymmetric expansion flow field inside a circular duct with a diameter of 22mm. To solve the compressible Navier-Stokes governing equations and account for turbulent viscous dissipation, the software implemented the standard k-epsilon turbulence model, chosen for its sufficiency, affordability, and robustness in internal high-speed configurations. Fluid viscosity properties were treated as temperature-dependent using Sutherland's three-coefficient viscosity model. The fluid domain was systematically evaluated under variable dynamic controls, specifically across a spectrum of Nozzle Pressure Ratios (NPR) ranging from 1.5 to 5. The passive control configurations incorporated annular quarter-circular ribs modeled across distinct fixed and dynamic geometric parameters, including variable rib radii (1mm, 2mm, 3mm, and 4mm), variable overall duct lengths (1D to 6D), and different spatial placement coordinates downstream from the base (0.5D, 1D, 1.5D, 2D, and 3D) evaluated under two separate structural orientations.

The CFD parametric sweeps by Mahaboobali et al.,[5] produced the following distinct performance trends:

A. **Orientation 1 performance (curved profile facing inflow):** Directing the incoming flow onto the curved contour of the rib prompts substantial base pressure recovery. Under a peak nozzle pressure ratio boundary (NPR=5), a 4mm rib radius situated at an axial coordinate of either 1D, 1.5D, or 2D forces a significant pressure expansion, driving the base pressure magnitude to nearly double the ambient atmospheric pressure (P_b/P_a≈1.7 to 1.8).

B. **Orientation 2 performance (flat profile facing inflow):** Reversing the internal device so that the straight vertical profile interceptor directly meets the core jet restricts the optimal geometric window exclusively to the 1.5D and 2D locations. Within these narrow operational limits, the CFD models show that the 4mm radius setup secures substantial recovery, reaching up to 80% and doubling ambient pressure gain.

C. **Ambient pressure balancing target (P_b = P_a):** When the engineering criteria require completely balancing the low-pressure base pocket with ambient atmospheric conditions, utilizing a specific 3mm rib radius at the 1D, 1.5D, or 2D coordinates consistently achieves this ambient threshold across both structural orientations in the CFD simulations.

D. **The zero-efficacy boundaries (0.5D and 3D):** Anchoring a rib too close to the nozzle trailing edge (0.5D) results in significantly lower base pressure recovery magnitudes and introduces severe flow field instabilities. Conversely, migrating the passive rib downstream to an axial distance of 3D yields no appreciable change or pressure enhancement, leaving the base flow completely independent of the control location. Furthermore, the CFD results

Citation: Khan A, Akhtar MN, Khan SA. Passive control of base pressure at sonic and supersonic Mach numbers. *Aeron Aero Open Access J.* 2026;10(3):180–184. DOI: 10.15406/aaoaj.2026.10.00266

show that reducing the rib height to a small 1mm or 2mm radius is fully inefficient across all coordinates, failing to suppress the sub-atmospheric vacuum.

Supersonic regime (M = 2.0) 2 mm static wake cylinder

Physical wind tunnel testing tracked high-speed supersonic jets expanding abruptly from a square converging-diverging (CD) nozzle into an enlarged square duct (area ratio of 9, length-to-width ratio L/W=10) in Asadullah et al. (2018). This experimental approach establishes a strict performance split governed by expansion states: The Over-Expanded Operating Deficit (NPR = 2 to 3): Under heavily over-expanded flow field boundaries, the 2 mm static wake cylinder is entirely ineffective. The total percentage increase in base pressure remains minimal across this low-pressure spectrum, dropping below 20 percent.

Transition Point (NPR = 6): As the pressure ratio increases and the system's level of over-expansion systematically drops, the micro-bluff body initiates pressure recovery, tracking a 14 percent increase in base pressure.

Perfect and Under-Expanded Peak Outcomes (NPR = 7.8 to 9): Once the supersonic flow transitions to perfectly expanded (NPR = 7.8) or under-expanded (NPR = 9) conditions, the passive cylinder engages with high efficiency. At NPR = 7.8, unmanaged base suction naturally hits its most severe baseline drop. Intercepting this state with the 2 mm cylinder directly counteracts the pressure deficit, culminating in a maximum base drag reduction of 59 percent at a high operating pressure of NPR = 9.

Discussions

The baseline vacuum mechanism

Across both high-speed compressible environments, abrupt cross-sectional area enlargement forces the boundary layer to separate at the step, growing into a free shear layer that isolates the central core stream from the dead-air corner region. This flow detachment creates a severe low-pressure suction deficit, building a powerful primary vortex closest to the blunt base. This baseline primary vortex operates as an efficient aerodynamic pump, continuously siphoning fluid and mass downstream away from the base wall. This persistent evacuation generates an intense sub-atmospheric suction state that drives base drag.

The fluid mechanics of passive interception

The physical justification for implementing fixed micro-geometric devices, such as internal annular ribs modeled via CFD by Mahaboobali et al.,[5] or physically tested static cylinders by Asadullah et al.,[6] is to establish a zero-power, unpowered flow management method to safely suppress this vacuum engine. When strategically positioned inside the wake region, these fixed structures function as localized mixing promoters. As the separating free shear layer strikes the control geometry, it deforms the dividing streamline and shears the high-speed flow boundaries. This direct geometric interaction intentionally forces the development of smaller, high-energy secondary vortices near the duct walls.

Unlike the large primary vortex which siphons air away, these secondary micro-eddies completely reverse the local directional trend of the wake. They successfully trap high-momentum fluid from the core jet stream and force-pump extra mass backward (upstream) directly into the dead-air recirculation bubble. This continuous mass injection meets the volumetric entrainment demands of the expanding jet, which then counteracts the vacuum effect, increases the base pressure magnitude (P_b), and actively minimizes overall drag.

Spatial and geometrical constraints

Synthesizing the data curves highlights three strict aerodynamic rules governing passive control viability:

A. **Boundary layer penetration:** To successfully modify the near wake, a device must maintain a sufficient structural profile height. Small 1mm and 2mm ribs analyzed by Mahaboobali et al.,[5] sit entirely buried within the slow-moving, low-energy acoustic boundary layer near the wall, leaving them unable to interact with the flow. Elevating the profile to a 4mm height allows the passive device to break past this low-energy boundary layer and actively hook into the high-momentum core jet streamlines, siphoning off the kinetic energy required to sustain the reverse upstream mass transfer.

B. **Proximity to the natural reattachment boundary:** The optimal operational window requires the passive controller to reside upstream or directly in the close neighborhood of the flow's natural reattachment point. The critical reattachment length where the shear layer meets the duct wall naturally settles between the 2D and 3D marks. Pushing a control device downstream to 3D seals it completely outside the recirculation bubble. Because the flow has physically reattached to the wall, the base pressure becomes independent of the control location, making upstream mass communication impossible.

C. **Upstream shock instabilities (0.5D Position):** Placing a device too close to the nozzle collar (0.5D) is insufficient for a stable shear layer to form. The passive device clashes directly with the initial unexpanded shock fan exiting the nozzle throat. This conflict generates violent wave reflections that bounce back into the base cavity, choking local throat alignment and provoking severe flow-field instabilities and rapid pressure oscillations.

Structural topology dynamics (rib orientations vs. wake cylinders)

The internal flow reviews reveal an extreme physics divergence based on structural geometric shapes:

a) Orientation 1 (Curved Face Inlet) in Mahaboobali et al.,[5] Promotes a gradual, progressive geometric compression of the sonic stream. This contour acts as a localized nozzle effect, smoothly deflecting flow momentum into stable secondary eddies over a broad optimization window (1D to 2D) while minimizing total stagnation pressure drops.

b) Orientation 2 (Flat Face Inlet) & Supersonic Wake Cylinders: Present an abrupt, blunt 90º vertical barrier or bluff-body obstacle directly to the oncoming stream. The high-speed fluid hits a sharp flat corner or cylinder wall, forcing instant, severe localized flow separation. This severe detachment generates much stronger, highly turbulent secondary vortices. For the supersonic cylinder in Asadullah et al. (2018), this brutal interaction directly weakens the primary base vortex by flushing it out at high pressure ratios, providing massive drag reduction.

The nozzle expansion barrier and the downstream shock train

A defining lesson from the supersonic Mach 2 review by Asadullah et al.,[6] is that passive bluff-body controllers are entirely hostage to the nozzle's global expansion state. At heavily over-expanded bounds

(NPR=2-3), the flow leaving the nozzle exit is physically choked and dominated by massive oblique shock waves. These severe shocks flush the flow away and fully block the base region, isolating the near-wake and rendering the static cylinder completely ineffective. Control effectiveness unlocks only at perfectly expanded or under-expanded pressure states (NPR = 6), as the exit oblique shocks systematically weaken.

Critically, wall pressure distribution tracks show that micro-scale controllers optimize base pressure while keeping global internal flow flux inside the duct nearly identical with and without control. However, at high-performance operating states (NPR=6, 7.8, and 9), the cylinder introduces localized downstream wave systems, triggering alternating increases and decreases in wall pressure (Pw) along the duct. Beyond an axial distance of X/W = 4 to 5, the control device causes a marginal decrease in wall pressure. This spatial tracking fluctuation confirms that perfectly expanded supersonic flow is never smoothly shock-free; instead, the entire internal duct flow field remains heavily dominated by an active internal shock train and complex shock-boundary layer interactions bouncing off the structural walls.

Critical analysis

Strengths of the study

A. **Practical aerospace application:** The research addresses base drag reduction, which is important for rockets, missiles, and high-speed aerospace vehicles.

B. **Use of passive control:** Passive control devices do not require external energy input, making them lightweight, reliable, and cost-effective.

C. **Systematic investigation:** Multiple cavity aspect ratios and NPR values were studied, allowing a detailed understanding of flow behavior.

D. **CFD-Based Analysis:** CFD provides detailed visualization of shock structures, recirculation zones, and pressure distributions that may be difficult to obtain experimentally.

Weaknesses of the study

a) **Limited mach number range:** The study mainly focuses on Mach 1.4, limiting the generalization of results to higher supersonic speeds.

b) **Lack of experimental validation:** The conclusions rely primarily on numerical simulations. Experimental testing would strengthen confidence in the results.

c) **Single area ratio:** Only Area Ratio 4 was investigated, whereas different area ratios may produce different flow characteristics.

d) **No direct drag measurement:** The study focuses on base pressure but does not directly calculate drag coefficient reduction.

Conclusion

A. **Dominant high-speed penalty:** At sonic and supersonic speeds (M = 1.0 to 2.0), rapid cross-sectional expansion triggers severe flow separation, forming a low-pressure primary vortex that acts as an aerodynamic vacuum siphoning mass away from the base wall, driving a drag penalty that accounts for up to 60% of net vehicle drag.

B. **Passive control efficacy:** Micro-scale passive modifications, such as internal annular quarter-circular ribs analyzed via CFD by Mahaboobali et al.,[5] or a 2 mm static wake cylinder physically tested by Asadullah et al. (2018), operate as highly reliable, zero-power mixing promoters that successfully recover base pressure without requiring moving parts or external energy inputs.

C. **The mixing mechanism:** These devices alter localized shear layers to intentionally generate high-energy secondary vortices. These secondary loops actively capture kinetic momentum directly from the main core jet and force-pump missing mass back into the recirculation bubble, satisfying wake-entrainment demands and suppressing the base vacuum.

D. **The expansion dependency:** Device performance is strictly hostage to the global Nozzle Pressure Ratio (NPR) and expansion states. Under heavily over-expanded bounds, strong oblique nozzle exit shocks dominate the field and isolate the wake pocket, rendering controls ineffective; peak drag reduction unlocks only across perfectly expanded or under-expanded operating windows.

E. **Strict geometric boundaries:** Actionable pressure recovery demands high geometric precision. Control devices must maintain sufficient structural profile height (4mm) to break past the low-energy wall boundary layer and capture core jet momentum. Furthermore, they must sit upstream or near the natural flow reattachment zone (1D to 2D); moving devices past 2D (at 3D) seals off the recirculation zone, making the flow completely independent of control locations.

F. **Downstream flow field integrity:** Utilizing micro-scale wake controllers successfully optimizes base pressure recovery and suppresses drag penalties while safely preserving the global internal flow flux down the pipe. However, downstream wall pressure tracks still show persistent spatial fluctuations, confirming that high-speed channels remain dominated by a highly active internal shock train and complex wave reflections.

Acknowledgement

None.

Conflict of interest

Author has no conflicts of interest.

Funding

None.

References

1. Sethuraman V, Khan SA. Base pressure control using micro-jets in supersonic flow regimes. *Int J Aviat Aeronaut Aerosp*. 2018;5(1).
2. Aabid A, Khan SA, Baig M. A critical review of supersonic flow control for high-speed applications. *Appl Sci*. 2021;11(15):6899.
3. Khan A, Rajendran P, Sidhu JSS. Passive control of base pressure: a review. *Appl Sci*. 2021;11(3):1334.
4. Khan A, Khan SA, Raja V, et al. Effect of ribs in a suddenly expanded flow at sonic Mach number. *Heliyon*. 2024;10(5):e30313.
5. Mahaboobali FAG, Khan A, Akhtar MN, et al. Passive control of base flow and impact of quarter rib radius and locations at sonic Mach number. *J Adv Res Numer Heat Transf*. 2025;30(1):47–81.
6. Asadullah M, Khan SA, Asrar W, et al. Passive control of base pressure with a static cylinder at supersonic flow. *IOP Conf Ser Mater Sci Eng*. 2018;370:012050.