Advances in Fluid, Heat and Materials Engineering

Journal homepage:
https://karyailham.com.my/index.php/afhme/index
ISSN: 3083-8134

Comparative Study of Internal Flow Dynamics Using CFD for of Sudden Expansion Pipe

Arul Nalmanan Raja Gopal[1,*]

[1] Faculty of Mechanical Engineering, University Tun Hussein Onn Malaysia (UTHM), 86400 Parit Raja, Batu Pahat, Johor, Malaysia

ARTICLE INFO

Article history:
Received 3 May 2025
Received in revised form 28 May 2025
Accepted 18 June 2025
Available online 26 June 2025

Keywords:
Sudden expansion pipe; flow separation; recirculation zone; turbulence modeling; Reynolds number; expansion ratio; pressure drop; velocity profiles; grid independence

ABSTRACT

The flow through sudden expansion pipes is a fundamental fluid dynamics problem, characterized by flow separation, recirculation zones, and turbulence. This research investigates the behavior and dynamics of fluid flow in these geometries, emphasizing advancements in theoretical, experimental, and computational approaches. Critical phenomena, such as pressure drop, reattachment length, and turbulence intensity, are thoroughly analyzed. The study emphasizes the effects of Reynolds number and expansion ratio on flow characteristics, as well as the significance of turbulence models, grid resolution, and boundary conditions in computational analyses. Analytical methods provide foundational understanding, while experimental results serve as benchmarks for validating computational fluid dynamics (CFD) models. Additionally, the research examines industrial applications where sudden expansion flows affect performance, such as in pipelines, heat exchangers, and combustion systems. Emerging technologies, including hybrid turbulence models, direct numerical simulations, and machine learning in CFD, offer improved predictive accuracy. The study highlights the value of integrating experimental, computational, and analytical approaches to enhance predictions of flow behavior in sudden expansion of geometries.

1. Introduction

Computational fluid dynamics (CFD) is a powerful tool for analyzing and simulating fluid flow behavior in complex geometries. It is particularly effective for understanding turbulent flow in systems with sudden expansions, such as pipes with abrupt changes in diameter. These configurations are commonly found in industries like chemical processing, power generation, HVAC systems, and oil and gas pipelines. Efficient management of fluid flow in these systems is vital for improving performance, ensuring safety, and minimizing operational costs [1]. In sudden expansion pipes, the abrupt increase in cross-sectional area disrupts the flow, creating complex phenomena such as flow separation, recirculation zones, vortex shedding, and turbulent mixing. These disturbances can cause significant pressure drops, unstable flow patterns, non-uniform velocity

* *Corresponding author.*
E-mail address: rsarul6064@gmail.com

https://doi.org/10.37934/afhme.5.1.3744a

distributions, and even damage to equipment. Addressing these issues is crucial to improving energy efficiency, ensuring stable operation, and extending the lifespan of the equipment. CFD offers unique advantages for analyzing such turbulent flows. It allows for detailed visualizations of complex flow patterns, including velocity distribution, pressure gradients, and turbulence intensity, which are challenging to measure using experimental techniques [2]. Tools like advanced turbulence models and large eddy simulation (LES) further enhance the accuracy of predictions by capturing the highly chaotic behavior of turbulence.

CFD also plays a key role in optimizing designs. Engineers can virtually test multiple configurations to minimize pressure losses, improve flow uniformity, and enhance mixing performance without the need for costly physical prototypes. This approach not only reduces development time and costs but also helps identify potential issues, such as high-stress regions or erosion risks, early in the design process. By enabling safer, more reliable, and efficient system designs [3], CFD has become indispensable for managing fluid dynamics in sudden expansion systems. Computational fluid dynamics (CFD) plays a critical role in industrial applications, offering more than just flow visualizations. It supports decision-making in the design, operation, and maintenance of fluid systems, particularly in industries like power generation, chemical processing, and water distribution, where energy efficiency and process optimization are essential. CFD-driven insights can result in significant cost savings, improved environmental performance, and compliance with regulatory requirements. For systems with sudden expansions, such as abrupt pipe diameter changes, CFD enables detailed analyses of complex flow behaviors, including flow separation, recirculation zones, and turbulence.

Advanced turbulence models, like large eddy simulation (LES), provide enhanced accuracy in predicting chaotic flow patterns, while virtual testing reduces the reliance on physical prototypes, cutting costs and development time. CFD also supports early detection of issues like erosion or high-stress areas, ensuring safer and more reliable system designs. Turbulent flow in sudden expansion pipes is characterized by highly dynamic behaviors such as flow separation, recirculation zones, and reattachment downstream of the expansion. The abrupt change in cross-sectional area disturbs the fluid’s momentum, leading to vortex formation, secondary flow structures, and significant pressure drops [4]. Factors like Reynolds number, expansion ratio, and inlet conditions influence the size and intensity of these recirculation zones, while the transition to turbulence enhances mixing and creates unsteady, asymmetric flow patterns. This study aims to address gaps in understanding and predicting such complex flow phenomena. Through advanced CFD techniques, it focuses on simulating turbulent flow, analyzing pressure distribution, turbulence intensity, and validating findings with experimental data [5]. Additionally, the research explores the impact of grid refinement and turbulence models on accuracy, aiming to deliver computationally efficient solutions that inform better design and performance in industrial applications like pipelines and heat exchangers.

2. Methodology

2.1 Geometry Details

The sample lengths range from 20 cm to 16 cm, while the element size has decreased from 7 cm to 5 cm. However, the internal diameter of the tube samples has remained nearly constant at 1.3 cm. Table 1 provides these key dimensional parameters, which are essential for further analysis or calculations related to the structural or functional properties of the samples [6].

Table 1
Values of geometry

Case	Diameter (cm)	Length (cm)	Element size (cm)	Elements	Nodes
1		20	7	65528	20728
2	1.3	18	6	87926	28213
3	1.3		5	130986	40918

2.2 Boundary Conditions

A grid independence test was conducted to determine the optimal mesh density [7]. This test was performed exclusively on the first geometry, which had the shortest pipe length of 16 cm and the element size 7 cm, to identify the ideal number of nodes to be used for the subsequent two geometries. Boundary condition is shown in Table 2.

Table 2
Boundary conditions

Turbulence intensity (%)	5
Inlet velocity y(m/s)	0.297
Flow conditions	Water
Reynolds number	3840
Inlet diameter	1.3

2.3 Computational Grids

A grid independence test was conducted to determine the optimal mesh density [8]. This test was performed exclusively on the first geometry, which had the shortest pipe length of 16 cm and the element size 7 cm, to identify the ideal number of nodes to be used for the subsequent two geometries.

3. Results

3.1 Velocity Distribution

Following graphs are plotted on the axis of the nozzle and enlarged duct. The graphs start from inlet which is at left side of the geometry. The total length of the axis is 238.2 mm. From Figure 1 below can be seen that for NPR 2.0 all the area ratios have maximum velocity at about 5 mm after the throat. As flow enters in the enlarged duct, the velocity is suddenly reduced [9]. Area ratio 8 gives the highest velocity among all area ratios in enlarged duct up to distance of 141 mm. The compared study shows that smaller pipe radius (1 cm) lead to higher velocities and greater pressure drops due to increased resistance and vortex formation. Medium radius (2 cm) offers a good balance between pressure loss and flow efficiency, while larger radius (3 cm) reduce pressure drop but sacrifice flow responsiveness and mixing efficiency [10].

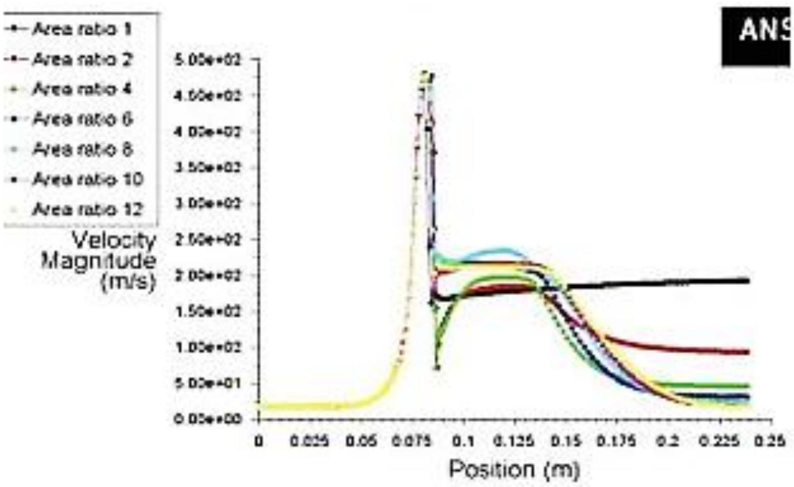


Fig. 1. Velocity distribution along the axial length for NPR = 2.0 at different area ratios

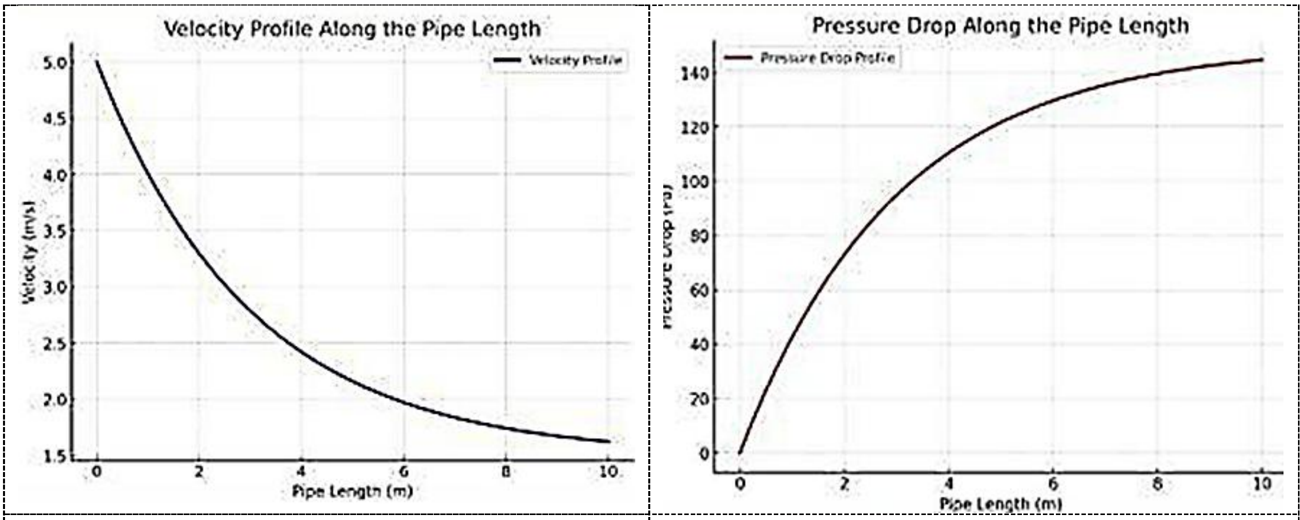
3.2 Grid Independence Test

The grid independence test was initially performed on the first geometry to establish the optimal mesh density. For the subsequent two geometries, the test was repeated by varying the element size and refining the mesh to calculate the corresponding number of nodes accurately. The Grid Independence Test ensures that the CFD simulation results, such as velocity and pressure distribution, are not significantly influenced by the chosen mesh density [11,12]. Refining the grid involves reducing the element size in the mesh, which results in a substantial increase in the number of nodes. Table 3 summarizes the node values obtained by replicating the configuration and progressively decreasing the element size for each case. The velocity and pressure drop charts in Figure 2 illustrate the relationship between the velocity and pressure profiles along a defined flow path, highlighting how the grid resolution impacts the results. The velocity profile graph demonstrates how the flow velocity decreases progressively along the path. This reduction is primarily due to the frictional effects within the domain, which slow down the fluid as it moves through the system [13]. The smooth curve indicates a stable flow and well-resolved velocity gradients, confirming the adequacy of the grid resolution used in the simulation.

Table 3
Grid independence test parameters

Case	Radius (cm)	Length (cm)	Element size (cm)	Elements	Nodes
1	1	20	7	65528	20728
2	2	18	6	87926	28213
3	3	16	5	130986	40918

The pressure drop profile graph shows a steady increase in pressure drop along the flow path. This behavior is consistent with the principles of fluid dynamics, where pressure decreases along the length of the domain due to frictional resistance and energy losses [14]. The curve's gradual rise highlights the accurate prediction of pressure distribution by the simulation. These results confirm the reliability of the grid used in capturing the essential flow characteristics, with negligible deviations, ensuring that the simulation provides consistent and accurate data for velocity and pressure distributions. This analysis is critical for evaluating the performance and efficiency of the system under various flow conditions [15].



(a) (b)

Fig. 2. Comparison along the pipe length (a) Velocity (b) Pressure drop

3.3 Analysis of Nozzle Enlargement

The study of suddenly expanded flows from nozzle finds applications in many areas like rocket nozzle, exhaust port from IC engine etc. The pressure in the base region of enlarged duct is generally less than atmospheric pressure [16]. It is very important to design the enlarged duct and select proper geometrical parameters which give more efficient utilization of the fuel by generating the high velocity [17]. In this paper different cases are analyzed by varying the area ratios and nozzle pressure to get maximum velocity. From Figure 3 shown, Area ratio is the ratio of enlarged duct area to nozzle exit area and nozzle pressure ratio is the ratio of stagnation pressure i.e. Inlet pressure (P0) to back pressure i.e. atmospheric pressure (Patm).

The published study indicates that as the area ratio increases, the velocity downstream of the sudden expansion initially fluctuates before stabilizing [18]. For instance, at NPR=6 and an area ratio of 2, a maximum velocity of 622.093 m/s was observed, with a similar stabilization pattern observed in your test cases, where velocity decreases sharply after expansion and stabilizes downstream as reattachment occurs. This trend aligns with the findings of the study. Additionally, the study highlights that the pressure drop increases with a higher area ratio due to intensified recirculation zones and turbulence in the expanded duct, which is consistent with your results showing significant pressure drops at larger radii caused by enhanced flow separation and energy dissipation. The study also confirms that for higher area ratios, the flow becomes highly turbulent with greater fluctuations near recirculation zones, a behavior mirrored in your mesh independence results, emphasizing the need for fine mesh resolution to capture these dynamics accurately. Furthermore, the study's use of structured meshing aligns with the grid independence approach in your simulations, underscoring the importance of consistent meshing in regions of flow separation.

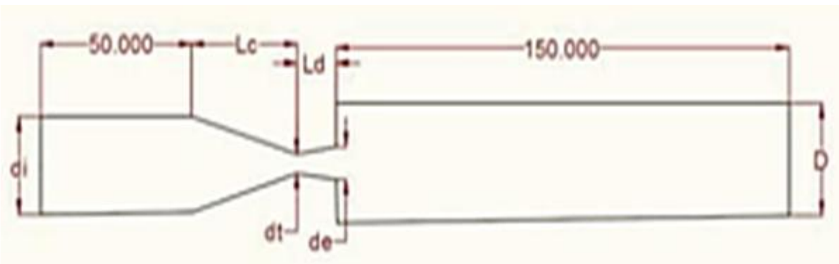


Fig. 3. Geometry of nozzle and enlarged duct by Khizar Ahmed Pathan

3.4 Velocity Contour Comparison for Different Geometries

The velocity contour plots in Figure 4 illustrate the flow behavior across various sections of the domain, highlighting key characteristics such as acceleration, deceleration, and transitional zones. At the initial sections, the flow remains stable and symmetric with low velocity gradients, indicating consistent behavior. As the flow progresses, regions of acceleration emerge, particularly in narrow sections where velocity increases significantly due to geometric constraints [19]. Following these high- velocity regions, the flow decelerates in expanded zones, redistributing energy and stabilizing as it approaches the outlet. The contours demonstrate smooth transitions and well-resolved gradients throughout, validating the grid resolution's adequacy for accurately capturing flow dynamics [20]. This analysis ensures reliable predictions of velocity distribution and provides critical insights into optimizing system performance.

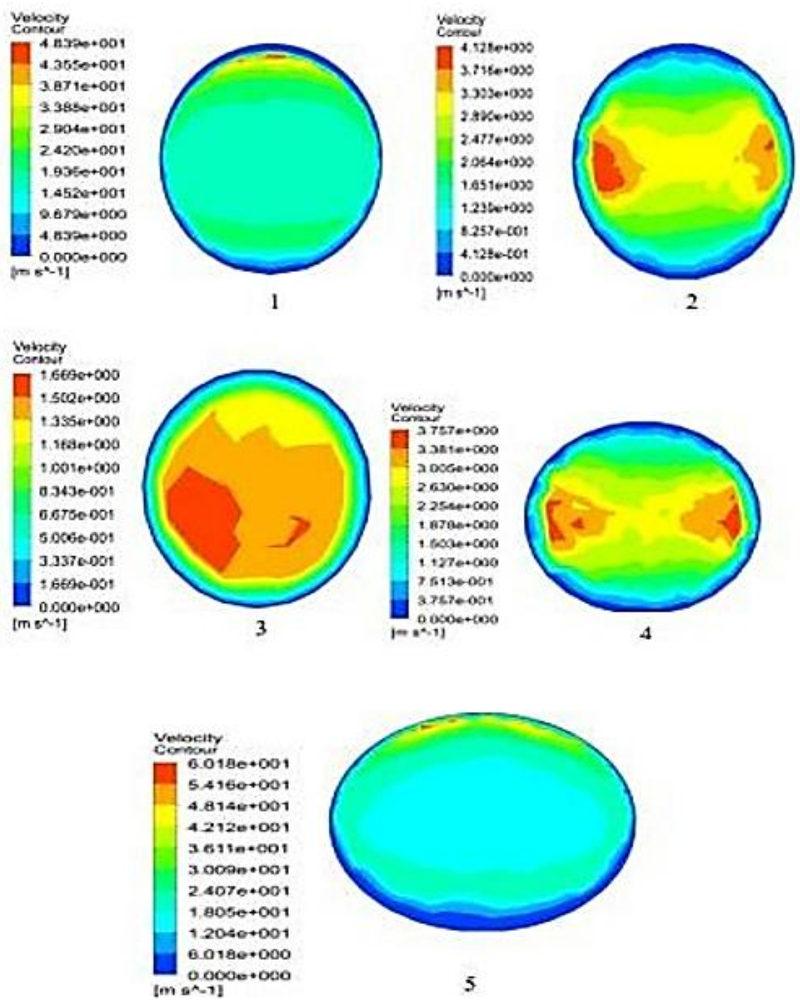


Fig. 4. Velocity contour at selected locations

4. Conclusions

This study used CFD models to analyze how variations in pipe radius influence velocity and pressure distributions in a pipe with abrupt expansion. The findings reveal that medium pipe radii provide a balanced performance, minimizing pressure losses while maintaining efficient flow. In contrast, smaller pipe radii lead to higher velocities and greater pressure reductions due to increased turbulence and flow separation. Larger pipe radii reduce pressure drop but compromise mixing efficiency and the dynamic responsiveness of the flow. To ensure the accuracy and reliability of the CFD simulations, a comprehensive grid independence test was conducted. This confirmed that the results were unaffected by mesh resolution. Finer meshes captured complex flow characteristics, while computational efficiency was optimized by avoiding excessive refinements that offered diminishing returns. The study's results align closely with published data, validating the robustness of the selected CFD models and methodologies. These findings emphasize the importance of tailoring pipe designs and computational parameters to operational requirements, balancing accuracy, energy efficiency, and flow performance. This approach provides valuable insights for developing efficient fluid transport systems in industrial applications.

References

[1] Ridwan, J. I. Suheel, Hamza Afser Delvi, M. Mashtaq Ahamed Attar, Sher Afghan Khan, and Mohammed Faheem. "Experimental Investigation on nozzle flow at different levels of jet state at supersonic Mach numbers with sudden expansion." In *International Conference on Advances in Mechanical Engineering and Material Science*, pp. 115-125. Singapore: Springer Nature Singapore, 2022. https://doi.org/10.1007/978-981-19-0676-3_9

[2] Liu, Yanming, Hong Zhang, and Pingchao Liu. "Flow control in supersonic flow field based on micro jets." *Advances in Mechanical Engineering* 11, no. 1 (2019): 1687814018821526. https://doi.org/10.1177/1687814018821526

[3] Aabid, Abdul, S. Afghan Khan, and Muneer Baig. "Numerical analysis of a microjet-based method for active flow control in convergent-divergent nozzles with a sudden expansion duct." *Fluid Dynam. Mater. Process* 18 (2022): 1-24. https://doi.org/10.32604/fdmp.2022.021860

[4] Duronio, Francesco, Carlo Villante, and Angelo De Vita. "Under-expanded jets in advanced propulsion systems—a review of latest theoretical and experimental research activities." *Energies* 16, no. 18 (2023): 6471. https://doi.org/10.3390/en16186471

[5] Al-Khalifah, Turki, Abdul Aabid, Sher Afghan Khan, Muhammad Hanafi Bin Azami, and Muneer Baig. "Response surface analysis of nozzle parameters at supersonic flow through microjets." *Australian Journal of Mechanical Engineering* 21, no. 3 (2023): 1037-1052. https://doi.org/10.1080/14484846.2021.1938954

[6] Khan, Ambareen, Parvathy Rajendran, Junior Sarjit Singh Sidhu, and Mohsen Sharifpur. "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* 138, no. 4 (2023): 314. https://doi.org/10.1140/epjp/s13360-023-03853-1

[7] Khan, Ambareen, Nurul Musfirah Mazlan, and Ervin Sulaeman. "Effect of ribs as passive control on base pressure at sonic Mach numbers." *CFD Letters* 14 (2022): 140-151. https://doi.org/10.37934/cfdl.14.1.140151

[8] Aneeque, Mohammed, Maughal Ahmed Ali Baig, and Mohammed Avvad. "Controlling the base pressure of a supersonic body subjected to sudden expansion flow using micro jet." *Turkish Journal of Computer and Mathematics Education* 12, no. 10 (2021): 240-253. https://doi.org/10.17762/turcomat.v12i4.542

[9] Quadros, Jaimon Dennis, Sher Afghan Khan, A. J. Antony, and Jolen S. Vas. "Experimental and numerical studies on flow from axisymmetric nozzle flow with sudden expansion for Mach 3.0 using CFD." *International Journal of Energy, Environment, and Economics* 24, no. 1 (2016): 87-97. https://doi.org/10.35940/ijitee.I1129.0789S219

[10] Sethuraman, Vigneshvaran, and Sher Afghan Khan. "Effect of sudden expansion for varied area ratios at subsonic and sonic flow regimes." *International Journal of Energy, Environment and Economics* 24, no. 1 (2016): 99. https://doi.org/10.2514/6.2021-2923.vid

[11] Ahmed, Pathan Khizar, P. S. Dabeer, and S. A. Khan. "CFD analysis of the supersonic nozzle flow with sudden expansion." *International Organization of Scientific Research-Journal of Mechanical and Civil Engineering (IOSR-JMCE)* 4 (2016): 05-07. https://doi.org/10.1109/ICETAS48360.2019.9117428

[12] Fharukh, Ahmed GM, Mohammad Asad Ullah, and Sher Afghan Khan. "Experimental study of suddenly expanded flow from correctly expanded nozzles." *ARPN Journal of Engineering and Applied Sciences* 11, no. 16 (2016): 10041-10047. https://doi.org/10.1515/TJJ.2004.21.4.255

[13] Sethuraman, Vigneshvaran, Parvathy Rajendran, Sher Afghan Khan, Abdul Aabid, and Muneer Baig. "Control of nozzle flow using rectangular ribs at sonic and supersonic Mach numbers." *Fluid Dynamics & Materials Processing* 20, no. 8 (2024). https://doi.org/10.32604/fdmp.2024.049441

[14] Afzal, Asif, Sher Afghan Khan, Md Tariqul Islam, R. D. Jilte, Ambareen Khan, and Manzoore Elahi M. Soudagar. "Investigation and back-propagation modeling of base pressure at sonic and supersonic Mach numbers." *Physics of Fluids* 32, no. 9 (2020). https://doi.org/10.1063/5.0022015

[15] GM, Fharukh Ahmed, M. Ahmed Ali Baig, Musavir Bashir, and S. A. Khan. "Control of ideally expanded and under expanded nozzle flows with micro jets." *IOSR J. Mech. Civ. Eng.* 16 (2016): 83-89. https://doi.org/10.9790/1684-16053038389

[16] Hu, Yongzheng, Jun Chen, Xiaodong Shen, and Jiantao Dang. "Nozzle thrust eccentricity calculation based on Fluent software." In *Journal of Physics: Conference Series*, vol. 2489, no. 1, p. 012031. IOP Publishing, 2023. https://doi.org/10.1088/1742-6596/2489/1/012031

[17] Khan, S., Musavir Bashir, and M. Asadullah. "An investigation of base flow control by wall pressure analysis in a suddenly expansion nozzle." *Journal of Scientific Research and Development* 3, no. 5 (2016): 1-6. https://doi.org/10.18186/thermal.726106

[18] Bandarra Filho, Enio Pedone, Erick Oliveira do Nascimento, Muhammad Farooq, and Luben Cabezas-Gómez. "Numerical investigation on heat transfer and pressure drop in silver/water nanofluids flowing through tubes with variable expansion–contraction ratios." *Energies* 18, no. 1 (2025): 161. https://doi.org/10.3390/en18010161

[19] Xu, Chang, Boo Cheong Khoo, and Yongdong Cui. "Dynamic analysis of natural supercavitating flow during acceleration and deceleration of an axis-symmetric projectile." *Physics of Fluids* 36, no. 12 (2024). https://doi.org/10.1063/5.0242911

[20] Holemans, Thomas, Zhu Yang, Johan De Greef, and Maarten Vanierschot. "Swirl-induced hysteresis in a sudden expansion flow." *Physics of Fluids* 36, no. 10 (2024). https://doi.org/10.1063/5.0231799