KARYA ILHAM PUBLISHING
CONNECTING MINDS, BRIDGING IDEAS, INSPIRING READERS

Advances in Fluid, Heat and Materials Engineering

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

CFD Study of Flow Through a Converging Nozzle: Effect of Bore Diameter

Nur Izzat Naqiuddin Nur Hisham[1,*]

[1] Department of Mechanical Engineering, Faculty of Mechanical Engineering and Manufacturing, Universiti Tun Hussein Onn Malaysia, 86400 Batu Pahat, Johor, Malaysia

ARTICLE INFO

Article history:
Received 22 October 2025
Received in revised form 12 December 2025
Accepted 17 December 2025
Available online 21 December 2025

Keywords:
CFD; converging nozzle; turbulent flow; pressure drop; bore diameter

ABSTRACT

This study investigates the effects of nozzle bore diameter on the flow characteristics of a turbulent, incompressible fluid through converging nozzles, focusing on velocity and pressure distributions. Computational Fluid Dynamics (CFD) simulations were conducted using ANSYS Fluent to analyse three different nozzle bore diameters: 4 mm, 6 mm, and 9 mm, under identical inlet conditions of 1 m/s uniform velocity and a zero-pressure outlet boundary. The primary objective was to quantify the impact of nozzle geometry on performance metrics such as velocity, pressure distribution, and overall pressure loss. The results demonstrate a significant relationship between bore diameter and flow behaviour. As the bore diameter decreases, the fluid experiences greater flow acceleration, leading to higher exit velocities. The 4 mm nozzle produced the highest exit velocity, which was 423.3% higher than the 9 mm nozzle and 130.9% higher than the 6 mm nozzle. However, this increase in velocity was accompanied by a substantial pressure drop, with the 4 mm nozzle exhibiting 96.5% lower pressure than the 9 mm nozzle and 81.7% lower pressure than the 6 mm nozzle. These findings suggest that smaller nozzles enhance jet velocity and momentum, making them ideal for high-intensity applications such as waterjet cutting and abrasive blasting. In contrast, the 9 mm nozzle, while producing the lowest exit velocity, demonstrated the least pressure drop, with 81.1% lower pressure than the 6 mm nozzle and 96.5% lower pressure than the 4 mm nozzle. This indicates that larger nozzles are more hydraulically efficient, making them suitable for applications prioritizing energy conservation and uniform flow distribution, such as fluid transport and pumping systems. The 6 mm nozzle, offering a balance between the higher jet velocity of the 4 mm and the hydraulic efficiency of the 9 mm nozzle, represents a middle ground for applications that require moderate velocity with reduced energy loss. This study highlights the trade-offs between velocity and pressure, allowing engineers to make informed decisions when selecting nozzle sizes for specific industrial applications.

1. Introduction

Converging nozzles are widely employed in various engineering applications such as abrasive water-jet machining, propulsion systems, surface finishing, jet grouting, and hydraulic energy devices. The geometry of these nozzles, particularly the bore diameter, has a significant impact on

* *Corresponding author.*
E-mail address: dd220023@student.uthm.edu.my

https://doi.org/10.37934/afhme.7.1.4048a

flow characteristics such as exit velocity, pressure drop, and turbulence intensity. Numerous studies have shown that nozzle bore diameter plays a crucial role in determining jet velocity and energy losses, making it a key design parameter in nozzle applications. For instance, smaller bores result in higher jet velocity but at the expense of increased pressure losses and energy consumption, while larger bores tend to provide more uniform flow profiles with reduced energy loss [1].

Previous research on converging nozzles has focused on both experimental and computational investigations to understand the effects of nozzle geometry on fluid dynamics. Early experimental studies primarily utilized particle image velocimetry (PIV), pressure transducers, and laser Doppler anemometry to characterize velocity profiles and pressure distribution in nozzle flows [2]. More recently, computational fluid dynamics (CFD) has been employed to simulate nozzle performance under varying operational conditions, providing deeper insights into flow behaviour and performance optimization [3,4]. These studies have demonstrated that nozzle geometry, especially bore diameter, significantly affects jet stability, energy efficiency, and overall performance [5,6].

However, despite the wealth of literature on nozzle performance, few studies have specifically isolated the effect of bore diameter while keeping all other variables constant [7]. By isolating bore diameter, this study aims to provide a clearer understanding of how this parameter influences velocity profiles, turbulence behaviour, and pressure drop in converging nozzles. The accurate prediction of flow through converging nozzles necessitates the use of appropriate turbulence models. Nozzle flows often exhibit complex turbulence patterns due to high velocity gradients and shear effects. The k–ε turbulence model is one of the most used models for nozzle simulations because of its simplicity and computational efficiency [8,9]. However, it has limitations in handling flows with strong velocity gradients or recirculation's [10]. Recent studies have proposed modifications to the standard k–ε model to improve its accuracy in nozzle simulations, including incorporating wall-function modifications and two-layer approaches [11,12].

For more complex flows, such as those involving flow separation or vortex shedding, more advanced models like Large Eddy Simulation (LES) or Reynolds Stress Models (RSM) are required to capture the finer details of turbulence dynamics [13,14]. However, these models come at a higher computational cost, making them less practical for design-oriented applications. The present study uses the standard k–ε model, which is sufficient for capturing the primary effects of bore diameter on nozzle performance under steady-state flow conditions.

CFD has become an invaluable tool for understanding and predicting fluid flow in nozzles. By numerically solving the governing Navier-Stokes equations, CFD provides detailed information on velocity fields, pressure distributions, and turbulence characteristics, which are difficult to measure experimentally [15,16]. CFD also allows engineers to visualize complex flow phenomena such as flow separation, vortex formation, and recirculation zones, which are important in nozzle design [17]. Recent advancements in CFD software, such as ANSYS Fluent, have made it easier to model and simulate complex nozzle flows. These simulations provide insights into the effects of nozzle geometry on flow behaviour, enabling engineers to optimize nozzle designs for specific applications [18,19]. Additionally, CFD is a cost-effective alternative to experimental testing, especially when multiple design iterations are required or when physical testing is impractical [20].

Understanding how nozzle bore diameter affects flow behaviour is crucial for many industries, including waterjet cutting, surface finishing, and propulsion systems. In these applications, nozzle efficiency directly impacts both performance and energy consumption. Smaller bores result in higher exit velocities, which are desirable for high-impact applications, but they also lead to increased pressure losses and higher energy consumption [1]. Larger bores, on the other hand, reduce pressure losses but may result in a less concentrated jet, which may not be ideal for applications that require high shear or momentum [2].

Given these trade-offs, this study aims to provide a comprehensive analysis of how bore diameter influences nozzle performance using CFD. By analysing three different bore diameters which is 4 mm, 6 mm, and 9 mm, this study will help engineers understand the relationship between nozzle diameter, jet velocity, pressure drop, and turbulence characteristics. The findings from this study will aid in the selection of nozzle diameters for specific industrial applications, improving energy efficiency and performance.

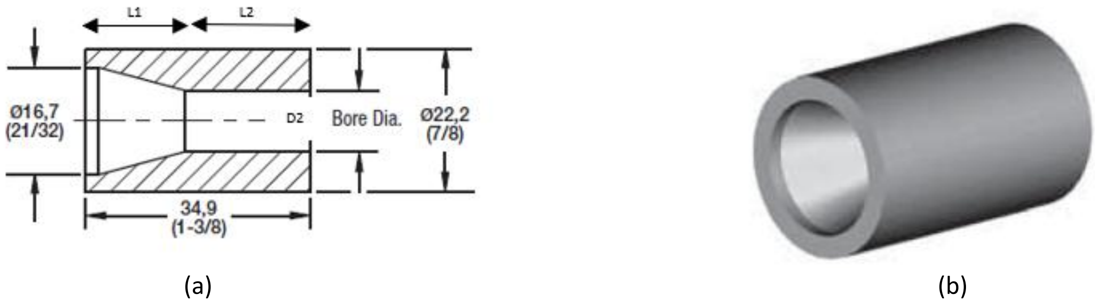
Despite extensive prior research on nozzle flows, much of the published literature focuses on complex multiphase, abrasive, or cavitating jets, leaving a comparative gap in the systematic analysis of how a single geometric parameter (bore diameter) affects performance under controlled, identical inlet conditions for single-phase water flow. Consequently, engineers are often required to select between smaller or larger nozzles without a clear quantitative understanding of the resulting changes in key performance metrics such as the exit velocity profile, pressure distribution along the converging section, total pressure losses, outlet turbulence intensity, and overall energy consumption implications. To address this gap, the present study conducts a systematic Computational Fluid Dynamics (CFD) analysis by modelling and comparing converging nozzles with bore diameters of 4 mm, 6 mm, and 9 mm using ANSYS Fluent. The specific objectives are to model steady turbulent water flow through these three geometries, perform a grid-independence study to ensure mesh-converged results, and quantitatively compare the velocity fields, pressure distributions, and total pressure losses to elucidate the effect of bore diameter on nozzle hydrodynamic performance.

2. Methodology

2.1 Geometry and Computational Domain

The converging nozzle geometries were based on the CL166 Series from the Kennametal Abrasive Flow Products catalogue, which is commonly used in abrasive flow machining applications. Figure 1 shows the design of the converging nozzle from the catalogue. Three bore diameters were selected for this study: 4 mm, 6 mm and 9 mm. These sizes represent typical small-to-medium nozzle exits for industrial flow control and jet applications. The inlet diameter and the overall length of the converging section were maintained constant across all three cases to isolate the effect of the throat diameter as the sole variable.

A three-dimensional, full-body model of the internal fluid volume was created for each nozzle. The use of a full 3D model, as opposed to exploiting symmetry, was adopted to capture any potential three-dimensional flow asymmetries that might arise in the turbulent flow field, thereby ensuring a more comprehensive and physically representative simulation. Figure 2 shows the model structure of the converging nozzle with the provided geometry in Table 1.



(a) (b)

Fig. 1. Design of convection nozzle (a) 2D design (b) 3D design [7]

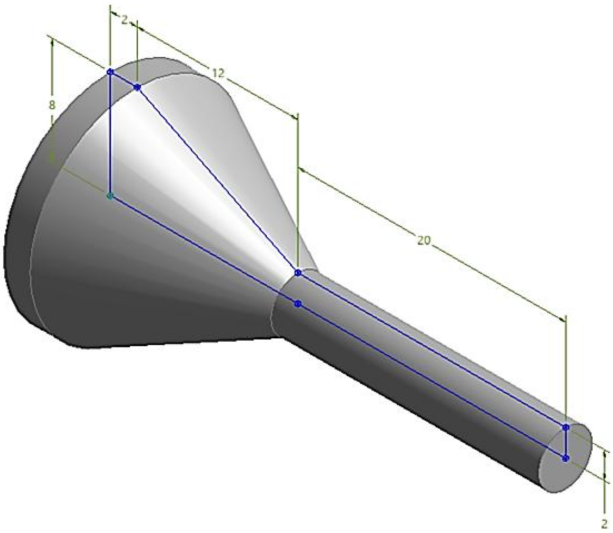


Fig. 2. Geometry structure of converging nozzle case A

Table 1
Detail Dimension of convection nozzle [7]

D1 (mm)	L1 (mm)	L2 (mm)	D2 (mm)
16	14	20	4
16	14	20	6
16	14	20	9

2.2 Mesh Generation and Grid Independence Test

Mesh generation was performed using an unstructured tetrahedron method, well-suited for curved internal surfaces and small throat regions. Body sizing controls were applied to specify the base element size in the converging section and a finer element size around the throat and exit, where high velocity gradients and strong shear were expected. Figure 3 shows the meshing of the model.

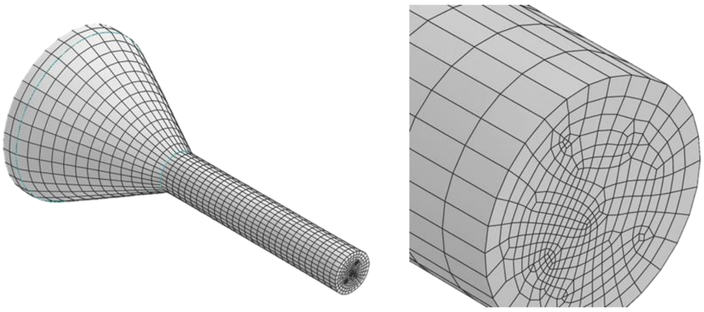


Fig. 3. Meshing of the model

2.3 Grid Independence Test

To verify that the simulation results were not affected by the mesh resolution, a grid-independence test was conducted. Several meshes with different numbers of nodes were generated for one reference geometry, and the resulting pressure difference between inlet and outlet was recorded.

2.4 Governing Equation

In CFD, the governing equations are derived from physical conservation laws: conservation of mass and momentum [5,13,18].

2.4.1 Continuity equation (mass conservation)

The continuity equation expresses conservation of mass in the flow field. For incompressible flow, it can be written as:

$$\nabla \cdot u = 0 \quad (1)$$

where u is the velocity vector. This equation ensures that there are no artificial sources or sinks of mass in the numerical solution, and that what flows into a control volume must either accumulate or flow out [13,18].

2.4.2 Momentum equation (Navier–Stokes)

Eq. 2 represents conservation of momentum, applying Newton's second law to a fluid element. It states that the change in momentum (left-hand side) arises from forces acting on the fluid: pressure gradients push the fluid, viscous stresses resist motion, and body forces like gravity add or remove momentum. In CFD, this equation is central because it governs how velocity fields evolve in response to forces it captures how pressure and friction (viscosity) shape the flow's behaviour, including acceleration, deceleration, and shear.

$$\rho\left(\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u)\right) = -\nabla p + \nabla \times \tau + \rho b \quad (2)$$

2.5 Boundary Conditions and Fluid Properties

The working fluid was specified as water at standard conditions, with a density of 998.2 kg/m³ and its corresponding dynamic viscosity. The boundary conditions were defined with an inlet set as a uniform axial velocity of 1 m/s and a turbulence intensity of 5%, specified with the hydraulic diameter, while the outlet was defined as a pressure outlet at 0 Pa gauge pressure; all other surfaces were stationary walls with a no-slip condition. For the numerical solution, the SIMPLE scheme was employed for pressure-velocity coupling, and second-order accuracy was ensured by using Second-Order Upwind schemes for the momentum, turbulent kinetic energy (k), and turbulent dissipation rate (ε) equations, along with second-order pressure interpolation and the Least Squares Cell Based method for gradient computation on the unstructured tetrahedral mesh. To promote stable convergence, under-relaxation factors were applied for pressure (0.3), momentum (0.7), turbulent kinetic energy (0.8), and turbulent dissipation rate (0.8).

3. Results and Discussion

3.1 Grid-Independence Verification

GIT was executed using velocity distribution to determine the appropriate mesh. Figure 4 is the velocity profile for each mesh sizing. The value of X is negative since the model throat were drawn at the origin. The relative error between two successive meshes was calculated using Eq. (3), and the calculated normalized relative error provided less than 5% for all the sampled. Grid-convergence was confirmed as pressure difference and exit velocity profiles changed is smaller when going from medium to fine mesh. This is shown in Table 2 where the calculation of pressure different is analyse. So, the medium mesh was selected for all subsequent runs since it will take less time to do the analysis.

$$\text{Error (\%)} = |\frac{\text{A}-\text{B}}{\text{A}}| \times 100 \quad (3)$$

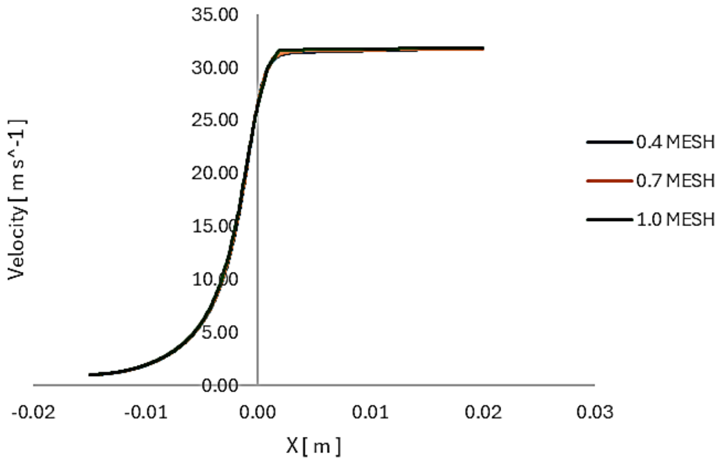


Fig. 4. Velocity profile for different meshing

Table 2
The results of pressure drop

Mesh	Nodes	Element size (mm)	Pressure inlet (Pa)	Pressure outlet (Pa)	Pressure different (Pa)	Error (%)
Coarse	5967	1.0	147527.36	0	147527.36	3.49
Medium	21783	0.7	149483.35	0	149483.35	1.26
Fine	125304	0.4	151395.63	0	151395.63	-

3.2 Velocity Field and Exit Profiles

From Figure 5, the simulation results demonstrate a clear and expected relationship between bore diameter and flow acceleration. As the bore diameter decreased, the flow acceleration increased, yielding higher exit velocities. The velocity field results clearly demonstrate the expected influence of bore diameter on flow acceleration within the converging nozzle. As the throat diameter decreases, the flow experiences a stronger contraction effect, resulting in a greater rise in axial velocity toward the exit. Among the three geometries, the 4 mm bore diameter produces the highest centreline and exit velocities. Its velocity profile is sharply peaked, indicating strong acceleration and a narrow high-momentum core. This behaviour also reflects elevated shear rates along the nozzle walls, consistent with the high velocity gradients commonly observed in small-diameter nozzles.

The 6 mm bore presents an intermediate behaviour, with moderate exit velocity and a less steep velocity gradient. Meanwhile, the 9 mm bore exhibits the lowest peak velocity but provides the most uniform velocity distribution across the nozzle exit. This smoother profile indicates less intense shear and a broader, more evenly distributed jet. These observations align with previous nozzle optimization studies, which emphasize the trade-off between jet concentration and energy requirements: smaller bores increase jet velocity and impact force, making them suitable for high-intensity applications, whereas larger bores promote more uniform flow and reduced energy losses [1,2]. Thus, the velocity results reaffirm that bore diameter governs both the magnitude and distribution of the exit jet velocity.

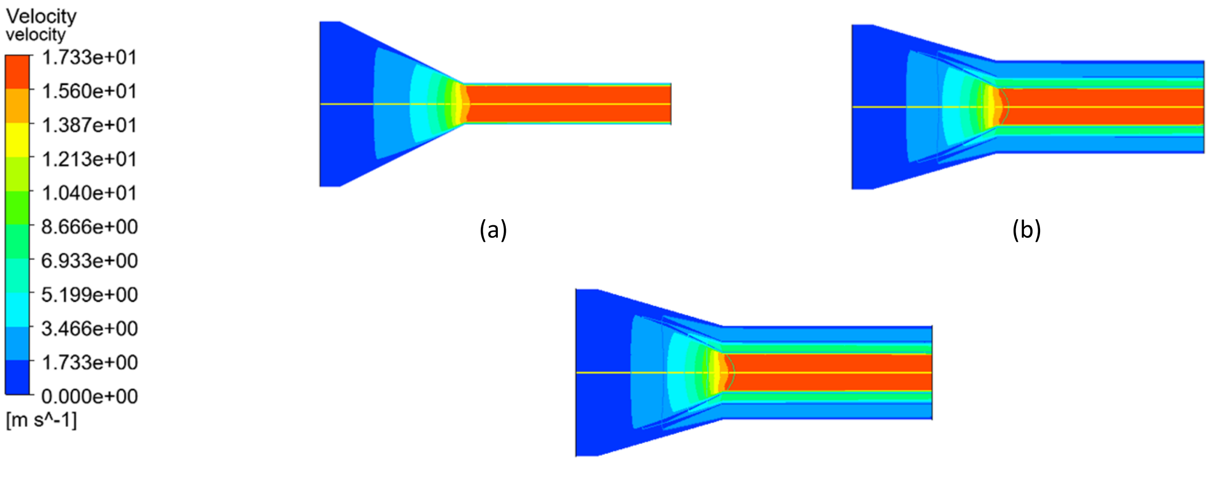


Fig. 5. Velocity contour of bore diameter (a) 4 mm (b) 6 mm (c) 9 mm

3.3 Pressure Distribution and Losses

From Figure 6, the contour shows pressures distribution along the nozzle further supports the relationship between bore size and hydraulic performance. The 4 mm nozzle shows the steepest pressure gradient from inlet to throat, reflecting significant energy conversion from pressure to kinetic energy. This sharp pressure drop corresponds directly to its high exit velocity and indicates a higher overall pressure loss, meaning more pumping energy is required to maintain the same inlet velocity.

In contrast, the 9 mm nozzle demonstrates a much gentler decline in pressure along the converging section. Due to its larger flow area, the fluid undergoes less acceleration, leading to the lowest pressure loss among the three geometries. This makes the 9 mm design more hydraulically efficient under identical inlet conditions. The 6 mm bore again behaves as an intermediate case, with moderate pressure gradients and pressure losses.

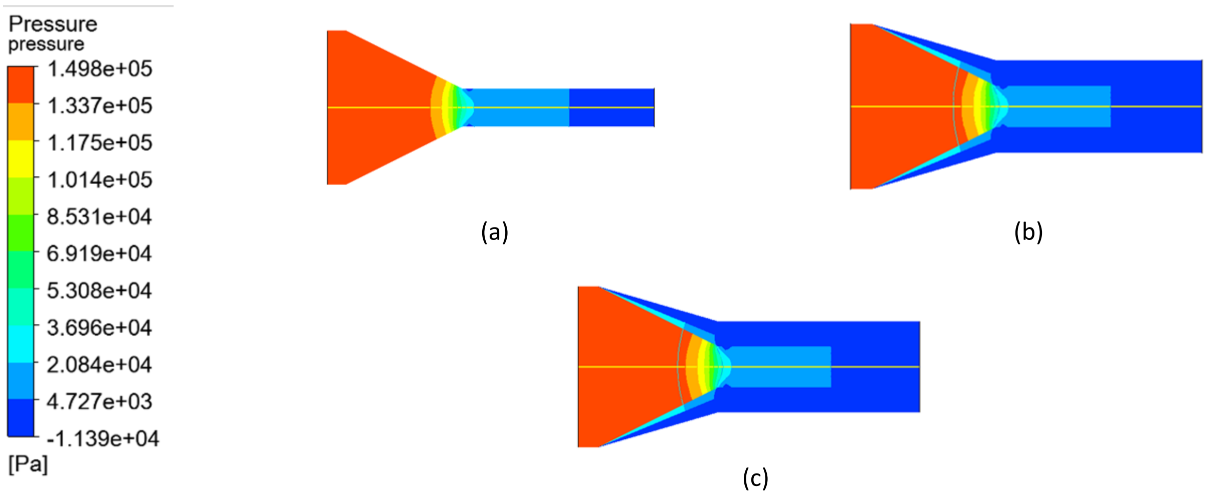


Fig. 6. Pressure contour of bore diameter (a) 4 mm (b) 6 mm (c) 9 mm

This inverse relationship between bore diameter and pressure drop is well-established in nozzle flow theory—higher acceleration demands higher pressure energy, while larger throats reduce velocity amplification and thus minimize pressure losses. These findings are consistent with published studies on nozzle performance, which highlight that reducing bore diameter increases both the exit velocity and the required pressure input, whereas larger diameters Favor energy savings and smoother fluid delivery.

4. Conclusions

The analysis of both velocity and pressure contours highlights the significant impact of nozzle bore diameter on flow characteristics. As the bore diameter decreases, both the exit velocity and pressure drop show a predictable trend. For the 4 mm nozzle, the velocity is the highest, with a dramatic 423.3% increase compared to the 9 mm nozzle, and a 130.9% increase over the 6 mm nozzle. This high velocity is complemented by the lowest pressure, with an 81.7% decrease compared to the 6 mm nozzle, and 96.5% lower than the 9 mm nozzle. This makes the 4 mm nozzle ideal for high-impact applications that require intense jet momentum, such as cutting, but at the cost of higher energy consumption due to the significant pressure drop.

The 6 mm nozzle provides a balance between high velocity and moderate pressure loss. It shows a 128.4% increase in velocity compared to the 9 mm nozzle, with a corresponding 81.1% lower pressure. This makes the 6 mm nozzle a good compromise for applications that require moderate jet velocity and reduced energy loss. The 9 mm nozzle exhibits the most uniform pressure distribution with the least pressure loss. However, it produces the lowest exit velocity, which is about 128.4% lower compared to the 6 mm nozzle and 423.3% lower than the 4 mm nozzle. This nozzle is ideal for applications that prioritize hydraulic efficiency and minimal energy consumption, providing a more uniform flow but with lower momentum.

In conclusion, the nozzle selection depends on the desired balance between velocity and pressure. Smaller nozzles like the 4 mm provide higher exit velocities for high-intensity applications, but with significant energy loss due to pressure drop. Larger nozzles like the 9 mm offer better energy efficiency but at the expense of lower velocity. The 6 mm nozzle represents a middle ground, offering a good balance of both velocity and pressure efficiency.

References

[1] Zhang, Bin, Chencheng Zhu, Jianxun Li, Hao Wang, Xiaolei Liu, and Kan Wang. "Optimization study on nozzle selection based on the influence of nozzle parameters on jet flow field structure." *Applied Sciences* 14, no. 19 (2024): 9098. https://doi.org/10.3390/app14199098

[2] Liu, Ying, Xiaoyang Chen, Jun Zhang, Lele Feng, Huiwen Liu, and Chong Hao. "Structural optimization design of ice abrasive water jet nozzle based on multi-objective algorithm." *Flow Measurement and Instrumentation* 97 (2024): 102586. https://doi.org/10.1016/j.flowmeasinst.2024.102586

[3] Ge, Hongen, Xin Zhang, and Yuqi Liu. "Numerical and experimental study on the jet behavior of ultrafine dry powder based on a supersonic nozzle." *ACS Omega* 9, no. 46 (2024): 46574-46587. https://doi.org/10.1021/acsomega.4c08542

[4] Luo, Yun, Jingyu Zang, and Hongxiang Zheng. "Flow field and gas field distribution of non-submerged cavitation water jet based on dual-nozzle with concentric configuration." *Water* 15, no. 16 (2023): 2904. https://doi.org/10.3390/w15162904

[5] Shen, Shui-Long, Pierre Guy Atangana Njock, and Annan Zhou. "Influence of nozzle structure on effectiveness of jet grouting operations and its optimal design." *Geoenergy Science and Engineering* 226 (2023): 211788. https://doi.org/10.1016/j.geoen.2023.211788

[6] Li, Jiadong, Yixiang Liao, Ping Zhou, Dirk Lucas, and Liang Gong. "Numerical simulation of flashing flows in a converging–diverging nozzle with interfacial area transport equation." *Processes* 11, no. 8 (2023): 2365. https://doi.org/10.3390/pr11082365

[7] Kennametal. *Abrasive Flow Products Catalog* (2009).

[8] Zhang, Xin, Chunyang Guo, Hongen Ge, and Longfei Jiao. "Analysis of internal flow field characteristics in the nozzle of the three-phase jet fire monitor." *Scientific Reports* 15, no. 1 (2025): 21968. https://doi.org/10.1038/s41598-025-05237-0.

[9] Said, Muhamad, Abouelmagd Abdelsamie Abdelsamie, Kareem Emara, and Momtaz Sedrak. "Impact of the jet velocity on the turbulent liquid jet." *Engineering Research Journal* 179 (2023): 249-273. https://doi.org/10.21608/erj.2023.323970

[10] Nwogu, Okey G. "A flexion-based approach for the simulation of turbulent flows." *Physics of Fluids* 32, no. 5 (2020). https://doi.org/10.1063/5.0007825

[11] Abd Halim, M. A., NAR Nik Mohd, MN Mohd Nasir, and M. N. Dahalan. "The evaluation of k-ε and k-ω turbulence models in modelling flows and performance of s-shaped diffuser." *International Journal of Automotive and Mechanical Engineering* 15, no. 2 (2018). https://doi.org/10.15282/ijame.15.2.2018.2.0399

[12] Olson, Britton, and Sanjiva Lele. "Large-eddy simulation of an over-expanded planar nozzle." In *41st AIAA Fluid Dynamics Conference and Exhibit*, p. 3908. 2011. https://doi.org/10.2514/6.2011-3908

[13] Khalid, Muhammad Waqas, and Muhammad Ahsan. "Computational fluid dynamics analysis of compressible flow through a converging-diverging nozzle using the k-ε turbulence model." *Engineering, Technology & Applied Science Research* 10, no. 1 (2020): 5180-5185. https://doi.org/10.48084/etasr.3140

[14] Lian, Xiao, Hao Xu, Lei Duan, and Tiezhi Sun. "Flow structure and parameter evaluation of conical convergent–divergent nozzle supersonic jet flows." *Physics of Fluids* 35, no. 6 (2023). https://doi.org/10.1063/5.0151556

[15] Yang, Yongfei, Weidong Shi, Linwei Tan, Wei Li, Songping Chen, and Bo Pan. "Numerical research of the submerged high-pressure cavitation water jet based on the RANS-LES hybrid model." *Shock and Vibration* 2021, no. 1 (2021): 6616718. https://doi.org/10.1155/2021/6616718

[16] Rhakasywi, Damora, Albi Hagi Rahmat, and Marcellino Rayhan Irwandi. "Study of nozzle injection performance using CFD." *Journal of Sustainable Mechanical Engineering* 2, no. 2 (2024): 51-55. https://doi.org/10.54378/josme.v2i2.9780

[17] Kumaran, T., Balaji Elangovan, S. Swetha, Raja Kannan, Murugu Nachippan Nachiappan, and Sarangapani Palani. "Optimization of the convergent-divergent nozzle tip and reducing the acoustic energy." *Turkish Journal of Engineering* 9, no. 3 (2025): 417-424. https://doi.org/10.31127/tuje.1561759

[18] Bruccoleri, Alexander R., Randy Leiter, Mark Drela, and Paulo Lozano. "Experimental effects of nozzle geometry on flow efficiency at low Reynolds numbers." *Journal of Propulsion and Power* 28, no. 1 (2012): 96-105. https://doi.org/10.2514/1.B34073

[19] Rahman, Mohammad Shajid, Godwin Fabiola Kwaku Tay, and Mark Francis Tachie. "Effects of nozzle geometry on turbulent characteristics and structure of surface attaching jets." *Flow, Turbulence and Combustion* 103, no. 3 (2019): 797-825. https://doi.org/10.1007/s10494-019-00047-7

[20] Jafar, Sarfraz Ali, Muhammad Tanveer, Asad Ali, Dilawar Ali, and Saqib Ali Awan. "Numerical investigation of effect of nozzle diameter on the performance of pelton turbine." *Annual Methodological Archive Research Review* 3, no 9 (2025): 601-613.