

Numerical Study of Nozzle Geometry and Back Pressure Effects on High-Pressure Waterjet Propulsion Systems

Fathin Muhammad Mahdhudhu^{1*}, Adilia Apriliana², James Julian³, Elvi Armadani⁴, Riki Hendra Purba⁵, Shirad Riezky Ilahi⁶

(Received: 23 June 2026 / Revised: 4 July 2026 / Accepted: 26 July 2026 / Available Online: 10 September 2026)

Abstract— The performance of Autonomous Underwater Vehicle (AUV) waterjet propulsion systems is highly dependent on nozzle geometry and environmental back pressure. Therefore, this study aims to analyze the effects of nozzle length, length-to-diameter (l/d) ratio, and back pressure on thrust, velocity, flow coefficient, and hydraulic loss. The study was conducted using Computational Fluid Dynamics (CFD) simulations. Three nozzle geometries (Cos, Conical, and Exponent) were evaluated at lengths of 40, 60, and 90 mm. The optimal geometry was then analyzed by varying the l/d ratio (1, 1/2, 1/3, and 1/4) under back pressure conditions from 0 to 1 MPa. The results indicate that the Cos nozzle with a 90 mm length yields the highest thrust and velocity. Furthermore, an l/d ratio of 1/3 demonstrates the optimal performance, achieving a near-ideal flow coefficient (0.999) and minimal hydraulic loss, particularly under low back pressure conditions. Ultimately, a Cos nozzle with a 90 mm length and an l/d ratio of 1/3 is highly recommended to maximize the propulsion efficiency of AUVs.

Keywords— High Pressure Waterjet Propulsion, Nozzle Optimization, CFD;

*Corresponding Author: fmahdhudhu@upnvj.ac.id

I. INTRODUCTION

Autonomous Underwater Vehicles (AUVs) are untethered robotic platforms designed to execute missions without real-time human intervention or physical surface connections [1]. The deployment of AUVs has expanded rapidly across various sectors, including oceanographic surveying, bathymetric mapping, and offshore infrastructure inspection. Market forecasts by Precedence Research indicate that the global AUV industry will expand from a valuation of USD 3.48 billion in 2025 to approximately USD 6.69 billion by 2033, reflecting a compound annual growth rate (CAGR) of 8.52%. As the operational scope of AUVs broadens, optimizing propulsion system performance has become critical, as these subsystems directly govern vehicle velocity, hydrodynamic stability, energy consumption, and overall mission endurance.

AUV applications, particularly for military purposes, require propulsion systems capable of high speeds, good mobility, and low acoustic noise. Advanced control systems enhance the maneuverability of waterjet-propelled vehicles [2]. These requirements pose significant challenges for conventional propeller-based propulsion systems, which have drawbacks in terms of

noise, vibration, and limited maneuverability. Although waterjet propulsion system is structurally more complex than conventional screw propellers, they offer significant advantages, including higher propulsive efficiency, lower noise and vibration, and better maneuverability [3].

These systems operate on the principle of water ingestion through intake ducts, followed by pressurization and high-velocity ejection through specially designed nozzles. Crucially, as an AUV descends to greater operating depths, the ambient hydrostatic pressure increases proportionally. This elevated external pressure acts as a back pressure at the nozzle exit, which fundamentally alters the jet's hydrodynamic characteristics, volumetric flow rate, and the resultant thrust and velocity vectors.

Many factors affect waterjet performance, one of which is nozzle geometry, including the nozzle type, outlet shape, outlet size, and transition from the duct to the nozzle. Optimization of inlet passage geometry significantly influences overall system efficiency [4]. Nozzle geometry has been shown to have a significant impact on hydraulic performance and thrust [5]. Nozzle performance in a waterjet system is greatly influenced by internal flow characteristics, which are generally represented by the flow coefficient (C_d). Geometric differences, particularly in outlet diameter and length, have been shown to affect velocity distribution, energy loss, and thrust, where a smaller inlet or outlet area can result in greater hydraulic losses [6, 7].

Extensive numerical evaluations of AUV waterjet systems have been conducted to identify optimal nozzle configurations. For instance, Jian et al. [8] demonstrated that cosine-profile nozzles generate superior momentum thrust compared to other geometries at optimized lengths, while also highlighting that smoother inner wall surfaces enhance thrust generation. Shifting focus to

Fathin Muhammad Mahdhudhu, UPN "Veteran" Jakarta, Indonesia. Email: fmahdhudhu@upnvj.ac.id

Adilia Apriliana, UPN "Veteran" Jakarta, Indonesia. Email: 2110313032@upnvj.ac.id

James Julian, UPN "Veteran" Jakarta, Indonesia. Email: james@upnvj.ac.id

Elvi Armadani, UPN "Veteran" Jakarta, Indonesia. Email: elviarmadani@upnvj.ac.id

Riki Hendra Purba, UPN "Veteran" Jakarta, Indonesia. Email: rikihendrapurba@upnvj.ac.id

Shirad Riezky Ilahi, UPN "Veteran" Jakarta, Indonesia. Email: 2410313025@mahasiswa.upnvj.ac.id

outlet geometries, Wang et al. [9] established that circular outlet profiles yield the most favorable hydraulic outcomes, improving flow velocity uniformity and boosting pump efficiency. From a flow stability perspective, Gu et al. [10] combined computational and experimental approaches to reveal that the convergence angle critically dictates throat flow stability. More recently, Wang et al. [11] highlighted the advantages of tapered geometries over flat or straight conical designs, noting their ability to produce highly stable jets with reduced energy dissipation.

However, despite these extensive investigations, a critical gap remains in the current literature. Previous studies predominantly isolate specific variables; they either focus solely on nozzle geometry under ideal surface conditions or analyze operating conditions without comprehensively mapping the geometric trade-offs. Consequently, there is a lack of understanding regarding how the length-to-diameter (l/d) ratio interacts with environmental back pressure to affect the delicate balance between absolute thrust generation and hydrodynamic efficiency. Designing a nozzle that performs optimally at the surface but fails to maintain efficiency and flow stability at greater depths remains a significant practical problem for AUV operations.

To address these limitations, the novelty of this study lies in the integrated optimization of AUV waterjet nozzles by simultaneously investigating the coupled effects of nozzle profile, nozzle length, length-to-

diameter ratio, and environmental back pressure. Unlike previous studies that examined these parameters separately, this work establishes a comprehensive design framework that identifies the optimal nozzle configuration for maintaining thrust, hydraulic efficiency, and flow stability under varying operating depths.

Therefore, the primary objective of the current study is to analyze how these geometric (geometric type of nozzle, nozzle length, and ratio length and inlet diameter) and environmental variations (back pressure) influence thrust, velocity, flow coefficient, and hydraulic loss. By identifying the optimal geometric configuration that maintains high efficiency and minimal hydraulic loss under varying back pressures, this study provides actionable design guidelines for enhancing the deep-water maneuverability and propulsion performance of AUVs.

II. METHOD

A. Boundary Design and Meshing

Three different types of nozzles are used in this study: the Cos Nozzle, the Exponent Nozzle, and the Conical Nozzle. The results of the study by Jian et al., [8] show that the Cos Nozzle produces the highest thrust momentum, followed by the Exponent Nozzle and the Con Nozzle.

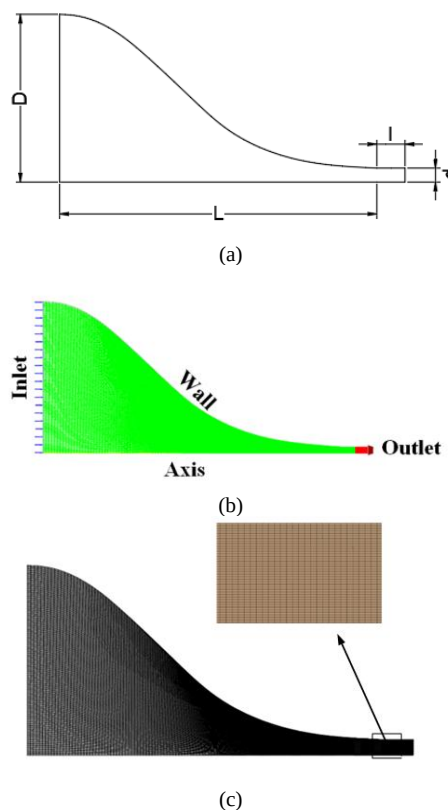


Figure 1. (a) Detailed Nozzle Dimension; (b) Boundary Condition; (c) Meshing

Figure 1. (a) shows the dimensional details of the nozzle, which include the nozzle inlet diameter (D), nozzle length (L), length of the nozzle outlet orifice (l), and nozzle outlet diameter (d). These parameters are used to define the length-to-diameter ratio (l/d) as the

primary variable in the analysis. The nozzle geometry was designed as a convergent nozzle to increase the acceleration of the fluid flow toward the outlet section.

Figure 1. (b) illustrates the boundary conditions in the CFD simulation performed using Ansys Fluent Student.

The boundary conditions are configured to represent a closed propulsion system, meaning that fluid flow occurs only within the nozzle channel without direct interaction with the external environment. The inlet is defined as the boundary where the fluid enters the domain under varying pressure conditions, while the outlet is modeled as a pressure outlet to represent back pressure conditions.

The nozzle wall is defined as a no-slip wall to account for friction losses along the nozzle surface, and the x-axis is used as the reference axis, representing rotational symmetry in the 2D axisymmetric formulation, so that the three-dimensional flow behavior around the nozzle can be efficiently represented by modeling only half of the nozzle.

Spatial discretization of the two-dimensional computational domain was achieved using a structured grid composed of quadrilateral finite-volume elements that conform precisely to the nozzle's physical boundaries. To accurately resolve steep velocity and pressure gradients, localized mesh refinement was applied near the solid walls and the outlet region. Furthermore, gradual transitions in cell sizing were implemented across the domain to preserve high mesh quality and ensure numerical stability, as illustrated in Figure 1(c).

B. Simulation Parameters and Conditions

The computational framework was established within the Ansys Fluent environment utilizing a multiphase flow model to account for potential cavitation. Liquid water was designated as the continuous primary phase, characterized by a density of 998.2 kg/m³, a dynamic viscosity of 1.003×10^{-3} kg/m·s, and a surface tension of 0.0728 N/m. The dispersed secondary phase, water vapor, was defined with a saturated vapor pressure of 3540 Pa, a density of 0.02558 kg/m³, and a viscosity of 1.26×10^{-6} kg/m·s, sharing the same surface tension properties.

The initial boundary conditions include a total pressure of 7 MPa at the inlet and a static pressure at the outlet ranging from 0 to 1 MPa, with the nozzle wall assumed to be under no-slip conditions. Cavitation is modeled using the Zwart–Gerber–Belamri model, which is commonly used in high-pressure flow simulations. Two-phase flow modeling is essential for accurately capturing cavitation phenomena in waterjet systems [12].

Meanwhile, the characteristics of turbulent flow are modeled using the $k-\omega$ SST model. The $k-\omega$ SST (Shear Stress Transport) turbulence model is used in place of the $k-\epsilon$ model due to its ability to predict the internal flow characteristics within the nozzle, particularly under conditions involving sharp pressure gradients and cavitation [13].

C. Key Performance Parameters of Nozzles

The flow coefficient (Cd) of a water jet nozzle is a dimensionless parameter that indicates the nozzle's maximum capacity to discharge fluid, which can be calculated using the following equation [14].

$$Cd = \frac{Q}{A \sqrt{2\Delta P}} \quad (1)$$

In a study by Zhang et al., [15] to relate thrust to hydraulic and geometric parameters, the nozzle is considered to be an orifice at the end of a pressurized pipe. Thus, the volumetric flow rate can be expressed using the flow coefficient Cd as follows:

$$F_N = (1 + 2C_d^2)\Delta p A_{out} \quad (2)$$

Average velocity is used in fluid flow analysis to determine the average velocity of the fluid passing through a cross-section in a flow system, which can be calculated using the equation:

$$V_{avg} = \frac{1}{A} \int_A V dA \quad (3)$$

Hydraulic loss is defined as the total pressure energy loss between the inlet and outlet, which can be calculated using the following equation [9]

$$\Delta h = \frac{P_{in} - P_{out}}{\rho g} \quad (4)$$

D. Variation

The nozzle is a vital component that links the hydraulic characteristics of the impeller to the momentum based thrust equation [16]. Differences in the shape and length of the nozzle alter the pressure and velocity patterns within the nozzle, thereby affecting energy efficiency and the velocity of the resulting jet [17]. With a longer nozzle, the resulting spray is more consistent and has a higher velocity compared to a shorter nozzle [18]. In Shinabuth's study et al., [19] It was found that a decrease in the nozzle l/d ratio typically widens the steam cone angle and improves atomization.

Accordingly, this study employs variations in nozzle length to determine whether such variations influence thrust, velocity, flow coefficient (Cd), and hydraulic loss (Δh) for cos nozzles, conical nozzles, and exponential nozzles. The most optimal length variation identified from the initial analysis is subsequently further developed through additional variations in the length-to-diameter ratio (l/d) and back pressure. This extended analysis aims to determine the most effective nozzle geometry by examining the combined effects of nozzle type, geometric configuration, and environmental influence, as represented by back pressure, using the same evaluation parameters.

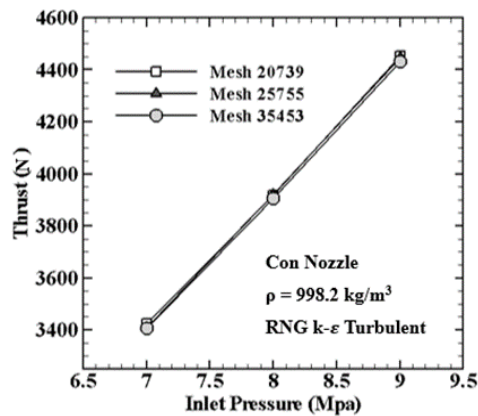
TABLE 1.
VARIATION PARAMETER

Variable	Parameter	Level	Responses
1	Nozzle Type	Cos, Conical, Exponent	Thrust, Cd, Δh , Velocity Average
2	Nozzle Length	40, 60, 90	
3	Ratio l/d	1/4, 1/3, 1/2, 1	
4	Back Pressure	0, 0.1, 0.3, 0.5, 1 Mpa	

III. RESULTS AND DISCUSSION

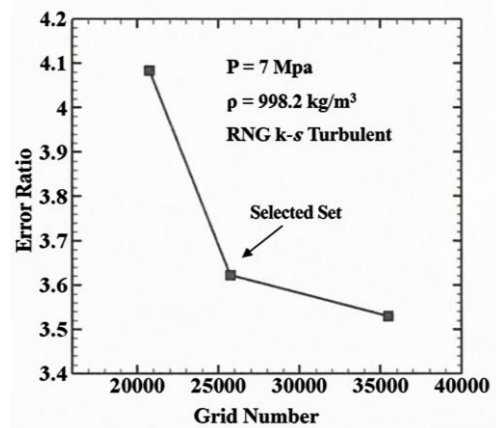
A. Mesh Sensitivity Analysis

Mesh sensitivity Analysis is crucial for ensuring that computational results such as pressure, velocity, and force are not affected by changes in the size or number of mesh elements, thereby reducing numerical errors caused by mesh selection. In addition, mesh sensitivity analysis helps determine the most efficient level of mesh density, in which the mesh is sufficiently fine to provide stable results but not excessive, so that the computational time remains realistic, and it constitutes an important



(a)

part of numerical verification and validation so that the CFD results can be accepted as a scientific reference. In Figure 2. (a), a mesh sensitivity analysis was performed by varying the number of mesh elements to 20,739, 25,755, and 35,453 under three inlet pressure conditions: 7, 8, and 9 MPa. The simulated thrust results were compared with experimental journal data to determine the error values. The simulation results show that changes in inlet pressure have a more dominant influence on thrust values and errors than variations in the number of mesh elements.



(b)

Figure 2. (a) Mesh Sensitivity Analysis; (b) Mesh Convergence

Before performing the main simulation, a mesh convergence test was conducted to ensure that the simulation results had converged with respect to the number of grid cells used. It can be seen that the error ratio decreases as the number of grid cells increases. Based on Figure 2. (b), the decrease in the error ratio is not significant between 25,000 and 35,000 grid cells, indicating that the solution has reached a converged state. Therefore, a grid of 25,000 cells was selected as the optimal mesh configuration for all CFD simulations in this study, taking into account the balance between result accuracy and computational efficiency.

B. Validation Data

The validation was performed by comparing the results of the numerical simulation with experimental data obtained from the study by Shi et al., [20] on various types of nozzles.

The parameter being compared is the reverse thrust of the fluid flow at the nozzle outlet. The simulation was performed using approximately 25,000 mesh elements for each inlet pressure variation to ensure consistency in the results and a uniform level of accuracy.

The validation results in Figure 3 show that the thrust trend obtained from the simulation is consistent with the trend in the reference journal. With an error rate of less than 10%, the numerical model used in this study can be considered calibrated and validated based on that reference.

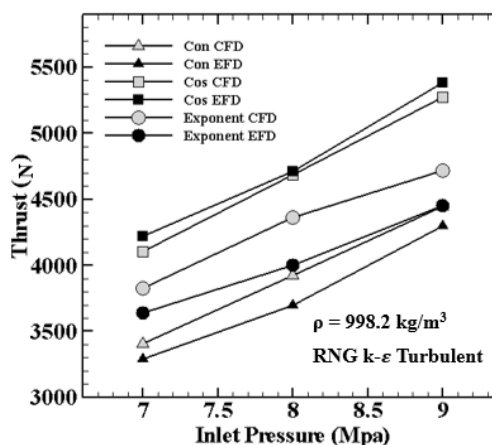
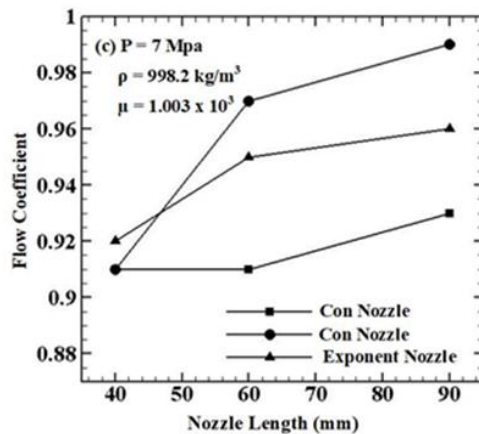


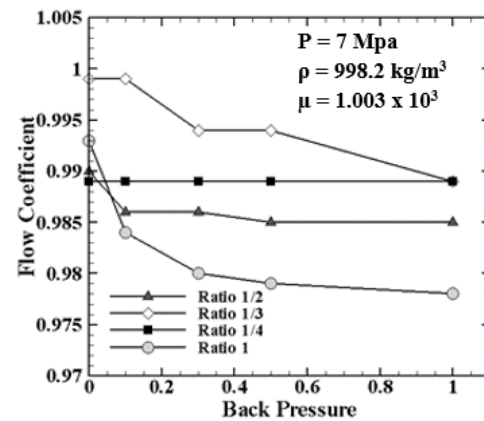
Figure 3. Comparison of Thrust values from previous research and present study

C. Comparison of Flow Coefficients Under Different Variations

The flow coefficient (C_d) is used to evaluate how effectively a nozzle conveys fluid with minimal pressure loss. C_d value of 1.0 indicates an ideal nozzle with no losses, while a C_d value < 1.0 indicates unavoidable losses due to friction, flow separation, or uneven velocity distribution at the outlet. Figure 6. (a) and Figure 6. (b) show the C_d characteristics for two different geometric variations. Graph (a) shows that the Nozzle Cos produces the highest C_d value, increasing from 0.90 to 0.99 as the nozzle length increases, followed by the Nozzle Exponent and Nozzle Con. In graph (b), the



(a)



(b)

Figure 4. (a) Flow Coefficient Length Variation; (b) Flow Coefficient Ratio Variation

In a study by Tharakan et al., [21] An increase in back pressure alters the flow structure around the vena contracta and the boundary layer, where the flow becomes more attached to the wall even though the effective cross-sectional area increases. However, the effects of viscosity and boundary layer thickening become dominant, especially at high l/d ratio, thereby increasing energy losses. Consequently, the effect of backpressure on the discharge coefficient is nonlinear and highly dependent on the geometric ratio. This is consistent with the CFD simulation results of this study, which show that the flow coefficient decreases in nozzles with a high l/d ratio.

D. Comparison of Thrust Under Different Variations

In the equation presented in the journal by Zhang et al., [15] thrust and C_d are related through the momentum of the jet exiting the nozzle. The distinction between net and gross thrust is critical in evaluating waterjet performance [22]. The C_d value indicates the flow rate and actual velocity that can be achieved at a given pressure difference, while thrust is the reaction force proportional to the flow momentum. As the C_d value increases, pressure loss within the nozzle is minimized, resulting in increased flow rate and exit velocity.

The simulation results show that the Cos nozzle produces the highest thrust at a length of 90 mm, with a value of 1047 N, which is significantly greater than that of the Conical and Exponent geometries at the same length, thus proving that the Cos nozzle geometry is the most efficient in increasing flow acceleration and fluid

effect of backpressure shows a different pattern in which all four l/d ratio have very high C_d values (0.978–1.000).

The 1/3 ratio demonstrates superior performance with a stable C_d value in the range of 0.989–0.999 across the entire backpressure range, indicating excellent resistance to changes in operating conditions. The 1/2 ratio decreases as backpressure increases, while the 1/4 ratio remains relatively stable at around 0.989. A comparison of the two graphs shows that optimizing the l/d ratio figure 4. (b) results in a higher and more stable C_d value compared to simply changing the length figure 4. (a), confirming that a 1/3 ratio provides the optimal flow coefficient for the waterjet AUV system.

momentum. Meanwhile, regarding the ratio variations, a 1/4 ratio produces a thrust of 16.456 N with a C_d of 0.989 and an outlet area of $8.08 \times 10^{-4} \text{ m}^2$, while a 1/3 ratio produces a thrust of 9,533 N with the highest C_d value (0.999) but a smaller outlet area ($4.57 \times 10^{-4} \text{ m}^2$).

This confirms that outlet area has a more significant effect on thrust than small variations in the C_d value, although the optimal combination of high C_d , outlet area, and pressure difference in a 1:3 ratio yields the best hydrodynamic efficiency with minimal hydraulic loss.

Research by Beifang Gu [10] has found that for a high-pressure waterjet with an inlet pressure of 20 MPa, the combination of C_d and nozzle exit area determines the thrust output, with the exit area being the factor more sensitive to changes in absolute thrust.

E. Comparison of Velocity Under Different Variation

Research on Con-strait nozzles shows that a small convergence angle (approximately $10\text{--}15^\circ$) and a throat length about three times the outlet diameter result in a more uniform velocity distribution and higher axial velocity at the outlet, thereby increasing the effective jet length. [22]. In a two-phase 2D waterjet nozzle simulation, an increase in backpressure reduces the jet exit velocity because the pressure difference between the inlet and outlet decreases. When the external ambient pressure at the outlet is increased at a constant flow rate, the jet momentum decreases, resulting in a reduction in thrust. This reduction is proportional to the decrease in the jet effective velocity [23].

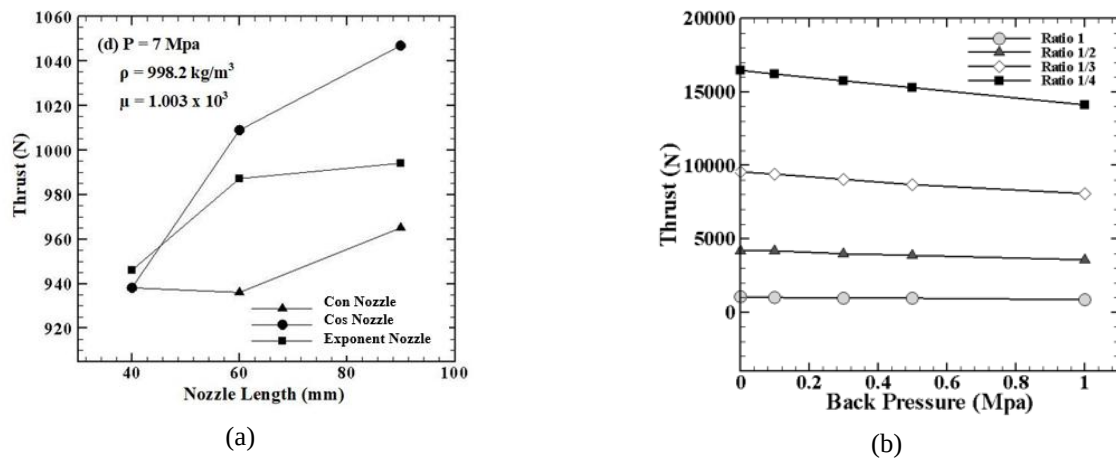


Figure 5. (a) Thrust Length Variation; (b) Thrust Ratio Variation

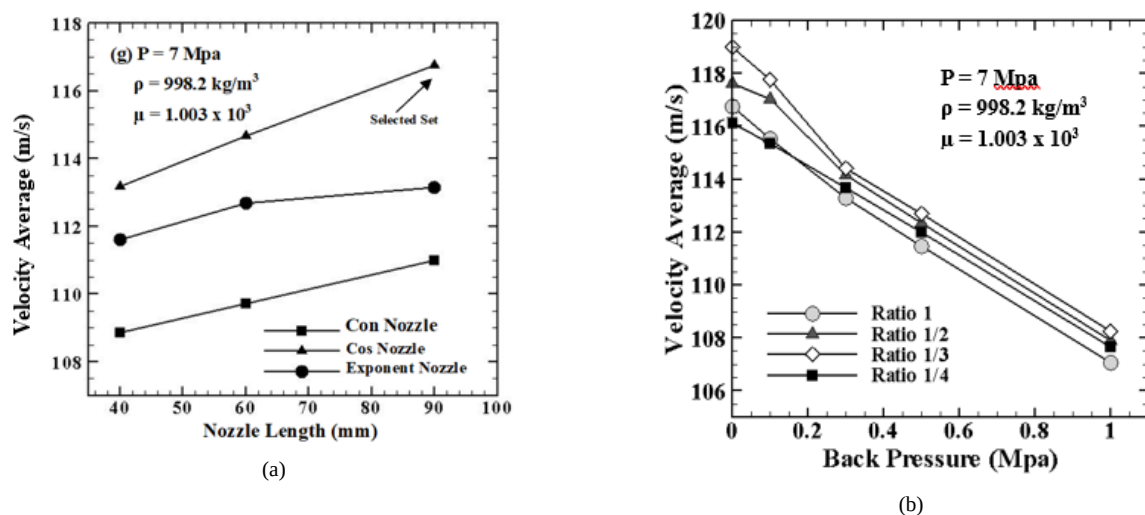


Figure 6. (a) Velocity Length Variation; (b) Velocity Ratio Variation

Among the different nozzle lengths, the Cos nozzle produced the highest velocity, increasing from 113.183 m/s at a length of 40 mm to 116.749 m/s at a length of 90 mm, demonstrating the effectiveness of the cos geometry in accelerating fluid flow. The Con nozzle and the Exponent nozzle exhibited lower velocities with more moderate increases as the nozzle length increased.

Meanwhile, in the variation of the l/d ratio versus back pressure, a consistent pattern is observed in which the highest velocity is achieved at low back pressure (0–0.1 MPa) and gradually decreases as the back pressure increases up to 1 MPa. The 1/3 ratio exhibits the best velocity performance, reaching 118.992 m/s at a backpressure of 0 MPa—higher than the other ratio—indicating that a geometry with an l/d ratio of 1/3 provides optimal conditions for fluid acceleration with minimal flow resistance.

This data indicates that there is an optimal outlet size that provides the best balance between flow and acceleration in the cos-nozzle design used. These findings differ slightly from the results obtained by Beifang Gu et al., [10] research on nozzle slotting, in which at a certain distance from the outlet, the jet velocity along the axial and vertical directions tends to increase as the outlet size is enlarged, because the flow can maintain a larger jet core and velocity decay occurs more slowly. The differences in this trend indicate that the effect of an increase in diameter on velocity depends

not only on the outlet diameter but is also strongly influenced by the nozzle geometry, pressure conditions, and nozzle length.

F. Comparison of Hydraulic Loss Under Different Variations

In the study by Wang et al., [9] on the hydraulic properties of waterjet nozzles, hydraulic loss is defined as the reduction in total pressure energy that occurs between the inlet and outlet of the nozzle, measured in terms of energy head. In terms of nozzle length variations, the 90 mm Cos nozzle has the lowest hydraulic loss at 0.155, which is significantly lower than that of the 90 mm Con nozzle and the 90 mm Exponent nozzle, confirming that the Cos geometry minimizes energy dissipation due to friction and turbulence.

Across the range of l/d ratio versus backpressure, the 1/3 ratio exhibits the lowest hydraulic losses, with values ranging from 0.007 to 0.034—significantly lower than those for the 1/2 and 1/4 ratio. Interestingly, the hydraulic loss at the 1/3 ratio reaches its lowest value at a backpressure of 0.5 MPa, indicating the presence of optimal flow conditions at a specific backpressure where the balance between the pressure gradient and flow velocity minimizes the loss.

The 1/4 ratio exhibits the highest hydraulic loss under all backpressure conditions, which explains why, despite producing high absolute thrust, its specific efficiency is

lower due to significant energy waste in the form of friction and turbulence losses. These results are consistent with the study by C. U. I. Xian et al., [25] which showed that abrupt changes in cross-sectional area

or excessively wide outlet openings increase energy losses; therefore, a smooth transition in cross-sectional area and an appropriate aspect ratio are crucial for reducing hydraulic losses.

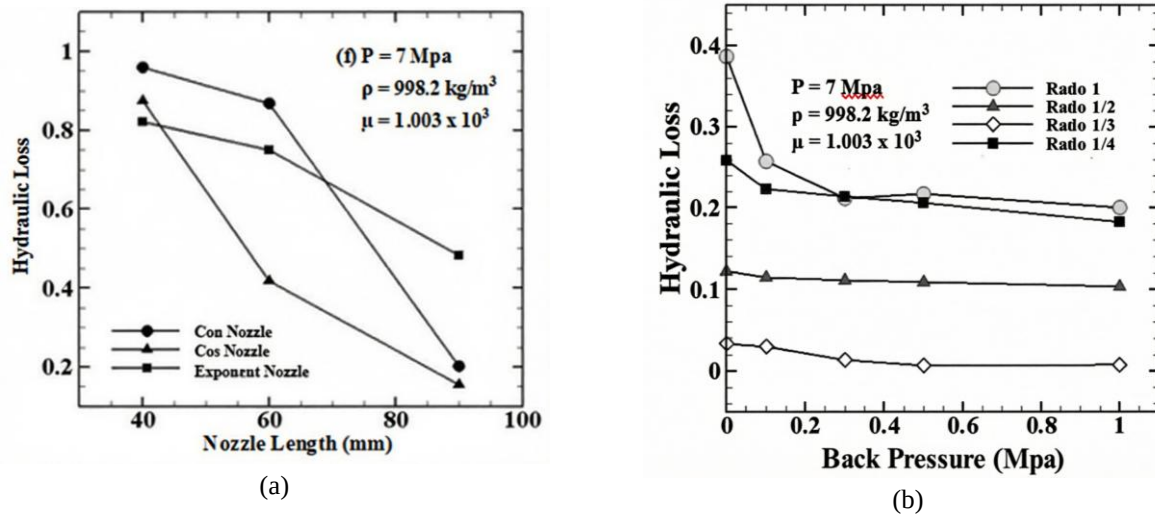


Figure 9. (a) Hydraulic Loss Length Variation; (b) Hydraulic Loss Ratio Variation

G. Velocity Contours

The velocity contours generated from the 2D CFD simulation show the velocity distribution for three types of nozzle geometries (cos, Con, and exponential) at different nozzle lengths with the same outlet diameter. In general, these geometries are capable of accelerating the flow from the inlet to the outlet. This comparison shows that variations in nozzle length affect the flow acceleration pattern. Nozzles with smoother contours (cos) tend to produce a more stable velocity distribution compared to conical and exponential nozzles.

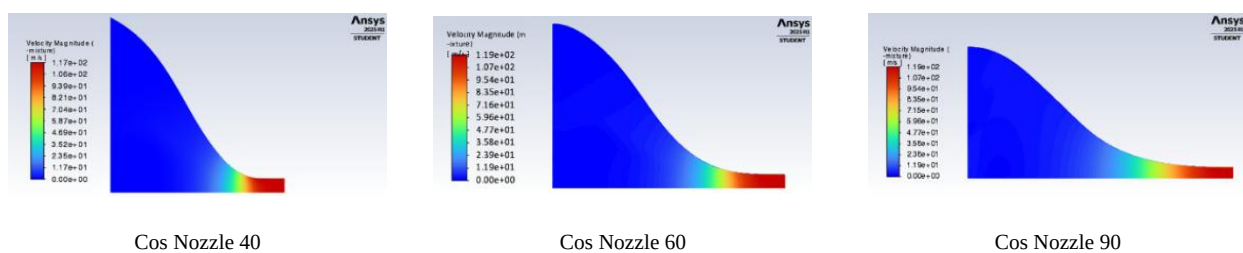
H. Discussion of the novel findings

The present study provides a more comprehensive understanding of the performance of AUV waterjet nozzles by simultaneously considering the combined effects of nozzle profile, nozzle length, length-to-diameter (l/d) ratio, and environmental back pressure. Unlike previous studies, which mainly investigated these parameters independently, the present work demonstrates that nozzle performance is governed by the interaction between geometric configuration and environmental conditions.

One of the most significant findings of this study is the identification of a hydrodynamic trade-off between maximum thrust and propulsion efficiency. The CFD results show that the l/d ratio of 1/4 generates the highest

absolute thrust due to its larger outlet area, which increases the mass flow rate. However, this configuration also produces the highest hydraulic loss because the abrupt geometric transition intensifies flow separation and energy dissipation. In contrast, the l/d ratio of 1/3 consistently maintains a near-ideal flow coefficient, the lowest hydraulic loss, and the highest average jet velocity across the investigated back-pressure range. These results indicate that maximizing thrust alone does not necessarily produce the most efficient propulsion system.

Another important contribution of this study is the incorporation of environmental back pressure into the nozzle optimization process. Most previous investigations evaluated nozzle performance under atmospheric outlet conditions or focused only on geometric modifications. In practical AUV operations, however, increasing operating depth increases the hydrostatic pressure acting at the nozzle outlet, thereby altering the pressure difference responsible for jet acceleration. The present results demonstrate that nozzle performance varies considerably with increasing back pressure and that the optimal configuration should therefore be evaluated under realistic underwater operating conditions rather than ideal laboratory conditions.



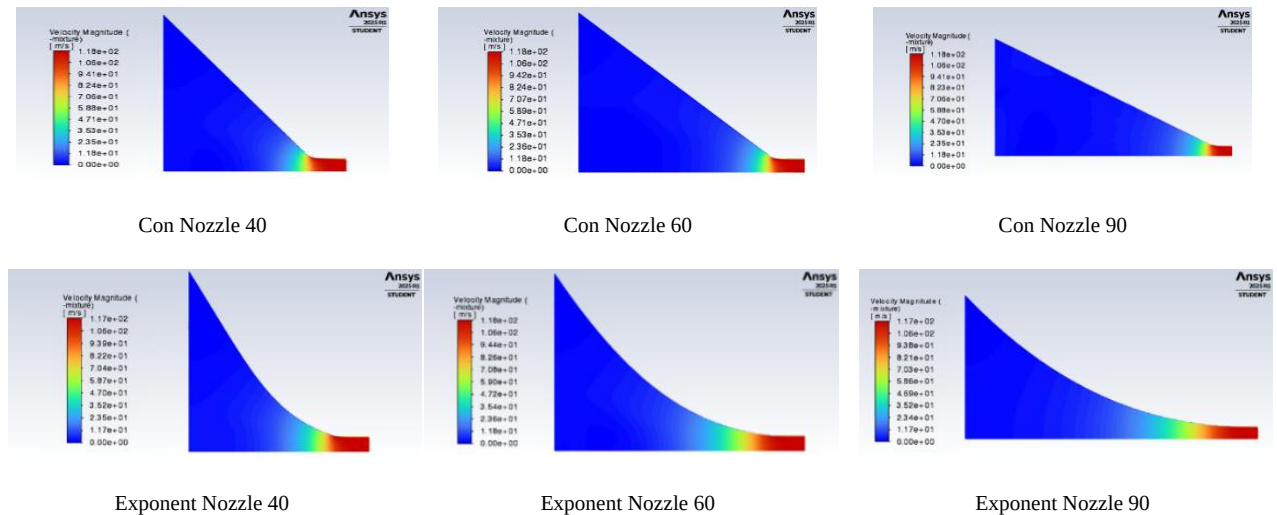


Figure 7. Velocity Contours Length Variation

IV. CONCLUSION

This study successfully analyzed the effects of nozzle geometry (axial length and length-to-diameter ratio) and environmental back pressure on the hydrodynamic performance of AUV waterjet propulsion systems. Regarding nozzle length, the Cos profile at 90 mm demonstrated superior performance, yielding the highest thrust (1047 N) and velocity by minimizing flow separation and energy dissipation compared to Conical and Exponent geometries.

Furthermore, the investigation into the length-to-diameter (l/d) ratio under varying back pressures revealed a critical hydrodynamic trade-off. While an l/d ratio of 1/4 generated the highest absolute thrust due to a significantly larger outlet mass flow rate, it incurred the highest hydraulic losses due to abrupt flow expansion. Conversely, the l/d ratio of 1/3 proved to be the most optimal configuration. It maintained a near-ideal flow coefficient (0.999) and minimal hydraulic loss (0.007–0.034) across the entire back pressure range (0 to 1 MPa), indicating excellent resistance to depth-induced pressure changes.

Therefore, a 90 mm Cos nozzle configuration with an l/d ratio of 1/3 is highly recommended for AUV applications. This specific geometry provides the best balance between propulsion efficiency, flow stability, and deep-water maneuverability, directly addressing the operational challenges of underwater vehicles.

ACKNOWLEDGEMENTS

The author also appreciates the Faculty of Engineering for providing facilities to conduct this research.

REFERENCES

- [1] R. B. Wynn *et al.*, "Autonomous Underwater Vehicles (AUVs): Their past, present and future contributions to the advancement of marine geoscience," *Mar. Geol.*, vol. 352, pp. 451–468, 2014, doi: 10.1016/j.margeo.2014.03.012.
- [2] Z. H. Gong, G. Q. Li, W. Xiong, J. Z. Li, Y. X. Yu, and J. Q. Yuan, "Modeling and simulation of the steering control system of marine water-jet propulsion unit," *Journal of Shanghai*

- Jiaotong University*, vol. 50, no. 11, pp. 1114–1118, 2016.
- [3] D. Ma, Y. Li, T. Ma, and A. M. Pascoal, "The state of the art in key technologies for autonomous underwater vehicles: a review," *Engineering*, 2025, doi: 10.1016/j.eng.2025.08.002.
- [4] W. X. Jiao, L. Cheng, D. Zhang, B. Zhang, Y. Su, and C. Wang, "Optimal design of inlet passage for waterjet propulsion system based on flow and geometric parameters," *Advances in Materials Science and Engineering*, vol. 2019, p. 2320981, 2019.
- [5] Y. sheng Yang, Y. chun Xie, and S. lin Nie, "Nozzle optimization for water jet propulsion with a positive displacement pump," *China Ocean Eng.*, vol. 28, no. 3, pp. 409–419, 2014, doi: 10.1007/s13344-014-0033-4.
- [6] G. Chen, "Jet Performance," *Eff. Spec. Nozzle Struct. Water Jet Perform.*, pp. 1–19, 2022.
- [7] C. Z. Xia, L. Cheng, C. Luo, W. Jiao, and D. Zhang, "Hydraulic characteristics and measurement of rotating stall suppression in a waterjet propulsion system," *Transactions of Famen*, vol. 22, no. 2, pp. 85–100, 2018.
- [8] L. Jian, L. Xiwen, Z. Zuti, L. Xiaohui, and Z. Yuquan, "Numerical investigation into effects on momentum thrust by nozzle's geometric parameters in water jet propulsion system of autonomous underwater vehicles," *Ocean Eng.*, vol. 123, pp. 327–345, 2016, doi: 10.1016/j.oceaneng.2016.07.041.
- [9] C. Wang *et al.*, "Numerical Simulation on Hydraulic Characteristics of Nozzle in Waterjet Propulsion System," pp. 1–21, 2019.
- [10] B. Gu, R. Hu, L. Wang, and G. Xu, "Study on the influence rule of high-pressure water jet nozzle parameters on the effect of hydraulic slotting," *Geofluids*, vol. 2022, Article ID 4510194, 2022, doi: 10.1155/2022/4510194.
- [11] Y. Wang *et al.*, "Influence of Nozzle Geometry and Operating Parameters on High-Pressure Water Jets," 2025, doi: 10.32604/fdmp.2025.072236.
- [12] W. X. Jiao, L. Cheng, J. Xu, and C. Wang, "Numerical analysis of two-phase flow in the cavitation process of a waterjet propulsion pump system," *Processes*, vol. 7, no. 10, p. 690, 2019.
- [13] C. Wang, J. Liang, and H. Huo, "Numerical study of effects of geometric parameters on the flow and cavitation characteristics inside conical nozzle of autonomous underwater vehicles," vol. 11, no. 2, pp. 1–12, 2019, doi: 10.1177/1687814019828584.
- [14] F. Zhang and S. Wang, "Effect of Internal Structure of Nozzle on Impingement Heat Transfer Performance of Single-beam Water Jet," vol. 61, no. 3, pp. 888–894, 2021.
- [15] Z. Zhang, S. Cao, X. Luo, W. Shi, and Y. Zhu, "Research on the thrust of a high-pressure water jet propulsion system," *Ships Offshore Struct.*, vol. 13, no. 1, pp. 1–9, 2017, doi: 10.1080/17445302.2017.1321713.
- [16] Y. Ni, W. Liu, Z. Shen, and X. Pan, "Thrust Characteristics and Nozzle Role of Water Jet Propulsion," vol. 10, no. 1, pp. 47–53, 2017.
- [17] P. Joshi, T. Gandhi, and S. Parveen, "Critical Designing and

- Flow Analysis of Various Nozzles using CFD Analysis,” vol. 9, no. 02, pp. 421–424, 2020.
- [18] G. Hu, W. Chen, and Z. Gao, “Flow Analysis of Spray Jet and Direct Jet Nozzle for Fire Water Monitor,” vol. 141, pp. 913–916, 2010, doi: 10.4028/www.scientific.net/AMR.139-141.913.
- [19] D. Shinabuth, T. Nagasawa, S. Sato, and H. Kosaka, “Study on Effect of Nozzle Hole Length to Diameter Ratio on Near-Field Diesel Spray Characteristics at High Density Conditions,” vol. 11, no. 4, pp. 159–168, 2020.
- [20] Z. Shi, S. Cao, X. Luo, and X. Tang, “Reverse thrust characteristics of water jet with different geometry nozzles,” *Indian J. Eng. Mater. Sci.*, vol. 20, no. 6, pp. 476–482, 2013.
- [21] T. J. Tharakan and T. A. Rafeeqe, “The role of backpressure on discharge coefficient of sharp edged injection orifices,” *Aerosp. Sci. Technol.*, vol. 49, pp. 269–275, 2016, doi: 10.1016/j.ast.2015.12.014.
- [22] A. Eslamdoost, L. Larsson, and R. Bensow, “Net and gross thrust in waterjet propulsion,” *Journal of Ship Research*, vol. 60, no. 1, pp. 1–14, 2016.
- [23] T. Jiang, Z. Huang, J. Li, Y. Zhou, and C. Xiong, “Effect of Nozzle Geometry on the Flow Dynamics and Resistance Inside and Outside the Cone-Straight Nozzle,” 2022, doi: 10.1021/acsomega.1c07050.
- [24] S. H. I. Zhaocun and Y. Jian, “Research on the Modeling and Simulation of Reaction Thrust Characteristics of Water Jet *,” pp. 80–83, 2015.
- [25] X. Cui, K. Shen, and J. Sun, “Numerical analysis of nozzles’ energy loss based on Fluent,” in *Proceedings of the International Workshop on Mechanical Engineering and Computer Science (Iwmecs)*, 2015, pp. 756–760.