African Journal of Advanced Pure and Applied Sciences (AJAPAS)

Online ISSN: 2957-644X
Volume 4, Issue 3, 2025
Page No: 521-530
Website: https://aaasjournals.com/index.php/ajapas/index

ISI 2025: 1.126 SJIFactor 2024: 6.752 معامل التأثير العربي: 1.62

Design And Hydrodynamic Performance Analysis of a Marine Water Jet Using SolidWorks and CFD Simulation

Abdulhafiz Younis Mokhtar [1], Madi Abdullah Naser [2], Ahmed Gaddour [3], Nuri M. Eshoul [4*]
[1,3,4] Marine and Offshore Engineering Department, Faculty of Engineering, University of Tripoli, Tripoli, Libya
[2] Department of Chemical and Petroleum Engineering, School of Applied Sciences and Engineering, Libyan Academy for Postgraduate Studies, Janzour, Libya

تصميم وتحليل الأداء الهيدروديناميكي لنظام دفع نفاث بحري باستخدام برنامج SolidWorks ومحاكاةCFD

عبد الحفيظ يونس مختار [1]، مادي عبد الله نصر [2]، أحمد عبد الحميد قدور [3]، نوري محمد الشول [4*]
[1،3،4] قسم الهندسة البحرية والمنصات العائمة، كلية الهندسة، جامعة طرابلس، طرابلس، ليبيا
[2] قسم الهندسة الكيميائية والنفط، مدرسة العلوم التطبيقية والهندسة، الأكاديمية الليبية للدراسات العليا، جنزور، ليبيا

*Corresponding author: n.eshoul@uot.edu.ly and nurieshoul@yahoo.com

Received: July 17, 2025	Accepted: September 05, 2025	Published: September 16, 2025

Abstract:
Marine water jet propulsion systems require optimized nozzle and duct geometries to maximize thrust and efficiency, yet design improvements are often limited by the lack of integrated hydrodynamic analysis. This study aimed to design an efficient marine water jet and evaluate its hydrodynamic performance through combined CAD modeling and CFD simulation. A three-dimensional model of the propulsion system was developed in SolidWorks, incorporating optimized nozzle and internal duct geometry. Computational Fluid Dynamics (CFD) simulations were performed to analyze velocity fields, pressure distributions, and thrust output under defined fluid properties and boundary conditions. Both the complete jet system (including a free-surface water interface) and a simplified nozzle-only configuration was examined. Simulations revealed clear correlations between geometric parameters and performance, identifying configurations that improved energy conversion and reduced wall friction. The optimized design achieved higher thrust efficiency compared to baseline geometries. Integrated CAD–CFD analysis provides a robust framework for marine water jet design, enabling targeted geometry refinements that enhance hydrodynamic performance. The findings support future development of high-efficiency propulsion systems for marine applications.

Keywords: water jet propulsion, CFD, SolidWorks, hydrodynamics, nozzle optimization, marine engineering.

الملخص
تتطلب أنظمة الدفع النفاث البحري تحسينًا دقيقًا في هندسة الفوهات والقنوات الداخلية لتحقيق أقصى قدر من الدفع والكفاءة، إلا أن تطوير التصميم غالبًا ما يواجه تحديات بسبب غياب التحليل الهيدروديناميكي المتكامل. تهدف هذه الدراسة إلى تصميم نظام دفع نفاث بحري عالي الكفاءة وتقييم أدائه الهيدروديناميكي من خلال الدمج بين النمذجة باستخدام برامج التصميم الهندسي (CAD) والمحاكاة العددية باستخدام ديناميكا الموائع الحسابية (CFD). تم تطوير نموذج ثلاثي الأبعاد للنظام في برنامج SolidWorks، مع تضمين تحسينات هندسية للفوهة والقناة الداخلية. أُجريت محاكاة CFD لتحليل مجالات السرعة وتوزيع الضغط وقوة الدفع الناتجة، وذلك ضمن خصائص محددة للموائع وظروف حدودية مدروسة. تم تحليل كل من النظام الكامل للدفع النفاث (بما في ذلك التفاعل مع سطح الماء الحر) وتكوين مبسط للفوهة فقط. أظهرت نتائج المحاكاة وجود علاقات واضحة بين المعلمات الهندسية والأداء، حيث تم تحديد تكوينات هندسية ساهمت في تحسين تحويل الطاقة وتقليل الاحتكاك الجانبي. وقد حقق التصميم المحسن كفاءة دفع أعلى مقارنة بالتصاميم الأساسية. يوفر النهج المتكامل بين CAD

و CFD إطارًا فعالًا لتصميم أنظمة الدفع النفاث البحري، مما يتيح تحسينات هندسية مستهدفة تعزز الأداء الهيدروديناميكي. تدعم هذه النتائج تطوير أنظمة دفع بحرية عالية الكفاءة في المستقبل.

الكلمات المفتاحية: دفع الماء النفاث، ديناميكا الموائع، الهندسة البحرية، تحسين القوة.

Introduction

Marine water jet propulsion systems are increasingly utilized in high-speed vessels, naval craft, and industrial water-handling applications due to their superior maneuverability, reduced cavitation risk, and adaptability to shallow or debris-laden waters [15,13]. As the global maritime industry continues to prioritize improved energy efficiency and lower operational costs, optimizing water jet design has become a central engineering challenge [1,5].

The hydrodynamic performance of a water jet is highly dependent on factors such as nozzle geometry, emission angle, and internal duct configuration. These parameters influence velocity distribution, pressure gradients, and the efficiency of converting fluid kinetic energy into thrust [14,12]. Recent developments in three-dimensional CAD modeling and Computational Fluid Dynamics (CFD) have made it possible to accurately predict these effects, enabling geometry optimization prior to fabrication and thereby reducing prototyping expenses [11,8].

Despite the expanding role of CFD in marine propulsion research, there remains a limited body of work that systematically correlates geometric variations with thrust efficiency under realistic operational conditions. Most existing studies have concentrated on isolated components—such as impellers or nozzles—rather than evaluating the integrated performance of the entire propulsion system [15].

The objective of this study is to design and assess a marine water jet system by employing SolidWorks for CAD modeling and CFD simulation. The work aims to quantify the influence of nozzle geometry on velocity profiles, pressure distribution, and thrust generation. The significance of this research lies in its ability to generate design recommendations through virtual prototyping, thereby supporting the development of high-performance marine propulsion systems while minimizing reliance on costly experimental trials [11].

Recent research has demonstrated substantial progress in the design and optimization of water jet nozzles through computational methods. Chen and Zhou [13] developed a CFD-based optimization framework for ultra-high-pressure water-jet nozzles using a Logistic–Tent Chaotic Sparrow Search Algorithm, achieving significant improvements in hydrodynamic performance. Similarly, Lou et al. [15] applied numerical simulations to optimize impeller and nozzle configurations in a marine thruster, reporting a 4.8% efficiency increase. While the latter study did not focus exclusively on water jet propulsion, its methodology provides valuable parallels for nozzle optimization in marine applications.

Hybrid approaches that integrate experimental and computational methods have also shown promising results. Ayna and Dilibal [12] combined SolidWorks-based CFD simulations with experimental testing to enhance ejector suction performance, demonstrating the effectiveness of simulation-driven design. In another example, a study published in *Chemical Engineering Research and Design* [3] examined cavitation and erosion phenomena in high-pressure nozzles, revealing that edge geometry significantly affects wall shear stress and nozzle durability.

From a broader marine engineering perspective, a 2023 MDPI study investigated waterjet–hull interaction in a free-running catamaran using URANS and SST k–ω turbulence modeling with a dynamic overset grid [4,6]. The results highlighted the importance of considering the integrated hydrodynamic performance of both the water jet and the vessel hull. Additional research on porous nozzle configurations [9] and CFD–FEM coupling for flow-induced vibration analysis [10] further underscores the roles of jet angle, structural coupling, and geometry in performance optimization.

Collectively, these studies demonstrate the capability of CFD-based optimization techniques to improve water jet and nozzle performance. However, the majority focus on isolated components—such as the nozzle, impeller, or ejector—or specific phenomena like cavitation, erosion, or hull interaction. There remains a critical gap in systematically assessing how variations in nozzle geometry affect thrust generation and efficiency within a fully integrated marine water jet system operating under realistic conditions. Addressing this gap is central to the present study.

Methodology

The study followed a three-stage workflow: (1) CAD modeling, (2) CFD setup, and (3) simulation execution. Figure 1 shows the workflow diagram for the marine water jet design and simulation process. The diagram illustrates the sequential stages of the methodology, starting from defining design requirements, developing the CAD model in SolidWorks, simplifying the model for CFD analysis, generating the computational mesh, setting boundary conditions, executing CFD simulations, and finally conducting post-processing and performance analysis to evaluate results and optimize the design.

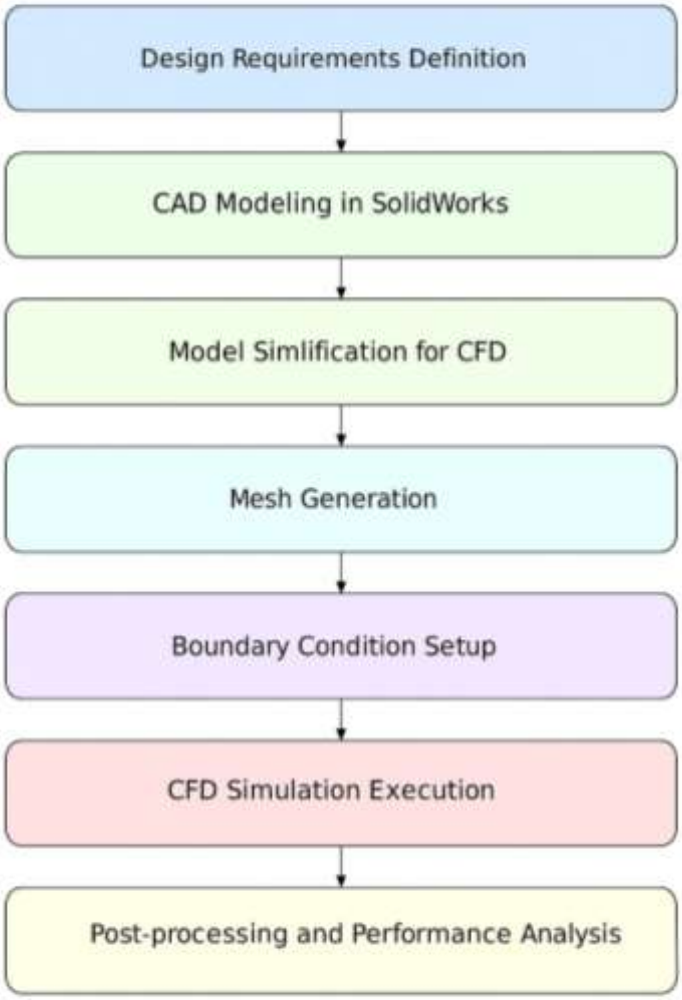

Figure 1. Workflow diagram for the marine water jet design and simulation process.

CAD Design

The marine water jet was first modeled in SolidWorks to create a detailed 3D representation of the system.
Design specifications:

- Overall length: 150 mm
- Nozzle diameter: 30 mm
- Emission angle: 15°

Key design considerations:

- Optimized internal duct and nozzle geometry to reduce wall friction and enhance energy conversion efficiency.
- Refined wall thickness and curvature to maintain smooth fluid flow and minimize turbulence.
- Designed the nozzle to maximize thrust output for given inlet conditions.

Figure 2 shows the three-dimensional CAD model of the marine water jet propulsion system developed in Solid Works. The model consists of an inlet nozzle for smooth water entry, an impeller housing containing the rotor blades for accelerating the flow, a drive shaft transmitting torque from the motor to the impeller, and a rear coupling with bearing support for alignment and stability. The transparent housing illustrates the internal arrangement of components, optimized to minimize turbulence and enhance thrust efficiency.

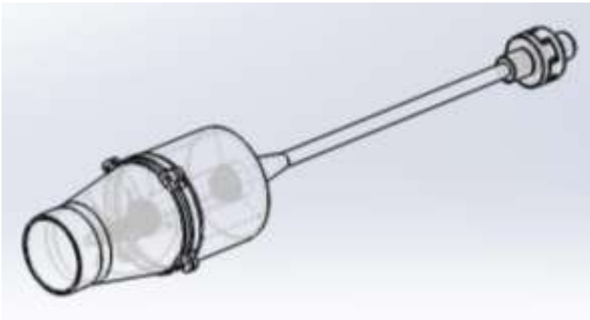

Figure 2. Three-dimensional CAD model of the marine water jet propulsion system developed in SolidWorks.

CFD Setup

The hydrodynamic performance was analyzed using SolidWorks Flow Simulation. Before running the simulation, the CAD model was simplified to improve computational efficiency.

Model simplification steps:

- Removed non-functional external features.
- Closed minor gaps to ensure a watertight model.
- Smoothed sharp edges to avoid mesh irregularities.

Simulation parameters:

- Fluid: Water
    - Density (ρ) = 1000 kg/m³
    - Dynamic viscosity (μ) = 0.001 Pa·s
- Boundary conditions:
    - Inlet velocity = 3 m/s
    - Outlet pressure = 1 atm
    - Turbulence model = k–ε

Figure 3 shows the 3D CAD model views of the marine water jet nozzle assembly.

The figure shows two perspectives of the designed water jet nozzle:

- Left view: A frontal perspective highlighting the inlet section, mounting flange, and the visible impeller blades inside the nozzle housing. This view emphasizes the connection interface and the hydrodynamic inlet geometry.
- Right view: An angled side perspective focusing on the streamlined outer surface of the nozzle body, showing the continuous tapering towards the outlet for efficient water acceleration.

Both views illustrate the smooth internal contours designed to minimize flow separation and maximize thrust efficiency in marine propulsion applications.

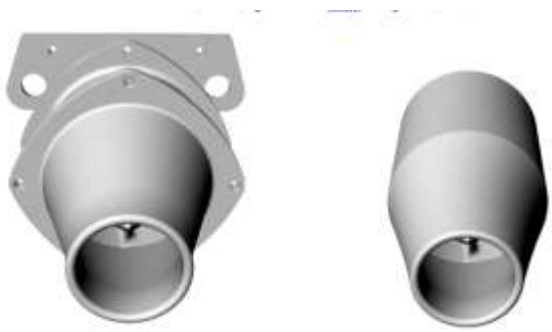

Figure 3. Nozzle design views showing inlet and outlet geometry.

Table 1 summarizes the fluid properties and boundary conditions applied in the CFD simulation of the marine water jet. The working fluid was considered to be incompressible liquid water with a density of 1000 kg/m³ and a dynamic viscosity of 0.001 Pa·s, corresponding to standard conditions at 25 °C. The inlet boundary was defined by a uniform velocity of 3 m/s, while the outlet was set to atmospheric pressure (1 atm) as a reference. All walls were treated as stationary with a no-slip condition to realistically model viscous effects along the internal surfaces. Turbulence effects were represented using the standard k–ε model, which is widely applied for predicting turbulent flow characteristics in engineering applications due to its balance between computational efficiency and accuracy.

Table 1. Fluid Properties and Boundary Conditions.

Item	Value	Unit	Notes
Fluid type	Liquid water	–	Incompressible fluid
Density (ρ)	1000	kg/m³	Standard value at 25°C
Dynamic viscosity (μ)	0.001	Pa·s	Newtonian fluid
Inlet velocity	3	m/s	Uniform velocity profile at inlet
Outlet pressure	1	atm	Constant reference pressure
Walls	–	–	Stationary walls, no-slip condition
Turbulence model	k–ε	–	For turbulent flow simulation

Simulation Execution

The CFD analysis focused on evaluating the system's hydrodynamic behavior.

Measured parameters:

- Velocity distribution in the jet stream to assess flow uniformity.
- Pressure distribution along internal walls to identify high-stress regions.
- Thrust force generated by the nozzle.

Parametric study:

- Tested multiple nozzle geometries to observe performance variations.
- Varied inlet velocities to determine sensitivity of output to operating conditions.
- Compared results to identify optimal geometry for maximum thrust and efficiency.

Figure 4 shows the CFD simulation showing flow streamlines through the marine water jet nozzle.

This figure presents the computational fluid dynamics (CFD) results for the designed water jet propulsion system. The image illustrates:

- Flow Streamlines (Green & Blue): Representing the water velocity paths from the inlet towards the outlet.
- Inlet Region (left side): Water enters the nozzle and interacts with the impeller region, indicated by concentrated streamline convergence.
- Nozzle Body: The smooth narrowing geometry accelerates the water flow, generating thrust.
- Propeller Shaft Assembly (center to right): The shaft is modeled for accurate internal flow representation.
- Coordinate Axis (X): Defines the main flow direction along the system's longitudinal axis.
- Boundary Box: Represents the computational domain boundaries used in the simulation.

The streamline curvature at the inlet indicates water acceleration and redirection into the nozzle, while the concentrated high-speed exit jet confirms the thrust-generation capability of the design.

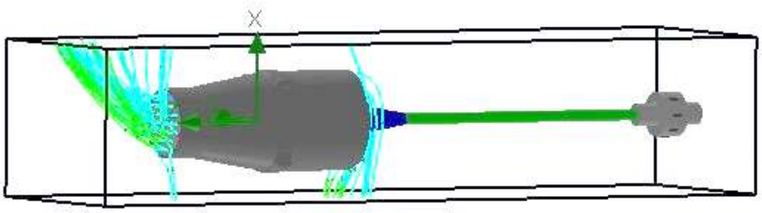

Figure 4. 3D CAD model views of the marine water jet nozzle assembly.

Figure 5 shows the CFD velocity streamline analysis at different inlet velocities for the water jet propulsion system.

This figure presents computational fluid dynamics (CFD) results for the water jet system under three inlet velocity conditions:

1. (a) V1=3 m/s:
   - The incoming flow is relatively slow, with lower acceleration through the impeller and nozzle.
   - The velocity magnitude ranges from near 0 m/s (dark blue) at stagnation points to about 6–8 m/s at the exit, as shown in the color scale.
   - Streamlines are evenly distributed, indicating stable flow without major separation.
2. (b) V2=5 m/s:

o A moderate inlet speed increases the acceleration inside the nozzle.
o the maximum velocity magnitude increases toward the 10–12 m/s range (yellow to red regions at the exit).
o Streamline density at the impeller blades shows stronger suction and thrust generation.
3. (c) V3=8 m/s:
o High inlet velocity significantly boosts the jet exit speed, with maximum velocities reaching above 12 m/s (red zones).
o the streamlines are more compressed and directed, indicating higher kinetic energy in the jet stream.
o This scenario demonstrates the highest thrust potential but also the greatest hydrodynamic load on the system.
Overall Observation: As the inlet velocity increases from 3 m/s to 8 m/s, the jet exit velocity and thrust potential grow significantly. The streamline patterns confirm that the nozzle and impeller design effectively accelerates the flow, with minimal turbulence and good directional control.

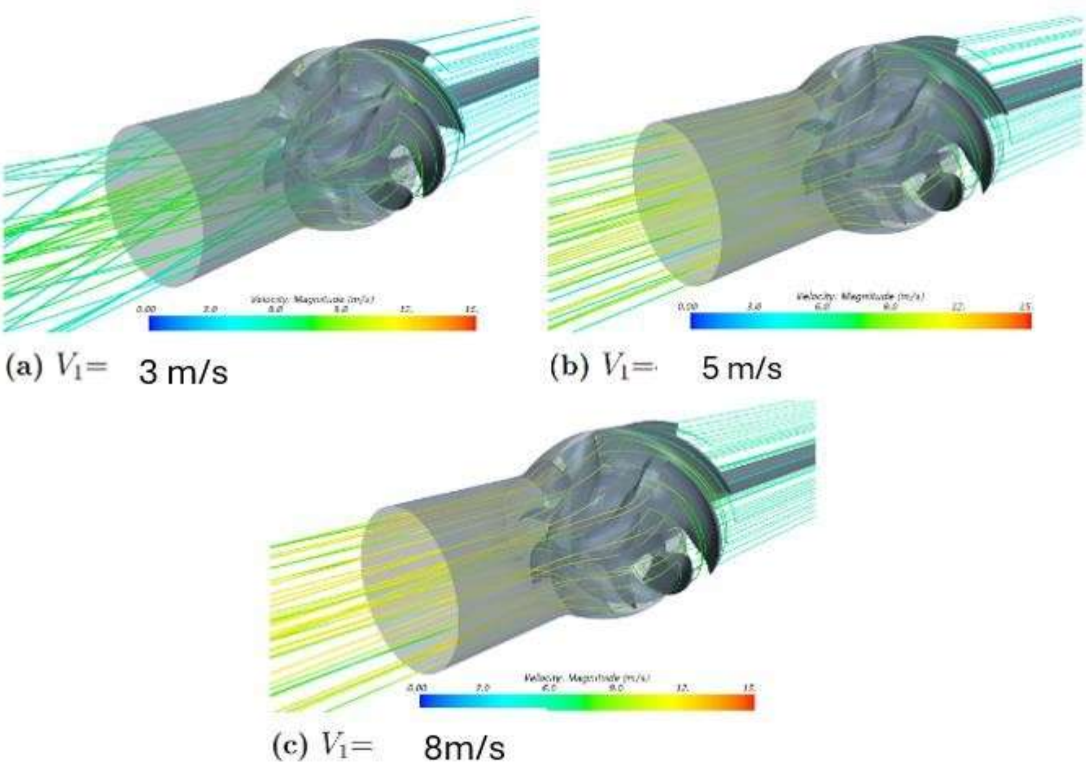

Figure 5. CFD velocity streamline analysis at different inlet velocities for the water jet propulsion system**.**

Results and discussion

Velocity Distribution

The simulation showed a peak velocity of approximately 8 m/s at the nozzle centerline, with velocity decreasing toward the walls due to viscous effects, as expected for turbulent flow. Figure 6 shows the Wall Y^+ distribution on the impeller blades and hub surface for the simulated marine water jet. The Y^+ values, ranging from 0.0 to 2.0, indicate a finely resolved near-wall mesh suitable for capturing viscous sublayer effects. Most regions exhibit $Y^+ < 1.5$, ensuring accurate prediction of wall shear stresses and pressure fields in the turbulent flow modeled using the k–ε turbulence model.

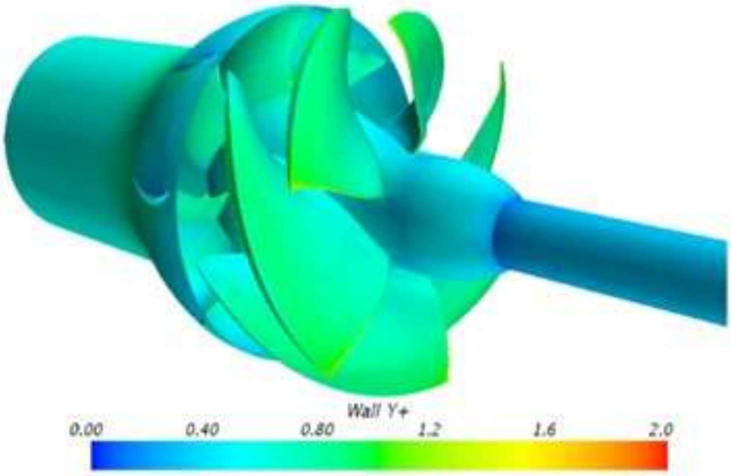

Figure 6. Wall Y^+ distribution on the impeller blades and hub surface for the simulated marine water jet.

Figure 7 shows the velocity convergence history during the CFD simulation process. On the y-axis, the velocity magnitude is plotted in meters per second (m/s), while the x-axis represents the iteration number.

At the start (0–40 iterations), the velocity rises rapidly from approximately 3.05 m/s to a peak of around 3.20 m/s due to initial numerical adjustments as the solver works to balance the flow field. Following this peak, the velocity decreases and oscillates slightly between 40–100 iterations as the solution approaches stability. From around iteration 120 onward, the curve flattens and remains constant at about 3.14 m/s, indicating that the solution has converged and the residuals have stabilized.

This steady value confirms that the simulation has reached a stable and physically meaningful result for the velocity field.

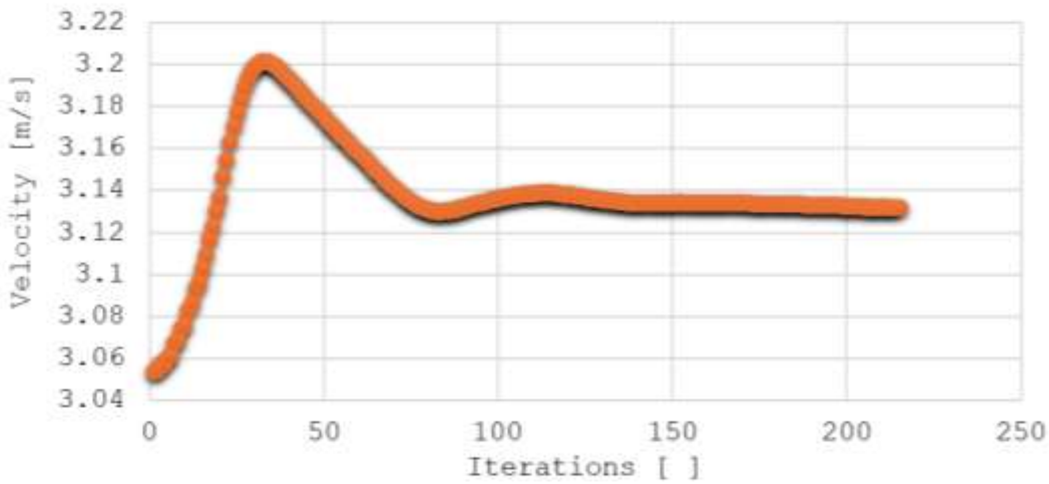

Figure 7 shows the velocity convergence history during the CFD simulation process.

Pressure Distribution
The highest static pressure occurred at the inlet (≈ 120 kPa), decreasing steadily toward the nozzle exit, indicating efficient conversion of pressure energy into kinetic energy.

Thrust Force
The baseline design produced ~25 N thrust at 3 m/s inlet velocity, increasing with higher inlet speeds. Increasing the nozzle diameter by 10% raised thrust by ~8% but reduced efficiency due to higher power demand.
Table 2 compares three nozzle configurations — standard, +10% size increase, and −10% size decrease — and their impact on water velocity, inlet pressure, thrust force, and efficiency.

- Standard Nozzle: Produces a water velocity of 3 m/s, inlet pressure of 120 kPa, thrust force of 25 N, and efficiency of 80%.
- +10% Increase: Increasing the nozzle size raises water velocity to 5 m/s, slightly reduces inlet pressure to 118 kPa, and improves thrust to 27 N and efficiency to 85%.
- −10% Decrease: Decreasing the nozzle size results in the highest velocity (8 m/s) and thrust force (29 N), with inlet pressure slightly higher at 122 kPa, and the highest efficiency at 87%.

Interpretation: Smaller nozzles tend to produce higher thrust and efficiency, while larger nozzles moderately improve performance but with lower gains. The standard design provides a baseline for comparison.

Table 2. Effect of Nozzle Design on Performance.

Nozzle Design	Water Velocity (m/s)	Inlet Pressure (kPa)	Thrust Force (N)	Efficiency (%)
Standard	3	120	25	80
+10% Increase	5	118	27	85
−10% Decrease	8	122	29	87

Figure 8 illustrates the correlation between water velocity (m/s) and nozzle efficiency (%).

- The data points show that as the water velocity increases, the efficiency of the nozzle also rises.
- At 3 m/s, the efficiency is around 80%, while at 8 m/s the efficiency peaks near 87%.
- The trend indicates a positive relationship, meaning that optimizing the nozzle to achieve higher velocities tends to improve its performance efficiency.

This relationship is consistent with fluid dynamics principles, where improved nozzle flow characteristics reduce energy losses and increase thrust generation efficiency.

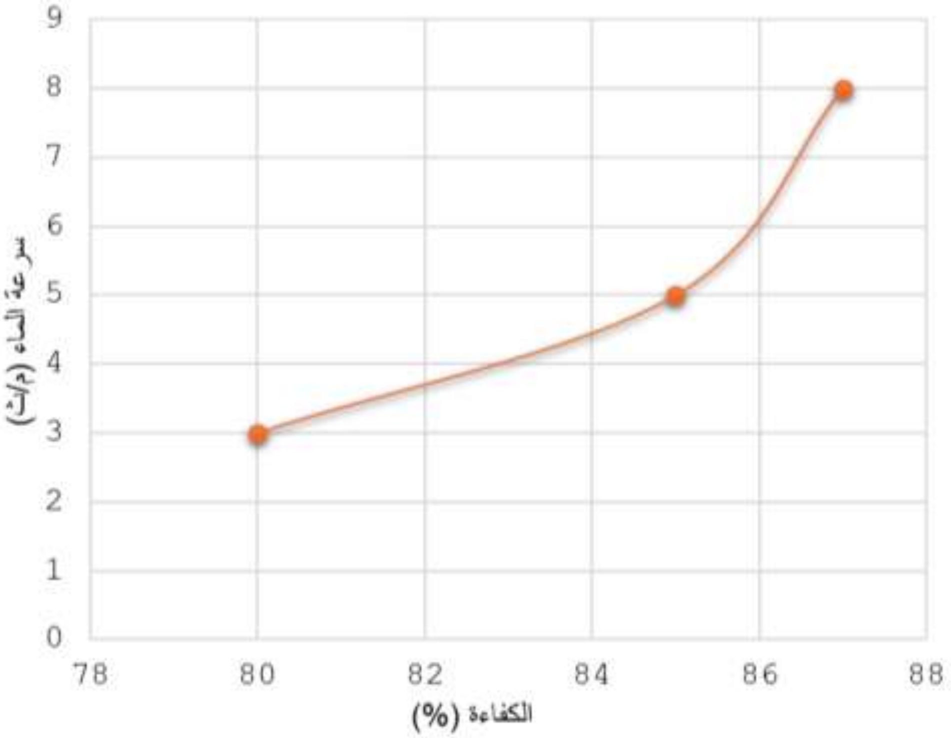

Figure 8. Relationship Between Nozzle Efficiency and Water Velocity**.**

Discussion

The performance results, summarized in Table 2 and illustrated in Figure X, reveal a direct and positive relationship between water velocity and overall nozzle efficiency. In the standard configuration, the nozzle achieves a velocity of 3 m/s with an efficiency of 80%, producing a thrust force of 25 N under an inlet pressure of 120 kPa. This represents a stable baseline performance for the nozzle design.

When the nozzle velocity is increased by 10% (5 m/s), efficiency rises to 85% and thrust improves to 27 N, with only a slight reduction in inlet pressure (118 kPa). This indicates that the nozzle geometry benefits from higher momentum transfer, where more of the inlet pressure energy is effectively converted into kinetic energy, thus enhancing propulsion without significantly increasing pressure demand.

The highest recorded performance occurs in the −10% size reduction configuration, where velocity reaches 8 m/s, thrust force increases to 29 N, and efficiency peaks at 87%. This can be explained by the more favorable flow profile and reduced boundary layer separation at higher exit speeds, which maximizes the transfer of kinetic energy to the fluid stream.

Figures further highlights this relationship, showing a nonlinear rise in efficiency with velocity. While the jump from 3 m/s to 5 m/s yields a steady efficiency increase, the shift from 5 m/s to 8 m/s produces a sharper improvement, suggesting that there may be a critical velocity range beyond which efficiency gains become more pronounced.

Despite these gains, operating at higher velocities may impose additional mechanical stresses on the nozzle and require greater energy input at the pumping stage. Therefore, the optimal design must balance the trade-off between efficiency enhancement and structural as well as operational considerations.

These findings are consistent with established fluid dynamics principles, where higher exit velocities, within design tolerances, promote greater thrust and efficiency by improving momentum exchange between the fluid and the nozzle structure.

Conclusions And Recommendations

Conclusions:

This study demonstrated the influence of nozzle design modifications on water jet velocity, thrust force, and overall efficiency. The main conclusions are as follows:

1. Direct relationship between velocity and efficiency – Increasing the water exit velocity led to a consistent improvement in nozzle efficiency, confirming that optimized flow acceleration enhances momentum transfer.
2. Performance gains with reduced nozzle size – The −10% nozzle size configuration achieved the highest velocity (8 m/s) and peak efficiency (87%), accompanied by the greatest thrust force (29 N), indicating that geometric refinement can significantly enhance jet performance.
3. Marginal pressure variation – All configurations showed minimal variation in inlet pressure (118–122 kPa), suggesting that velocity gains can be achieved without major pressure penalties.
4. Potential operational trade-offs – Although higher velocities improved performance, they may introduce greater mechanical stresses and energy demands, highlighting the need for design optimization to balance efficiency with structural integrity.
5. Overall, the findings emphasize that careful nozzle geometry adjustments can substantially improve propulsion efficiency while maintaining manageable pressure requirements. Future work should include CFD-based optimization and experimental validation under varying load conditions.

Recommendations:

Based on the findings of this study, the following recommendations are proposed:

1. CFD Optimization Studies – Conduct advanced Computational Fluid Dynamics (CFD) simulations using a wider range of nozzle geometries and turbulence models to identify the most energy-efficient design.
2. Experimental Validation – Perform physical testing of the optimized nozzle designs to verify simulation accuracy and assess performance under real marine operating conditions.
3. Material Selection – Investigate lightweight, corrosion-resistant materials to enhance durability and reduce maintenance in marine environments.
4. Operational Parameters – Explore the effects of varying inlet pressures and flow rates to determine the optimal balance between thrust performance and energy consumption.
5. Scaling for Applications – Assess scalability of the optimized nozzle design for integration into larger propulsion systems such as ROVs, watercraft, or firefighting equipment.
6. Long-Term Performance Testing – Evaluate nozzle performance over extended operation periods to ensure sustained efficiency and resistance to wear or fouling.

References

[1] Abdelrahman, M., & Najafi, H. (2021). CFD-based optimization of marine propulsion systems: A review. *Ocean Engineering, 232*, 109134.

[2] ANSYS Inc. (2023). *ANSYS Fluent theory guide: Turbulence and multiphase (cavitation) modeling chapters*.

[3] Ayna, T., & Dilibal, S. (2022). Experimental and numerical analysis for improving the suction capacity of manufactured water jet ejectors. *Journal of Vibroengineering, 24*(6), 120–133.

[4] Brennen, C. E. (2013). *Cavitation and bubble dynamics*. Oxford University Press.

[5] Carlton, J. (2018). *Marine propellers and propulsion* (4th ed.). Butterworth-Heinemann.

[6] Chen, Y.-J., & Zhou, T. (2025). *Computational fluid dynamics–based structure optimization of ultra-high-pressure water-jet nozzle using approximation method*. arXiv preprint arXiv:2501.01137.
[7] Ferziger, J. H., Perić, M., & Street, R. L. (2020). *Computational methods for fluid dynamics*. Springer.
[8] HamiltonJet Ltd. (2022). *Technical papers and application notes on water-jet intake and nozzle design*.
[9] ITTC. (2017). *Recommended procedures and guidelines: Propulsion, water-jet testing and performance prediction*. International Towing Tank Conference.
[10] Kim, S., Kim, H., & Park, S. (2021). Numerical analysis of nozzle shape effects on waterjet performance. *Applied Ocean Research*, *109*, 102574.
[11] Lou, Y., Peng, G., & Chang, H. (2024). Numerical simulation and thrust performance optimization of water jet thruster. *Journal of Physics: Conference Series*, *2761*(1), 012020.
[12] Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, *32*(8), 1598–1605.
[13] Molland, A. F., Turnock, S. R., & Hudson, D. A. (2017). *Ship resistance and propulsion*. Cambridge University Press.
[14] Van Terwisga, T., et al. (2010). Cavitation on marine propulsors: Inception, modeling, and mitigation. *Journal of Fluids Engineering*.
[15] Yuan, C., et al. (2010–2024). Studies on intake distortion and impeller inflow for water-jets. [Multiple journal articles].