International Journal of Innovation in
Mechanical Engineering and Advanced Materials
Vol. 6 (No. 1), 2024, pp. 38-45
Journal homepage: publikasi.mercubuana.ac.id/index.php/ijimeam
DOI: 10.22441/ijimeam.v6i1.27652

Design and Analysis of a Vertical Axis Ocean Current Turbine Tunnel Using SolidWorks Computational Fluid Dynamics

Hardi Gunawan [1,2,]*, Nanang Ruhyat [1] and Sentot Novianto [3]

[1]Department of Mechanical Engineering, Universitas Mercu Buana, Meruya Selatan, Jakarta 11650, Indonesia
[2]Department of Welding Engineering, SMKN 53 Jakarta, Cengkareng, Jakarta 11730, Indonesia
[3]Department of Mechanical Engineering, Universitas Trisakti, Petamburan, Jakarta 11440, Indonesia
*Corresponding Authors: hardi97gunawan@gmail.com (HG)

Abstract

The development of renewable energy in the marine power generation sector presents a promising approach to producing electrical energy in a sustainable and environmentally friendly manner. Indonesia, with its vast oceanic territory, holds significant potential for harnessing marine energy. However, the relatively slow speed of ocean currents in the region, typically ranging from 0.1 m/s to 1.5 m/s, poses a challenge to the efficiency of marine power generation. To overcome this limitation, this research focuses on the design and analysis of a vertical-axis ocean current turbine tunnel aimed at increasing the speed of ocean currents, thereby enhancing the overall efficiency of energy production. The study combines a thorough literature review with experimental research methods, utilizing SolidWorks Computational Fluid Dynamics (CFD) software to simulate the tunnel's impact on ocean current velocity. The simulations reveal that the tunnel construction significantly boosts current speeds, increasing them from 1.0 m/s to 1.7 m/s, and from 1.5 m/s to 2.6 m/s. This increase in velocity directly translates to higher kinetic energy available for conversion into electrical power by the turbine. Moreover, the study shows that the tunnel construction contributes to a more uniform flow of ocean currents, as evidenced by the Reynolds numbers obtained—100.250 at a current speed of 1.0 m/s and 150.375 at 1.5 m/s. These values, being below 2000, indicate laminar flow conditions within the tunnel, which are beneficial for optimizing turbine performance by reducing turbulence and ensuring a stable energy output. The findings underscore the effectiveness of the tunnel design in improving the efficiency of vertical-axis ocean current turbines, making it a viable solution for enhancing renewable energy production in regions with low ocean current speeds.

Article Info:
Received: 11 June 2024
Revised: 7 July 2024
Accepted: 13 July 2024
Available online: 31 August 2024

Keywords:
Vertical-axis ocean current turbine; renewable energy; tunnel construction; CFD; laminar flow

1. Introduction

Energy availability is a critical issue that plays a central role in achieving economic, ecological, and social development goals, all of which are integral to efforts such as the Renewable Energy Based on Industrial Development (RE-BID) initiative. Renewable energy sources, particularly marine energy, present a promising solution to the growing energy demands of the modern world [1]. Indonesia, with its vast oceanic territory covering two-thirds of the nation's area, is particularly well-positioned to harness marine energy. The country's unique geographical configuration, characterized by numerous straits, provides substantial potential for generating electricity from ocean currents, wherein the kinetic energy of flowing water is converted into electrical power through turbines. This potential is vital given the increasing energy needs driven by rapid population growth and the gradual depletion of fossil fuel resources [2].

However, the development of Marine Current Power Plants (MCPPs) in Indonesia faces several challenges, particularly due to the country's limited capacity for marine energy technology [3]. The ocean currents in most Indonesian waters are relatively slow, generally less than 1.5 m/s, with a few exceptions where the current speed can reach up to 5 m/s in certain straits [4]. This poses a significant challenge for the efficient operation of marine turbines, which typically require higher flow velocities for optimal performance. While there has been considerable research on marine energy technologies, a significant gap exists in applying these technologies to the specific conditions of

How to cite:

H. Gunawan, and N. Ruhyat, "Design and analysis of a vertical axis ocean current turbine tunnel using SolidWorks computational fluid dynamics," *Int. J. Innov. Mech. Eng. Adv. Mater*, vol. 6, no. 1, pp. 38-45, 2024

Indonesian waters. Much of the existing research has been conducted in regions with stronger and more consistent ocean currents, which are not representative of the majority of Indonesia's marine environments. Additionally, most studies have focused on horizontal-axis turbines, which may not be the most effective design for areas with lower current speeds and fluctuating flow conditions [5].

A critical gap also lies in the design and implementation of flow augmentation technologies, such as tunnels or diffusers, which can significantly enhance the performance of turbines in low-speed current environments [6]. While augmented diffusers have been shown to improve turbine efficiency in some cases, there is a lack of comprehensive studies exploring their application to vertical-axis turbines, especially those designed for the specific conditions of Indonesian waters. Advances in Computational Fluid Dynamics (CFD) have allowed for more accurate simulations of fluid flow in complex environments, which is essential for optimizing turbine designs [7]. Research has demonstrated that factors such as turbulence intensity and non-uniform inflow profiles can greatly impact the performance of ocean current turbines [8]. Studies using CFD have investigated various design modifications, including the addition of flanged channels and augmented diffusers, to improve turbine efficiency [9]. However, most of these studies are focused on environments with higher current speeds or use simplified models that do not fully account for the complexities of Indonesian waters.

Turbines equipped with flanged channels have been shown to significantly enhance power output in tidal flow applications [10]. Additionally, the use of augmented diffusers has proven to be more efficient than using turbines without such enhancements [11]. This evidence highlights the importance of incorporating tunnel constructions that can increase the speed of ocean currents, thereby boosting turbine performance. However, there remains a notable gap in research specifically focused on the design of tunnel constructions for vertical-axis ocean current turbines that are tailored to the unique geographical and oceanic conditions of Indonesia.

To address these gaps, the present study aims to design and simulate a vertical-axis ocean current turbine with a tunnel construction specifically tailored to the conditions found in Indonesian marine environments. The tunnel is intended to increase the speed of ocean currents as they enter the turbine, thereby enhancing turbine performance. Through CFD simulations, this research seeks to provide a deeper understanding of the potential benefits of tunnel-augmented vertical-axis turbines in low-speed marine environments. This study represents a crucial step toward developing more efficient and sustainable marine current power plants in Indonesia.

2. Methods

The research methodology for this study was divided into two main components: the literature review and the simulation-based experimental research.

The literature review serves as the foundational stage of the research, where relevant materials are gathered and analyzed to provide theoretical support for the study. This stage involves a thorough review of existing research, focusing on previous studies related to marine current turbines, tunnel constructions, and flow augmentation techniques. The goal is to identify gaps in the current knowledge, particularly those related to the application of these technologies in Indonesian waters, and to establish a solid theoretical framework that guides the subsequent experimental work.

The experimental component of the research is carried out using simulation methods, specifically through the application of SolidWorks 2018 software for Computational Fluid Dynamics (CFD) analysis. This stage involves the design and testing of a vertical-axis ocean current turbine tunnel. The process begins with the creation of a detailed construction design of the tunnel using SolidWorks software. The design phase is followed by a series of simulations that replicate the conditions of Indonesian sea currents. These simulations are carefully controlled to test the impact of the tunnel on the speed of ocean currents as they enter the turbine, with the primary objective of enhancing the turbine's performance.

To ensure the replicability of the study, all materials and tools used in the research are clearly identified. The key equipment includes the SolidWorks CFD application, version 2018, which is used to model and simulate the fluid dynamics of the tunnel and turbine system. The simulations are based on real data from Indonesian sea currents, ensuring that the results are relevant and applicable to the specific geographical context of the study.

The research design is methodically structured, beginning with the construction of the vertical-axis ocean current turbine tunnel model, followed by the execution of simulations using SolidWorks Simulation. Each step is carefully documented, and the design parameters, including tunnel dimensions and flow conditions, are systematically adjusted to observe their effects on the speed of ocean currents and the resulting turbine performance. The results of these simulations provide insights into the potential benefits of using tunnel constructions in vertical-axis ocean current turbines, particularly in regions with lower current speeds like those found in Indonesia.

The design of the tunnel construction can be seen in figure 1.

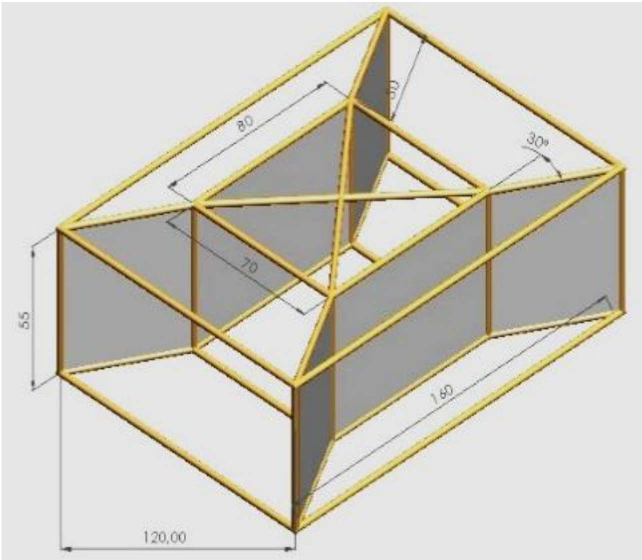

Figure 1. Tunnel

Meanwhile, the dimensions of the tunnel construction design are shown in Table 1.

Table 1. Tunnel Dimensions

Specifications	Size
Tunnel length	160 cm
Width in tunnel	70 cm
Tunnel Outer Width	120 cm
Tunnel height	50 cm
Length tunnel flow chamber	80 cm
Tunnel opening angle	30º

2.3. Calculation

To calculate the Reynolds number, it is essential to know the fluid's density, the velocity of the current, the length of the tunnel, and the dynamic viscosity. These factors significantly influence the flow characteristics within the tunnel, making the Reynolds number a crucial parameter in this context. The calculation is performed using the following equation [12].

$$Re = \frac{\rho v p}{\mu} \tag{1}$$

Where:

ρ = density (Kg/m^3)
v = velocity (m/s)
p = long (m)
μ = viscosity dynamics (Kg/ms)

Fluid flow discharge is a key parameter used to calculate the velocity of fluid flow. The discharge, or flow rate, can be determined using the following equation:

$$Q = A.V \tag{2}$$

Where:

Q = discharge flow (m^3/s)
A = cross-sectional area (m^2)

V = velocity (m/s)

In this study, the continuity equation was applied to determine the flow rate of fluid within the tunnel designed by the author. According to this principle, for a solid, incompressible fluid, the discharge (flow rate) remains constant at every point along the tunnel. This implies that the flow rate of the fluid is inversely proportional to the cross-sectional area of the tunnel. In simpler terms, as the cross-sectional area through which the fluid flows increase, the velocity of the fluid decreases; conversely, as the cross-sectional area decreases, the velocity of the fluid increases. This relationship is mathematically represented by the continuity equation:

$$A_1V_1 = A_2V_2 \tag{3}$$

Where:

A_1 and A_2 are the cross-sectional areas at two different points along the tunnel (m^2).
V_1 and V_2 are the fluid velocities at those respective points (m/s).

2.4. Tunnel simulation

The simulation process in this study utilizes SolidWorks Flow Simulation, a Computational Fluid Dynamics (CFD) tool, which is employed to predict fluid flow patterns, heat transfer, chemical reactions, and other related phenomena by solving complex mathematical equations and models. CFD provides a powerful method for analyzing the behavior of fluid flows within the tunnel and its impact on the performance of the ocean current turbine. The calculations involved in this simulation process generally rely on fundamental equations governing energy, momentum, and continuity [13].

The design simulation specifically incorporates current velocity data reflective of actual ocean currents, as presented in Table 2. This data is crucial for ensuring that the simulation accurately represents real-world conditions, allowing for a more precise analysis of how the tunnel construction affects fluid dynamics and turbine efficiency.

Table 2. Ocean current variation input

Ocean Current	Units
1.0	m/s
1.5	m/s

In the tunnel construction simulation, the primary parameters under investigation include the current speed and the continuity of fluid flow within the tunnel. The objective is to evaluate whether the tunnel construction can effectively increase the velocity of the ocean current, thereby enhancing the turbine's efficiency.
The simulation process is divided into four key stages: Wizard, Setup, Solver, and Result.

1. **Wizard:** In this initial stage, the relevant condition data is entered into the simulation software to ensure that the virtual experiment closely mirrors real-world conditions. This setup is crucial for achieving accurate and reliable results.
2. **Setup:** This stage involves configuring the specific parameters of the simulation, such as defining the computational domain, setting up boundary conditions, and specifying the regions of interest within the tunnel. Careful setup is essential to accurately model the physical environment and the fluid dynamics within the tunnel.
3. **Solver:** During this stage, the simulation software processes the input data using numerical methods to solve the equations governing fluid flow, energy, and momentum. This computational step is where the software calculates the effects of the tunnel construction on the current velocity.
4. **Result:** Finally, the simulation results are analyzed and interpreted. This stage includes visualizing the flow patterns, current velocity distributions, and other key outcomes that indicate whether the tunnel construction has successfully increased the current speed, thus potentially improving turbine efficiency.

During the setup stage, various settings are configured to address the specific problems under analysis. The simulation focuses on fluid flow at two different speed variations: 1.0 m/s and 1.5 m/s. This stage involves crucial steps, including defining the computational domains, configuring rotating regions, setting boundary conditions, and establishing the goals to be achieved. These configurations ensure that the simulation accurately reflects real-world conditions and provides meaningful data for analysis.

In the subsequent solver stage, the SolidWorks Flow Simulation application performs numerical calculations. At this point, the simulation applies key formulas, including the Reynolds number calculation, fluid flow discharge, and the continuity equation. Once these inputs are set, the simulation process is initiated by running the tool.

The final stage is the result stage, where the outcomes of the numerical calculations are displayed. This stage involves visualizing the results through various tools such as Cut Plot, Flow Trajectories, Surface Plot, and Goals. These visualizations allow for a detailed analysis of the simulation results. Additionally, the desired data can be easily exported to MS Excel for further examination and reporting.

3. Results and Discussion

This study focuses on analyzing the results of the simulation for the tunnel construction of an ocean current power plant, tailored to the specific characteristics of sea currents in Indonesia. The simulation, conducted using SolidWorks Computational Fluid Dynamics (CFD), aims to evaluate the effectiveness of the tunnel design. The primary objective is to determine the performance parameters of the tunnel construction, specifically its ability to enhance current velocity, and to assess the results derived from the associated calculations [14].

The calculation of the Reynolds number, a critical parameter for understanding fluid flow behavior, was also included in the analysis. The outcomes of these calculations are presented in Table 3, offering valuable insights into the flow dynamics within the tunnel.

Table 3. Reynolds number calculation

Velocity	Reynolds Number
1.0 m/s	100.250
1.5 m/s	150.375

In Table 3, the Reynolds number is calculated to be 100.250 at a current velocity of 1.0 m/s, and 150.375 at a current velocity of 1.5 m/s. These values indicate that the flow within the tunnel is laminar, as the Reynolds number is below the critical threshold of 2000.

Further calculations using the fluid flow discharge formula were performed to determine the discharge value of the ocean current entering the tunnel of the vertical-axis ocean current turbine. The results of these calculations are presented in Table 4, providing additional insights into the flow characteristics within the tunnel.

Table 4. Fluid flow discharge calculation

Velocity	Debit
1.0 m/s	0.63 m^3/s
1.5 m/s	0.91 m^3/s

While the results of the calculation in finding the value of the continuity equation, the calculation results can be seen in table 5.

Table 5. Continuity calculation

Velocity	Continuity
1.0 m/s	1.71 m/s
1.5 m/s	2.57 m/s

The continuity equation demonstrates that the speed of the incoming current has increased, with the initial velocity of 1.0 m/s rising to 1.71 m/s, and at an initial speed of 1.5 m/s, it increases to 2.57 m/s. This indicates that the tunnel design for the vertical-axis ocean current turbine effectively boosts the velocity of the sea current.

In the context of a vertical-axis ocean current turbine, this increase in current speed is crucial. The kinetic energy of the fluid flow, which is converted into mechanical energy by the turbine, is directly proportional to the velocity of the fluid. Therefore, an increase in flow velocity leads to a corresponding increase in kinetic energy, enabling the turbine to generate more power. This enhancement in current speed is essential for maximizing the efficiency and output of the turbine.

3.1. Discussion experimental result tunnel

In a previous study by Matheus Nunes et al. (2020), a systematic review of augmented diffusers applied to horizontal-axis ocean current turbines demonstrated that turbines equipped with augmented diffusers achieved significantly higher efficiency compared to those without diffusers [11]. Similarly, research on tidal turbines with flanged channels revealed a notable improvement in power output performance [10]. Building on these findings, the current analysis focuses on the design of a vertical-axis ocean current turbine tunnel using 3-dimensional simulation through SolidWorks Computational Fluid Dynamics (CFD). This analysis aims to determine the most suitable design concept and assess its impact on increasing ocean current speeds.

In contrast to this study, Roman Gabl et al. (2022) conducted a 2-dimensional simulation using a cross-flow turbine with a venturi design [15]. The current research, however, specifically investigates the vertical-axis turbine with a tunnel construction. The goals of this simulation include evaluating the effectiveness of the tunnel in enhancing current velocity.

The results from the tunnel construction simulation at a current velocity of 1.0 m/s were analyzed using CFD, with the cut-plot velocity illustrated in Figure 2. These results provide crucial insights into how the tunnel design impacts the speed of the ocean currents, contributing to the overall efficiency of the vertical-axis ocean current turbine.

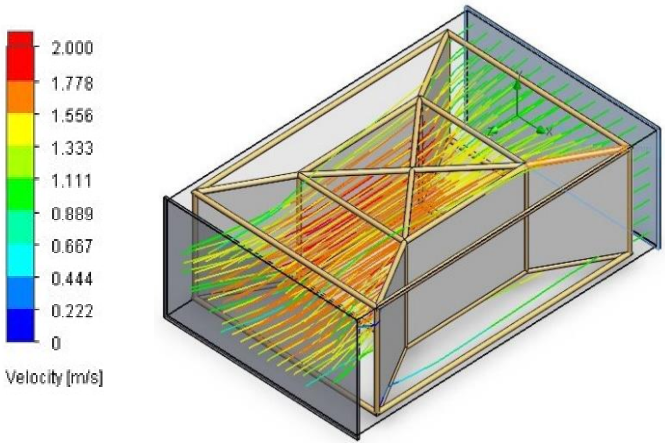

Figure 2. Cut-plot velocity (m/s) V 1,0 m/s

The simulation of the vertical-axis ocean current turbine tunnel demonstrates a significant increase in the speed of the ocean current after passing through the tunnel construction. Initially, the ocean current had a velocity of 1.0 m/s, as indicated by the green flow area in the simulation image. Upon entering the middle section of the tunnel, the current speed visibly increases, transitioning to an orange area, which corresponds to a velocity increase to 1.8 m/s. This increase is a direct result of the reduction in the cross-sectional area of the flow within the tunnel, consistent with the continuity equation. According to this principle, the volumetric flow rate must remain constant, so when the cross-sectional area decreases, the velocity of the current must increase to maintain the same flow rate. These simulation results highlight the effectiveness of the tunnel design in accelerating ocean currents, which in turn has the potential to enhance the performance of vertical-axis ocean current turbines.

The methodology employed to calculate the increase in ocean current velocity through the tunnel construction is based on the continuity equation and other fundamental fluid mechanics concepts. The process begins by defining the system and the assumptions, including the initial and final cross-sectional areas and the corresponding current velocities.

In a separate simulation with an initial current velocity of 1.5 m/s, the results further demonstrate the effectiveness of the tunnel design. The Computational Fluid Dynamics (CFD) simulation revealed the cut-plot velocity, as shown in Figure 7, illustrating the increase in current speed as it flows through the tunnel. These findings underscore the tunnel's capability to enhance current velocities, which is crucial for optimizing the performance of vertical-axis ocean current turbines.

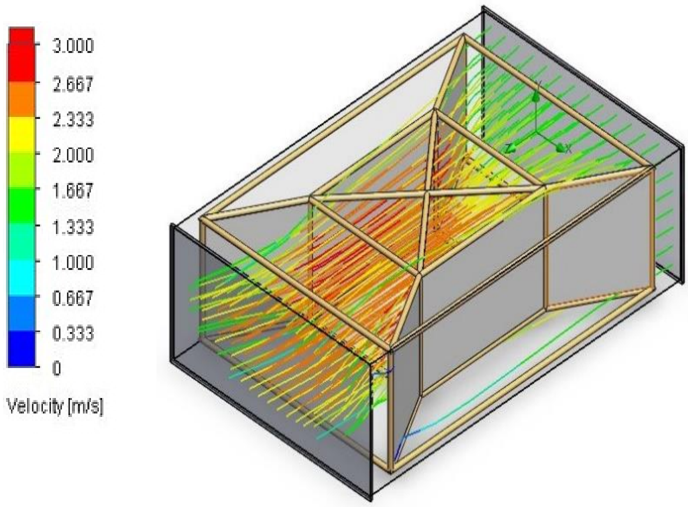

Figure 3. Cut-plot velocity (m/s) V 1,5 m/s

The simulation results demonstrate a significant increase in ocean current speed after entering the tunnel construction. Initially, the current had a velocity of 1.5 m/s, represented by the green flow area in the simulation. After passing through the tunnel, the current speed increased to 2.6 m/s, as indicated by the orange flow area in the color graph. According to the continuity equation, which states that the volumetric flow rate must remain constant, this increase in velocity corresponds to a ratio of the cross-sectional area of 1.73. Computational simulations, such as those performed using Computational Fluid Dynamics (CFD), are crucial for modeling and verifying these theoretical calculations. These simulations account for boundary conditions and velocity distributions within the tunnel, ensuring that the increase in ocean current speed is accurately calculated and supporting an effective tunnel system design.

The experimental data from the study further supports these findings. Analysis of two experiments with different ocean current speed variations indicates a consistent increase in current speed as it enters the tunnel. This acceleration is a key factor in improving the performance of the vertical-axis ocean current turbine installed in the tunnel. As the speed of the ocean current increases, the kinetic energy available for conversion into mechanical energy by the turbine also rises, leading to higher energy output. The experimental data also reveal a positive correlation between ocean current speed and turbine performance, confirming that higher current speeds result in more optimal electricity generation by the turbine.

The results of the tunnel construction data analysis are presented in the form of graphs, which provide a clear comparison of the simulation outcomes for different ocean current speeds. These graphs are essential for visually interpreting the simulation data, allowing for an easier understanding of how the tunnel construction affects ocean current velocities under varying conditions.

The comparative graph of the two simulations, which were conducted at different initial ocean current velocities, reveals an interesting pattern of speed increases. For an initial ocean current velocity of 1.0 m/s, the graph shows a relatively modest but noticeable increase in speed. In contrast, when the initial ocean current velocity is 1.5 m/s, the graph indicates a more pronounced acceleration. This suggests that stronger currents tend to experience greater acceleration within the tunnel. This phenomenon can be attributed to factors such as the increased thrust generated by faster-moving currents and potential synergistic effects with other environmental factors, which collectively enhance the performance of the tunnel in boosting current velocities.

4. Conclusions

The research and design of the vertical-axis ocean current turbine tunnel have shown that the tunnel construction effectively increases the speed of ocean currents as they pass through the system. The tunnel successfully raises the current velocity from 1.0 m/s to 1.8 m/s, and from 1.5 m/s to 2.6 m/s. This enhancement in current speed has the potential to significantly boost the performance of vertical-axis ocean current turbines, as the turbine's efficiency is closely tied to the velocity of the ocean currents. The increase in flow velocity results in greater kinetic energy available for conversion into electrical power, thereby improving the turbine's overall output.

Additionally, the study confirms that the tunnel construction contributes to a more uniform flow of ocean currents, which is reflected in the calculated Reynolds numbers. With Reynolds numbers of 100.250 at a current speed of 1.0 m/s and 150.375 at 1.5 m/s, the flow within the tunnel remains laminar. This laminar flow is beneficial for consistent turbine operation, as it minimizes turbulence and enhances the efficiency of energy conversion. The findings underscore the effectiveness of tunnel construction in optimizing the performance of vertical-axis ocean current turbines.

References

[1] W.M. Rumaherang, R. Ufie, W.M.E. Wattimena, and B.G. Tentua, "Karakteristik turbin propeller sumbu horisontal pembangkit listrik tenaga arus laut," *Archipel. Engineering.*, vol. 1, no. 1, pp. 90–95, 2018.

[2] A. Kasharjanto, E. Erwandi, E. Marta, Z. Irawanto, D. Rahuna, and S.J.M. Cahyadi, "Study of the motion performance of marine current power plant turbine floaters due to ocean current forces under moored conditions," *Kapal: Jurnal Ilmu Pengetahuan dan Teknologi Kelautan*, vol. 20, no. 3, pp. 343–349, 2023.

[3] R.M. Lopulalan, "Desain blade turbin pembangkit listrik tenaga arus laut di Banyuwangi berbasis CFD," *Jurnal Teknik ITS*, vol. 5, no. 2, pp. 424–430, 2016.

[4] S.E. Yulianto," Analisa putaran dan torsi turbin arus laut akibat pengaruh variasi diameter dan jumlah blade," Undergraduate thesis, Institut Technology Sepuluh Nopember, 2015.

[5] H. Li, L. Yu, and Y. He, "Modified BEM methods for the marine current turbine performance analysis under non-uniform inflow condition," *Proc. 5th International Symposium on Marine Propulsors (SMP'17),* 2017.

[6] T. Blackmore, W.M.J. Batten, and A.S. Bahaj, "Influence of turbulence on the wake of a marine current turbine simulator," Proceedings of the Royal Society A: Mathematical, Physical and Engineering Sciences, vol. 470, no. 2170, 2014.

[7] N. Ruhyat, "Comparison of the dryer air inlet position on the spray dryer with a double condenser to produce a rotating flow throughout the drying chamber: CFD analysis," *International Journal of Innovation in Mechanical Engineering & Advanced Materials*, vol. 5, no. 1, 2023.

[8] P. Mycek, B. Gaurier, G. Germain, G. Pinon, and E. Rivoalen, "Experimental study of the turbulence intensity effects on marine current turbines behaviour. Part II: Two interacting turbines," *Renewable. Energy*, vol. 68, pp. 876–892, 2014, doi: 10.1016/j.renene.2013.12.048.

[9] W. Hua-Ming, Q. Xiao-Kun, C. Lin, T. Lu-Qiong, and W. Qiao-Rui, "Numerical study on energy-converging efficiency of the ducts of vertical axis tidal current turbine in restricted water," *Ocean Engineering*, vol. 210, p. 107320, 2020, doi: 10.1016/j.oceaneng.2020.107320.

[10] M. Maduka and C. W. Li, "Numerical study of ducted turbines in bi-directional tidal flows," *Engineering Applications of Computational Fluid Mechanics*, vol. 15, no. 1, pp. 194–209, 2021, doi: 10.1080/19942060.2021.1872706.

[11] M. M. Nunes, A. C. P. Brasil Junior, and T. F. Oliveira, "Systematic review of diffuser-augmented horizontal-axis turbines," *Renewable and Sustainable Energy Reviews,* vol. 133, p. 110075, 2020, doi: 10.1016/j.rser.2020.110075.

[12] A. Febrianto and A. Santoso, "Analisa Perbandingan torsi dan RPM turbin tipe Darrieus terhadap efisiensi turbin," *Jurnal Teknik ITS*, vol. 5, no. 2, 2017, doi: 10.12962/j23373539.v5i2.19414.

[13] R. B. A. Nugraha, N. Prayogo, M. Kartidjo, and S. Nugroho, "Analisis computational fluid dynamic (CFD) dalam perancangan turbin arus laut sumbu vertikal (vertical axis ocean current turbine, VAOCT)," *Journal Kelautan Nasional.*, vol. 8, no. 3, p. 135, 2013, doi: 10.15578/jkn.v8i3.6233.

[14] Syahdan S. Maulana, "Analisis optimasi aliran fluida dalam pipa kondensasi water from atmosphere generator berbasis computational fluid dynamic (CFD)," Thesis, Universitas Negeri Semarang, 2019.

[15] R. Gabl, J. Burchell, M. Hill, and D.M. Ingram, "Sensitivity analysis of a Venturi shaped structure for cross-flow turbines," *Engineering Applications of Computational Fluid Mechanics*, vol. 16, no. 1, pp. 2243–2269, 2022.