Jurnal Polimesin
Department of Mechanical Engineering
State Polytechnic of Lhokseumawe
http://e-jurnal.pnl.ac.id/polimesin

e-ISSN	: 2549-1999	No.	: 2	Month	: April
p-ISSN	: 1693-5462	Volume	: 22	Year	: 2024

Article Processing Dates: Received on 2023-08-29, Reviewed on 2024-03-07, Revised on 2024-03-24, Accepted on 2024-04-24 and Available online on 2023-04-30

Harnessing CFD simulations and CAD design to optimize hydropower efficiency of a propeller turbine in the Maliringan River: a case study

David Nathanlius, Levin Halim*, Bagus Arthaya
Department of Electrical Engineering, Parahyangan Catholic University, Bandung,40141, Indonesia
*Corresponding author: halimlevin@unpar.ac.id

Abstract

This research focuses on improving hydropower production by designing a turbine for the Maliringan River in Kalimantan Selatan. The rotational speed and torque will be the focus of maximizing power generation. This research used Computational Fluid Dynamics (CFD) to understand how the fluid flows and how efficiently the turbine works. The simulations helped us see how the fluid flowed, the pressure differences, and the speed of the water inside the turbine. The software COMSOL Multiphysics imitates how the fluid behaves and interacts in real-life situations. The utilization of Solidworks played a crucial role in the turbine's design process, facilitating an accurate representation of the turbine's geometry and the subsequent fabrication of a prototype propeller turbine, featuring an outer diameter measuring 0.27 meters and an inner diameter measuring 0.113 meters. The methodology resulted in a power efficiency of 76.45%, showcasing the possibility of significant enhancements in the efficiency of hydropower generation. The broader ramifications of this study emphasize the feasibility of tailor-made turbines for local hydropower initiatives, thereby supporting Indonesia's renewable energy plan by providing sustainable and efficient energy alternatives. This study emphasizes the collaborative utilization of Computer-Aided Design (CAD) and CFD technologies in the progression of turbine technology, thereby establishing a basis for future investigations in hydropower optimization.

Keywords:
Propeller, COMSOL, CFD, reaction turbine.

1 Introduction

The development of renewable energy sources has advanced significantly, especially in developed countries like the United States and Germany. According to data from the U.S. Energy Information Administration[1], in 2018, approximately 11% of the electricity supply in the United States was derived from renewable energy sources out of the total electricity generation[2]. These renewable energy sources include hydropower, biomass, geothermal, wind, and solar[3]. Several nations already heavily depend on renewable energy sources for their electricity generation. In April 2017, Germany reported that 85% of its electricity came from renewable sources[4].

Indonesia has great potential for developing new and renewable energy sources such as solar, wind, biomass, ocean waves, hydro, and geothermal[5]. However, using renewable energy in Indonesia only accounts for 6.8% of the total energy produced[6]. One of the renewable energy sources is hydropower, which involves harnessing the energy from flowing water to produce electricity [7]. As a continuously flowing resource, water offers a significant natural energy source. It can be derived from rivers' flowing water or ocean waves. This energy can be converted into electricity, providing a continuous power supply throughout the day [8]. Hydropower accounts for 30% of the world's energy production [9]. There are two main types of hydropower plants: Large Hydroelectric Power Plants (LHPP) and Micro/Mini Hydropower Plants (MHP) [10].

In generating electricity from hydropower, a water turbine converts the potential energy from the water source into kinetic energy, which rotates the turbine wheel connected to the generator shaft, producing electricity [11], [12]. However, the potential energy in different water sources varies, making it essential to choose the appropriate water turbine for each specific water source [13]. Moreover, despite Indonesia's substantial hydropower potential, only around 10% of this capacity is exploited due to difficulties accessing water sources[14].

Optimizing hydropower output in Indonesia might boost energy security, reduce fossil fuel use, and reduce environmental impacts, contributing to sustainable development[15], [16]. Economically, hydropower efficiency can cut electricity prices, increase energy independence, and enhance rural growth by providing stable power[17], [18]. Hydropower, a clean energy source, helps reduce greenhouse gas emissions and climate change[19]. To guarantee sustainability, hydropower projects must carefully examine ecological implications such as water usage and biodiversity conservation[20]. This research aligns with these goals by optimizing the Maliringan River turbine design to convert water to energy efficiently. The study focuses on enhancing hydropower efficiency, making Indonesia's energy infrastructure more sustainable and eco-friendly.

For optimal electricity generation through hydropower, the water turbine must rotate the generator shaft above a certain minimum speed while providing enough torque to overcome the generator's detent torque[21]. Detent torque is a force that momentarily prevents the generator shaft from rotating before it starts to move. This resisting force briefly stops the shaft from turning until it overcomes the obstacle and rotates[22], [23]. Despite efforts to optimize water turbine design, there are still some losses during the energy conversion process. This means the actual power output differs from the mathematically calculated output[6].

In recent years, scientists have been intensively studying how to improve water turbine designs. Their research has focused on making water turbines more efficient, mainly by examining the design of the turbine blades[24]. Additionally, researchers have used Computational Fluid Dynamics (CFD) to analyze the power efficiency of Francis turbines[25]. By looking at the features of the water sources, the turbine blades, guide vanes, and turbine casing can be improved to boost power efficiency through CFD analysis. This allows the determination of fluid flow speeds within the water turbine.

The broad potential of hydropower in Indonesia and its underexploited capacity warrants a focused turbine efficiency study. Recognizing these general opportunities and difficulties to this specific study target emphasizes the need for novel turbine designs customized to local conditions. This water turbine design and implementation study uses the Maliringan River to close this gap. This approach improves water kinetic energy use and models responsible and sustainable hydropower use in Indonesia. We hope to contribute to renewable energy optimization and solve hydropower development's technical and environmental issues by focusing on propeller turbine design.

Considerable progress has been made in the design and optimization of water turbines, with a substantial body of research making valuable contributions to this dynamic study area. Significantly, Nunes et al. conducted an empirical study on diffuser-enhanced propeller hydrokinetic turbines, showcasing the potential for higher efficiency through novel design alterations [26]. The design and hydrodynamic performance of horizontal axis micro-hydrokinetic river turbines were investigated by Wang et al., who offered valuable insights into the design parameters that optimize energy capture[27]. The research study by Ghimire et al. emphasizes the significance of customizing Francis turbine designs for micro-hydropower projects. This tailoring of turbine configurations is crucial to optimize power generation, as it allows the turbines to suit the distinctive features of each specific site[28]. Previous research has laid a strong foundation for advancements that harness hydrokinetic energy by studying turbine efficiency and design. Our research builds on this foundation by designing and evaluating a prototype water turbine tailored to the Maliringan River.

This research aims to create a prototype water turbine for the Maliringan River to improve hydropower generation. Using computer simulations, the study looks at how the liquid flows through the turbine and how well it generates power. The project aims to find the best type of turbine based on its specific speed, test a propeller-based prototype, and see how well it converts energy. It should be noted that this propeller turbine does not include adjustable blades that will be adjusted to the equations used for calculations. The study aims to help make hydropower better, solve problems with getting to water sources, and optimize turbine designs for similar areas. This research will focus on taking energy from a specific water source, the Maliringan River in Desa Muara Hungi, Kalimantan Selatan, Indonesia. The investigation will involve utilizing a water source with a 3-meter head. A water turbine will convert the potential energy from the water source into kinetic energy generated by the turbine wheel's rotation. The turbine design will be conducted using SolidWorks, a Computer-Aided Design (CAD) software, and simulations will be performed using COMSOL Multiphysics software. The performance of the designed propeller turbine will be tested to determine its rotational speed.

2 Research Methods

The previous research used as a reference in this study[29] discusses a Kaplan-type water turbine whose power efficiency is influenced by the design of its turbine blades. Also, it addresses the issue of hazardous gas emissions causing significant climate change impacts on the environment.

Hydropower generated by water turbines can utilize constant flow of water in rivers as a steady energy source. Therefore, hydropower plants can be found worldwide, and turbine-generated power can reach up to 95% efficiency [29].

The design of turbine blades significantly impacts the efficiency produced by a Kaplan turbine. Therefore, blade design must be optimized to maximize energy absorption. The design of these Kaplan turbine blades utilizes the characteristics of a water source with a flow rate of 5 m3/s and a head (H) of 6 meters[29].

Research on Francis turbine design[30] is also available, exploring the foundational principles of designing such turbines using a methodology incorporating Computational Fluid Dynamics (CFD) analysis. The study's results indicate that the accuracy level achieved using the CFD method is over 90%, making CFD a suitable approach for this research. However, CFD analysis is not employed in the turbine design process; instead, it is used to determine the fluid flow velocity within the turbine to calculate the power generated.

In conducting this research, the stages of turbine design are carried out in Fig. 1.

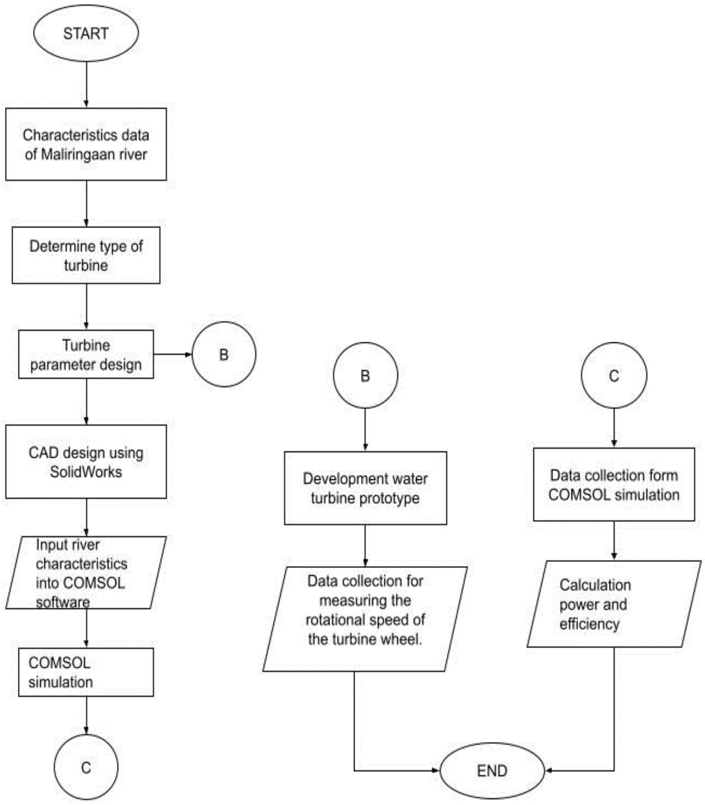

Fig. 1. Method flowchart.

Fig. 1 illustrates the methodology flowchart employed in this study, explained as follows:

1. The selection of the water-energy source to be studied is followed by data collection of its characteristics, including flow velocity, water discharge, and head. The factors such as flow velocity (v), water discharge (Q), and head (H) were measured. Different discharge rates and head pressures were considered to create various energy potential scenarios and improve the turbine's design.
2. Determination of the suitable turbine type is based on the characteristics of the obtained water-energy source. The specific speed was calculated using the obtained data ($Q$and H), which will be the main parameter for selecting a turbine type.
3. Establishment of turbine parameters for the design, which include the number of turbine blades, blade spacing, blade height, blade length, inner turbine diameter, and outer turbine diameter. These parameters were used to developa CAD model, which is essential for the simulation and prototype.
4. Using the identified parameters as a guide, the design process utilizes Computer-Aided Design (CAD) software like SolidWorks. Furthermore, a prototype of the turbine was developed during the design phase.
5. Utilizing the COMSOL Multiphysics program to input the water energy source's properties as fluid properties into the simulation.
6. Execution of simulations with COMSOL Multiphysics based on the CAD design and the characteristics of the water energy source. The CAD model of the turbine was brought into COMSOL Multiphysics for running simulations to examine the fluid velocity and pressure distribution.
7. Several simulations will be conducted at different flow rates and head conditions to grasp how well the turbine can operate fully.
8. The fluid velocity data obtained from the simulation was utilized to determine the generated power and turbine efficiency.
9. Conducting a trial of the turbine wheel rotation speed measurement system (prototype testing). The prototype was set

up, where its speed, force, and energy generation were assessed in different scenarios. The prototype underwent various tests with varying water flows and speeds to confirm the accuracy of the simulation findings and improve the design using real-world data.

2.1 Determining Turbine Type

Three parameters are used as guidelines in determining the most suitable type of turbine for specific operating conditions [22]. Three essential parameters are considered: water flow rate, head (vertical waterfall distance), and the minimum rotational speed for the power plant's generator. These factors determine the turbine's specific speed, helping pinpoint the ideal turbine type for a given water resource.

The specific speed is a parameter used to classify types of water turbines based on the turbine speed at which it can generate 1 unit of power for each head unit[23]. The unit of specific speed is not mentioned because specific speed is a dimensionless parameter or a parameter without dimensions. The specific speed is indicated in Eq. 1.

$$n_q = \frac{n\sqrt{Q}}{H^{\frac{3}{4}}} \quad (1)$$

n_q is specific speed (dimensionless)
n is generator rotational speed (rad/sec)
Q is flowrate (m^3/s)
H is head (m).

Utilizing this specific speed allows the identification of the best turbine type for achieving maximum power efficiency[23]. Fig. 2 illustrates the classification of turbines.

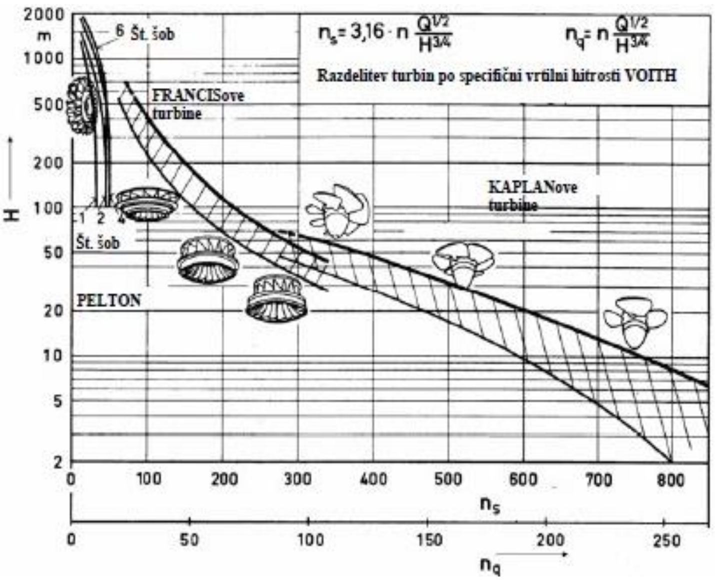

Fig. 2. Graphic curve for turbintype [31].

2.2 Computational Fluid Dynamics

CFD, or Computational Fluid Dynamics, is a numerical method solving fluid flow and heat transfer equations in a defined geometric space. It's a highly effective technique, widely applicable in various industries[32].

Using numerical algorithms, CFD addresses problems in fluid flow systems. It provides user-friendly problem-solving through an interface for inputting parameters and obtaining instant results. The CFD algorithm involves three stages: pre-processor, solver, and post-processor.

2.3 Rotational Speed and Torque Calculation

To optimize energy conversion in the turbine wheel, the rotational speed and torque of the water turbine were calculated using theoretical equations and computer simulations (CFD). Eq. 1 was used to determine the rotational speed based on the turbine's specific speed, which considers the Maliringan River's flow rate and head (drop in water level).

To calculate torque (T in Nm), we use the formula $T = P/\omega$, where P is power (W), and ω is the angular velocity (rad/s), which we calculate from rotational speed. We used COMSOL Multiphysics software to run Computational Fluid Dynamics (CFD) simulations. These simulations helped us accurately model the fluid flow inside the turbine and confirm our theoretical calculations, giving us a deep understanding of how the turbine works.

In this study, the research focused on the flow rate and head of water, as they directly impact the energy that the turbine can convert. The turbine's rotational speed was also critical, as it affected the mechanical energy output. In the prototype testing, these parameters will be varied to consider various operational scenarios. We specifically looked at how adjusting the flow rates and head pressures affected the efficiency and power output of the turbine. These experiments allowed us to fine-tune the turbine's design for optimal performance in real-world conditions like those in the Maliringan River.

3 Results and Discussion

The simulated turbine will be designed using real-world data from the Maliringan River in Muara Hungi Village, South Kalimantan. The specific characteristics used for the design are detailed in Table 1, which contains measurements collected in 2018.

Table 1. River characteristic

Flow rate	0.26 m^3/s
Flow velocity	0.55 m/s
Head	3 m
Density	997.0 kg/m^3
Gravity	9.8 m/s^2

3.1 Turbine Specification

The determination of the specific speed of the turbine is carried out by assuming that the generator has a frequency of 50Hz and a total of 6 poles, resulting in a minimum rotational speed for the generator to produce electrical energy of 1000 RPM or 16.67 Revolutions Per Second (RPS). Derived from the rotational speed and the attributes of the Maliringan River, the specific turbine speed calculated using Eq. 1 is 223.97.

Fig. 3 shows that the propeller or Kaplan turbine type is the most suitable for the specific speed value that the turbine generates based on the characteristics of the Maliringan River.

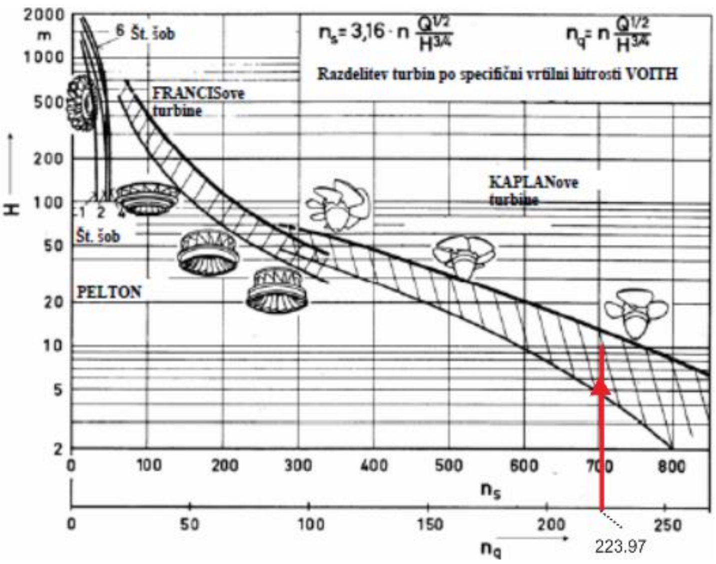

Fig. 3. Determine the type of turbine[31].

In the design of the propeller turbine runner, the characteristic factors σ and δ need to be determined beforehand. The characteristic factor σ can be determined using Eq. 2.

$$\sigma = \frac{2\pi\sqrt{\pi Q}}{(2gH)^{\frac{3}{4}}} = 1.42 \quad (2)$$

Subsequently, it is used to determine the ratio between the inside diameter (*D.N.)* and the outside diameter (D_a), the number of turbine blades (*z*), and the characteristic factor δ using the curves shown in Fig. 4.

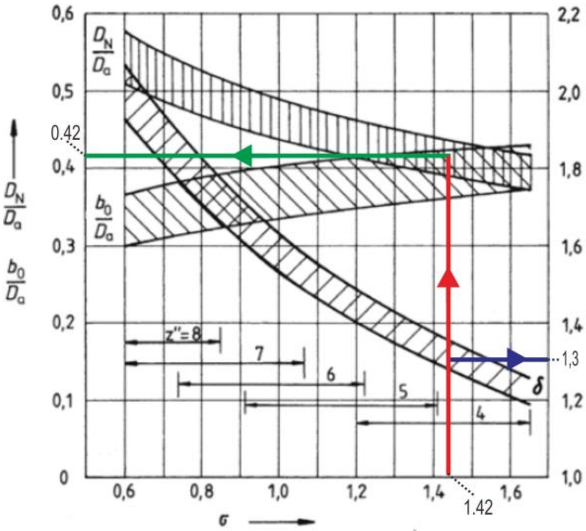

Fig. 4. Determine the parameters of turbine[31].

Based on these lines, the values are approximately δ ≈ 1.3, the ratio between *D.N.* and D_a ≈ 0.42, and the number of propeller turbine blades is four (Eq. 3).

$$D_a = \frac{2\delta}{\sqrt{\pi}}\frac{Q^2}{2gH}^{1/4} = 0.27\ m \quad (3)$$

Given that the ratio between the inside diameter D_N and the outside diameter $D_a$is 0.42, the inside turbine diameter is calculated as shown in Eq. 4.

$$\frac{D_N}{D_a} = 0.42 \quad (4)$$
$$D_N = 0.113\ m$$

In this propeller turbine, a total of four blades are used, so the distance between the blades at the inside diameter and the outside diameter can be determined using the Eq. 5 and Eq. 6.

$$t' = \frac{\pi D_N}{z} = 0.088\ m \quad (5)$$

$$t'' = \frac{\pi D_a}{z} = 0.212\ m \quad (6)$$

The angle formed by the guide vane is the same as the angle formed by the turbine runner blade, where θi = θe = 45°. Thus, the length of the blade at the inside and outside diameters of the turbine runner can be determined using the Eq. 7 and Eq. 8.

$$l' = \frac{t'}{cos\theta_e} = 0.125\ m \quad (7)$$

$$l'' = \frac{t''}{cos\theta_e} = 0.30m \quad (8)$$

The suitable hub height of the propeller turbine, considering the distance between the blades and the length of the blades, can be determined Eq. 9.

$$s' = \sqrt{(l'')^2 - (t'')^2} = 0.212m \quad (9)$$

3.2 Turbine Design

The turbine design is based on the specifications that were previously specified. Fig. 5 illustrates the propeller turbine design.

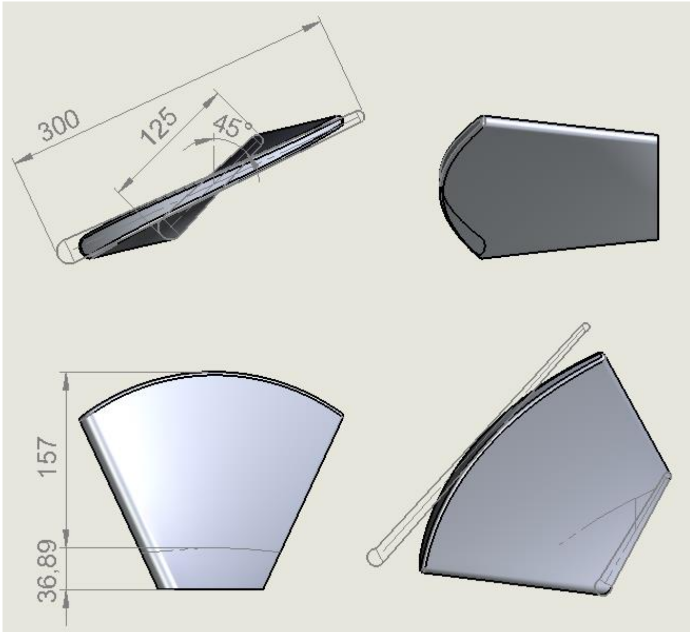

Fig. 5. Design of propeller turbine runner blade.

In Fig. 6 of the turbine runner hub, four holes connect the blades and the hub. These holes have a depth of 36.89 mm, resulting in an offset of 36.89 mm in the design of the turbine runner blades. The orthogonal projection of the propeller turbine runner shown in Fig. 7.

The length of the chord (T.B.) and the chord angle (S.B.) for the blade at the inside diameter and outside diameter are used to determine the distance between the blades. Eq. 10 represents the distance between the blades at the inside diameter, whereas Eq. 11 represents the distance at the outside diameter.

$$t' = TB_N\pi\frac{SB}{360}\csc\left(\frac{SB}{2}\right) = 0.88\ m \quad (10)$$

$$t' = TB_N\pi\frac{SB}{360}\csc\left(\frac{SB}{2}\right) = 0.212\ m \quad (11)$$

For calculating the guide vane angle, we can apply the Eq. 12–Eq. 15.

$$t' = \frac{\pi D_N}{z} = 0.0433\ m \quad (12)$$

$$t'' = \frac{\pi D_a}{z} = 0.106\ m \quad (13)$$

$$l' = \frac{t'}{cos45} = 0.062\ m \quad (14)$$

$$l' = \frac{t''}{cos45} = 0.15\ m \quad (15)$$

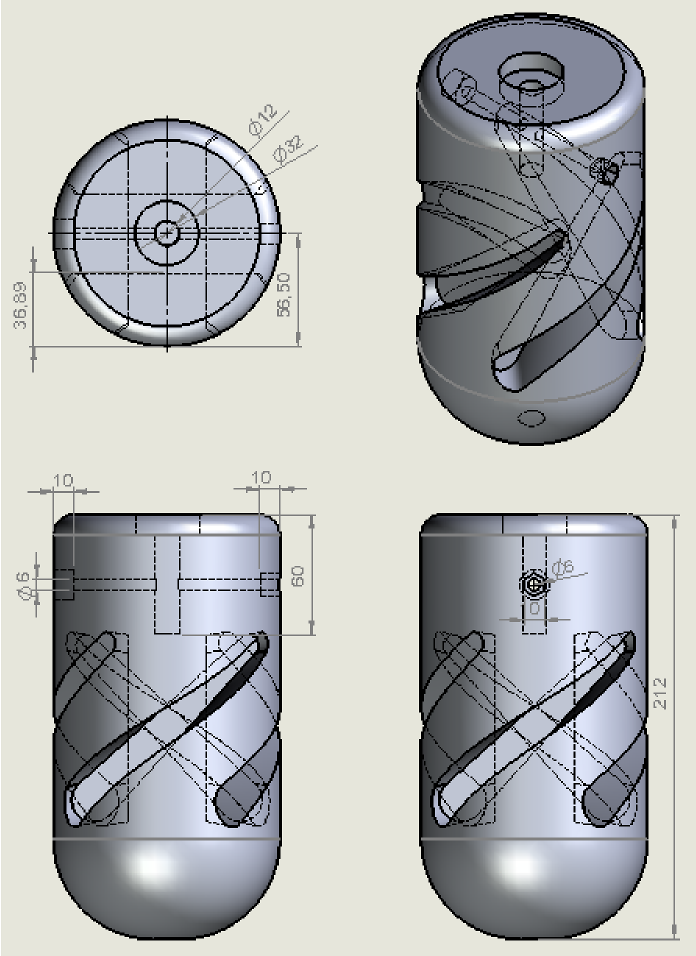

Fig. 6. Hub design of propeller turbine.

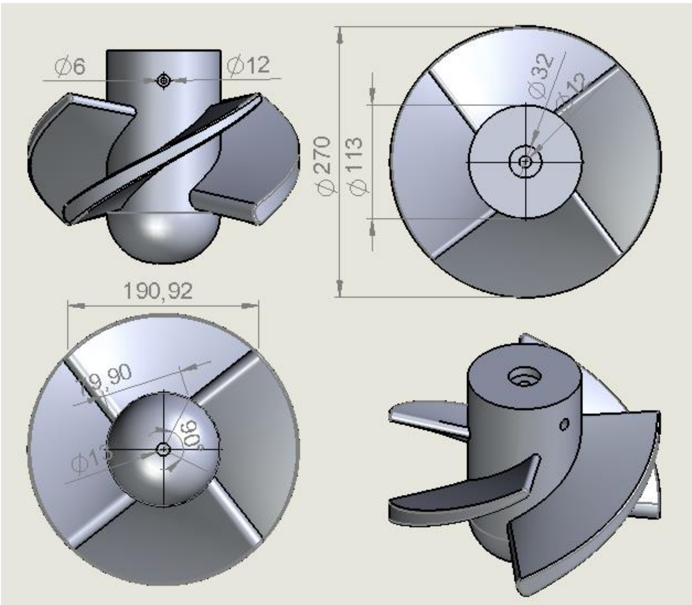

Fig. 7. The orthogonal projection of the propeller turbine runner.

The suitable hub height of the guide vane, considering the distance between the vanes and the length of the vanes, is as Eq. 16.

$$s' = \sqrt{(l'')^2 - (t'')^2} = 0.106\,m \qquad (16)$$

The guide vanes in the design of this propeller turbine do not rotate like the turbine runner blades but remain stationary. This is because the function of the guide vanes is to direct the flow of water so that it impinges perpendicularly on the blades of the turbine runner. Design of the guide vane shows in Fig 8-10.

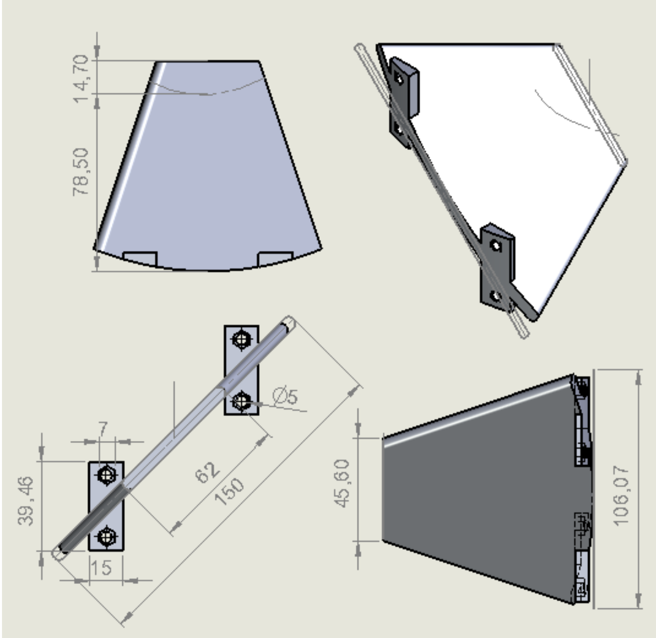

Fig. 8. Design of the guide vane blades.

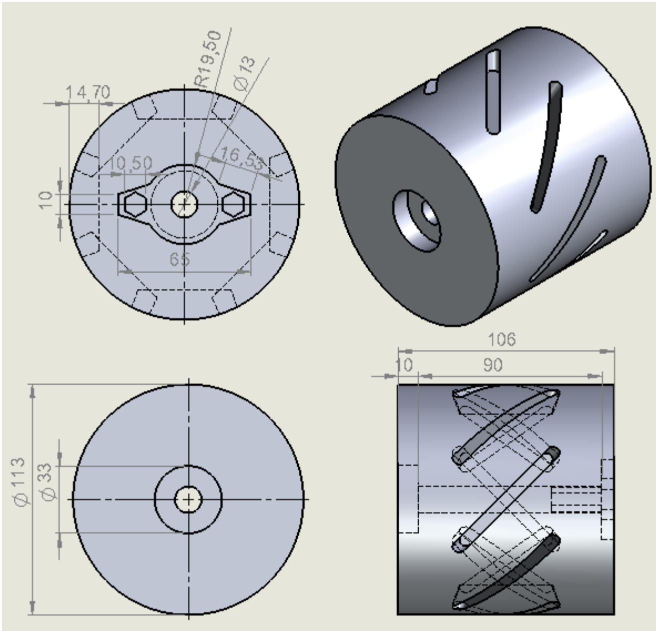

Fig. 9. Hub design of guide vane baldes.

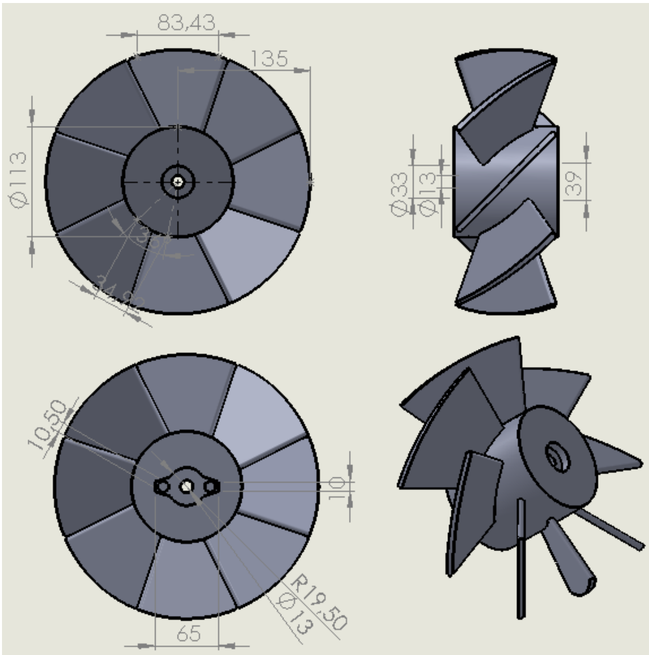

Fig. 10. Design of guide vane.

3.3 The Design of a Propeller Turbine House

The propeller turbine house used in this study was designed with a tubular channel that acts as a conduit for fluid to flow toward the propeller turbine runner. Fig. 11 illustrates the fluid conduit pipe leading to the propeller turbine runner.

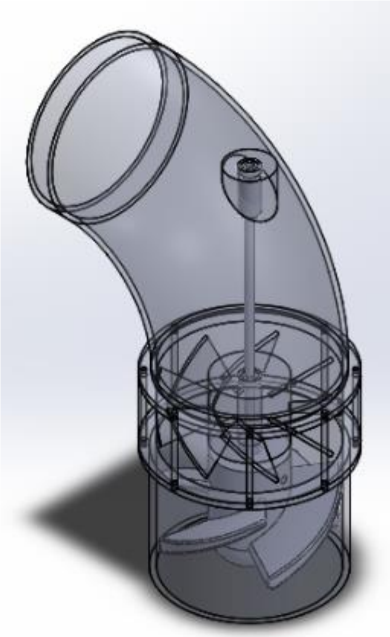

Fig. 11. 3D assembly of propeller turbine.

The prototype design of the turbine runner and guide vanes were carried out using a 3D printer, with parts being printed for both the turbine runner and the guide vanes. The 3D printer utilized in this research was the CREALITY ENDER 3-D. The 3D printing process for these two components involved creating CAD designs for each part, which were then imported into Ultimaker Cura 4.0 software to estimate material usage and determine printing quality. The material used in this 3D printer was Polylactic Acid (PLA). In the prototype design of the propeller turbine house, three types of PVC pipes were used: one elbow pipe, one connecting socket, and one PVC pipe with a length of 0.5 meters, each having a diameter of 11 inches. In the prototype design of the turbine runner using a 3D printer, the turbine runner was divided into two parts: the hub and the propeller turbine blades. Around the hub of the propeller turbine, holes were created to connect the blades with the hub, eventually forming the turbine runner (Fig. 12).

Fig. 12. Assembly turbine with runner.

Fig. 13 is the 3D-printed result of the propeller turbine blades. Four blades were printed to match the design's blade count. These blades were connected to the turbine runner hub using iron adhesive.

Fig. 13. 3D print of guide vane.

The overall result of the guide vanes (Fig. 14) used in the propeller turbine prototype is shown in Fig. 15.

Fig. 14. Guide vane hub.

Fig. 15. Theoverall result of guide vane in propeller turbine.

Fig. 16 shows the bevel gear in the propeller turbine house. The propeller turbine prototype combines the propeller turbine runner, guide vanes, and the propeller turbine house. Fig. 17 illustrates the prototype of the propeller turbine based on the characteristics of the Maliringan River in Muara Hungi Village, South Kalimantan.

Fig. 16. Bevel gear.

Fig. 17. The result of prototype propeller turbine.

The power generated by the designed propeller turbine based on the characteristics of the water source from the Maliringan River, as shown in Table 1, and the propeller turbine design using Eq. 17.

$$P_{id} = \rho QgH = 994.65\ W \qquad (17)$$

The result shown in Eq. 17 represents the flow rate entering the turbine house. The turbine house used has a diameter of 0.2794 meters. Referring to Table 1, where the flow velocity of the water in the Maliringan River is 0.55 m/s, the flow rate (Q) used in determining the ideal power is 0.0339 m^3/s.

In calculating the actual power, the velocity triangle on the turbine blade needs to be analyzed. This analysis of the velocity triangle was performed using Computational Fluid Dynamics (CFD) simulations. The simulations were carried out to obtain the fluid velocity flowing inside the propeller turbine, allowing the inlet and outlet fluid velocities to be determined.

The simulation results indicate that a numerical simulation approach was employed, revealing consistent and replicable outcomes when the simulation was repeated with identical parameters. This simulation was conducted using the COMSOL Multiphysics software. In the initial simulation process, meshing is necessary, which involves dividing the geometric area of the turbine's surface into several subdivisions or simple geometric shapes such as triangles, quadrilaterals, hexagons, or tetrahedra[33]. The entire turbine surface is divided into tetrahedral shapes in the meshing process. This is done so that variations in the quantity sought can be calculated based on simple function equations present in COMSOL Multiphysics, requiring meshing for each element on the turbine surface[33].

Fig. 18 shows the flow vectors, representing the motion of the fluid flow occurring inside the propeller turbine housing. These vectors indicate the direction of the water flow entering the turbine housing and colliding with its interior components. Based on the three simulations conducted, identical graphs were obtained. This graph illustrates the flow velocity for each position inside the turbine, as shown in Fig. 19. The graph depicts the velocities occurring within the turbine house. Using this graph, the fluid velocity at a specific position in the propeller turbine can be determined.

Based on Fig. 20, which originates from the simulation, the resulting data consists of the fluid's inlet velocity (c_e) and outlet velocity (c_e), with values of c_i = 7.66 m/s and c_e = 5.42 m/s. With these values of c_i and c_e, the actual power generated by the turbine can be obtained using Eq. 18.

$$P_{ak} = \rho Qu(c_{iu} - c_{eu}) = 760.47\ W \qquad (18)$$

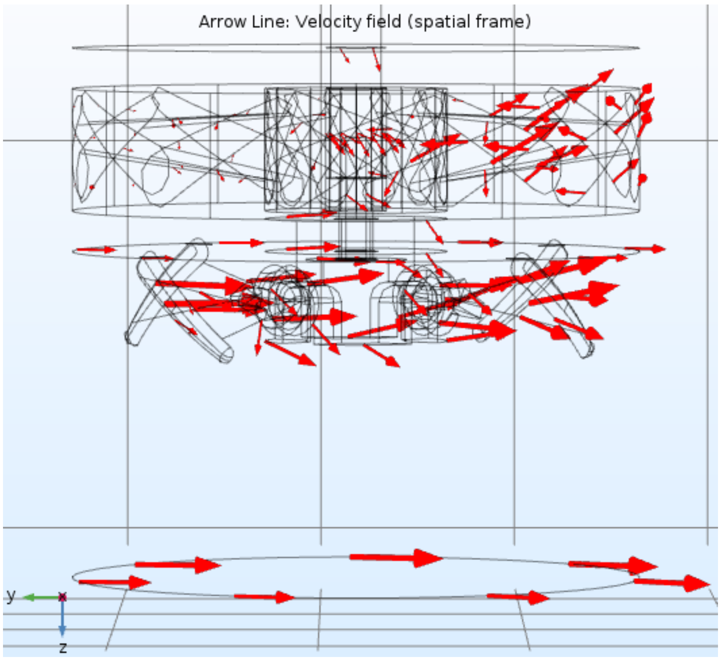

Fig. 18. Velocity vector result.

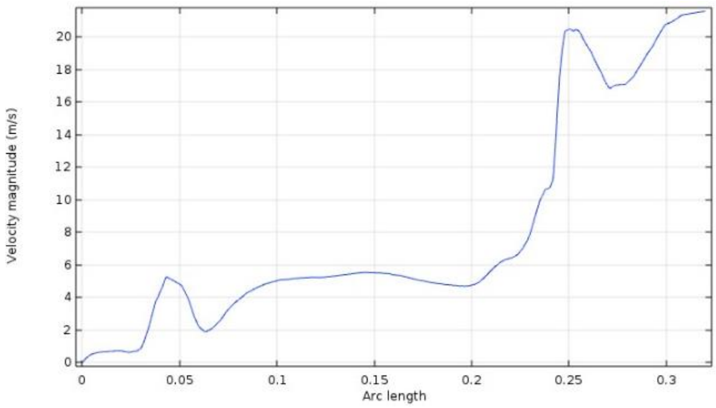

Fig. 19. The fluid velocity.

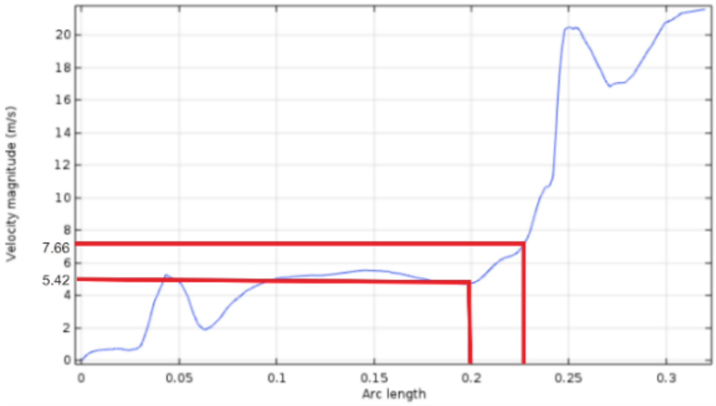

Fig. 20. $C_i$and $C_e$value.

Therefore, we can conclude the efficiency using Eq. 19, the result is as follows:

$$n_t = \frac{P_{ak}}{P_{id}} = 76.45\ \% \qquad (19)$$

The efficiency denotes the mechanical power output achievable by the prototype propeller turbine. The calculated efficiency of 76.45% surpasses that of various comparable studies. For instance, according to reference[34], the efficiency of a tubular propeller turbine is reported at 64%. Likewise, reference[35]indicates an efficiency of 57.35% for low-headed turbine propellers with axial flow.

Testing the prototype propeller turbine involved a comparative analysis of three rotational speed measurement datasets of the turbine shaft. The shaft, actuated by a drive motor, was subjected to measurements from three sources: speed data acquired from the drive motor's encoder, speed data from the encoder mounted on the propeller turbine, and measurements obtained through a tachometer. Fig. 21 presents the rotational speed values captured from the three measurement types under varying PWM configurations.

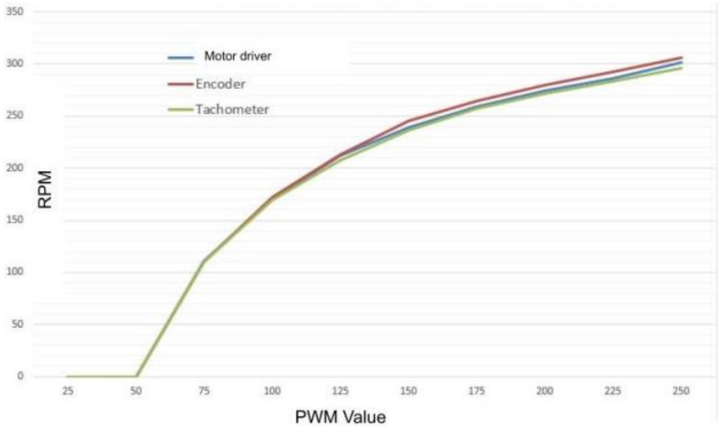

Fig. 21. Speed measurement results of the propeller turbine.

Assessing a turbine's performance requires careful rotational speed and torque analysis. These factors strongly influence the efficiency of the turbine wheel's energy conversion. To understand their impact, we have calculated their role in optimizing the turbine's operating performance.

The faster the turbine spins (RPM), the faster the turbine wheel moves, which depends on the water flow. Using the specific speed value of 223.97 and the formula for speed, we calculated that the turbine should spin at approximately 2772.88 RPM. This high speed shows that the turbine harnesses the river's kinetic energy effectively.

Conversely, torque, measured in Newton meters (Nm), measures the turbine's ability to create rotational force. This force is crucial for the turbine's power generation capacity. The torque was derived from the generated power of 760.47 Watts and the turbine's rotational speed, which were used to calculate its angular velocity (ω). The angular velocity (ω) was approximately 92.43π rad/s, and the torque was calculated using Eq. 20.

$$T = \frac{P}{\omega} = \frac{760.47}{92.43\pi} = \frac{8.23}{\pi}\ Nm \qquad (20)$$

The turbine effectively converts the river's water energy into mechanical energy, essential for generating electricity. This conversion results in a torque value close to 2.62 Nm, roughly 8.23/π Nm.

4 Conclusion

A custom-designed prototype propeller turbine for the Maliringan River was developed. The turbine's efficiency was optimized for Indonesia's diverse waterways using speed calculations. Computer simulations showed that the turbine could generate up to 760.47 Watts of power under ideal conditions, while in real-world scenarios, it could produce 994.65 Watts. This represents an impressive 76.45% efficiency in converting water power into electricity. Rigorous prototype testing showcases remarkable rotational speed measurement system accuracy, with a minimal 3.77% maximum reading error. Comparative analyses against the encoder and drive motor compared to the tachometer reveal 98.38% and 97.15% accuracy, respectively. This efficiency surpasses that of various comparable studies, such as the efficiency reported by Samora et al. (64%) and Pribadyo et al. (57.35%) for similar turbine configurations.

This research suggests adapting turbine designs to other rivers with unique characteristics, integrating ecological and socio-economic assessments into renewable energy projects, and using advanced simulation techniques for more accurate performance predictions. However, this study has drawbacks. Detailed insights from a specific river system indicate caution when generalizing findings to other contexts. This research could be expanded by investigating more hydrological conditions and including more environmental effect assessments. Prototype testing's practical challenges emphasize the need for field-testing solid methods under diverse settings by comparing our findings to earlier studies conducted by researchers like Nunes et al. on diffuser-enhanced propeller turbines and Ghimire et al. on tailored Francis turbine designs. This research expands on the practical knowledge of enhancing turbine efficiency through targeted CFD simulations and prototype testing, showcasing the transformation of theoretical ideas into optimized real-world applications.

References

[1] "What role does renewable energy play in the United States?." Accessed: Jun. 14, 2019. [Online]. Available: https://www.eia.gov/energyexplained/renewable-sources/

[2] V. P. Sharma, D. K. Somwanshi, K. Jain, and R. K. Satankar, "A Literature Review on Renewable Energy Resource and Optimization," in *IOP Conference Series: Earth and Environmental Science*, IOP Publishing, 2022, p. 12002.

[3] B. Sireesha and Y. Nagaraja, "An Efficient Power Management Algorithm for a Micro Grid," *TELKOMNIKA Indonesian Journal of Electrical Engineering*, vol. 15, no. 1, Jul. 2015, doi: 10.11591/telkomnika.v15i1.7912.

[4] "Menilik Perkembangan Energi Terbarukan di Negara Maju." Accessed: May 12, 2019.[Online]. Available: https://www.kompasiana.com/zasorayya/59986ec3b6848368836904d2/menilik-perkembangan-energi-terbarukan-di-negara-maju?page=all

[5] Z. Anisa, A. Apprianda, H. Novianto, and I. Rachman, "Micro-Hydro Power Plants (MHPP): Technical and analytical studies in creating experimental learning media for physics students," *Momentum: Physics Education Journal*, pp. 53–64, Jan. 2021, doi: 10.21067/mpej.v5i1.4876.

[6] C. Anton, "Penggunaan Energi Terbarukan di Indonesia Hanya 6,8%." Accessed: May 12, 2019.[Online]. Available: https://ekbis.sindonews.com/berita/1115774/34/penggunaan-energi-terbarukan-di-indonesia-hanya-68

[7] M. Kiraga, "Hydroelectric Power Plants and River Morphodynamic Processes," *Journal of Ecological Engineering*, vol. 22, no. 7, pp. 163–178, Jul. 2021, doi: 10.12911/22998993/139068.

[8] M. N.S., G. N.A., and A. G.A., "Role of Renewable Energy Sources in the World," *Journal of Renewable Energy, Electrical, and Computer Engineering*, vol. 2, no. 2, p. 63, Nov. 2022, doi: 10.29103/jreece.v2i2.8779.

[9] International Energy Agency, "Hydropower Special Market Report," 2021.[Online]. Available: www.iea.org/t&c/

[10] B. Mayor, I. Rodríguez-Muñoz, F. Villarroya, E. Montero, and E. López-Gunn, "The role of large and small scale hydropower for energy and water security in the Spanish Duero basin," *Sustainability (Switzerland)*, vol. 9, no. 10, Oct. 2017, doi: 10.3390/su9101807.

[11] A. del M. Sabroso, "Simulation And Analysis Of A Combined Energy & Water System," Bachelor Thesis, Universidad Carlos III de Madrid, 2019.

[12] S. Hadi, R. J. Apdila, A. H. Purwono, E. P. Budiana, and D. D. D. P. Tjahjana, "Performance of the drag type of Horizontal Axis Water Turbine (HAWT) aseffect of depth to width ratio of blade," in *International Conference On*

Engineering, Science And Nanotechnology 2016, Solo, Jan. 2017. doi: 10.1063/1.4968257.
[13] A. Kumar, *Hydropower Engineering For Diploma Level Courses*, 3rd ed. 2008.
[14] BPS, "Sungai yang Daerah Pengalirannya Lebih dari 100 km2 , 2015." Accessed: Jun. 14, 2019. [Online]. Available: https://www.bps.go.id/statictable/2017/11/14/1984/rata-rata-harian-aliran-sungai-tinggi-aliran-dan-volume-air-di-beberapa-sungai-yang-daerah-pengalirannya-lebih-dari-100-km2-2015.html
[15] R. Syahputra and I. Soesanti, "Renewable energy systems based on micro-hydro and solar photovoltaic for rural areas: A case study in Yogyakarta, Indonesia," *Energy Reports*, vol. 7, pp. 472–490, Nov. 2021, doi: 10.1016/j.egyr.2021.01.015.
[16] Y. D. Herlambang *et al.*, "The effect of variations in electric load on the performance of a 3 kW Micro Hydro Power Plant using an undershot waterwheel," *J. Polimesin*, vol. 21, no. 2, pp. 214–218, 2023.
[17] E. Quaranta*et al.*, "Hydropower Case Study Collection: Innovative Low Head and Ecologically Improved Turbines, Hydropower in Existing Infrastructures, Hydropeaking Reduction, Digitalization and Governing Systems," *Sustainability*, vol. 12, no. 21, p. 8873, Oct. 2020, doi: 10.3390/su12218873.
[18] D. Bazzana, G. Gilioli, and B. Zaitchik, "Impact of hydropower development on rural livelihood: An agent-based exploration," *J Clean Prod*, vol. 275, p. 122333, Dec. 2020, doi: 10.1016/j.jclepro.2020.122333.
[19] L. Sun, D. Niu, K. Wang, and X. Xu, "Sustainable development pathways of hydropower in China: Interdisciplinary qualitative analysis and scenario-based system dynamics quantitative modeling," *J Clean Prod*, vol. 287, p. 125528, Mar. 2021, doi: 10.1016/j.jclepro.2020.125528.
[20] J. Geist, "Editorial: Green or red: Challenges for fish and freshwater biodiversity conservation related to hydropower," *Aquat Conserv*, vol. 31, no. 7, pp. 1551–1558, Jul. 2021, doi: 10.1002/aqc.3597.
[21] Y. Fan, D. Zhang, and J. Li, "A Control Scheme for Variable-Speed Micro-Hydropower Plants," *Sustainability*, vol. 10, no. 11, p. 4333, Nov. 2018, doi: 10.3390/su10114333.
[22] W. Paryatmo, *Turbin Air*. Universitas Pancasila , 2007.
[23] W. Arismunandar, "Penggerak Mula Turbin ITB." Bandung, 2004.
[24] C. Abeykoon and T. Hantsch, "Design and analysis of a Kaplan turbine runner wheel," *Proc. 3rd World Congr. Mech. Chem. Mater. Eng., Rome, Italy*, p. 1â, 2017.
[25] H. Akin *et al.*, "A CFD aided hydraulic turbine design methodology applied to Francis turbines," in *4th International Conference on Power Engineering, Energy and Electrical Drives*, IEEE, 2013, pp. 694–699.
[26] M. M. Nunes, R. C. F. Mendes, T. F. Oliveira, and A. C. P. Brasil Junior, "An experimental study on the diffuser-enhanced propeller hydrokinetic turbines," *Renew Energy*, vol. 133, pp. 840–848, Apr. 2019, doi: 10.1016/j.renene.2018.10.056.
[27] W.-Q. Wang, R. Yin, and Y. Yan, "Design and prediction hydrodynamic performance of horizontal axis micro-hydrokinetic river turbine," *Renew Energy*, vol. 133, pp. 91–102, Apr. 2019, doi: 10.1016/j.renene.2018.09.106.
[28] A. Ghimire, D. Dahal, A. Kayastha, S. Chitrakar, B. S. Thapa, and H. P. Neopane, "Design of Francis turbine for micro hydropower applications," in *Journal of Physics: Conference Series*, IOP Publishing, 2020, p. 12019.
[29] C. Abeykoon and T. Hantsch, "Design and Analysis of a Kaplan Turbine Runner Wheel," Jun. 2017. doi: 10.11159/htff17.151.
[30] H. Akin *et al.*, "A CFD aided hydraulic turbine design methodology applied to Francis turbines," in *4th International Conference on Power Engineering, Energy and Electrical Drives*, 2013, pp. 694–699. doi: 10.1109/PowerEng.2013.6635694.
[31] D. F, *Turbin Pompa dan Kompressor*. Erlangga, 1997.
[32] H. Lomax, T. H. Pulliam, D. W. Zingg, and T. A. Kowalewski, "Fundamentals of computational fluid dynamics," *Appl. Mech. Rev.*, vol. 55, no. 4, pp. B61–B61, 2002.
[33] M. Tabatabaian, *COMSOL5 for Engineers*. Mercury learning and information, 2015.
[34] I. Samora, V. Hasmatuchi, C. Münch-Alligné, M. J. Franca, A. J. Schleiss, and H. M. Ramos, "Energy production with a tubular propeller turbine," in *IOP Conference Series: Earth and Environmental Science*, Nov. 2016, p. 102001. doi: 10.1088/1755-1315/49/10/102001.
[35] Pribadyo, H. Hadiyanto, and J. Jamari, "Study of low head turbine propellers axial flow for use of micro-hydropower plant (MHP) in Aceh, Indonesia," *J PhysConfSer*, vol. 1524, no. 1, p. 012019, Apr. 2020, doi: 10.1088/1742-6596/1524/1/012019.