Advances in Technology Innovation, vol. 11, no. 1, 2026, pp. 90-100

Effect of Weir Height on Undershot Water Wheel Turbine Performance

Alfathi Adam, Luther Sule, Rustan Tarakka*

Department of Mechanical Engineering, Hasanuddin University, Gowa, Indonesia

Received 21 May 2025; received in revised form 22 October 2025; accepted 19 December 2025

DOI: https://doi.org/10.46604/aiti.2026.15168

Abstract

Undershot water wheel turbines are suitable for operation in low-elevation regions and locations with very low water head. Turbines with curved blade configurations are commonly implemented to optimize efficiency. This research examines how variations in weir height influence the performance of undershot turbines when operating under limited water flow conditions. A turbine with six blades is tested experimentally and numerically using seven weir heights ranging from 0.02 m to 0.14 m. The study used both experimental and computational fluid dynamics approaches, employing the dynamic mesh model. The results indicate that applying a curved weir as a passive flow-control device can significantly enhance turbine efficiency. At a weir height of 0.10 m, the maximum efficiency reached 80.83% based on the experimental approach and 84.69% based on the computational approach. These findings demonstrate the potential of weir-assisted undershot turbines for energy harvesting in shallow rivers under low-flow conditions.

Keywords: computational fluid dynamics (CFD), turbine efficiency, undershot turbine, weir height

1. Introduction

Renewable energy plays a crucial role in achieving global sustainability goals, with various technologies such as wind turbines and water turbines (both vertical and horizontal axis) under active development to meet the increasing demand. Among these, hydropower stands out because water covers approximately 65–75% of the Earth's surface and is about 800 times denser than air, enabling significantly higher energy output compared to wind-based systems [1]. Globally, the installed hydropower capacity has reached around 1360 GW, including 254 GW in Europe, contributing to an annual electricity output of approximately 4250 TWh [2]. Water is a vital element for human survival, especially in areas with river systems. The undershot water wheel turbine can be operated in lowland areas and at low-head waterfalls, making it highly suitable for use in river flows near residential areas. In particular, the curved blade design is commonly used in water turbines because it offers higher efficiency [3-4].

The development of water turbines involves varying the number of blades to achieve maximum efficiency. Research results indicate that the highest efficiency, 74.22%, is achieved with a turbine featuring 6 blades [5]. This performance difference is due to better water distribution, which increases torque and reduces friction on the blades. In this study, the 8-blade turbine produced slightly less power than the 6-blade turbine, while the 12-blade turbine had the lowest output. Experiments also show that aluminum, with lower density and greater thickness, gives higher efficiency than denser materials [6].

The speed at which a water turbine rotates is largely determined by variations in water head and flow rate. Similar studies have been conducted on the Sagebien and Zuppinger water turbine models to reduce the high take-off costs. One solution that

* Corresponding author. E-mail address: rustan_tarakka@yahoo.com

can be applied is the use of a weir, which can be adjusted to enhance turbine efficiency. The weir in an undershot water wheel functions to regulate the level or height of the water flowing into the wheel from below. By ensuring an optimal water height, the wheel can rotate more efficiently, generating maximum mechanical energy and reducing unnecessary energy losses [7-8]. However, previous studies have not investigated the role of adjustable weirs or dynamic flow control in undershot water wheel turbines. This study addresses this gap by analyzing the influence of weir height variation and validating the results using both experimental and computational fluid dynamics (CFD) approaches.

The geometry and slope angle of the weir face play a crucial role in optimizing the water flow directed to the wheel, allowing it to operate efficiently. The curved shape of the weir face, following the wheel's circumference, helps direct the water flow optimally to the blades, thereby enhancing the wheel's rotational efficiency [9]. The slope angle of the weir face also affects the speed and distribution of water flow, impacting the wheel. An overly steep angle can result in energy loss, while a too-gentle slope may reduce the thrust force on the blades. Therefore, the correct angle and geometry ensure that all blades receive a uniform water flow, optimizing the energy generated [10].

This strategy holds great promise as a solution to increase power in low-speed water flows, especially in conditions with relatively low flow rates. The development of such systems has significant potential in creating more efficient hydropower technologies in locations with limited water resources. However, to ensure its effectiveness, full-scale testing is required to further investigate the practical operation and performance of the system under real-world conditions.

One aspect that needs to be explored in full-scale trials is the use of an adjustable weir. While this concept promises efficiency improvements, its implementation presents technical challenges that need to be addressed, such as designing a mechanism that allows the weir to be adjusted without disrupting the water flow. The dynamic adjustment of the weir in response to changes in flow rate requires a reliable mechanical system and efficient maintenance procedures to ensure optimal long-term performance.

Through full-scale testing and overcoming these technical challenges, a better understanding of system behavior under various operational scenarios can be achieved. The results from this research and testing will be invaluable in developing more practical and efficient solutions, enabling more optimal and sustainable utilization of water resources in low-flow regions.

Previous studies have investigated the use of weirs to improve undershot water wheel performance. Nevertheless, most of these works did not systematically examine the effects of different weir heights or validate their findings through both experimental measurements and CFD simulations. The novelty of this study lies in the comprehensive evaluation of seven weir height configurations (0.02–0.14 m), supported by both experimental and numerical validation. From a practical perspective, the weir in this study functions as a passive flow-control device. Unlike nozzles, which can provide precise regulation but require expensive and complex construction, the weir offers a more straightforward and cost-effective alternative for directing water toward the turbine blades. Although its installation requires additional work beneath the channel, it enables undershot turbines to harness low river discharges more effectively, which is particularly valuable for rural applications and during dry-season low-flow conditions. Therefore, the objective of this study is to evaluate the influence of weir height variations on the performance of a six-blade undershot turbine using both experimental and CFD approaches.

2. Methodology

This research was carried out to assess the effectiveness of an undershot flow-driven water wheel with specially designed curved blades. The turbine is designed with six blades, each having a thickness of 0.002 m. With a total mass of 2.75 kg, the turbine features an outer span of 0.5 m and an inner core diameter of 0.02 m. These specifications were selected to optimize the turbine's ability to generate energy from the water flow.

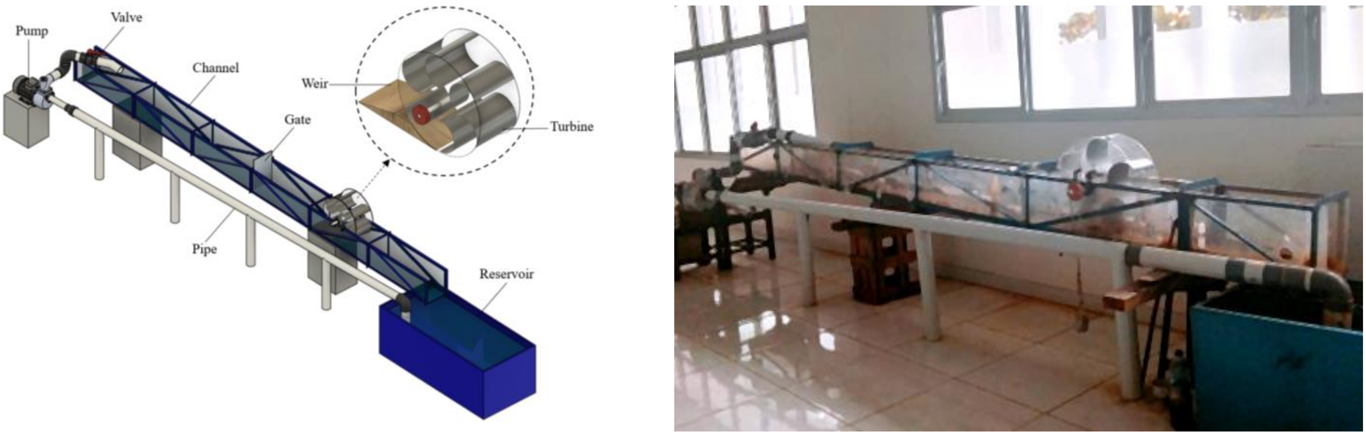
The variation in weir height was applied to examine its effect on the water flow entering the turbine. This study utilized weir heights varying from 0.02 m to 0.14 m, with incremental steps of 0.02 m, resulting in a total of seven variations. Each weir was designed with an upstream slope angle of 15° [11-12] to maintain the uniformity of the flow characteristics. The weir crest measured 0.1 m in length, and its downstream face was designed with a 0.25 m curvature radius to match the turbine's contour, thereby optimizing water flow toward the blades.

The research process was conducted using two approaches: experimental and computational. The experimental test setup is shown in Fig. 1. In this experiment, variations in weir height were tested while keeping the turbine operating at fixed rotational speeds ranging from 30 rpm down to 10 rpm in decrements of 5 rpm.

For each variation in weir height tested, the turbine was loaded by suspending a weight on the turbine shaft. A series of weights ranging from 0.5 kg to 2.5 kg was applied in steps of 0.5 kg to test the system's performance. The purpose of applying these weights was to observe the changes in torque occurring in the water turbine due to the differences in load applied under each weir height variation.

The water flow rate in the channel was adjusted by gradually opening or closing the water valve until the turbine's rotation reached the desired speed, approximately 0.0080 to 0.0116 m^3/s. The discharge was determined using the volume collection method, in which the time required to fill a 20 L container was recorded with a digital stopwatch (0.01 s resolution). The container volume accuracy was estimated at ±0.2 L (±1%), while the human reaction time in operating the stopwatch introduced an additional uncertainty of about ±0.2 s.

The overall uncertainty of the flow rate measurement was therefore estimated at approximately ±2%. The turbine speed was measured using a handheld laser digital tachometer, with a manufacturer-stated accuracy of ±0.05%. Torque and output power were subsequently calculated from the measured speed and the applied shaft load. Fig. 1 shows the experimental setup, complemented by a photograph of the prototype installation to provide a clearer representation of the configuration.



(a) Schematic diagram of the experimental setup (b) Photograph of the prototype installation

Fig. 1 Overview of the experimental setup

The computational approach in this study is conducted to investigate the fluid dynamics phenomena in the water turbine. The goal is to understand the flow characteristics, the generated torque, and the visualization of flow patterns. In the initial stage, a 3D symmetric geometry model of the water turbine was created using Fusion 360 software. This modeling was performed by considering the dimensions and shape of the turbine to match real-world conditions [13].

The model was then imported into Ansys Fluent software. The simulation domain was divided into two main sections: the rotating zone surrounding the turbine and the stationary zone, which includes the water channel, inlet, outlet, and channel walls [14-15]. The domain scheme and the boundary conditions applied are presented in Fig. 2.

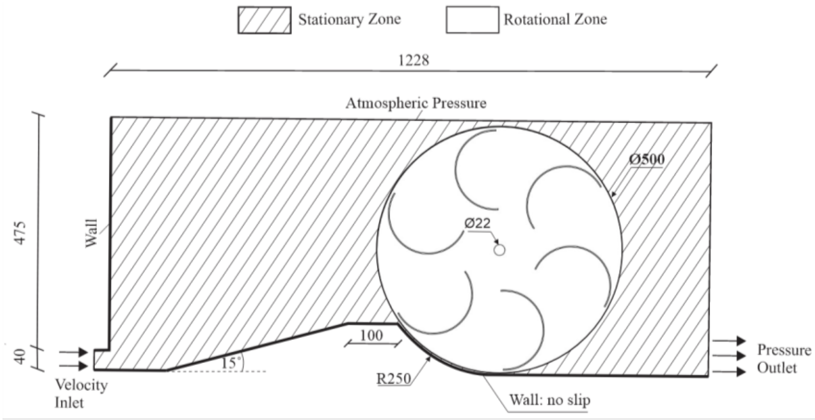


Fig. 2 Schematic of the domain and boundary conditions

Meshing was performed using polyhedral elements, with element sizes ranging from 0.001 to 0.01 m depending on the geometric complexity of each region. To ensure numerical accuracy, a grid independence test was conducted using five mesh densities ranging from approximately 0.3 million to 1.45 million elements. As shown in Fig. 3, the predicted turbine torque converged as the mesh density increased. The difference in torque between the medium mesh (832455 elements) and the fine mesh (approximately 1.45 million elements) was less than 0.2%, confirming that the medium mesh was sufficient and was therefore adopted for all subsequent simulations.

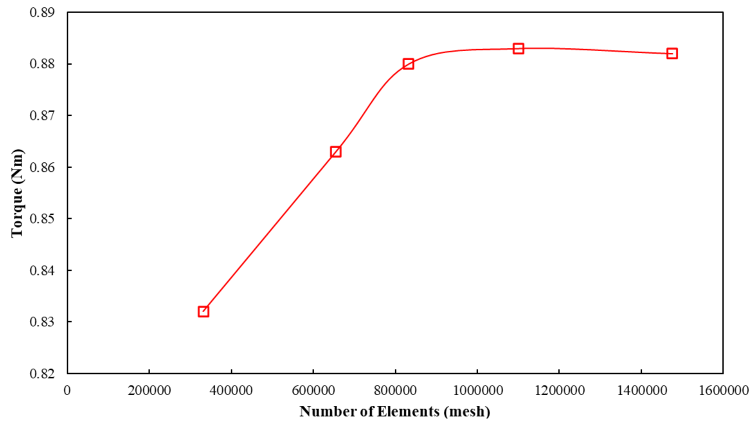


Fig. 3 Grid independence test: torque variation with mesh element size

The boundary conditions in the CFD setup were made consistent with the experimental conditions. The inlet velocity was prescribed according to the measured flow rates in the experiment (0.0065–0.0075 m^3/s), while the turbine angular velocity was set in the range of 10–30 rpm, consistent with the experimental operating conditions. The outlet was defined as a pressure outlet at atmospheric pressure, and all walls were modeled as no-slip boundaries [16-17].

The simulation was performed under unsteady conditions utilizing the VOF-based multiphase modeling [18]. To model the two-phase fluid flow consisting of water and air, the SST k-ω turbulence model was adopted, as it is well-suited for simulating turbulent flows around rotating bodies [19-20]. A dynamic mesh was incorporated to enable the turbine to rotate at a defined angular velocity throughout the simulation [21-22]. The torque produced by the turbine was calculated using the report solution feature. A total of 1000 time steps were executed in the simulation, each having a duration of 0.003 seconds.

Validation of the CFD model was conducted by comparing the predicted torque with the experimental results. At the medium mesh density, the CFD simulation produced a torque of 0.88 Nm, which showed good agreement with the experimental value of 0.83 Nm. The deviation between the simulation and the experiment ranged from 3.06% to 5.81%, which is within an acceptable range for numerical simulations of rotating flows. This agreement indicates that the CFD model is capable of reliably reproducing the hydrodynamic behavior of the turbine and can be used to complement the experimental findings in evaluating the influence of weir height variations.

Turbine efficiency indicates the fraction of the motion energy of the flowing water that is effectively converted into mechanical power to rotate the turbine [23]. In this research, the mechanical power produced by the turbine is calculated from the torque generated by the flowing water:

$$P_b = T \cdot \omega \tag{1}$$

where P_b represents the turbine power (Watts), T denotes the torque (Nm), and ω is the angular velocity of the turbine's rotation (rad/s). The efficiency of the turbine can be determined by the following formula:

$$\eta = \frac{P_b}{P_{in}} \times 100\% \tag{2}$$

where P_{in} represents the kinetic power of the incoming water flow, calculated based on the flow rate and velocity. This efficiency reflects the proportion of kinetic energy that can be utilized to generate torque and rotation in the turbine.

3. Results and Discussion

The correlation between power and turbine efficiency shows that efficiency tends to increase as turbine speed increases, with maximum efficiency being achieved at 30 rpm. The application of a weir also has a significant impact on turbine efficiency. According to the results presented in Table 1, it is evident that no weir (0 m) results in relatively low turbine efficiency, at 35.44% based on the computational results and 33.71% based on experimental measurements. Increasing the weir height consistently improves efficiency until it reaches an optimal height of 0.10 m, where the efficiency reaches 84.69% in the computation and 80.83% in the experiments. This improvement is mainly due to the weir's ability to guide the incoming flow more uniformly across the turbine blades, thereby enhancing torque transfer and reducing hydraulic losses. Beyond this optimal point, efficiency begins to decrease, as seen at weir heights of 0.12 m and 0.14 m. The reduction is caused by excessive upstream water accumulation, which creates a non-uniform velocity distribution and turbulence, leading to suboptimal blade loading and additional energy losses.

Table 1 Relationship between turbine power and turbine efficiency at 30 rpm with variations in weir height

Weir height	Flow rate	Turbine power		Turbine efficiency		Deviation
		Experiment	CFD	Experiment	CFD	
(m)	(m^3/s)	(watt)	(watt)	(%)	(%)	(%)
No weir	0.0116	2.62	2.75	33.71	35.44	4.86
0.02	0.0096	2.62	2.70	39.69	40.94	3.06
0.04	0.0087	2.62	2.73	44.89	46.84	4.17
0.06	0.0083	2.62	2.78	51.30	54.47	5.81
0.08	0.0076	2.62	2.76	66.15	69.64	5.02
0.10	0.0071	2.62	2.74	80.83	84.69	4.57
0.12	0.0074	2.62	2.76	72.80	76.79	5.20
0.14	0.0080	2.62	2.73	60.28	62.92	4.20

In addition, the trend of the curved graphs for each variation in weir height, as shown in Fig. 4, reveals that an increase in turbine efficiency does not always lead to a linear increase in turbine power. This phenomenon indicates a limitation in the relationship between efficiency and turbine power. In specific conditions, raising the weir height can enhance efficiency in changing water's energy into mechanical work, as the flow pattern becomes more streamlined and energy losses caused by turbulence are minimized.

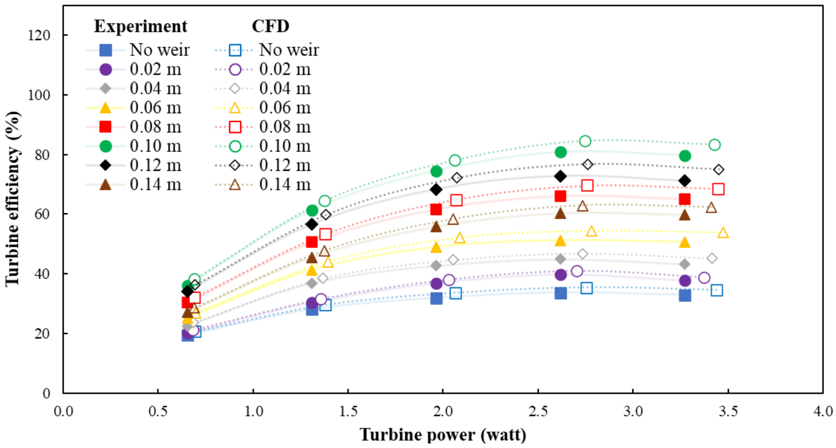


Fig. 4 Relationship between turbine power and efficiency at 30 rpm

However, variations in weir height also influence the velocity profile and the effective flow rate of water reaching the turbine blades. The irregular velocity distribution and fluctuations in effective flow result in the total kinetic energy available for conversion into mechanical power not increasing in direct proportion, despite the improvement in energy efficiency. In some cases, an increase in efficiency occurs alongside a reduction in incoming flow, meaning the total output power may not increase significantly or may even decrease. Therefore, the turbine performance in this study demonstrates a trade-off between efficiency and power, where optimizing one parameter does not always result in a proportional improvement in the other.

The relationship between weir height and turbine efficiency can be seen in Fig. 5. At higher weir heights, the potential energy of the water is also greater. This means that water falling from a higher elevation carries more energy, allowing the turbine to generate more power. Additionally, although the water flow slightly decreases with an increase in weir height, this condition helps the turbine operate more efficiently. The slightly lower flow makes the water stream more stable and controlled, reducing energy waste and mechanical losses occurring in the turbine.

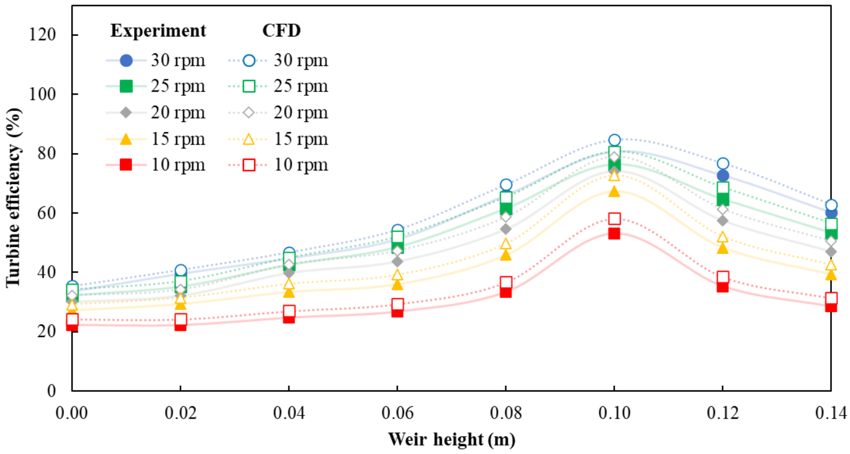


Fig. 5 Relationship between weir height and turbine efficiency at different rotational speeds

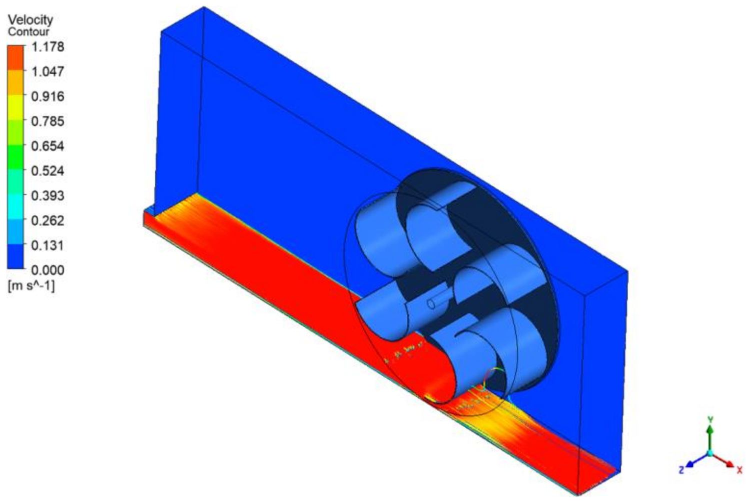
When compared to previous studies, the performance of the present turbine design is superior. For example, the work reported in [5] achieved a maximum efficiency of 74.22% without the use of a weir, whereas the curved weir configuration in this study reached 84.69%. This nearly 10 percentage point increase highlights the novelty and effectiveness of incorporating a weir to optimize undershot turbine operation under low-head conditions. Unlike the study by Helmizar, which developed a turbine wheel for medium-head conditions between 2.5 and 5 m, the present work focuses on ultra-low-head flows (<0.2 m). This contrast highlights that while Helmizar's design is suitable for small-scale hydropower sites with higher head differences, the curved weir approach in this study offers an effective solution for shallow rivers where conventional turbines are less applicable [24].

The increase in efficiency at higher weir heights is also attributed to the way the turbine converts energy. When water falls from a higher elevation, the turbine can better utilize that energy because the higher water pressure enables the turbine to rotate faster and more efficiently. Therefore, even though the water flow decreases, the turbine can more effectively harness the available energy.

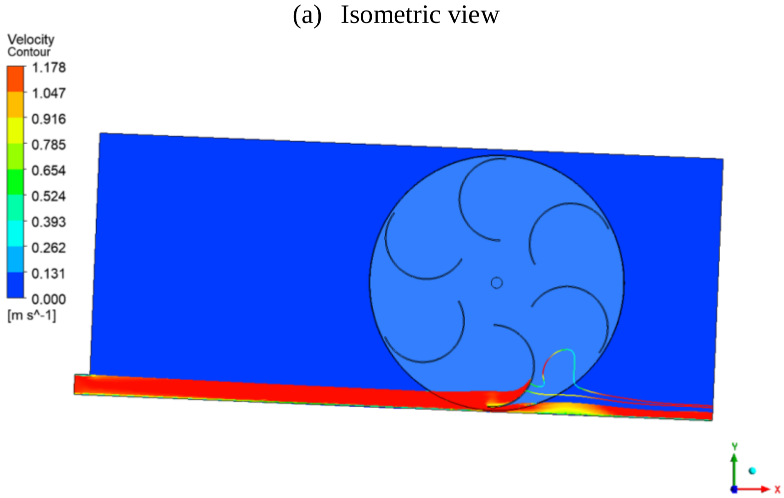
This higher efficiency can also occur because, at higher weir heights, the design and performance of the turbine are more optimal in handling a steady and controlled water flow. The application of a higher weir acts as an excellent flow control, helping to improve turbine efficiency. With the increased water pressure due to the higher weir, the turbine can more effectively utilize energy, allowing it to spin faster and more efficiently.

Turbine efficiency reaches its maximum value at a weir height of 0.10 m. After this point, at 0.12 m and 0.14 m, efficiency begins to decline due to increased flow resistance and energy distribution imbalance, requiring a larger water flow to maintain the same turbine rotation. From a practical point of view, however, a weir higher than 0.10 m is not recommended in rivers with high sedimentation, as sediment buildup upstream may block the flow and reduce long-term reliability. Thus, 0.10 m can be considered the optimal height for balancing efficiency and feasibility.

The flow visualization in the undershot water turbine with no weir is shown in Fig. 6. The simulation results indicate a stable flow with a uniform velocity distribution along the channel. This flow stability is crucial to ensure that the water flows smoothly toward the turbine blades without disruptive turbulence. Irregular flow can reduce the efficiency of energy conversion and increase energy losses due to friction and vortices.



(a) Isometric view

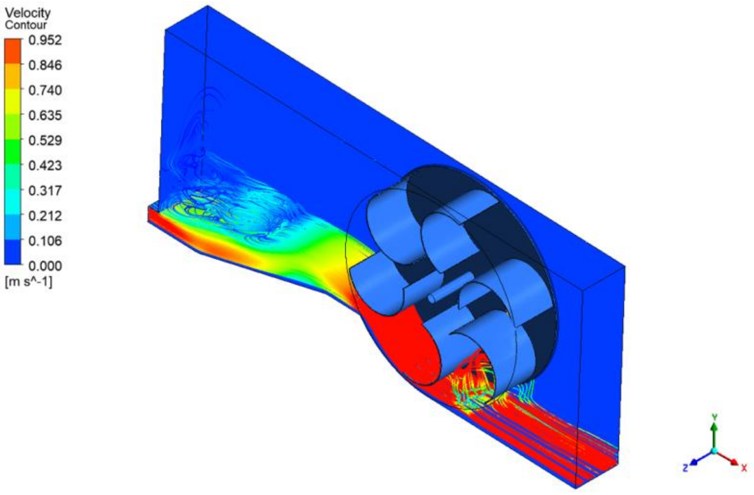


(b) Side view

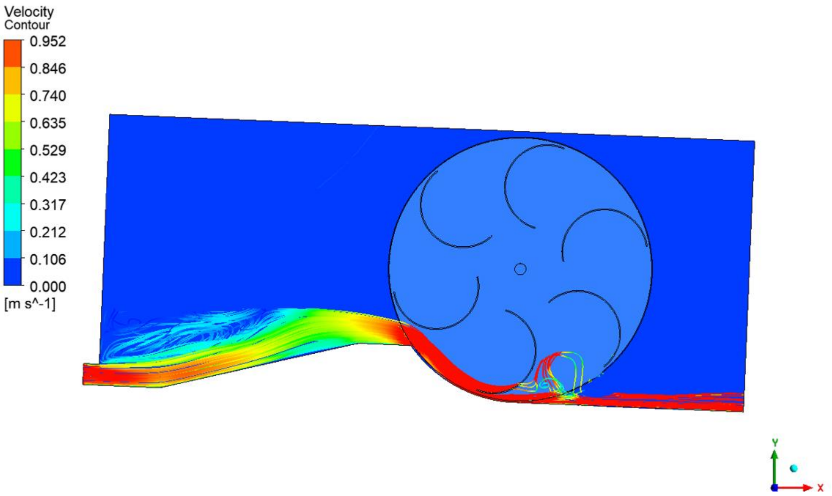
Fig. 6 Velocity contour of the undershot water turbine without weir (color scale in m/s)

The velocity contours, represented as streamlines, show the direction and speed of the flow along the channel, making it easier to understand how water moves through the turbine and allowing for the identification of regions with both high and low speeds, as well as the potential for vortex formation. The red contour indicates the initial velocity of water provided to the turbine. With a water velocity of 1.178 m/s, the flow remains stable, exhibiting a uniform velocity profile along the channel, which facilitates smooth water movement toward the turbine blades. However, when the water strikes the turbine blades, there is a noticeable decrease in velocity, particularly behind the blades, marked by a change in the contours to green, orange, and yellow, ranging from 0.654 to 1.047 m/s. This decrease in velocity occurs because, after passing through the blades, the flow experiences a sudden change in direction, causing some of the flow to lose energy and slow down. In addition, small vortices form behind the blades, which eventually merge back with the main flow. After a short time, the velocity of the water returns to normal, as indicated by the contours turning red again.

In Fig. 7, showing a weir height of 0.10 m, an increase in turbulence is observed before the flow reaches the turbine. This is caused by pressure differences creating a backflow area. Although turbulence increases, the flow entering the blades remains stable and does not disrupt energy conversion.



(a) Isometric view



(b) Side view

Fig. 7 Velocity contour of the undershot water turbine with weir height 0.10 m (color scale in m/s)

The red contour represents a water velocity of 0.952 m/s, corresponding to the initial velocity set in the simulation. The resulting turbulence, which becomes more evident, shows that higher velocity reduction creates a backflow area upstream of the weir, marked by contours shifting from yellow to blue, ranging from 0.212 to 0.740 m/s.

However, when the flow reaches the crest of the weir, the water velocity increases again, indicated by the contour returning to red as it strikes the blades. This further confirms that the application of a weir not only enhances low water potential but also serves as a flow control that optimizes turbine performance. Turbulence develops behind the blades as the water velocity changes upon impact, indicated by the contour shifting to green, with velocities between 0.529 and 0.740 m/s. Subsequently, the velocity of the water begins to stabilize, as shown by the contour returning to red, indicating a more stable flow.

The turbine maintains its efficiency as the kinetic energy of the incoming flow remains sufficient to generate power. In general, although increasing the weir height tends to elevate turbulence and vortex formation behind the blades, it simultaneously enhances the stability of the flow approaching the blades. This improved stability supports efficient turbine operation. Moreover, a higher weir height reduces the required water discharge to achieve the same power output, allowing the system to remain effective even under lower flow conditions.

A similar conclusion was reported by Quaranta and G. Müller [7] and by Quaranta and Revelli [8], who emphasized the significance of appropriate weir configurations in improving the operational performance of turbines. In undershot water wheel systems, the weir functions to regulate the upstream water level feeding the wheel from below [7]. Maintaining an optimal water level enables a more effective conversion of potential energy into mechanical energy. A controlled and directed flow not only improves rotational efficiency but also reduces energy losses caused by flow disturbances or turbulence [8].

4. Conclusions

The performance of the turbine was evaluated through both computational and experimental approaches by varying the weir height. The results demonstrated that the presence of a weir significantly influences flow behavior and enhances the efficiency of energy conversion.

(1) The implementation of a weir improves turbine efficiency by increasing upstream water pressure and concentrating the flow direction toward the turbine blades. As the weir height increases, turbine efficiency increases until an optimal condition is reached.

(2) The highest efficiency was achieved at a weir height of 0.10 m, reaching 84.69% in the computational analysis and 80.83% in the experimental results at a flow rate of 0.0071 m^3/s. Beyond this height, efficiency tends to stagnate or decrease due to increased flow resistance.

(3) Based on the performance results, a weir height of 0.10 m with a slope angle of 15° and a downstream curvature radius of 0.25 m is therefore recommended. The curvature radius, which corresponds to half of the turbine diameter, allows the weir contour to follow the turbine geometry and effectively direct the flow toward the blades.

(4) From a practical perspective, the weir acts as a passive flow-control device and offers a simpler and more cost-effective alternative to nozzles, which require more complex and expensive construction. This makes the proposed weir design suitable for undershot turbines operating in shallow rivers with low flow velocities, particularly in rural areas and during dry-season conditions.

For future work, the development of dynamically adjustable weirs is recommended to enable the turbine to adapt to varying river discharges under seasonal fluctuations.

Acknowledgments

The authors extend their gratitude to the Head and Staff of the Fluid Machinery Laboratory, Faculty of Engineering, Hasanuddin University.

Conflicts of Interest

The authors declare no conflict of interest.

References

[1] C. H. Achebe, O. C. Okafor, and E. N. Obika, "Design and Implementation of a Crossflow Turbine for Pico Hydropower Electricity Generation," Heliyon, vol. 6, no. 7, article no. e04523, 2020.

[2] E. Quaranta, "Lubricant Oil Consumption and Opportunities for Oil-Free Turbines in the Hydropower Sector: A European Assessment," Energies, vol. 16, no. 2, article no. 834, 2023.

[3] B. Kirke, "Hydrokinetic Turbines for Moderate Sized Rivers," Energy for Sustainable Development, vol. 58, pp. 182-195, 2020.

[4] G. Macara, M. Capelo, J. Ferreira, and D. Covas, "Further Experimental Analysis of Undershot Water Wheels Towards the Development of a Prototype Model," Urban Water Journal, vol. 22, no. 1, pp. 1-23, 2024.

[5] L. Sule, A. A. Mochtar, and O. Sutresman, "Performance of Undershot Water Wheel with Bowl-Shaped Blades Model," International Journal of Technology, vol. 11, no. 2, pp. 278-287, 2020.

[6] E. Y. Setyawana, D. H. Praswanto, and M. Bahri, "Effect of Blade Materials on the Undershot Water Turbine Performance," Journal of Applied Research and Technology, vol. 22, no. 2, pp. 266-273, 2024.

[7] E. Quaranta and G. Müller, "Optimization of Undershot Water Wheels in Very Low and Variable Flow Rate Applications," Journal of Hydraulic Research, vol. 58, no. 3, pp. 1-5, 2019.

[8] E. Quaranta and R. Revelli, "Gravity Water Wheels as a Micro Hydropower Energy Source: A Review Based on Historic Data, Design Methods, Efficiencies and Modern Optimizations," Renewable and Sustainable Energy Reviews, vol. 97, pp. 414-427, 2018.

[9] E. Quaranta and R. Revelli, "Optimization of Breastshot Water Wheels Performance Using Different Inflow Configurations," Renewable Energy, vol. 97, pp. 243-251, 2016.

[10] D. Adanta, Budiarso, and Warjito, "The Effect of Channel Slope Angle on Breastshot Waterwheel Turbine Performance by Numerical Method," Energy Reports, vol. 6, pp. 606-610, 2020.

[11] F. Malekzadeh, F. Salmasi, J. P. Abraham, and H. Arvanaghi, "Numerical Investigation of the Effect of Geometric Parameters on Discharge Coefficients for Broad-Crested Weirs with Sloped Upstream and Downstream Faces," Appl Water Science, vol. 12, no. 5, article no. 110, 2022.

[12] L. Jiang, M. Diao, H. Sun, and Y. Ren, "Numerical Modeling of Flow Over a Rectangular Broad-Crested Weir with a Sloped Upstream Face," Water, vol. 10, no. 11, article no. 1663, 2018.

[13] M. Asim, S. Muhammad, M. Amjad, M. Abdullah, M. A. Mujtaba, M. A. Kalam, et al., "Design and Parametric Optimization of the High-Speed Pico Waterwheel for Rural Electrification of Pakistan," Sustainability, vol. 14, no. 11, article no. 6930, 2022.

[14] E. Quaranta and R. Revelli, "CFD Simulations to Optimize the Blade Design of Water Wheels," Drinking Water Engineering and Science, vol. 10, no. 1, pp. 27-32, 2017.

[15] D. Adanta, D. P. Sari, I. Syofii, K. Sahim, E. Martides, Y. Radiansah, et al., "Pico Scale Undershot Waterwheel for Ultra-Low-Head: Analytical, Experimental and CFD Method," Renewable Energy Focus, vol. 48, article no. 100532, 2024.

[16] K. H. Kim, D. H. Kim, S. J. Hong, and S. M. Lee, "Computational Fluid Dynamics Analysis and Validation with Field Test of 1 MW Hydropower Turbine System," Energies, vol. 18, no. 3, article no. 628, 2025.

[17] R. Pienika, G. Usera, and H. M. Ramos, "Simulation of a Hydrostatic Pressure Machine with Caffa3D Solver: Numerical Model Characterization and Evaluation," Water, vol. 12, no. 9, article no. 2419, 2020.

[18] A. Benavides-Morán, L. Rodríguez-Jaime, and S. Laín, "Numerical Investigation of the Performance, Hydrodynamics, and Free-Surface Effects in Unsteady Flow of a Horizontal Axis Hydrokinetic Turbine," Processes, vol. 10, no. 1, article no. 69, 2022.

[19] C. S. Bang, Z. A. Rana, and S. A. Prince, "CFD Analysis on Novel Vertical Axis Wind Turbine (VAWT)," Machines, vol. 12, no. 11, article no. 800, 2024.

[20] R. V. C. Ramalho, M. J. M. Pereira Filho, M. J. S. Sena, R. L. S. G. Mendes, S. U. Neto, D. E. S. e Souza, et al., "Design of Low-Cost Axial-Flow Turbines for Very Low-Head Micro-Hydropower Plants," Processes, vol. 13, no. 6, article no. 1865, 2025.

[21] J. Betancour, L. Velásquez, A. Rubio-Clemente, and E. Chica, "Performance Simulation of Water Turbines by Using 6-DoF UDF and Sliding Mesh Methods," Journal of Applied Research and Technology, vol. 21, no. 2, pp. 181-195, 2023.

[22] M. Zhao, Y. Zheng, C. Yang, Y. Zhang, and Q. Tang, "Performance Investigation of the Immersed Depth Effects on a Water Wheel Using Experimental and Numerical Analyses," Water, vol. 12, no. 4, article no. 982, 2020.

[23] O. R. Alomar, H. M. Abd, M. M. M. Salih, and F. Aziz Ali, "Performance Analysis of Pelton Turbine Under Different Operating Conditions: An Experimental Study," Ain Shams Engineering Journal, vol. 13, no. 4, article no. 101684, 2022.

[24] H. Helmizar, "Turbine Wheel - A Hydropower Converter for Head Differences Between 2.5 and 5 m," Ph.D. Dissertation, Faculty of Engineering and the Environment, University of Southampton, 2016.