Computational Fluid Dynamics of Savonius Water Turbine for Hydrokinetic Pico Hydro Systems

Rinasa Agistya Anugrah[1*], *Fajar* Yuafiq[2], *Mirza* Yusuf[1], *Mohamad Heerwan Bin* Peeie[3], and *Aan* Yudianto[4]

[1]Department of Automotive Engineering Technology, Faculty of Engineering, 55183 Universitas Muhammadiyah Yogyakarta, Indonesia.
[2]Student of Department of Automotive Engineering Technology, Faculty of Engineering, 55183 Universitas Muhammadiyah Yogyakarta, Indonesia.
[3]Faculty of Mechanical & Automotive Engineering Technology, Universiti Malaysia Pahang Al-Sultan Abdullah, Pahang, Malaysia.
[4]Department of Mechanical, Aerospace and Civil Engineering, The University of Manchester, Manchester, United Kingdom

Abstract. Hydrokinetic pico hydro systems reflect an expectant route for exploiting renewable energy from lowly water flows. The Savonius water turbine, known for its self-starting power and modest construction which is particularly equal for such several applications in agricultural sectors for example irrigation pump, smart farming system, and post-harvest processing reduce operational costs and carbon emissions. This simulation study serves a Computational Fluid Dynamics (CFD) analysis on a four-bladed Savonius water turbine by utilizing Ansys Fluent 2024 RI Software. A three-dimensional turbine model was developed and divided into more than 2.7 million small parts to ensure accurate simulation results. Five types of inlet velocities—0.5 m/s, 0.75 m/s, 1.0 m/s, 1.25 m/s, and 1.5 m/s—were tested to see how they affect the flow shape, flow course, and turbulence level. The results show that the higher the inlet speed, the greater the speed change, the more complex the flow shape, and the higher the turbulence level, especially in the central area of the turbine. When the speed is low, the flow remains calm with a small flow separation, while high speed causes the flow to become unstable and a larger flow mixing occurs.

1 Introduction

The acceleration of the global transition to renewable energy has triggered an increase in research focused on significantly developing efficient technologies to sustainably utilize local resources [1]. In addition, this also minimizes the existing environmental impact. Picohydro hydrokinetic systems that generally operate with power capacities below 5 kW among various clean energy alternatives show great potential as innovative solutions for rural electrification [2] and environmentally friendly decentralized energy provision [3]. These

* Corresponding author: rinasaanugrah@umy.ac.id

systems harness the kinetic energy of free-flowing water without the need for large-scale infrastructure such as dams, making them suitable for low-head rivers, irrigation canals, and tidal channels.

In agriculture, water turbines can be applied in several practical and sustainable ways. One of the main applications is pico-hydropower generation. The turbines are installed in irrigation canals or small rivers produce electricity for lighting, water pumps, greenhouses, barns, and post-harvest facilities. This provides a reliable off-grid energy source for rural farming areas. Water turbines can also be used as mechanical or electrical drivers for irrigation pumps, enabling water distribution to higher or distant fields without relying on diesel fuel or grid electricity. This application is especially beneficial in remote agricultural regions with continuous water flow. In modern farming systems, small-scale water turbines support smart agriculture technologies by supplying power to soil moisture sensors, water level controllers, automatic gates, and IoT-based monitoring systems. Even low-power turbines are sufficient to operate these devices continuously. Additionally, turbine-generated energy can be utilized in post-harvest processing, such as rice milling, grain drying, feed chopping, and small agro-processing units, helping farmers reduce operational costs and carbon emissions. Water turbines may also support aquaculture systems by powering aerators and water circulation in fishponds. Overall, water turbines offer a renewable, low-emission, and cost-effective energy solution for agriculture, enhancing energy independence, productivity, and sustainability, particularly in areas with stable irrigation or natural water flows.

Savonius water turbines feature superior characteristics in the hydrokinetic category, which include self-start capabilities, omnidirectional operation, as well as simple constructions that can be made from locally sourced materials [4]. In addition to these advantages, Savonius turbines generally exhibit a lower power coefficient compared to lift-based turbines. This condition then prompted various studies focusing on geometric modifications, flow control devices and hybridization strategies to improve aerodynamic performance and energy efficiency [5], [6]. Efforts to improve the performance of Savonius turbines have been the focus of several recent studies that integrate experimental approaches and numerical simulations. Sanjaya et al. [7] had reported which the installation of a type D cylinder with an angle of 53° on the upstream side was able to increase the maximum power coefficient by up to 24.56% with the results of Computational Fluid Dynamics (CFD) for showing predictions a good fit for the experimental data. Meanwhile, Cabalo and Marcelo [8] had optimized the Savonius hydrokinetic turbine for irrigation pipeline applications in the Philippines through CFD simulations and found that a three-blade configuration with an overlap ratio of 0.05 resulted in the highest torque and power coefficients among the tested variations.

Muhyiddin et al. [3] had conducted a paper review of CFD simulations on the vertical axis hydrokinetic turbines (VAHKTs) by identifying performance assessment, numerical insights, and parameter optimization as the main research themes. In another similar study, Muhyiddin et al. [9] also highlights research topics focusing on blade profile geometry, overlap ratios, and rotor staging configurations in the development of Savonius hydrokinetic turbines (SHT). In addition, Zakaria and Ibrahim [10] through a study of the interaction of the build and the multi-rotor arrangement showed that the optimal placement of the turbine in an array can improve the performance of the system by up to 11%, due to the vortex interaction formed between the rotors. Then research from the material and structural side there was Zakaria A. et al. who found that aluminum provides an optimal balance between mechanical strength, light mass, and corrosion resistance, so it is suitable for use as a material for Savonius turbine blades. Furthermore, another study [11] investigating the effect of blade count on VAHKT showed that increasing blade count can increase torque and output power, as a larger surface area allows for more efficient fluid momentum transfer against rotors.

The comparative Computational Fluid Dynamics (CFD) approach has proven that three-dimensional simulations are able to represent the characteristics and complexity of fluid flow more accurately than two-dimensional simulations, resulting in more reliable estimates of torque and power coefficients [12]. A hybrid configuration concept integrating the Savonius and Darrieus rotors has been developed and evaluated to improve self-start capabilities and maintain energy conversion efficiency under a wide range of flow speed conditions [6]. In the context of turbine design and optimization research, CFDs serve as an essential analytical instrument, as they allow in-depth observations of velocity distributions, turbulence patterns, and flow wake dynamics [8], [12]. Furthermore, CFD provides the ability to evaluate complex interactions between hard-to-reach flow structures through experimental approaches, as well as support systematic parametric studies in assessing the influence of blade geometry, overlap ratios, and variations in operating conditions [13], [14]. In addition, recent CFD research has confirmed the importance of the phenomena of vortex formation, flow separation, and turbulence intensity in determining the torque performance and power efficiency of turbines [8], [6].

This study aims to provide a high-resolution CFD simulation and analysis of a Savonius-type water turbine to be applied to a pico-scale hydrokinetic power, by using Ansys Fluent 2024 R1 software. The three-dimensional model of the turbine is mesh with more than 2.7 million elements to ensure high numerical accuracy. The simulation was carried out in a transient condition with a realized k–ε turbulence model. Therefore, it was able to represent the characteristics of the flow more realistically. The focus of this investigation includes the contour analysis of velocity, flow smoothness, and turbulence intensity, which aims to provide an in-depth understanding of hydrodynamic behavior and support turbine design optimization for sustainable small-scale hydropower plants. This study uses the CFD method which provides important information to improve the performance of Savonius turbines in the use of small hydroelectric power plants, which supports the development of sustainable power generation technologies especially in rural area and agricultural sectors.

2 Research Methods

This study uses the CFD numerical simulation method to analyze the hydrodynamic performance of Savonius water turbines, designed for hydro-pico generation applications. The technique consists of three main stages: model development, numerical simulation setup, and analysis of simulation results and validation.

2.1 Turbine Geometry and Model Development

The three-dimensional model of the conventional four-blade Savonius, shown in Figure 1, was created using the SolidWorks CAD software platform. The turbine design is based on standard geometric parameters researched by other researchers in previous literature [8], including blade height, diameter, and overlap ratios, which are suitable for low-speed water flows but with different configurations and variations. The computational domain encompasses inlet, outlet, and side boundaries to represent a controlled waterway environment with sufficient clearance between turbines, as well as domain walls to minimize the effect of blockage. Additionally, we installed a pair of deflectors in front of it to enhance its efficiency, as previous research has shown that this has a positive effect [15].

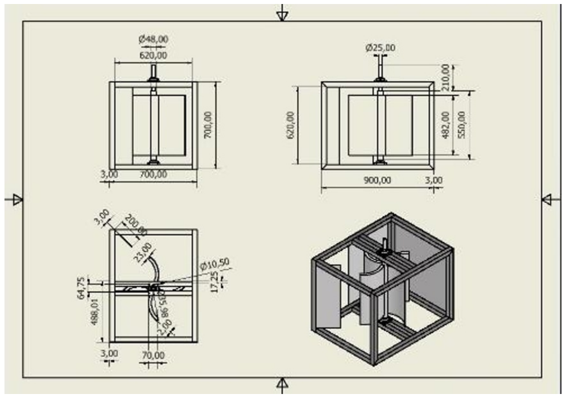


Fig. 1. Geometry of Savonius water turbine

2.2 Mesh Generation

The turbine model was discretized using an unstructured hexahedral mesh with local refinement around the blade surfaces to accurately capture boundary layer behavior and flow separation phenomena. Additional mesh refinement was applied in the wake region to resolve vortex structures and turbulence effects. The final mesh contained more than 2.7 million elements, which was determined through a mesh independence study to ensure accuracy while maintaining computational efficiency (Figure 2).

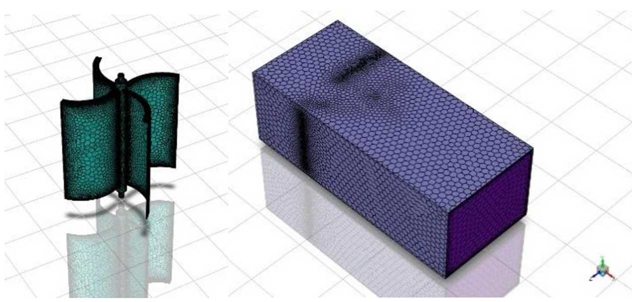


Fig. 2. Meshing details of computational domain with blade profile

2.3 Governing Equations and Turbulence Model

The simulations were conducted using Ansys Fluent 2024 R1, solving the three-dimensional, transient Reynolds-Averaged Navier–Stokes (RANS) equations for incompressible flow. The realizable k–ε turbulence model was selected for its proven capability in capturing rotational flows and predicting separation around bluff bodies [7], [5]. The governing equations were discretized using a second-order upwind scheme for momentum and turbulence quantities, and a first-order implicit time-stepping scheme for temporal discretization.

2.4 Boundary Conditions

A uniform velocity inlet was applied at the upstream boundary (Figure 3), with water velocity values representative of low-speed hydrokinetic sites (0.5–1.5 m/s). The outlet boundary was set to a constant static pressure of 0 Pa (gauge). The turbine shaft was modeled using a moving mesh approach with a specified rotational speed to replicate realistic turbine operation. Blade surfaces were treated as no-slip walls, and the domain side walls were specified as symmetry boundaries to reduce artificial flow confinement.

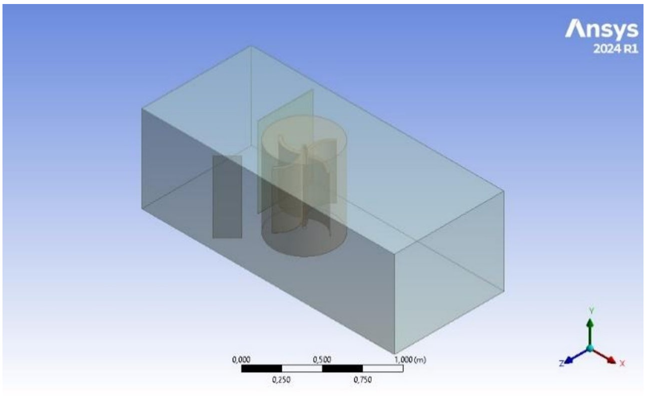


Fig. 3. Rotating region and domain

2.5 Solution Strategy

In the solution strategy, the research employs a transient simulation to capture the unsteady nature of the flow region, vortex, and wake interactions during turbine rotation, with an iteration per time step of 20 and a total of 300-time steps.

2.6 Post-Processing and Data Analysis

Post-processing of the data is carried out using Ansys CFD-Post to extract velocity contours, flow patterns, as well as the distribution of turbulence intensity at specific time intervals. These parameters are then analyzed to evaluate the hydrodynamic behavior, vortex formation process, and flow building characteristics, which play an important role in understanding the torque generation mechanism and efficiency of the Savonius turbine. In addition, the results of the transient flow simulation are also averaged in phases during the full rotation of the rotor, with the aim of identifying stable flow patterns as well as consistent performance characteristics.

3 Results and Discussions

The results of the CFD simulation show a clear correlation between the speed of incoming water and the hydrodynamic characteristics of the Savonius water turbine. As the flow speed increases, noticeable changes are observed in the speed contour, tilt pattern, and distribution of turbulence intensity around the rotor blades and in the downstream wake area. Transient CFD simulations of the Savonius water turbine revealed that the inlet velocity significantly affects the hydrodynamic characteristics, especially in terms of the velocity, flow, and turbulence intensity contours. There are five variations of inlet velocities which were investigated: 0.5 m/s, 0.75 m/s, 1.0 m/s, 1.25 m/s, and 1.5 m/s.

3.1 Velocity Contours

The results of the velocity contour are shown in Figure 4 below. At 0.5 m/s, the velocity contours indicate mild acceleration of the flow along the advancing blade and limited recirculation on the returning side. The wake region is narrow, with minimal velocity loss downstream, suggesting lower energy extraction potential but stable flow. At 0.75 m/s, the high-velocity region near the advancing blade becomes more pronounced, and the wake widens slightly due to stronger vortex shedding. At 1.0 m/s, a sharper velocity gradient develops between the upstream and downstream sides, indicating increased torque generation. Flow separation is more apparent on the returning blade, leading to a more complex wake structure. At 1.25 m/s, the contours reveal high-speed zones extending further along the advancing blade, while the returning blade exhibits a larger low-speed recirculation zone. This condition enhances pressure differences but also increases turbulence in the wake. At 1.5 m/s, the high-velocity regions dominate the advancing side, with intense acceleration and more energetic vortex shedding. The wake becomes highly disturbed, showing significant kinetic energy dissipation downstream.

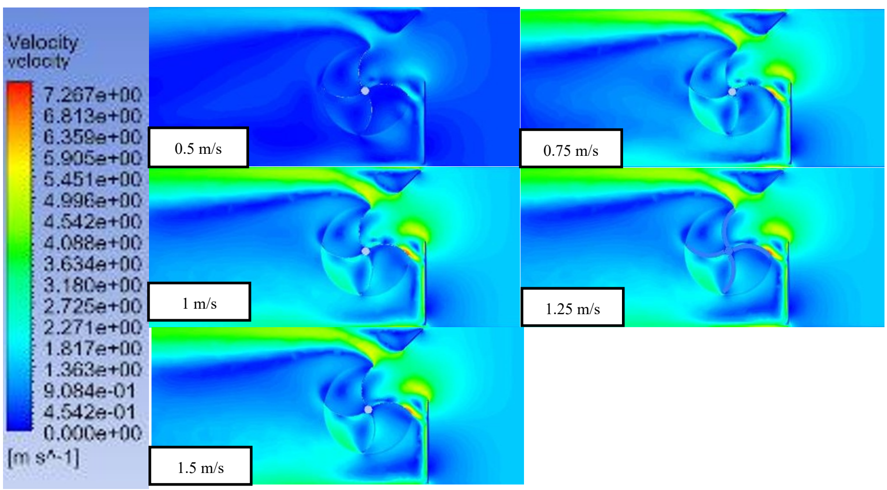


Fig. 4. Water velocity contours at 0.5; 0.75; 1; 1.25; 1.5 m/s

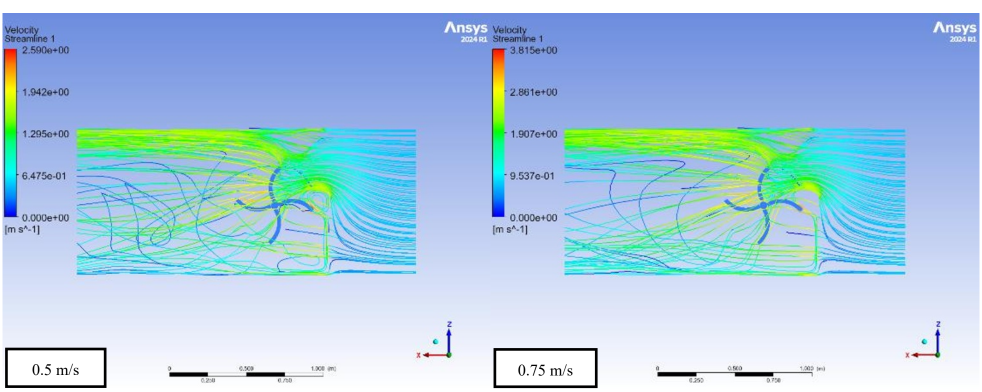
At lower inlet velocities, the velocity contour plots indicate relatively smooth acceleration of the fluid along the concave blade surfaces, with a moderate velocity gradient between the advancing and returning blades. The wake region is relatively narrow, and vortex formation is limited, resulting in reduced energy dissipation. The streamlines at these conditions remain largely attached to the blade surfaces on the advancing side, while mild flow separation occurs near the returning blade due to adverse pressure gradients. Turbulence intensity in this regime is generally low, concentrated in small regions behind the blades where minor vortices are shed.

High-velocity zones form along the blades as they move forward, while the blades' return motion creates a broader area of recirculation. Therefore, the speed contours show an increasingly noticeable difference between the upstream and downstream sides of the rotor as the inflow speed increases. This indicates a stronger pressure gradient and the potential for increased torque generation. Moreover, the upstream area also appears to be increasingly wide, with more intense vortex shedding and complex flow structures. The flow streamlines plot shows a more obvious curvature and release of the flow from the blade surface in line with this phenomenon. It can indicate the increased flow instability and energy exchange within the rotor domain.

3.2 Velocity Streamline

The current line speed (Figure 5) provides an in-depth overview of the fluid flow characteristics and the interaction between the inlet water and the turbine blades at various inlet speeds. At low flow speeds, the current line appears to be approaching the blade moving forward with a gentle curvature, indicating the presence of a smooth acceleration of fluid along the concave surface of the blade. The fluid flow generally remains attached to the blade surface until it reaches the downstream side, with minor deviations produced by the release of small-scale vortex from the blade tip. The wake region looks narrow and extends downstream with a relatively stable flow pattern under these conditions, indicating a low rate of kinetic energy loss. The flow line still shows a smooth pattern and sticks firmly to the forward blade at a speed of 0.5 m/s, with only a small vortex forming around the end of the blade. Therefore, the flow wake remains stable and narrow. At 0.75 m/s, curvature of the streamlines around the advancing blade increases, and small separation bubbles appear on the returning side. At 1.0 m/s, streamlines exhibit clear detachment from the returning blade surface, forming distinct recirculation zones in the near wake. Vortex interaction between the blades becomes more visible. At 1.25 m/s, the flow pattern becomes increasingly asymmetric, with strong curvature and multiple vortex cores forming downstream. Streamline distortion indicates enhanced turbulence and mixing. At 1.5 m/s, streamlines are highly distorted and intertwined in the wake, with large-scale vortex shedding dominating the downstream flow field. This unsteady pattern indicates both increased torque potential and higher structural loading on the blades.

As the inlet velocity increases, the streamline patterns undergo significant changes. The streamlines on the advancing side bend more sharply toward the blade surface, illustrating a higher local acceleration and stronger pressure gradients. On the returning side, the adverse pressure conditions promote early flow separation, resulting in the formation of larger recirculation zones. This separation leads to a more complex wake structure with alternating vortical patterns that are shed downstream.



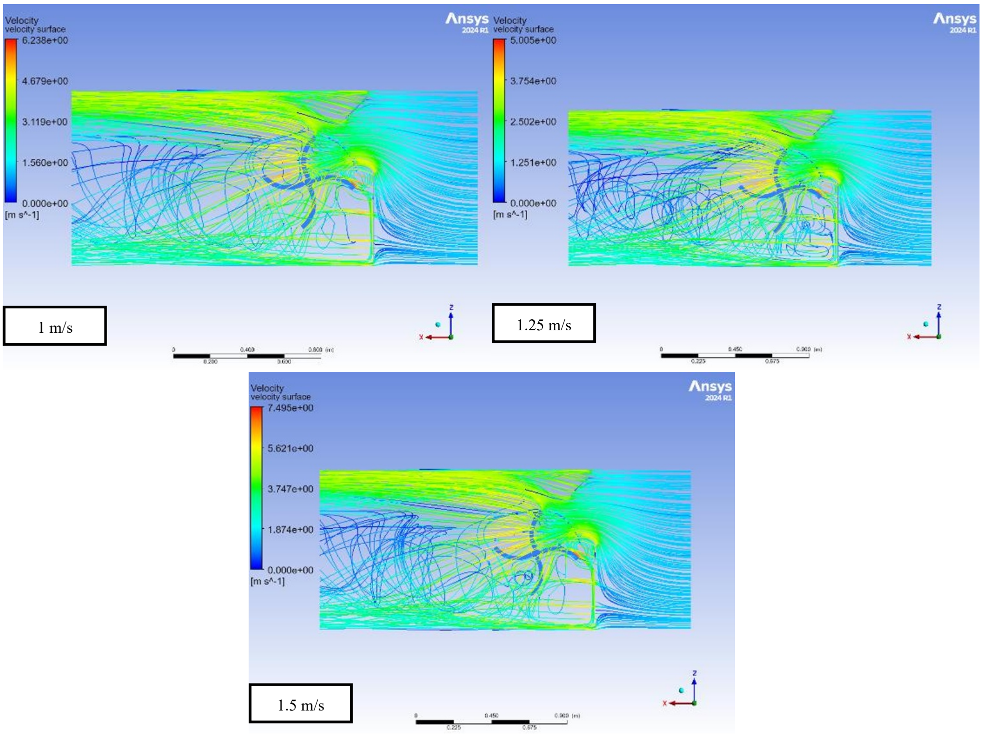


Fig. 5. Velocity streamlines at 0.5; 0.75; 1; 1.25; 1.5 m/s

At the highest tested inlet velocities, the streamlines depict intensified turbulence in the wake, with highly distorted and intertwined flow paths extending farther downstream. The interaction between the shedding vortices from both blades creates regions of strong mixing, which are evident in the swirling streamline patterns. These unsteady flow features indicate higher energy extraction potential but also suggest increased unsteady loading on the rotor blades, which may have implications for structural integrity and fatigue life.

The evolution of the streamline patterns with inlet velocity highlights the balance between beneficial flow acceleration and detrimental flow separation in Savonius turbine operation. The higher inlet velocities also introduce more pronounced wake instabilities, while amplifying the kinetic energy available for conversion. This matter can influence the overall efficiency and mechanical stresses experienced by the turbine.

3.3 Turbulence Intencity

The turbulence intensity distribution follows a similar trend, with higher inlet velocities leading to a significant increase in turbulence levels, particularly in the blade tip regions and the wake (Figure 6). This elevated turbulence promotes greater mixing in the downstream flow, which, while beneficial for energy extraction in some configurations, may also contribute to increased energy dissipation and reduced downstream flow quality. The observed flow phenomena align with previous studies that highlight the influence of inlet velocity on the onset of large-scale vortex structures in Savonius-type turbines [8], [10].

 https://doi.org/10.1051/bioconf/202519902009

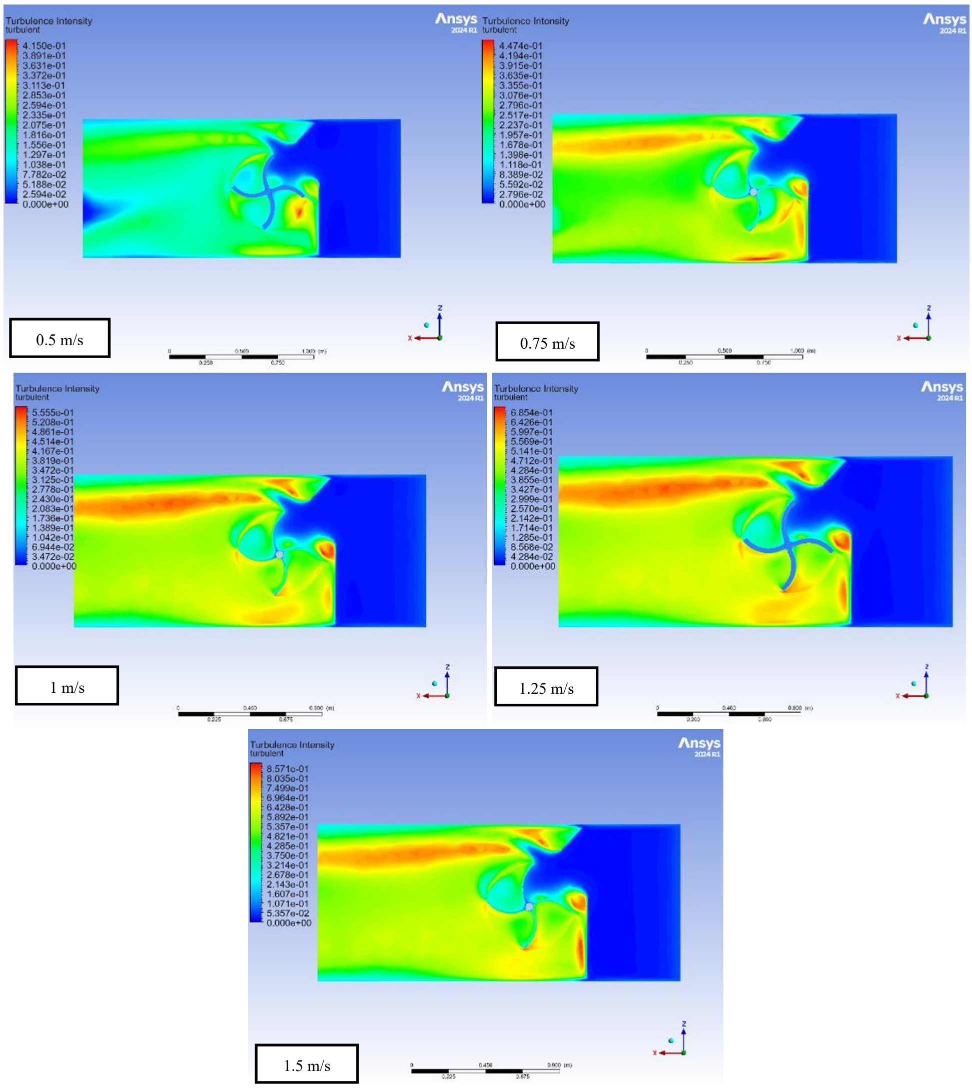


Fig. 6. Turbulence Intensity at velocity variations with 0.5; 0.75; 1; 1.25; 1.5 m/s

At a speed of 0.5 m/s, the turbulence intensity was still low and limited to a narrow region behind the blade that it prompts that the energy loss due to turbulent mixing is minimal. As the speed increases to 0.75 m/s, the turbulence began to increase gradually, specifically at the blade ends and downstream because of the appearance of stronger vortices. After that, at a speed of 1.0 m/s, the level of turbulence increases significantly, particularly in the rear blade and the area afterwards as vortex shedding becomes more active. Moreover, at a speed of 1.25 m/s, a high-intensity turbulent zone extends in the downstream region. It made signal more intense fluid mixing and increased kinetic energy dissipation. Finally, at a speed of 1.5 m/s, the turbulence intensity reached its peak which the turbulent region was extending far downstream and illustrating the strong flow instability and complex vortex interactions around the rotor.

The results showed that increasing the inflow speed can increase the acceleration of the fluid along the front side of the blade, expand the flow making area, and increase the intensity of turbulence downstream. At low speeds (0.5–0.75 m/s), the flow pattern is still stable, with limited wake-up disturbances. Meanwhile, at moderate velocity (1.0 m/s), a stronger vortex release and moderate turbulence growth begin to be seen. As the speed increases to 1.25–1.5 m/s, the flow wake pattern becomes more unstable, the flow line looks more distorted, and the intensity of turbulence increases sharply. This state was able to affect the downstream flow quality and reduce the turbine performance stability.

These findings recommend that the inlet velocity has an important role in determining the flow pattern and wake behaviour around the Savonius water turbine. Therefore, the selection of the right operating speed on a hydrokinetic pico hydro system is important to increase the efficiency of energy harvesting with the stability of flow and environmental impact.

3.4 Applications in Agriculture

This study employs computational fluid dynamics (CFD) to obtain detailed insights into the flow behavior, pressure distribution, and torque generation of the Savonius water turbine, which are essential for improving its overall performance and operational efficiency. The CFD-based analysis enables systematic optimization of turbine geometry and operating conditions for application in small-scale hydropower systems, particularly those integrated into irrigation channels and low-head water resources commonly found in agricultural environments.

The findings of this study demonstrate the potential of Savonius water turbines as a reliable and environmentally friendly energy solution for rural and agricultural sectors, where access to conventional electricity infrastructure is often limited. By harvesting low-velocity water flows without significantly disturbing irrigation functions, the proposed turbine supports energy self-sufficiency in farming activities, including irrigation pumping, post-harvest processing, and smart agriculture systems. Therefore, the outcomes of this research contribute to the advancement of sustainable power generation technologies that align with the needs of modern agriculture and rural development.

4 Conclusion

CFD analysis of Savonius water turbines at inlet velocity variations between 0.5 m/s to 1.5 m/s showed that flow velocity has a significant influence on the hydrodynamic characteristics of the turbine. At the lowest speed (0.5 m/s), the fluid flow appears smooth and stable, with a current line attached to the blade surface, a narrow flow build, and a low turbulence intensity. As the flow velocity increases to 0.75 m/s and 1.0 m/s, the velocity gradient becomes sharper, followed by the separation of light streams and the formation of small to medium-sized vortices in the wake region. At higher entry speeds (1.25 m/s and 1.5 m/s), the velocity contour shows a wide high-speed zone on the forward side of the blade, while the returning side of the blade shows a larger recirculation region. In addition, the intensity of turbulence shows a significant increase at such high speeds, with the high-energy turbulent zone extending further downstream of the rotor. Overall, the results of this analysis confirm that the increase in inflow speed not only accelerates flow dynamics and complicates the building structure but also increases the intensity of turbulence and magnifies the instability of the flow around the turbine.

Further research can be focused on the analysis of the influence of material variations and blade counts on the performance efficiency of Savonius turbines. This study may include considerations of production costs, material resistance to corrosion and wear, and turbine performance at a wide range of low to medium flow speeds. Experimental testing plans may involve the use of composite materials to compare the energy efficiency generated as well as their durability under different operational conditions. Moreover, further research can explore variations in configuration or blade count to optimize turbine performance in various water flow conditions. Thus, the results of this study are expected to be able to serve more efficient and durable solution for small-scale hydrokinetic applications, which is relevant for areas with low water discharge, so that it can minimize operational and maintenance costs.

Acknowledgements

The authors would like to acknowledge support from the Department of Automotive Engineering Technology, Faculty of Engineering, Universitas Muhammadiyah Yogyakarta (UMY). And big thanks for valuable support from PT Sahla Inovatif Teknik, a privat company in Yogyakarta who has facilitated simulation labs and as a research partner. This research was funded by Universitas Muhammadiyah Yogyakarta through the Internal Research Grant Program based on Contract Number: 16/R-LRI/I/2023.

References

1. A. Jebelli, N. Lotfi, M. Saeid Zare, and M. C. E. Yagoub, "A comprehensive review of effective parameters to improve the performance of the Savonius turbine using a computational model and comparison with practical results," *Water-Energy Nexus*, vol. 7, pp. 266–274, Dec. 2024, https://doi.org/10.1016/j.wen.2024.11.002.
2. N. A. M. Rais, M. F. M. Basar, E. Z. Ahmad, and M. I. M. N. Rizan, "Reliability Study of Ultra Z-Blade Water Turbine for Pico-Hydro System with Low Head and Low Flow Water Resources," *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, vol. 117, no. 1, pp. 132–142, May 2024, https://doi.org/10.37934/arfmts.117.1.132142.
3. M. Mohammed, S. Sarip, S. A. Aziz, and W. A. Mustafa, "Systematic Review of Computational Fluid Dynamics Modelling and Simulation Techniques Employed in Vertical Axis Hydrokinetic Turbines," *Journal of Advanced Research in Applied Mechanics*, vol. 117, no. 1, pp. 51–71, May 2024, https://doi.org/10.37934/ARAM.117.1.5171.
4. S. Puppala, P. P. Singh, and D. Potnuru, "A Techno-Economic Analysis of Solar/Wind/Hydrokinetic Turbine-Based Green Hydrogen and Electricity-Generated Standalone DC Microgrid for Remote Area Applications," *International Journal of Mathematical, Engineering and Management Sciences*, vol. 10, no. 5, pp. 1365–1396, Oct. 2025 https://doi.org/10.33889/IJMEMS.2025.10.5.065.
5. M. Mohammadzadeh Kowsari, H. Niazmand, and M. M. Tokarev, "Bed configuration effects on the finned flat-tube adsorption heat exchanger performance: Numerical modeling and experimental validation," *Appl Energy*, vol. 213, pp. 540–554, Mar. 2018, https://doi.org/10.1016/J.APENERGY.2017.11.019.
6. M. M. Kamal, A. Abbas, T. Alam, R. Khargotra, and T. Singh, "Influence of Water Flow Speed on the Torque Behaviour of the Hybrid HKT Having Straight and Helical Bladed Savonius Rotor," *CFD Letters* , vol. 16, no. 9, pp. 114–125, Sep. 2024, https://doi.org/10.37934/CFDL.16.9.114125.
7. M. Al-Ghriybah and D. H. Didane, "Performance Improvement of a Savonius Wind Turbine using Wavy Concave Blades," *CFD Letters*, vol. 15, no. 9, pp. 32–44, Aug. 2023, https://doi.org/10.37934/CFDL.15.9.3244.
8. J. L. Cabalo and A. L. Marcelo, "Optimization of Savonius type hydro-kinetic turbine for irrigation canal in the Philippines using computational fluid dynamics,"

IOP Conf Ser Earth Environ Sci, vol. 1500, no. 1, 2025, https://doi.org/10.1088/1755-1315/1500/1/012016.

9. M. Mohammed, S. Sarip, S. A. Aziz, W. A. Mustafa, and A. A. Ajmi, "Savonius Hydrokinetic Turbine: A Bibliometric Analysis," *Journal of Advanced Research in Applied Mechanics*, vol. 133, no. 1, pp. 36–51, Feb. 2025, https://doi.org/10.37934/ARAM.2.1.3651.
10. A. Zakaria and M. S. N. Ibrahim, "Velocity Pattern Analysis of Multiple Savonius Wind Turbines Arrays," *CFD Letters*, vol. 12, no. 3, pp. 31–38, 2020, https://doi.org/10.37934/cfdl.12.3.3138.
11. Thoharudin, T. H. A. Santosa, H. N. Irawan, and H. T. Waloyo, "Investigation of Blade Number Effect on the Centrifugal Blower as a Small-Scale Water Turbine," in *Journal of Physics: Conference Series*, Institute of Physics, 2025. https://doi.org/10.1088/1742-6596/2989/1/012008.
12. A. A. Najib, D. H. Didane, M. R. Behery, and H. A. Kabrein, "Comparison of 2D and 3D Simulations on Predicting the Performance of a Savonius Wind Turbine," *CFD Letters*, vol. 16, no. 7, pp. 71–88, Feb. 2024, https://doi.org/10.37934/CFDL.17.6.7188.
13. T. G. Shanegowda, C. M. Shashikumar, V. Gumtapure, and V. Madav, "Comprehensive analysis of blade geometry effects on Savonius hydrokinetic turbine efficiency: Pathways to clean energy," *Energy Conversion and Management: X*, vol. 24, p. 100762, Oct. 2024, https://doi.org/10.1016/J.ECMX.2024.100762.
14. J. Priyadumkol *et al.*, "CFD modelling of vertical-axis wind turbines using transient dynamic mesh towards lateral vortices capturing and Strouhal number," *Energy Conversion and Management: X*, vol. 26, p. 101022, Apr. 2025, https://doi.org/10.1016/J.ECMX.2025.101022.
15. H. Mirmanto, A. A. Athallasyah, and T. Yuwono, "Numerical Study of the Effect of Deflector Length on the Water Channel Wall on the Performance of Savonius Water Turbine," *CFD Letters*, vol. 18, no. 2, pp. 183–202, 2026, https://doi.org/10.37934/CFDL.18.2.183202.