

Numerical Investigation of Multi-Stage Hybrid Darrieus-Savonius Hydrokinetic Turbine Performance with Fibonacci Blades and Darrieus Solidity Variations

Dimas Rio Javani¹, Sutardi^{2✉}

(1,2) Department of Mechanical Engineering, Institut Teknologi Sepuluh Nopember, Surabaya, Indonesia

Abstract: Indonesia's increasing energy demand and fossil fuel crisis have accelerated the transition toward New and Renewable Energy (NRE). With an NRE potential of 3,600 GW and utilization still below 2%, hydrokinetic turbines offer a promising solution for harnessing water flow energy. This study numerically investigated the performance of a multi-stage hybrid Darrieus-Savonius hydrokinetic turbine using Computational Fluid Dynamics (CFD). The proposed design combined Fibonacci spiral-based Savonius blades with NACA 0018 Darrieus blades. Simulations were conducted within a Tip Speed Ratio (TSR) range of 0.2–2.3 and solidity ratios of 0.5–1.1 to determine the optimum turbine performance. The results showed that blade solidity significantly influenced energy extraction efficiency in a non-linear manner. The optimum condition was achieved at a solidity ratio of 0.95 and TSR = 1, producing a maximum power coefficient (C_p) of 0.082. Flow visualization indicated stable negative pressure regions and higher local flow acceleration, which reduced flow separation around the blades. In contrast, a solidity ratio of 1.1 at TSR 0.8 caused stronger blockage effects and wake interactions, increasing hydrodynamic drag and reducing turbine performance. These findings demonstrate that solidity ratios close to unity can improve torque stability and turbine efficiency under low TSR conditions.

Article history:

Received: 22 May 2026

Revised: 16 June 2026

Accepted: 18 June 2026

Published: 25 June 2026

Keywords:

CFD, Hybrid Turbine, Moment Coefficient (C_m), Power Coefficient (C_p), Fibonacci Spiral, Solidity.

This is an open-access article under the CC-BY-SA License.



How to cite: Javani, D. R., & Sutardi, S. (2026). Numerical Investigation of Multi-Stage Hybrid Darrieus-Savonius Hydrokinetic Turbine Performance with Fibonacci Blades and Darrieus Solidity Variations. *RESWARA: Jurnal Riset Ilmu Teknik*, 4(3), 120–135. <https://doi.org/10.70716/reswara.v4i3.544>

INTRODUCTION

National energy demand continues to rise in tandem with rapid population growth, urbanization, and industrial expansion. Concurrently, pressing environmental challenges namely escalating carbon emissions, the depletion of fossil reserves, and climate change have accelerated the strategic transition toward New and Renewable Energy (NRE). Indonesia possesses an immense renewable energy potential estimated at 3,600 GW, encompassing solar, wind, hydro, bioenergy, and ocean resources. However, current utilization stands at a mere 14.8 GW, representing less than 3% of this vast capacity (Kementerian ESDM, 2023). This stark disparity highlights a critical opportunity for developing scalable renewable technologies, with hydrokinetic energy conversion systems standing out as a highly viable pathway.

Hydrokinetic turbines offer a practical means of converting the kinetic energy of natural water currents directly into electricity without the need for large-scale dams (Raditya Yudistira et al., 2021). These systems are broadly categorized by their rotational axis into horizontal and vertical configurations, with vertical-axis or cross-flow hydrokinetic turbines being preferred for shallow and

constrained river channels due to their design simplicity and multidirectional operation (Kadiri et al., 2012). To overcome the inherent hydrodynamic limitations of standalone designs, recent attention has shifted toward hybrid Savonius–Darrieus configurations (Rendi & Hartadi, 2022). By integrating a drag-driven Savonius rotor with a lift-driven Darrieus rotor, these hybrid systems leverage the high starting torque of the former and the superior high-speed efficiency of the latter, thereby ensuring self-starting capability alongside steady operational performance (Ghafoorian et al., 2025; Maldar et al., 2021).

Extensive research has targeted the geometric optimization of these hybrid configurations. For instance, the symmetrical NACA 0018 hydrofoil has been proven to yield better initial torque compared to thinner or thicker profiles like the NACA 0015 and NACA 0021 (Kanyako & Janajreh, 2014). Furthermore, maintaining a three-bladed Darrieus assembly balances rotor stability and efficiency, avoiding the excessive drag and torque ripples associated with higher blade counts (Hartanto, 2022). Regarding the Savonius sub-unit, a two-bladed layout typically achieves higher hydrodynamic efficiency than three- or four-bladed designs (T.G. et al., 2024). More recently, biomimetic blade geometries, such as those derived from the Fibonacci spiral, have been introduced to smoothen fluid flow, suppress flow separation, and improve pressure distribution across the rotor (Damota et al., 2022; Galdo et al., 2024). Another pivotal geometric driver is the solidity ratio, which dictates the blade chord area relative to the overall turbine swept area (Mohan Kumar et al., 2019). Changes in solidity directly govern torque behavior and power extraction limits under varying flow conditions (Chhin, 2020; Irawan et al., 2024; Rogelio et al., 2025).

Despite these developments, a clear gap remains in understanding the combined effects of these parameters. Previous studies have generally restricted their focus to either investigating baseline H-Darrieus solidity variations (Chhin, 2020; Rogelio et al., 2025) or assessing standalone biomimetic Savonius rotors in isolation (Galdo et al., 2024). Crucially, literature addressing the simultaneous hydrodynamic interaction between non-linear solidity changes in a Darrieus rotor and a multi-stage Fibonacci-spiral Savonius rotor is virtually non-existent. Most hybrid designs still rely on standard semi-circular Savonius blades, leaving it unclear how a combined biomimetic-solidity configuration alters localized flow acceleration, wake propagation, and structural drag under low-velocity hydrokinetic conditions.

To address this gap, this study establishes a comprehensive numerical framework to evaluate the hydrodynamic performance of a multi-stage hybrid Darrieus–Savonius hydrokinetic turbine using transient 3D Computational Fluid Dynamics (CFD). The primary objective is to clarify the non-linear relationship and identify the optimal performance threshold of the Darrieus chord solidity ratio (ranging from 0.5 to 1.1) when coupled with a Fibonacci-based Savonius rotor across a broad Tip Speed Ratio (TSR) spectrum of 0.2–2.3. By analyzing key performance metrics, specifically the Power Coefficient (C_p), Moment Coefficient (C_m), and localized pressure and velocity fields this work uncovers the physical mechanisms governing flow attachment, flow acceleration, and wake-blade interactions. Ultimately, these insights provide actionable design guidelines for maximizing energy extraction efficiency in low-velocity river flows.

RESEARCH METHOD

To resolve the complex fluid-structure interactions of the hybrid rotor, a numerical framework using 3D transient Computational Fluid Dynamics (CFD) was established across a wide spectrum of Tip Speed Ratios (TSR 0.2–2.3). The primary data extraction focuses on monitoring the generated torque over time to calculate macro-level performance parameters via power (C_p) and moment (C_m) coefficients. Simultaneously, localized hydrodynamic behavior is evaluated using pressure contours to identify lift-to-drag distributions, alongside velocity contours to map flow acceleration, flow separation, and wake propagation behind both the Darrieus and Savonius stages.

In this framework, the Darrieus solidity ratio is systematically varied from 0.5 to 1.1 based on a fundamental trade-off in cross-flow turbine physics. Lower solidity utilizes smaller blade chords that reduce structural drag but weaken self-starting capabilities under low fluid velocities. Conversely, higher solidity maximizes energy capture at low speeds but risks generating severe blockage effects and heavy wake interactions that can choke the fluid flow. Evaluating this specific spectrum is essential to identify the optimal threshold where fluid momentum extraction is maximized before hydrodynamic penalties dominate.

While this transient CFD approach provides comprehensive flow insights, certain inherent limitations must be acknowledged. The numerical model relies on Reynolds-Averaged Navier-Stokes (RANS) turbulence closures, which struggle to perfectly replicate highly chaotic, isotropic turbulence under extreme dynamic stall conditions. Furthermore, the simulation assumes an idealized fluid domain and rigid blade structures, thereby neglecting real-world river turbulence intensities, floating debris, and fluid-structure interactions (FSI) such as blade deflection. Despite these physical constraints, this framework remains a highly reliable and standard tool for mapping non-linear turbine performance trends before moving into physical prototyping.

Turbine Geometry

The hydrokinetic turbine used in this study is a hybrid turbine that combines the Darrieus and Savonius types, as shown in Figure 1. The hybrid configuration was chosen to leverage the advantages of each turbine type: the Darrieus turbine's ability to generate high lift and good efficiency at certain operating speeds, and the Savonius turbine's high starting torque, which allows it to begin rotation at low flow speeds.

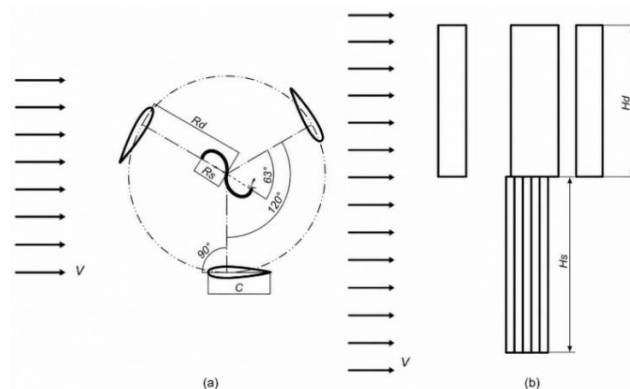


Figure 1. Design of a Savonius-Darrieus turbine (a) Top view (b) Side view

In this study, the Darrieus section uses a three-bladed NACA 0018 hydrofoil profile due to its symmetrical shape, which provides good performance under bidirectional hydrokinetic flow conditions. The three-blade configuration was selected to maintain rotational stability and structural balance. Meanwhile, the Savonius section employs a two-bladed Fibonacci-based design, which offers improved hydrodynamic characteristics compared to conventional Savonius blades, particularly in terms of drag distribution and initial torque performance. The complete dimensions of the hydrokinetic turbine used in this study are presented in Table 1.

Table 1. Dimensions of a hydrokinetic turbine

	Darrieus	Savonius
Turbine Radius (Rd, Rs)	500 mm (Rd)	148.6 mm (Rs)
Span Length (Hd, Hs)	1000 mm (Hd)	1000 mm (Hs)
Chord Length (C)	166.6–366.6 mm	-
Solidity Ratio (σ)	0.5–1.1	-
Blade Thickness (t)	-	5 mm
Number of Blades (N)	3	2
Blade Type	NACA 0018	Fibonacci

The main variation applied in this study is the chord length of the Darrieus blades, which directly affects the solidity ratio (σ). Solidity is an important parameter because it determines the interaction between the turbine blades and the fluid flow. In this research, the solidity ratio is calculated from the ratio between the total chord length of all blades and the rotor sweep radius, formulated as follows:

$$\sigma = \frac{N \cdot c \cdot H}{A} \quad (1)$$

Where N denotes the number of blades utilized, c represents the blade chord length in meters (m), H is the span length in meters (m), and A refers to the turbine swept area in square meters (m²).

The proposed hydrokinetic system combines a lift-driven Darrieus rotor with NACA 0018 hydrofoils and a multi-stage drag-driven Savonius rotor based on a Fibonacci spiral geometry. To optimize the interaction between both stages, the Darrieus solidity ratio was varied from 0.5 to 1.1 by adjusting the blade chord length. This range was selected to evaluate the trade-off between low solidity, which reduces drag but weakens self-starting performance, and high solidity, which enhances energy capture at low flow velocities but may induce blockage effects and stronger wake interactions. Therefore, investigating this range is essential to determine the optimum solidity that maximizes energy extraction while minimizing hydrodynamic losses. The resulting geometric configurations are shown in Figure 2.

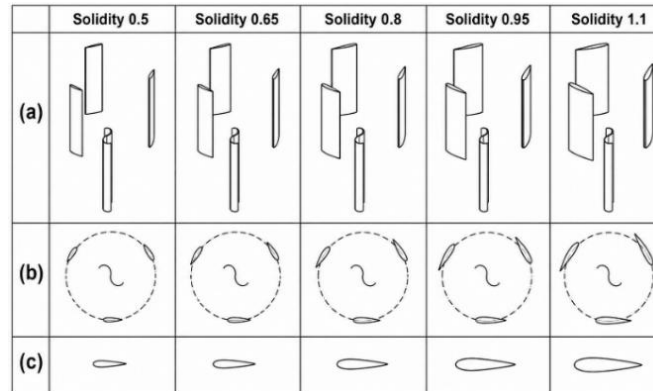


Figure 2. Variations in solidity (σ) for a multi-stage Darrieus-type hydrokinetic turbine: (a) isometric view, (b) top view, and (c) blade cross-section (hydrofoil).

Computational Domain

The hydrokinetic turbine model designed using SolidWorks software was then exported to ANSYS SpaceClaim for the creation of the simulation domain. In this simulation, the computational domain is divided into two main parts: the rotating domain and the stationary domain, as shown in Figure 3. The rotating domain is the area containing the turbine rotor. This domain is modeled as a cylinder with a diameter slightly larger than the outer diameter of the turbine blades to ensure there is free space between the blades and the domain boundary, allowing the flow distribution around the blades to be accurately captured.

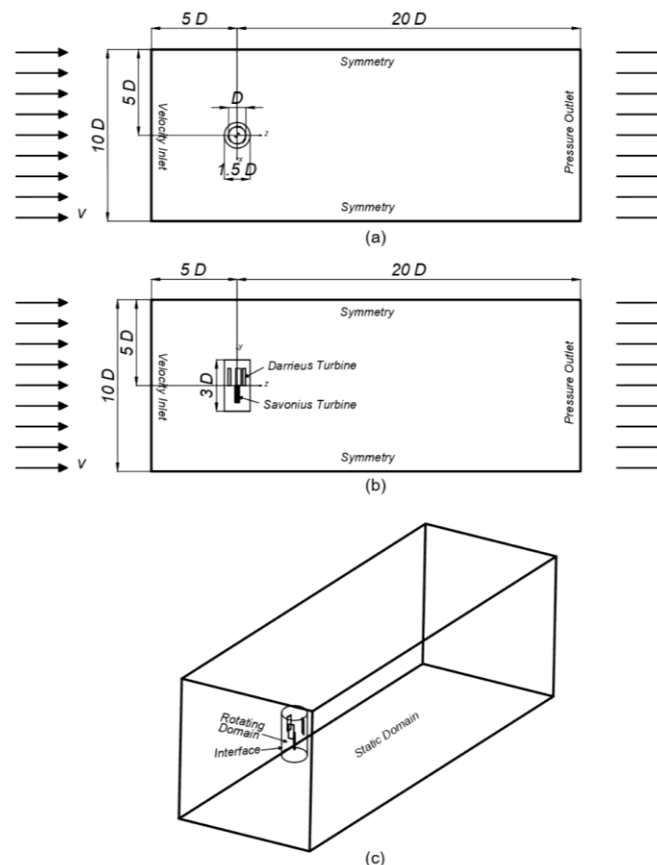


Figure 3. Computational domain (a) Top view (b) Side view (c) Isometric view

Meanwhile, the static domain serves as the fluid region surrounding the turbine, representing free-flow conditions. This domain is made larger than the rotating domain so that the interaction between the turbine and the fluid can be fully represented without being affected by wall boundaries. These two domains are connected via an interface that serves to merge the fluid in the static and rotating domains, ensuring a continuous flow between them.

Meshing

Meshing is the critical preprocessing step of discretizing the continuous fluid domain into millions of small, finite elements, which enables the solver to compute the governing Navier–Stokes equations numerically across every grid localized point. This grid generation is highly vital for capturing transient flow behaviors, such as flow separation and vortex shedding, with high geometric and computational accuracy. The detailed mesh sizes and refinement configurations assigned to each specific computational region in this simulation are summarized in Table 2.

Table 2. Data Meshing	
Boundary name	Meshing size (mm)
Darrieus	25
Savonius	5
Interface	200

In this study, the meshing method used was an unstructured mesh with tetrahedral elements, as shown in Figure 4. Tetrahedral elements were chosen because of their flexibility and ability to accurately represent the turbine's complex geometry.

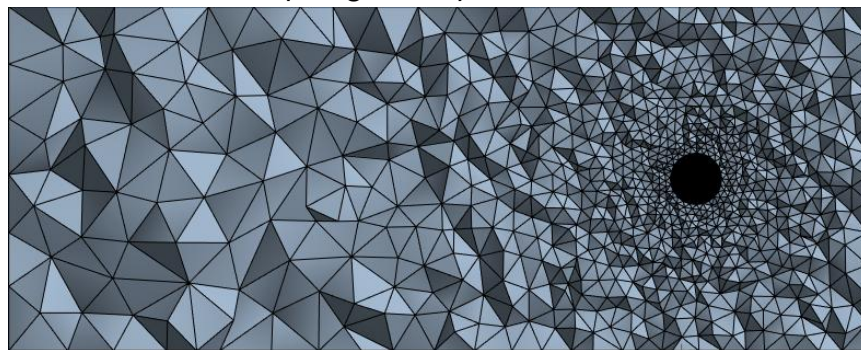


Figure 4. Meshing across the entire domain

Figure 5 shows the meshing settings for this simulation. In the rotating domain, the mesh is refined around the turbine blade surfaces to improve the accuracy of the simulation results in the primary focus area. Conversely, in the static domain, the mesh is coarser because it serves only as an inlet and outlet flow channel, which does not require the same level of detail as the turbine blade area.

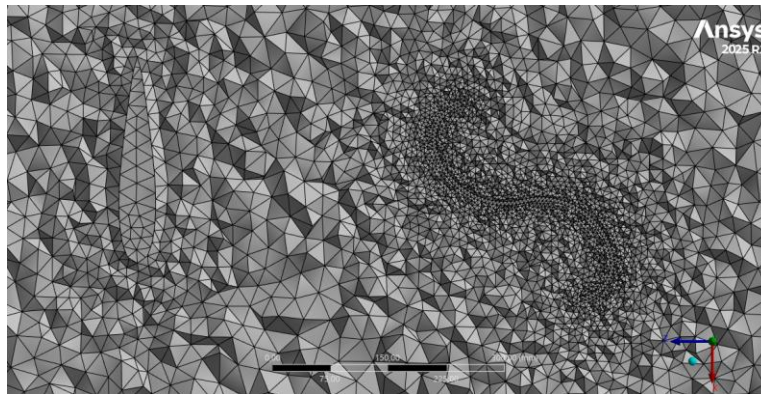


Figure 5. Meshing around the turbine blades

Once the meshing process is complete, a mesh quality check is performed to ensure that the simulation results obtained are valid and accurate. One of the primary parameters used to assess mesh quality is skewness. Skewness measures the extent to which mesh elements deviate from their ideal shapes (such as a perfect cube or tetrahedron). A low skewness value indicates good mesh quality, while a high skewness value may indicate that the mesh elements have distortions that could affect the simulation results.

Skewness mesh metrics spectrum					
Excellent	Very good	Good	Acceptable	Bad	Unacceptable
0-0.25	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00
Orthogonal Quality mesh metrics spectrum					
Unacceptable	Bad	Acceptable	Good	Very good	Excellent
0-0.001	0.001-0.14	0.15-0.20	0.20-0.69	0.70-0.95	0.95-1.00

Figure 6. Mesh quality recommendations (Kumar et al., 2023)

Figure 6 presents the recommended skewness and orthogonality values for CFD simulations according to ANSYS guidelines. The mesh used in this study achieved an average skewness of 0.23 and a maximum value of 0.61, both of which are within the acceptable limit of 0.85, indicating good mesh quality and stable simulation accuracy. In addition, the orthogonal quality showed a minimum value of 0.38 and an average value of 0.76, demonstrating that the mesh elements were sufficiently well-shaped and balanced to accurately represent the fluid domain without significant distortion. The total mesh consisted of approximately 2.5 million elements, which provided a suitable balance between computational accuracy and efficiency.

Turbulent Model

The turbulence model selected for this simulation is the $k-\omega$ Shear Stress Transport (SST) model, which is a two-equation model based on eddy viscosity. This model was chosen for its superior ability to predict flow phenomena in the near-wall region and in separated flow, which are highly relevant for hybrid hydrokinetic turbine applications (Lanzafame et al., 2014; Zamani et al., 2016).

The $k-\omega$ SST model combines the advantages of the standard $k-\epsilon$ and $k-\omega$ models. The $k-\omega$ model is used in the boundary layer to maintain a robust and accurate formulation near the wall, while the $k-\epsilon$ model is applied to the outer (free-stream) layer to avoid sensitivity issues related to free-stream conditions (Menter, 1994). This combined formulation allows the $k-\omega$ SST model to function as a Low-

Reynolds model without requiring additional damping functions. The transport equations for turbulent kinetic energy (k) and specific dissipation rate (ω) are expressed as follows (Wilcox, 2010)

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_j} \right] - \beta^* \rho \omega k [1 + \alpha_1 M_t^2 (1 - F_1)] + P_k + (1 - F_1) \overline{p'' d''} \quad (2)$$

$$\frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_j \omega)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_j} \right] + 2(1 - F_1) \frac{\rho \sigma_\omega}{\omega} \frac{\partial k}{\partial x_j} \frac{\partial \omega}{\partial x_j} - \beta \rho \omega^2 + (1 - F_1) \beta^* \alpha_1 M_t^2 \rho \omega^2 - \frac{\rho}{\mu_t} (1 - F_1) \overline{p'' d''} + \alpha \frac{\omega}{k} P_k \quad (3)$$

where P_k represents turbulence production. The transition between the k - ω and k - ϵ models is controlled by the blending function (F_1), which is defined as:

$$F_1 = \tanh \left\{ \min \left[\max \left(\frac{\sqrt{k}}{\beta^* \omega d}, \frac{500\nu}{\omega d^2} \right), \frac{4\rho\sigma_\omega k}{CD_{k\omega} d^2} \right] \right\} \quad (4)$$

In this formula, d is the distance to the nearest wall surface. To calculate the eddy viscosity (μ_t), the following equation is used:

$$\mu_t = \frac{\alpha^* a_1 \rho k}{\max(a_1 \omega, \Omega F_2)} \quad (5)$$

The constant a_1 is set to 0.31, and the function F_2 is defined by the following equation:

$$F_2 = \tanh \left[\max \left(\frac{2\sqrt{k}}{\beta^* \omega d}, \frac{500\nu}{\omega d^2} \right) \right]^2 \quad (6)$$

The use of the k - ω SST algorithm yields more accurate results than classical turbulence models such as the standard k - ϵ model, particularly in capturing the opposing pressure gradients on the turbine blades (Daróczy et al., 2015).

Boundary Condition

A boundary condition is a condition applied to the boundaries of the domain being analyzed in a simulation or numerical modeling, with the aim of describing the interaction between the system and its surrounding environment or the physical boundaries of the system.

In the context of hydrokinetic turbine simulations, boundary conditions play a crucial role in defining how the fluid interacts with the turbine surface, as well as how conditions along the domain boundaries (e.g., the turbine surface, inflow, and outflow) are specified. The boundary conditions for this simulation are presented in Table 3.

Table 3. Boundary conditions in the simulation

Boundary name	Boundary Type	Boundary Condition	Location
Inlet	Velocity Inlet	1 m/s	Located at the front of the domain (fluid inlet)
Outlet	Pressure Outlet	0 (Pa)-gauge pressure	Located at the rear of the domain (fluid outlet)
Symmetry	Stationary walls	No shear stress (0 Pascal)	Located at the top, bottom, left, and right of the domain
Darrieus Turbine	Rotating wall	No-slip	Inside the rotating domain
Savonius Turbine	Rotating wall	No-slip	Inside the rotating domain

Rotating Domain	Rotating Condition	Interior	The rotating fluid flow around the turbine
Static Domain	Fixed Condition	Interior	Non-rotating outer fluid region

To evaluate the hydrodynamic performance of the designed hybrid turbine, several quantitative parameters are employed as indicators of energy extraction efficiency and torque characteristics. The primary parameters include the Tip Speed Ratio (TSR), Moment Coefficient (C_m), and Power Coefficient (C_p). The relationship between the blade's linear velocity and the free-stream velocity is defined as the TSR (λ) through the following equation:

$$TSR(\lambda) = \frac{\omega \cdot R}{V} \tag{7}$$

Furthermore, the turbine's capability to generate rotational torque is expressed as the Moment Coefficient (C_m), while the overall efficiency in converting the kinetic energy of water into mechanical power is represented by the Power Coefficient (C_p) through the following relationship:

$$C_m = \frac{T}{\frac{1}{2} \cdot \rho \cdot A \cdot R \cdot V^2} \tag{8}$$

$$C_p = \frac{P}{\frac{1}{2} \cdot \rho \cdot A \cdot V^3} \tag{9}$$

The parameters utilized in the aforementioned calculations include T , representing the average torque value in Newton-meters (N.m), and ρ as the fluid density in kilograms per cubic meter (kg/m^3). The turbine swept area is symbolized by A in square meters (m^2), while R refers to the turbine radius for both the Darrieus (R_d) and Savonius (R_s) types in meters (m). Lastly, V denotes the free-stream velocity or inlet velocity, specified in meters per second (m/s).

Run Calculation

During the Run Calculation stage, ANSYS Fluent performs the calculation process to generate a solution based on the conditions previously defined in the simulation. This process is crucial because it calculates the fluid flow conditions within the prepared domain. The numerical model parameters used in the simulation are presented in Table 4.

Table 4. Numerical Model Parameters

Parameter	Description
Fluid	Water ($\rho = 998.2 \text{ kg/m}^3$)
Initialization	Hybrid Initialization
Turbulence model	SST K- ω
Solution method	SIMPLE (Semi-Implicit Method for Pressure Linked Equations)
Simulation	Transient
Time Step Size	The time step size was set to represent 10° of rotor rotation per time step.
Max Iterations	250
Convergence criteria	$1\text{e-}6$

The accuracy of the transient simulation results in this study is highly dependent on the determination of the time step size. The time step is determined such that it is capable of capturing

flow phenomena at every 10° of rotor rotation. By utilizing the angular velocity (ω) in rad/s, the time step size (Δt) is calculated using the following equation:

$$\Delta t = \frac{\Delta \theta \cdot \pi}{180 \cdot \omega} \quad (10)$$

Where $\Delta \theta$ represents the azimuthal angle increment of 10° and ω denotes the angular velocity of the rotor. The determination of this value aims to maintain convergence stability and ensure that the transient flow characteristics surrounding the Darrieus and Savonius blades can be observed in detail at each angular position.

RESULTS AND DISCUSSION

This section presents and discusses the research results for each case variation analyzed. In the initial section, the model validation results are presented with reference to previous studies on hybrid turbines, both those conducted using numerical methods and experimental methods. The analysis is conducted by examining performance graphs such as the Power Coefficient (C_p) and Moment Coefficient (C_m), and is supported by flow contour visualizations to understand the flow characteristics around the rotor.

Validation

The numerical model in this study was validated using the research conducted by Saini (Saini & Saini, 2020) by replicating the turbine geometry and simulation conditions in ANSYS Fluent. The reference study compared the performance of a single Darrieus turbine and a hybrid Darrieus–Savonius turbine using both 3D CFD simulations and experimental testing. The simulations were performed under a constant water flow velocity of 1 m/s with variations in Tip Speed Ratio (TSR).

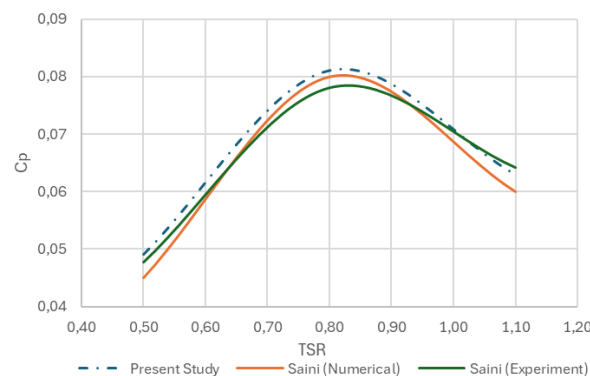


Figure 7. C_p vs TSR Validation Plot

Figure 7 presents the comparison of the C_p –TSR relationship between the present study and Saini’s results. In general, all curves show similar trends, where the C_p value increases with TSR and reaches its maximum value at approximately TSR 0.8. The maximum C_p obtained in this study is very close to Saini’s numerical result, with a deviation of only 1.40%. This small difference indicates that the numerical model developed in this study is capable of accurately predicting the flow behavior and fluid–structure interaction around the turbine. Therefore, the CFD model can be considered well validated and suitable for further performance analysis of the hybrid hydrokinetic turbine.

Grid Independence Test

A grid independence test was conducted to ensure that the simulation results were not significantly affected by the number of mesh elements used. The analysis evaluated five mesh configurations ranging from 1 million to 3 million elements under identical simulation conditions at solidity (σ) 0.95 and TSR 0.8.

Table 5. Grid Independence Test

Number of Elements	Average C_p	Error (%)
1.059.026	0.0769	-
1.515.912	0.0737	4.25
2.045.733	0.0719	2.49
2.547.571	0.0710	1.28
3.054.061	0.0708	0.31

The results of the study show that an increase in mesh density leads to a decrease in the average C_p value, as shown in Table 5. Significant convergence was observed between the 2.5 million and 3 million element meshes, where the error difference decreased to only 0.31%. This indicates that the numerical solution had reached mesh independence and that further refinement would provide only negligible improvements in accuracy.

Based on these results, the mesh with approximately 2.5 million elements was selected for all simulations because it provided a good balance between computational accuracy and simulation efficiency.

Characteristics of C_m -TSR and C_p -TSR

The solidity ratio is an important parameter in determining the hydrodynamic performance of a hybrid hydrokinetic turbine because it affects the interaction between the fluid flow and the turbine blades. In this study, solidity variation was achieved by modifying the chord length of the Darrieus blades, where increasing chord length produced higher solidity values. Variations in solidity directly influenced both the Moment Coefficient (C_m) and Power Coefficient (C_p) across different Tip Speed Ratio (TSR) conditions.

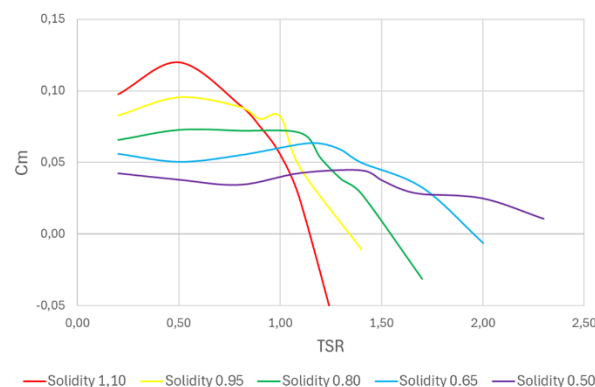


Figure 8. Graph of the Effect of the Solidity Ratio on the Relationship Between the Moment Coefficient (C_m) and the Tip Speed Ratio (TSR)

The results of the study show that turbines with high solidity ratios ($\sigma = 1.1$ and 0.95) produce higher C_m values at low TSR values, as shown in Figure 8, indicating better self-starting capability due

to stronger blade–fluid interaction. However, as TSR increased, the C_m values decreased significantly because of higher drag and blade interaction effects. In contrast, lower solidity ratios ($\sigma = 0.65$ and 0.5) produced lower initial torque but maintained more stable performance at higher TSR values. The intermediate solidity ($\sigma = 0.8$) demonstrated a balanced behavior between initial torque and operational stability.

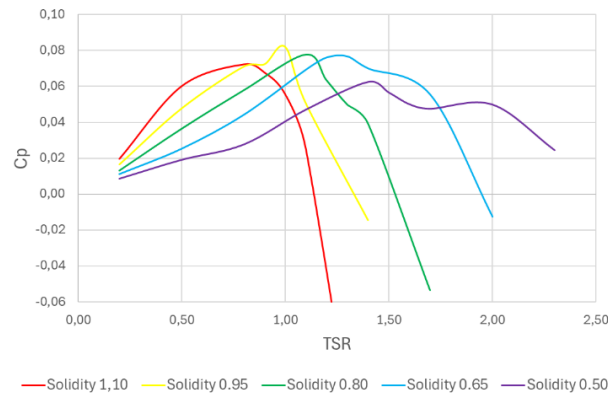


Figure 9. Graph showing the effect of the solidity ratio on the relationship between the Power Coefficient (C_p) and the tip speed ratio (TSR)

A similar trend is also observed in the C_p –TSR relationship, as shown in Figure 9. Turbines with high solidity reach their maximum C_p value at lower TSR values because the larger blade area generates greater thrust at low rotational speeds. Meanwhile, low solidity turbines reached optimum C_p at higher TSR due to lower flow resistance, which allowed the rotor to rotate faster. The highest turbine performance was achieved at a solidity ratio of 0.95 with $TSR = 1$, producing a maximum C_p of approximately 0.082. This condition indicates an optimal balance between lift and drag forces, enabling the turbine to extract energy from the fluid flow more efficiently. Beyond the optimum TSR, C_p decreased for all solidity variations, indicating reduced effectiveness of blade–flow interaction at excessively high rotational speeds.

Hydrodynamically, the optimum performance achieved at a solidity ratio of 0.95 and $TSR = 1$ represents a critical equilibrium between hydrodynamic lift and structural drag. At a high solidity ratio of 1.1, the closely spaced blades create a severe blockage effect, which deflects incoming water away from the rotor and drops the local fluid velocity significantly. Conversely, at lower solidity ratios (0.5 and 0.65), the blade surface area is insufficient to capture enough fluid momentum. At the optimum solidity of 0.95, the blade chord length provides the ideal surface area to intercept incoming flow velocity without choking the turbine. These findings align well with trends observed by (Chhin, 2020; Saini & Saini, 2020), which concluded that increasing solidity close to unity drastically enhances torque capabilities at low operational speeds. However, by pairing this Darrieus configuration with a biomimetic Fibonacci-spiral Savonius rotor, the system effectively mitigates the typical torque ripples often reported in standard hybrid setups, such as those investigated by (Irawan et al., 2024).

Visualization of velocity and pressure contours

Pressure and velocity contours are important parameters in hydrodynamic turbine analysis because they describe the interaction between the fluid flow and the blade surface. Pressure contours illustrate the pressure differences responsible for generating lift and drag forces, while velocity

contours show flow acceleration, wake formation, and flow separation around the turbine blades. Together, these parameters provide a comprehensive understanding of the turbine's energy extraction mechanism and performance efficiency.

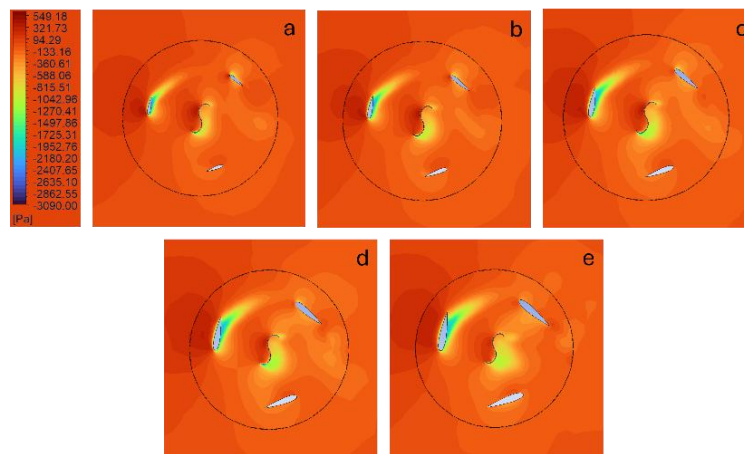


Figure 10. Pressure contours at an azimuthal angle of 290 degrees for a turbine with a solidity of (a) 0.5, (b) 0.65, (c) 0.8, (d) 0.95, and (e) 1.1

Figure 10 shows the pressure contour distribution at an azimuth angle of 290° for different solidity ratios. At low solidity (0.5) and high TSR (1.4), the negative pressure region is still limited, indicating that the blade area is insufficient to maximize lift generation. As solidity increases to 0.65 and 0.8, the negative pressure region becomes larger and more stable, showing improved energy extraction capability. The optimum condition occurs at a solidity ratio of 0.95 and $TSR = 1$, where the pressure gradient is strongest and the flow remains attached to the blade surface, resulting in the highest C_p value. However, at solidity 1.1 and $TSR = 0.8$, hydrodynamic performance decreases due to blockage effects caused by excessive blade density, which increases drag and weakens the pressure gradient.

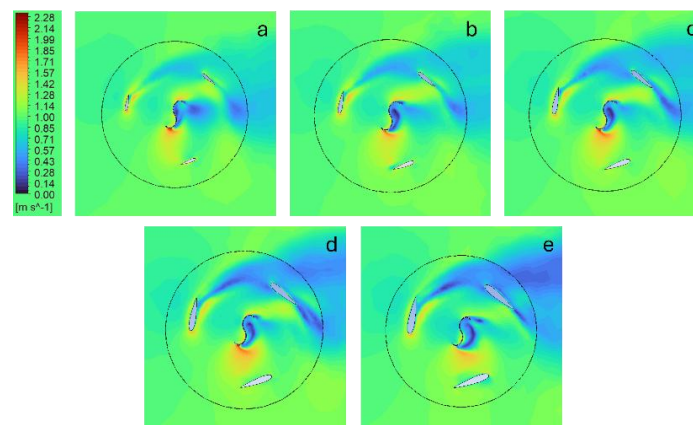


Figure 11. Velocity contours at an azimuth angle of 290 degrees for a turbine with a solidity of (a) 0.5, (b) 0.65, (c) 0.8, (d) 0.95, and (e) 1.1

Figure 11 presents the velocity contours at the same azimuth position. At solidity 0.5, the maximum velocity reaches 2.19 m/s, but the interaction between the flow and blade surface is still weak. Increasing solidity to 0.65 and 0.8 improves local flow acceleration and enhances the low-pressure distribution along the blade surface. The best performance is achieved at solidity 0.95 and

TSR = 1, where the maximum velocity reaches 2.32 m/s and the flow remains attached to the blade, producing stable torque and the highest Power Coefficient (C_p). In contrast, solidity 1.1 causes severe blockage effects, reducing the maximum velocity to 1.69 m/s and increasing drag losses, which ultimately lowers the turbine efficiency.

The velocity and pressure contours strongly corroborate this behavior at an azimuth angle of 290°. At the 0.95 solidity configuration, the localized pressure gradient along the NACA 0018 hydrofoils is at its strongest and remains remarkably stable. The fluid accelerates up to 2.32 m/s around the active blades, effectively delaying flow separation and ensuring that the fluid remains attached to the blade suction side. This localized flow acceleration minimizes energy losses from severe wake interaction and vortex shedding behind the multi-stage system, whereas high solidity (1.1) chokes the flow velocity down to 1.69 m/s, shifting the fluid behavior toward high hydrodynamic drag.

Connecting these simulation trends to real-world applications offers valuable insights for hydrokinetic deployment in low-velocity river environments, which are typical across many regions in Indonesia. In actual river channels, water flow speeds are often low and highly variable. A turbine configured with a high solidity ratio near unity (0.95) ensures that the system can reliably self-start and operate efficiently under low-velocity conditions without requiring complex external starter motors. Furthermore, the multi-stage integration of the Fibonacci Savonius rotor and the optimized Darrieus blades provides smoother torque delivery throughout a full rotation cycle. This structural stability reduces cyclic fatigue on the turbine shaft and bearings, making it highly suitable for long-term, low-maintenance power generation in remote or rural run-of-river installations.

CONCLUSION

Based on the 3D transient numerical investigations, this study confirms that the hydrodynamic performance of the multi-stage hybrid hydrokinetic turbine is highly dictated by the non-linear interaction between blade solidity and the Tip Speed Ratio (TSR). The optimal operating threshold is achieved at a solidity ratio of 0.95 and TSR = 1, yielding a maximum Power Coefficient (C_p) of 0.082 and a stable peak Moment Coefficient (C_m) of 0.082. Contour analysis reveals that this optimal configuration induces localized fluid acceleration up to 2.32 m/s, which effectively stabilizes the pressure gradient, delays flow separation, and minimizes wake-interaction losses. Conversely, exceeding this limit with a high solidity of 1.1 triggers severe blockage effects that choke the local velocity down to 1.69 m/s and drastically increase hydrodynamic drag.

In terms of practical engineering benefits, these findings demonstrate that utilizing a high solidity ratio close to unity (0.95) paired with a biomimetic Fibonacci Savonius rotor provides a highly effective solution for low-velocity, run-of-river environments typical in rural regions. This specific configuration ensures excellent self-starting capabilities without requiring external electrical starters, while simultaneously delivering smoother torque throughout the rotation cycle. Consequently, this structural stability minimizes cyclic fatigue on the turbine shaft and bearings, offering a viable blueprint for developing low-maintenance, long-lasting decentralized power generation systems.

For future research directions, it is highly recommended to conduct physical experimental fluid tests in a water flume channel to validate the transient numerical trends observed in this study. Additionally, future works should explore a broader range of geometric parameters, such as variations in the multi-stage axial overlap ratio, the implementation of dynamic fluid-structure interaction (FSI)

to assess blade structural deflections, or evaluating alternative asymmetrical hydrofoil profiles for the Darrieus section to further suppress dynamic stall penalties.

ACKNOWLEDGMENT

The authors would like to express their sincere gratitude to Prof. Ir. Sutardi, M.Eng., Ph.D., IPM., AEng. for his invaluable guidance, insightful suggestions, and continuous support throughout the completion of this research. The authors also gratefully acknowledge the support and facilities provided by Institut Teknologi Sepuluh Nopember, which greatly contributed to the successful completion of this study. Furthermore, appreciation is extended to all colleagues, laboratory staff, and family members for their encouragement and assistance during the research and journal preparation process.

REFERENCES

- Chhin, S. (2020). *Numerical simulations on hybrid Savonius–Darrieus wind turbine with varying airfoil shape and solidity ratio* [Master's thesis, Institut Teknologi Sepuluh Nopember].
- Damota, J. B., García, J. D. D. R., Casanova, A. C., Miranda, J. T., Caccia, C. G., & Galdo, M. I. L. (2022). Analysis of a nature-inspired shape for a vertical axis wind turbine. *Applied Sciences*, 12(14), Article 7018. <https://doi.org/10.3390/app12147018>
- Daróczy, L., Janiga, G., Petrasch, K., Webner, M., & Thévenin, D. (2015). Comparative analysis of turbulence models for the aerodynamic simulation of H-Darrieus rotors. *Energy*, 90, 680–690. <https://doi.org/10.1016/j.energy.2015.07.102>
- Galdo, M. I. L., Rodríguez García, J. D. D., Couce Casanova, A., Blanco Damota, J., Caccia, C. G., Rebollido Lorenzo, J. M., & Telmo Miranda, J. (2024). Enhanced performance of a hydrokinetic turbine through a biomimetic design. *Journal of Marine Science and Engineering*, 12(8), Article 1312. <https://doi.org/10.3390/jmse12081312>
- Ghafoorian, F., Hosseini Rad, S., & Moghimi, M. (2025). Enhancing self-starting capability and efficiency of hybrid Darrieus–Savonius vertical axis wind turbines with a dual-shaft configuration. *Machines*, 13(2), Article 87. <https://doi.org/10.3390/machines13020087>
- Hartanto, D. (2022). *Analisis numerik pengaruh konfigurasi passive flap terhadap performansi vertical axis hydrokinetic turbine pada kanal terbuka* [Undergraduate thesis, Institut Teknologi Sepuluh Nopember].
- Irawan, E. N., Shibuya, K., Yamashita, K.-I., & Fujita, G. (2024). Solidity effects on small hybrid Savonius–Darrieus wind turbine performance: Experimental analysis with NREL S809 airfoil. In *2024 10th International Conference on Applied System Innovation (ICASI)* (pp. 163–165). IEEE. <https://doi.org/10.1109/ICASI60819.2024.10547931>
- Kadiri, M., Ahmadian, R., Bockelmann-Evans, B., Rauen, W., & Falconer, R. A. (2012). A review of the potential water quality impacts of tidal renewable energy systems. *Renewable and Sustainable Energy Reviews*, 16(1), 329–341. <https://doi.org/10.1016/j.rser.2011.07.160>
- Kanyako, F., & Janajreh, I. (2014). Numerical investigation of four commonly used airfoils for vertical axis wind turbine. In M. O. Hamdan, H. A. N. Hejase, H. M. Noura, & A. A. Fardoun (Eds.), *ICREGA'14–Renewable energy: Generation and applications* (pp. 443–454). Springer. https://doi.org/10.1007/978-3-319-05708-8_35
- Kementerian Energi dan Sumber Daya Mineral. (2023). *Presiden Joko Widodo: Indonesia kaya potensi EBT, manfaat bagi masa depan bumi*. <https://www.esdm.go.id/id/media-center/arsip-berita/presiden-joko-widodo-indonesia-kaya-potensi-ebt-manfaat-bagi-masa-depan-bumi>

- Kumar, R., Kumar, L., Mirjat, N. H., & Harijan, K. (2023). Simulation study on the effect of cover tilt angle of solar still on its productivity. *VFAST Transactions on Mathematics*, 11(2), 63–76. <https://doi.org/10.21015/vtm.v11i2.1586>
- Lanzafame, R., Mauro, S., & Messina, M. (2014). 2D CFD modeling of H-Darrieus wind turbines using a transition turbulence model. *Energy Procedia*, 45, 131–140. <https://doi.org/10.1016/j.egypro.2014.01.015>
- Maldar, N. R., Ng, C. Y., & Oguz, E. (2021). A review of the hybrid Darrieus-Savonius turbine for hydrokinetic applications. In *2021 Third International Sustainability and Resilience Conference: Climate Change* (pp. 424–431). IEEE. <https://doi.org/10.1109/IEEECONF53624.2021.9668166>
- Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8), 1598–1605. <https://doi.org/10.2514/3.12149>
- Mohan Kumar, P., Sivalingam, K., Lim, T.-C., Ramakrishna, S., & Wei, H. (2019). Strategies for enhancing the low wind speed performance of H-Darrieus wind turbine. *Clean Technologies*, 1(1), 185–204. <https://doi.org/10.3390/cleantechnol1010013>
- Raditya Yudistira, D., Nindito, D. A., & Saputra, R. H. (2021). Kinerja turbin hidrokinetik tornado Savonius. *Jurnal Teknika: Jurnal Teoritis dan Terapan Bidang Keteknikan*, 4(2), 181–186. <https://doi.org/10.52868/jt.v4i2.2732>
- Rendi, & Hartadi, B. (2022). Pengaruh kelengkungan deflektor turbin Savonius modifikasi untuk turbin hidrokinetik. *Al Jazari: Jurnal Ilmiah Teknik Mesin*, 7(1). <https://doi.org/10.31602/al-jazari.v7i1.7106>
- Rogelio, M. O., García, J. C., Gómez, L. C., Vera-Wilimek, I., Jaramillo, Ó. A., & Ramirez, J. O. D. (2025). Influence of profile geometry on the self-starting capability of an H-Darrieus turbine. *International Journal of Renewable Energy Development*, 14(4), 694–702. <https://doi.org/10.61435/ijred.2025.61124>
- Saini, G., & Saini, R. P. (2020). Comparative investigations for performance and self-starting characteristics of hybrid and single Darrieus hydrokinetic turbine. *Energy Reports*, 6, 96–100. <https://doi.org/10.1016/j.egyr.2019.11.047>
- T. G., S., Shashikumar, C. M., Gumtapure, V., & Madav, V. (2024). Comprehensive analysis of blade geometry effects on Savonius hydrokinetic turbine efficiency: Pathways to clean energy. *Energy Conversion and Management: X*, 24, Article 100762. <https://doi.org/10.1016/j.ecmx.2024.100762>
- Wilcox, D. C. (2010). *Turbulence modeling for CFD* (3rd ed.). DCW Industries.
- Zamani, M., Nazari, S., Moshizi, S. A., & Maghrebi, M. J. (2016). Three-dimensional simulation of J-shaped Darrieus vertical axis wind turbine. *Energy*, 116, 1243–1255. <https://doi.org/10.1016/j.energy.2016.10.031>