

Design and Optimisation of Cross-flow Turbine for Enhanced Efficiency in Electricity Generation Using Free-Flow Water from Reinforced Tank

Sharvika Pasubathy^{1,2}, Zuliazura Mohd Salleh^{1*,3}, Muhammad Alif Z. Yusri¹, Izuan Amin Ishak⁴, Mohd Fairuz Marian⁵, Nik Normunira Mat Hassan⁶

¹Department of Mechanical Technology, Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia

²Suite 13.01, Penthouse, Wisma E & C, No.2, Lorong Dungun Kiri, Damansara Heights, 50490 Kuala Lumpur, Malaysia

³Energy Technologies Research Group (EnRG), Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia

⁴Department of Mechanical Engineering Technology, Faculty of Engineering Technology, Universiti Tun Hussein Onn Malaysia, 84600 Pagoh, Johor, Malaysia

⁵Department of Professional and Skill Development, Faculty of Technical and Vocational Education, Universiti Tun Hussein Onn Malaysia, 86400 Parit Raja, Johor, Malaysia

⁶Engineered Materials and Structures Research Group, Faculty of Mechanical Engineering, Universiti Teknologi MARA, 40450 Shah Alam, Selangor, Malaysia

ARTICLE INFO

Article history:

Received 20 February 2026

Revised 06 July 2026

Accepted 31 July 2026

Published 15 September 2026

Keywords:

Crossflow turbine

Angle of blades

Number of blades

Ansys CFD

Electricity generation

DOI:

10.24191/jmeche.v23i3.10844

ABSTRACT

Urban and industrial water systems are neglected renewable energy source, even as demand for energy has risen and sustainability targets push back on fossil fuel power generation. In order to fill this research gap, this study aims to design and optimise the vertical free-fall water flow from reinforced concrete (RC) tanks for electricity generation through the crossflow hydro turbine. The impact of blade quantity (15, 25, and 30 blades) and blade angles (20°, 25°, 30°, 35°, 40°, and 45°) on turbine performance was evaluated. A 3D model of the turbine was created in SolidWorks, including detailed blade-geometry considerations to optimise the turbine. The pressure distribution and the efficiency of energy conversion were investigated using computational fluid dynamics (CFD) simulation in ANSYS Fluent based on 36 configurations of turbine, where the inlet velocity was kept constant at 18.26 m/s (corresponding to a head of 0.8 m). The results indicated that both a low blade angle (20°–25°) and a high blade number (30 blades) showed the highest performance, and the 30-blade, 25° configuration generated the maximum pressure of 319,336.3 Pa, torque of 3257.230 Nm, angular velocity of 22.687 rad/s and power output of 73.896 kW. The results serve as a comparative benchmark for future experimental and practical studies into the generation of small-scale hydropower. The

^{1*} Corresponding author. E-mail address: zulia@uthm.edu.my

<https://doi.org/10.24191/jmeche.v23i3.10844>

correct positioning of the outlet of the RC tank with respect to the turbine is identified as a critical parameter for reducing losses and improving efficiency in successive design versions.

Introduction

As the world moves towards cleaner energy systems, the need for renewable technologies that can meet the growth in electricity demand and environmental pressures has increased. Hydropower, which harnesses the kinetic and potential energy of flowing water to produce electricity, is among the most well-developed and dependable renewable energy sources with low emissions (Mohsin et al., 2023; Kumar et al., 2011). While many large hydropower stations still have a significant share of the market, recent advances have focused on small, decentralised hydro-energy projects that can harness low-head and non-river water.

The energy landscape in Malaysia offers ample motivation to look at such alternatives. In 2022, about 81% of the country's electricity generation was from fossil fuels, adding weight to concerns about carbon emissions and the security of energy supplies (Grid System Operator, 2025). Although there are policy commitments to diversify the energy mix, the penetration of renewable energies is limited (Centre for Global Sustainability, 2024). As micro-hydros are embedded in existing water infrastructure, including reinforced concrete (RC) storage tanks, there is an opportunity for this technology to be adopted more widely in Malaysia to diversify the country's energy mix without further land acquisition, major civil works or natural water bodies (Kementerian Ekonomi Malaysia, 2023).

The reinforced water tank, which is frequently used in residential and commercial structures, releases water into vertical channels, resulting in uniform free flow conditions. This controlled discharge could be used for micro-hydropower to generate small-scale electricity. To realise this potential, a turbine technology developed to work under low head, variable flow and non-riverine conditions is required. A crossflow turbine, or Banki-Mitchell turbine, has demonstrated promise in such conditions, as its simple geometry and ease of manufacture make it accessible, and it is effective over a wide range of flow variations (Ho-Yan & Lubitz, 2011; Quaranta et al., 2022).

The significance of geometrical parameters, such as blade angle, number of blades and nozzle configuration, on the performance of crossflow turbines is apparent from a considerable amount of literature. In the study by Choi et al. (2008), it was found that the runner blade geometry and the nozzle shape have a significant effect on the internal flow characteristics and the hydraulic efficiency. In a similar manner, Chen & Choi (2013) reported that the guide vane angles are important in optimising the flow incidence and minimising the hydraulic loss. Recent studies have furthered optimisation by using computational methods. CFD and FDA-based analysis by Prabowoputra et al. (2023a, 2023b) revealed that certain combinations of blade number and blade angle maximise turbine output. Further experimental evidence also backs up these claims, such as Lillahulhaq et al. (2022), who reiterate the effects of the blade attack angle on the performance of the turbine. A trend of ongoing innovation can be observed, with new designs for use in gravitational vortex systems expanding the operational range of the crossflow-based turbines (Alfeuz et al., 2024).

Despite this progress, there is a large research gap. Most of the existing optimisation studies consider conventional hydropower applications like river-based or channel-based flow and rarely consider free-flow water from raised storage tanks where water is released through a short vertical pipe with limited head but high velocity. The flow profile at the outlet of an RC tank is unique, and some issues it poses are not covered completely by the existing literature on crossflow turbines, such as inlet velocity alignment, blade angle

appropriateness, and flow-through behaviour in the two interaction stages in the turbine. A conceptual explanation of this application is given in Fig. 1, where a turbine-generator system is integrated into a residential building with an elevated RC water tank.

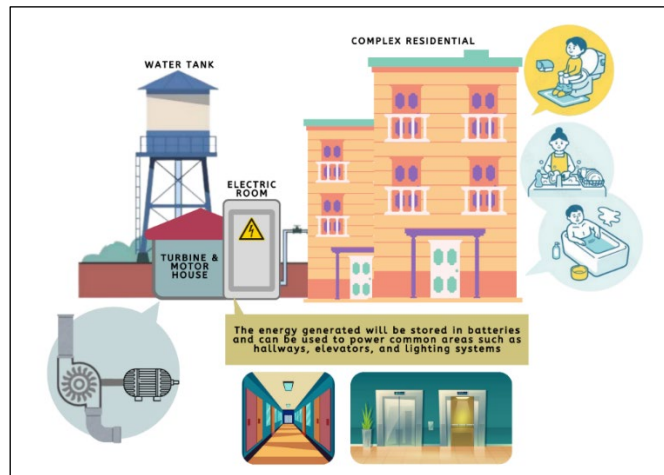


Fig. 1. Conceptual illustration of a crossflow turbine integrated with an elevated RC water tank for free-flow power generation in a residential building.

The present study, therefore, aims at designing and optimising a crossflow turbine especially for free-flow water of a reinforced tank system. The study compares the different performances of the turbine with various blade angles and blade numbers using SolidWorks for geometric modelling and ANSYS Fluent for CFD analysis at velocity conditions typical of the water outlet of a domestic water tank. The scope includes conceptual design, simulation-based assessment and performance prediction, particularly focusing on finding the optimum turbine configuration for micro-generation applications.

Material and Methods

A computational design and simulation-based approach was used to examine the crossflow turbine performance operating under free-flow discharge from a reinforced tank system. Different turbine configurations with varying numbers of blades and blade angles were created using SolidWorks and analysed using ANSYS CFD to check pressure distribution, torque output, angular velocity, and power output for some representative outlet velocity conditions. In this way, systematic performance comparison can be done, and an optimised turbine configuration can be identified to generate small-scale energy production (Homa et al., 2022).

Model and geometry

Using SolidWorks, the models and geometry of the crossflow turbines were designed. There were 18 designs created in total, with the combination of 3 different numbers of blades and 6 different angles. The completed design of a crossflow turbine with 15 blades and an inlet angle of 20° is illustrated in Fig. 2, and a cross-section of a crossflow turbine is shown in Fig. 3. The turbine blades' detailed configuration is shown in Fig. 4, which denotes the differences in the blade angles.

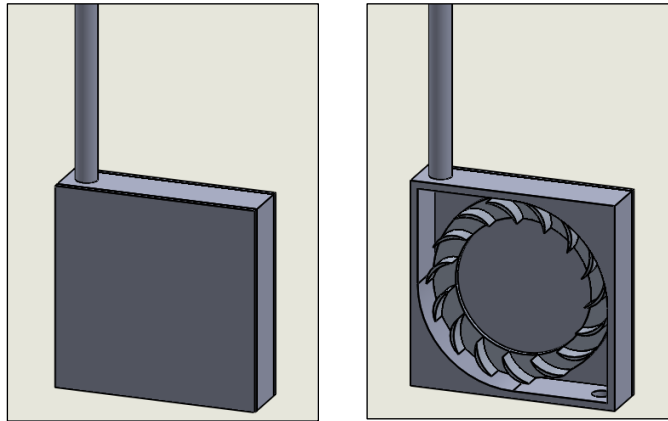


Fig. 2. Completed design of crossflow turbine featuring 15 blades with an inlet angle of 20° .

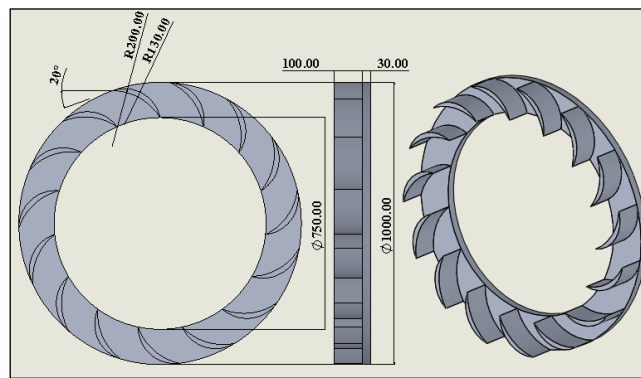


Fig. 3. Cross section of crossflow turbine featuring 15 blades with an inlet angle of 20° .

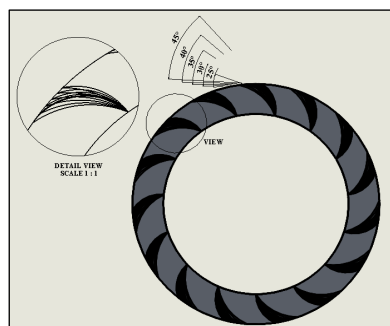


Fig. 4. Crossflow turbine featuring 15 blades with a consequent arrangement of inlet angles of 20° , 25° , 30° , 35° , 40° , and 45° .

Computational domain and mesh configuration

A three-dimensional geometry of the crossflow turbine system of the inlet pipe, turbine casing and runner was created in SolidWorks and imported into ANSYS Fluent. The tetrahedral meshing with prismatic inflation layers (PBL: 8 layers, growth rate = 1.2 and first-layer height = 0.2 mm) was applied to all the solid walls to capture velocity gradients close to the wall, which is essential for the prediction of pressure and torque. Blade surfaces (0.003–0.005 m) and leading/trailing edges were refined, whereas coarser elements (0.010–0.015 m) were used in the casing and pipe areas. Specifically, curvature capture was turned on, and defeaturing was used to eliminate unnecessary details for computational efficiency.

The mesh contained a total of 264,014 tetrahedra and 58,922 nodes. Quality assessment was performed, and the average skewness was 0.2315 (max 0.78), and the average orthogonal quality was 0.82 (min 0.62), both of which are within acceptable CFD limits (skewness < 0.85, orthogonal quality > 0.15). To assess the torque variation, a grid independence study was performed for a typical configuration (25 blades, 25°), and the results showed an average difference of less than 2% between the torque obtained from the present configuration and a refined mesh (1.2 million elements). To allow for valid comparative evaluation, identical mesh specifications were applied uniformly across all blade-angle and blade-quantity configurations. The mesh was relatively coarse compared to the high-resolution turbine simulation studies (usually 1–5 million elements) but was coarse enough to allow for ease of computational feasibility for the 36 simulation cases ($\approx$ 4 hours each) while maintaining the objective of the comparative study in which the relative ranking validity has been preserved throughout the application of the same mesh. Future studies using turbulent modelling should use highly resolved meshes with improved near-wall resolution.

Boundary conditions

Boundary conditions are crucial in CFD because they specify the physical conditions at the domain boundaries. In this simulation, the parameters are given in Table 1 and Table 2:

Table 1. Parameters of 3D model

Parameters	Specifications
External diameter of turbine	1000 mm
Internal diameter of turbine	750 mm
Number of blades	15, 25, 30
Inlet blade angle	20°, 25°, 30°, 35°, 40°, 45°
Length of pipe	17000 mm
Outer diameter of pipe	120 mm
Inner diameter of pipe	100 mm

Table 2. Parameters of Ansys simulation

Parameters	Specifications
Inlet velocity	18.26 m/s
Density of water	1000 kg/m ³
Acceleration due to gravity	9.8 m/s ²
Viscous	k-SST

Governing equations

This flow simulation was carried out using ANSYS Fluent software with a pressure-based solver under steady-state conditions. The software is based on the conservation equations for mass and momentum in incompressible Newtonian flow. These equations constitute the basic mathematical structure that can be

used to solve the velocity and pressure fields inside the computational domain as well as to determine the hydrodynamic forces, torque, and power output. For incompressible flow, the continuity equation (conservation of mass) becomes:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (1)$$

where u , v , and w are the velocity components in the x , y , and z directions (m/s). This equation is imposed on the velocity field to ensure that the divergence of velocity is zero everywhere in space, which is assumed for water as the working fluid because it has no compressibility.

The momentum equation (Navier–Stokes equation) solved by Fluent is:

$$\rho \left(\frac{\partial V}{\partial t} + V \cdot \nabla V \right) = -\nabla P + \mu \nabla^2 V + \rho g \quad (2)$$

The present simulations are steady state; the transient term is equal to zero ($\partial V / \partial t = 0$); so, the equation becomes:

$$\rho (V \cdot \nabla V) = -\nabla P + \mu \nabla^2 V + \rho g \quad (3)$$

where ρ is the fluid density (kg/m³), P is the pressure (Pa), μ is the dynamic viscosity (Pa·s), and g is the gravitational acceleration vector (m/s²). The left-hand term is the convective acceleration, and the right-hand terms represent pressure forces, viscous diffusion, and gravitational body forces, respectively.

The governing equations were discretised by the finite volume method in the ANSYS Fluent software package. The second-order interpolation scheme was used for the discretisation of the pressure field. The second-order approximation for this method is obtained by using the pressure gradient for each cell to reconstruct pressure at the cell faces:

$$P_f = P_c + \nabla P_c \cdot \Delta r \quad (4)$$

where P_f is the pressure at the cell face (Pa), P_c is the pressure at the cell centroid (Pa), ∇P_c is the pressure gradient at the cell centroid, and Δr is the vector from the cell centroid to the cell face. Given that the second-order scheme is capable of capturing pressure variations across the entire computational domain and alleviates the problem of numerical diffusion, it is considered more accurate than the first-order scheme. This is especially helpful in simulations with complex flow patterns like those around the crossflow turbine blades.

The momentum equations were computed and written in a discretised form by using the second-order upwind scheme. The second-order upwind method is different from the first-order upwind method as it attempts to reconstruct a value of the transported variable on the cell face by using the cell-centred value and the spatial gradient of the transported variable:

$$\phi_f = \phi_U + \nabla \phi_U \cdot \Delta r \quad (5)$$

where ϕ_f is the transported variable at the cell face, ϕ_U is the value of the transported variable in the upstream cell, $\nabla\phi_U$ is the gradient of the transported variable in the upstream cell, and Δr is the vector from the upstream cell centroid to the cell face. For momentum transport, ϕ represents the velocity components (u , v , or w). This higher-order interpolation gives an improvement in the resolution of velocity gradients and eliminates numerical diffusion, making it ideal for resolving the behaviour of the flow around the turbine blades.

The second-order pressure interpolation together with the second-order upwind momentum discretisation makes sure that the numerical solution is sufficiently accurate with respect to the spatial direction and that artificial diffusion is reduced. To compare the effects of blade geometry in these schemes, the same schemes were used in all the simulation cases.

Analytical formulations

The hydrodynamic performance of the crossflow turbine runner was analysed by both theoretical formulations and CFD-coupled calculations. The pressure field was calculated using ANSYS Fluent simulation, and the primary quantities such as hydrodynamic force, torque, angular velocity and power output were derived from the calculated pressure field.

For all of the CFD simulations in this study, the inlet velocity was calculated using Torricelli's theorem, which is based on the basic principle of conservation of energy:

$$v = \sqrt{2gH} \quad (6)$$

This velocity was uniformly applied as a boundary condition on the inlet cross-sectional area ($A = \pi(0.1)^2/4 = 0.007854 \text{ m}^2$), which was the same for all the tested configurations in this comparative study. In CFD literature related to hydraulic turbines, it is common to adopt a theoretical inlet velocity to be used as a reference for comparison with other geometric analyses. The efflux equation for Torricelli was presented by Epple et al. (2017) and proved to be valid as a theoretical relation for hydraulic head-velocity by experiments employing analytical momentum equations and CFD models. This method has been used by Sammartano et al. (2016) and Sinagra et al. (2016) for crossflow turbine CFD to provide a standardised starting point for comparative validation of designs, which has the benefit of reducing computational complexity whilst maintaining the ability to rank design alternatives. Benzon et al. (2015) provides a particularly relevant precedent in their study of Pelton injector geometries, which explicitly used an ideal jet with a constant velocity profile as a standard inlet condition to rule out the effects of upstream hydraulic variability and allow direct attribution of the differences in the performance of the various injector geometries to the geometries themselves. Quaranta & Revelli (2017) also show how the Torricelli term $\sqrt{2gH}$ is used as a theoretical reference to normalise hydraulic performance measures to carry out comparisons among different water wheels, while Reis & Carvalho (2018) clearly state the use of the Torricelli velocity (a theoretical inlet boundary condition) in the OpenFOAM CFD simulation of an Archimedes screw turbine.

It is important to note that the aim of this study is not to determine the absolute field performance of a specific installation with a known set of pipe characteristics, but to perform a comparative geometric analysis under controlled theoretical conditions. In a design application for a specific site, practitioners would use proper correction factors (e.g., Darcy-Weisbach friction loss, minor loss coefficient) to modify the velocity at the inlet based on actual pipe characteristics, as is typical in hydraulic turbine design (Quaranta & Revelli, 2017; Sammartano et al., 2016). The idealised inlet condition used here is therefore not a mistake, but rather a methodological decision well-grounded in the experience of other comparative

turbine CFD studies that has a direct bearing on the main goal of this study, namely, to isolate the effects of the blade geometry.

Hydrodynamic loading created during an interaction between the incoming water jet and the turbine blades produces the rotational motion and torque production. In the present work, the resultant force on the blade surface was computed based on the pressure–area relationship:

$$\text{Force}, F = P \times A \quad (7)$$

where F is the resultant hydrodynamic force (N), P is the spatially averaged pressure on the blade surface obtained from CFD (Pa), and A is the projected interaction area of the blade surface (m²). The exact hydrodynamic force is computed as the surface integral of local pressure over the blade geometry:

$$\vec{F} = \int_S P \hat{n} dS \quad (8)$$

where $\hat{n}$ is the unit outward normal at each surface element dS and P is the local CFD pressure. This expression reduces to $F = P \times A$ under two assumptions: (1) the pressure is represented by its spatial average, $P = \frac{1}{A} \int_S P dA$; and (2) the direction of the resultant force is given by the projected area normal to the dominant flow incidence, i.e., $\vec{F} \approx P \cdot A_{proj}$. If the same surface integral, same mesh, same inflow conditions and the same solver settings are kept for all configurations, the relation between the surface integral and the product of the pressure and the area is linear, and hence the relative ranking between the hydrodynamic loading, the torque and the power output of the blades is also preserved.

The application of pressure-based force metrics that are simplified can be justified based on the literature available in the context of comparative and parametric CFD studies. Low-fidelity 2D pressure and flow response metrics are used explicitly by Tiwari et al. (2024) for parametric design sweeps of a Francis turbine runner, with 3D simulation only being used for cross-validation of top-performing candidates; this highlights that simplified pressure-based representations can be used to identify relative performance trends during geometric optimisation. Marsh et al. (2016) also use a CFD pressure field to determine hydrodynamic loading to compare the blade configurations for vertical-axis tidal turbines, but do not report actual load magnitudes; only relative trends based on the pressure-derived loading are used to infer relative structural performance trends across the blade configurations. Bowman et al. (2021) also shows that the approach of combining the blade-resolved CFD pressure and viscous forces into parameterised force coefficients to evaluate the performance in a reduced-order fashion is valid, with the reduced-order quantity demonstrating a thorough performance comparison of the parent high-fidelity simulation.

The same philosophy is followed in the present investigation to a similar end: the aim is to provide a relative assessment of the effects of blade angle and blade quantity on torque generation and on the power output for the same operating conditions, but not on the prediction of the magnitude of the absolute structural loads. As a result, a comparative performance evaluation can be made in a consistent manner for the various simulated output configurations using the pressure–area formula in a computationally efficient way. It is worth highlighting that the current system is not a fully coupled fluid–structure interaction (FSI) analysis. In this work, hydrodynamic flow behaviour and loading characteristics under pressure are the focus, and the deformations of the blades, transient structural responses, and two-way coupling between fluid and solid are not considered. A rotational moment is generated around the turbine shaft when the hydrodynamic force on the blade surface acts as a force to cause it. The torque was calculated from:

$$\text{Torque, } \tau = F \times r \quad (9)$$

where τ is the generated torque (N·m), F is the resultant hydrodynamic force on the blade surface (N), and r is the radial distance from the turbine centre to the blade interaction point (m). The exact torque around the turbine shaft can be computed from the surface integral of the moment that each of the elements of the pressure force contributes:

$$\vec{\tau} = \int_S (\vec{r} \times P \hat{n}) dS \quad (10)$$

where $\vec{r}$ is the position vector from the shaft centre to each surface element dS , and $P \hat{n} dS$ is the local pressure force vector. This simplifies to $\tau = F \times r$ if two consistent assumptions are made: (1) the pressure force resultant F is at a characteristic radial distance r (e.g., blade centroid or mid-span), and (2) the force vector is perpendicular to the radial vector, giving maximum moment. In the current comparative context, in which blade geometry changes while r is taken consistently throughout the configurations, the simplified moment-arm formulation is sufficient to determine the relative levels of torque without having to integrate the entire surface.

This moment-arm method is common for the calculation of pressure-induced blade loading for comparative turbine CFD work. To do this, Naseem et al. (2022) used torque as the key performance parameter in a parametric study of crossflow turbine blade design parameters and directly compared the trends of torque in different geometries. Likewise, Kim et al. (2021) took CFD-calculated torque as the criterion to compare the blade numbers of gravitational water vortex turbines. In the current study, where r is kept equal for all of the blade configurations, the comparative rankings of torque take on the same values as the hydrodynamic force and are thus interpretable and consistent for blade angle and blade quantity effects.

A kinematic angular velocity, which was idealised, was calculated from the tangential flow velocity obtained from CFD with the assumption of perfect momentum transfer at the blade surface:

$$\omega_{ideal} = \frac{V_{max}}{r} \quad (11)$$

where ω_{ideal} is the idealised angular velocity (rad/s), V_{max} is the maximum tangential flow velocity from CFD (m/s), and r is the turbine runner radius (m). It should be noted that ω ideally corresponds to the angular velocity of the turbine runner solid body, which is controlled by the unsteady torque balance:

$$I \frac{d\omega}{dt} = \tau_{fluid} - \tau_{load} \quad (12)$$

where I is the moment of inertia of the rotating assembly, τ_{fluid} is the hydrodynamic torque, and τ_{load} is any external resisting torque. The quantity $\omega_{ideal} = V_{max}/r$ is not an actual predicted shaft angular velocity, but an idealised kinematic upper bound, based on the velocity triangle condition at the runner inlet, under steady-state, no-load and perfect momentum transfer conditions where the peripheral blade speed equals the tangential flow velocity. This condition represents the theoretical no-load (runaway) speed limit and not the best efficiency operating point, which Sammartano et al. (2015) have determined to be where the inlet particle velocity is about twice the peripheral velocity ($\omega_{opt} \approx V_{max}/(2r)$). The adoption of $\omega_{ideal} = V_{max}/r$ here is only a consistent kinematic reference for comparative assessment of the different blade geometries, but not for optimum or actual operating speed. Because all simulations were performed with the same inlet, the calculated ideal ω trends present a common relative measure of the

rotational response for these various blade configurations, but do not necessarily represent an experimentally verified prediction of shaft speed or transient rotor-dynamics modelling.

The mechanical power output of the turbine runner was calculated to be:

$$Power = \tau \times \omega \quad (13)$$

where P is the mechanical power (W), τ is the torque (N·m), and ω is the angular velocity (rad/s). This basic definition of mechanical power of a rotating machine is used for any performance analysis on a turbine. To find out the best geometry and to rank the blade configuration in different parametric designs, Velichkova et al. (2023) and Woldemariam & Lemu (2019) used $P = \tau\omega$ in their turbine studies conducted using CFD. In the current work, the power output combines both loading (torque) and rotation (angular velocity) responses and is used to address scenarios where a configuration might have a high torque but a low angular velocity, or the other way around. Under constant CFD conditions (inflow conditions, mesh topology, and solver settings), all the torques and angular velocities are comparable, and reliable geometric performance ranking can be carried out without the need for transient rotor dynamics or coupled shaft modelling.

Result and Discussion

Three turbine blade configurations (15 blades, 25 blades, 30 blades) were investigated at six blade angles (20°, 25°, 30°, 35°, 40°, 45°). The results were presented in a systematic manner and critically discussed with the aim of preserving the analytical clarity and technical relevance. The effect of the interaction between these two parameters (blade number and blade angle) was investigated in detail, and the combined effect of the two on turbine performance and efficiency optimisation was highlighted.

Pressure distribution characteristics

This type of pressure distribution on turbine blades offers insight into the hydrodynamic loading and momentum exchange mechanisms of crossflow turbine performance. The crossflow turbine is a type of impulse turbine and is based on the idea that effective deceleration and redirection of the jets during blade interaction produce the local development of pressure, which is a measure of energy transferred (Lukeš et al., 2025). The distinction is important when interpreting the values in Table 3, since in impulse machines, the amount of static pressure at the runner inlet and outlet remains about the same, and a change in pressure on the blade surfaces is the result of dynamic (velocity-induced) loading.

Table 3. Overall pressure data

Blade angle	Pressure (Pa)		
	15 blade	25 blade	30 blade
20	312274.2	317972.6	318383.0
25	310146.1	314490.1	319336.3
30	308471.8	310623.6	313017.5
35	306981.6	301797.1	309907.6
40	300051.1	299924.3	304522.6
45	296327.1	305641.4	296672.7

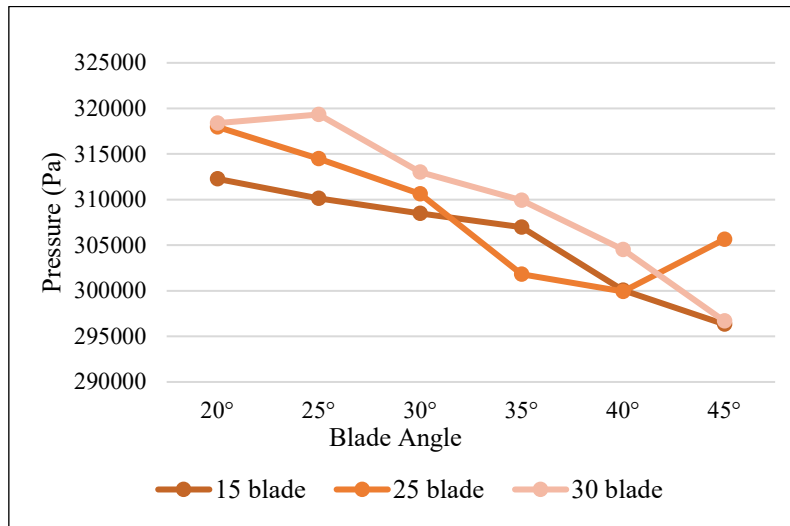


Fig. 5. Overall pressure data.

As the number of blades was increased, the pressure magnitudes on the blade surfaces became greater, especially at moderate blade angles as shown in Fig. 5. The peak pressure for the 30-blade configuration was 319,336.3 Pa at 25°, while it was 312,274.2 Pa at 20° for the 15-blade configuration. This is due to the rise in the frequency of jet–blade interaction that enables stagnation pressure on the leading surface of the blade to increase during the first phase of energy extraction. This behaviour is similar to the results obtained by Assefa and Tesfay (2025a), who have undertaken parametric CFD simulations and shown that the hydraulic efficiency increased by around 6 percentage points when the number of blades was increased from 24 to 28 and that this effect was attributed to better continuous momentum transfer and steadier pressure gradients across the runner (Assefa & Tesfay, 2025a). Likewise, Nishi et al. (2017) found that for an undershot crossflow runner, increasing the number of blades resulted in higher mean torque and lower torque fluctuations, corresponding to more stable jet–blade engagement during each revolution.

The number of blades does not, however, have a linear relationship with pressure. At 45°, the 25-blade configuration (305,641.4 Pa) exceeded both the 15-blade (296,327.1 Pa) and 30-blade (296,672.7 Pa) configurations. At high blade angles, this reversal indicates that an excessive number of blades creates an interblade flow blockage and a narrower passage that does not increase the development of productive pressure. Choi et al. (2008) reported similar characteristics in their CFD studies of crossflow hydraulic turbines, where the suboptimal blade geometry induced recirculating flow inside the runner passages, leading to significant hydraulic loss in both the energy extraction stages. In addition, Wu et al. (2022) showed that for a hydrokinetic crossflow rotor, further increasing the number of blades from 2 to 4 reduced the P_c from 37.5% to 30.2%, highlighting that the number of blades beyond the optimum for a particular application will result in blockage and interaction losses, which will offset gains from increased frequency of engagement.

Pressure development was highly affected by the blade angle. Pressure dropped steadily as the number of blades was increased, from 20° up to 40°; then there was a slight increase for the 25-blade configuration at 45° (305,641.4 Pa). An angle of 20° to 30° was found to generate a balance in pressure distribution and cause good jet incidence while keeping the possibility of flow separation to a minimum, whereas angles in the high range of 35° to 45° increased the turbulence and hydraulic losses, thereby making the transfer of momentum less effective (Haryono et al., 2019; Lillahulhaq et al., 2022). These trends are well supported by recent optimisation studies. Assefa and Tesfay (2025b) found that the optimum blade entry angle of 15°

and a water admission angle of 60° provided the maximum hydraulic efficiency of 84.91%, whereas the conventional combination of higher angles (30° , 90°) experienced dispersion of the jet inside the chamber and high internal energy losses. Choi et al. (2008) also measured the influence of blade angle changes on the pressure distributions and found that the changes in pressure were large, but the changes in local velocity were relatively small, again indicating that the development of pressure is dominated by the flow incidence geometry.

Repeated pressure peaks that occur at fixed places on the blade for all configurations show stagnation zones where there is a significant velocity drop at the impact zone, which is closely related to torque production (Tuapetel et al., 2025; Prabowoputra et al., 2023a). These stagnation zones also depend on the geometry of the blade profile, as shown by Assefa and Tesfay (2025c), with the aerodynamic profiles providing lower exit pressure and reducing the shedding of vortices for better flow attachment and a coherent pressure difference between the surfaces of the blades. In addition, Wu et al. (2022) found that the torque yields were asymmetric at varying rotational positions for different surfaces of the blades, further emphasising the need to understand the averaged values of pressure with the flow behaviour at the instant. Significantly, the findings showed that there is no direct link between turbine performance and pressure level; that is, the highest pressure (319,336.3 Pa) was recorded for the 30-blade configuration at 25° , but this does not necessarily correspond to the highest power output, since high pressure concentration may increase hydraulic losses and structural loading (Assefa & Tesfay, 2025a; Monsalve et al., 2024). In the subsequent torque, angular velocity and power analyses in the following section, this compromise is eliminated by applying both pressure-driven loading and rotational response.

Torque performance

Torque production was found to be dependent both on the number of blades and the blade angle, with the specific relation between these variables varying by configuration and not necessarily being monotonic. Generally, the more blades there were on the turbine, the more torque was produced at the lower blade angles (20° – 30°), indicating more transfer of momentum from the water jet to the runner (Fig. 6). The torque values are shown in Table 4 for all the configurations.

The 30-blade configuration had the highest torque value of 3257.230 Nm at 25° , which was 3.0% greater than that of the 15-blade configuration (3163.490 Nm) at 25° . This enhancement is brought about by the greater interaction of the blades with the jet, which enables the higher jet kinetic energy to be utilised in the crossflow process in both stages. Assefa and Tesfay (2025a) employed parametric CFD simulations to examine the effect of blade number (from 24 to 28) on hydraulic efficiency, finding a similar result, with a slight improvement in hydraulic efficiency due to the increased number of blades, which resulted in more continuous transfer of momentum and steadier torque generation around the circumference of the runners. Tuapetel et al. (2025) also found the torque to be dependent on the blade angle; moderate angles were found to create hydraulic losses due to the blade angle.

Table 4. Torque (Nm)

Blade angle	Torque (Nm)		
	15 blade	25 blade	30 blade
20	3185.197	3243.321	3247.507
25	3163.490	3207.799	3257.230
30	3146.412	3168.361	3192.779
35	3131.212	3078.330	3161.058
40	3060.521	3059.228	3106.131
45	3022.536	3117.542	3026.062

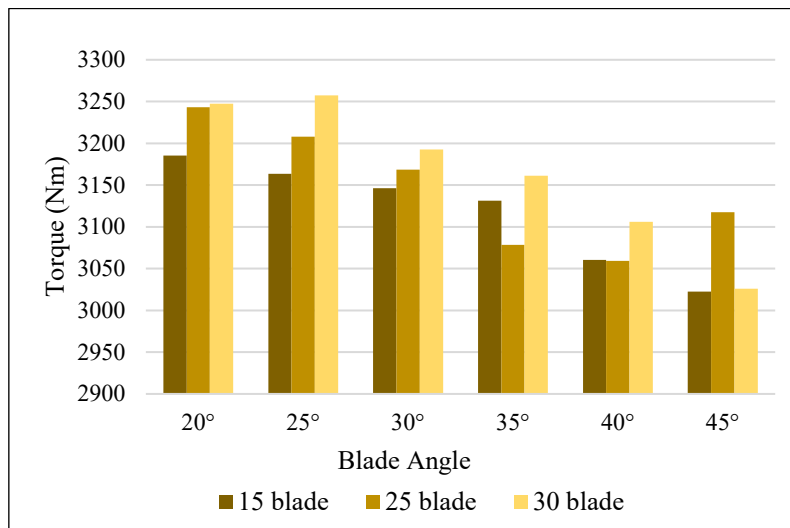


Fig. 6. Torque versus angles for 3 different blade numbers.

However, the higher number of blades is not always beneficial in terms of torque. The 25-blade configuration (3078.330 Nm) was found to be lower than both the 15-blade (3131.212 Nm) and the 30-blade (3161.058 Nm) configurations at 35°, and this indicates that interblade blockage at higher angles can overcome the engagement advantage. With CFD analysis, Choi et al. (2008) showed that recirculating flows may occur in the runner passages due to suboptimal blade angles, leading to hydraulic losses in both stages of the turbine, which get more severe as the number of blades increases. The literature also reported that increasing the number of blades too much might cause blockage, meaning that the number of blades must be carefully optimised and not maximised (Maulana et al., 2023).

The angle of the blade was a key factor in torque generation. Optimum blade angles of 20° to 25° were found to give maximum torque values for all of the configurations, giving an optimum jet incidence angle and momentum redirection. The decreased torque at high angles (35° to 45°) is believed to be caused by increased flow separation and energy dissipation, which aligns with the results of experimental studies on crossflow turbines by Alfeuz et al. (2024), as well as with the observations made by Lillahulhaq et al. (2022), who observed that higher blade angles led to lower efficiency because of early flow separation.

One remarkable deviation from the monotonic behaviour is the torque value for 25 blades at 45° of 3117.542 Nm as compared to 3059.228 Nm at 40°, a 1.9% improvement. This behaviour is more likely a result of complex interactions between the unsteady flows at extreme angles of attack of the blades than a simple secondary impulse mechanism. Intracycle angular velocity simulations by Dave et al. (2022) revealed that large apparent angles of attack create trailing-edge vortices and dynamic stall phenomena that cause non-monotonic fluctuations in torque; Wu et al. (2022) described the regions of torque surface cancellation at high angles of attack, where the contributions from the upper and lower surfaces partially compensate for each other, causing transient oscillations rather than a systematic recovery. The local increase observed for the 25-blade case is thus taken to be a transient unsteady phenomenon particular to this configuration, and conditions like these are generally avoided in practical designs for reasons of associated flow instability and risk of structural vibrations.

Angular velocity behaviour

Generally, velocity showed an inverse trend with blade number; in most cases, the 15-blade configuration had a higher velocity compared to the 25- and 30-blade configurations (Fig. 7). The full values of the angular velocities for all configurations are presented in Table 5.

Table 5. Angular velocity (rad/s)

Blade angle	Angular velocity (rad/s)		
	15 blade	25 blade	30 blade
20	22.804	22.463	22.368
25	23.644	22.284	22.687
30	22.960	23.264	22.685
35	22.506	22.890	21.311
40	23.414	22.018	22.896
45	24.101	22.742	23.386

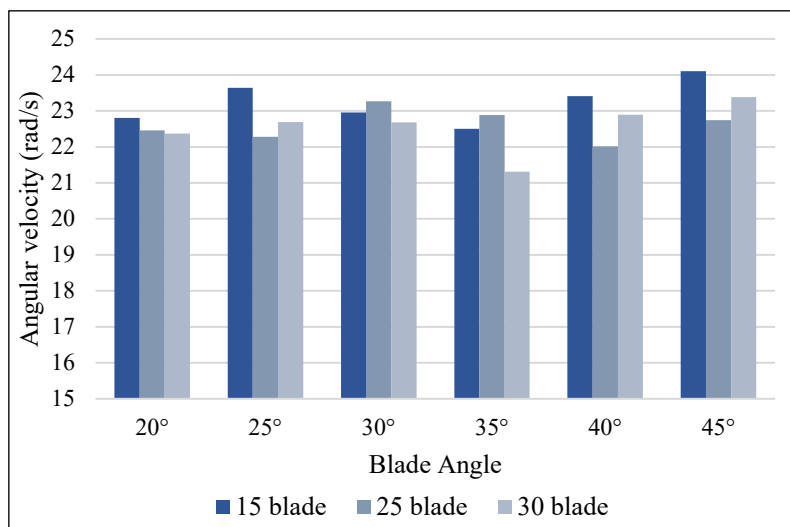


Fig. 7. Angular velocity versus angles for 3 different blade numbers.

It was observed that the maximum angular velocity obtained by the 15-blade configuration of the wind turbine was 24.101 rad/s at 45°, whereas the 25-blade and 30-blade wind turbine configurations had maximum angular velocities of 22.742 rad/s at 45° and 23.386 rad/s at 45°, respectively. This trend is in good agreement with the smaller blade area and lower flow blockage of the 15-blade runner, which will allow the runner to accelerate more freely under the same flow conditions. It must be emphasised, however, that this inverse relationship does not necessarily occur for all combinations. However, Sahu et al. (2025) found that for a particular geometry, the maximum performance of the turbine was obtained at 20 blades and 600 rpm, which shows that the optimum number of blades and speed depend on the application and are dependent on loading conditions, admission geometry, and runner inertia.

In general, the number of blades was found to have an inverse effect on the angular velocity due to increased hydrodynamic resistance. This behaviour is typical in the design of crossflow turbines, where increasing the density of blades leads to a reduction in the flow passage area, which in turn reduces the ability to accelerate the turbine's rotation. The effect of the blade angle on the angular velocity was less

systematic than in torque. Moderate angles (20°–30°) were the most effective in balancing force generation and flow continuity.

Interestingly, the 15-blade configuration was found to exhibit a monotonic increase in the angular velocity with increasing blade angle (from 22.804 rad/s at 20° to 24.101 rad/s at 45°). The counterintuitive trend is explained by the reduced blade surface area of low-blade-count runners; flow separation at high angles causes less cumulative drag if there are fewer blades to experience the separated flow. In the 30-blade configuration, this same separation causes a compounding drag effect, which leads to a minimum angular velocity of 21.311 rad/s at 35° out of all the various cases. Minimisation of peak apparent angles of attack by intracycle control was shown to reduce dynamic stall and enhance rotational consistency (Dave et al., 2022), highlighting the effect of angular velocity on the separation behaviour. The angular velocity values stated are kinematic estimates based upon maximum tangential flow velocity and under idealised assumptions of momentum transfer and exclude transient rotor dynamics, bearing friction, and shaft inertia.

Power output performance

Power output ($P = \tau \times \omega$), the most complete expression of turbine response, combines loading capacity and rotation in a single measure, and is directly related to the effectiveness of the energy-conversion process. Complete power output values for all configurations are shown in Table 6.

Table 6. Power (kW)

Blade angle	Power (kW)		
	15 blade	25 blade	30 blade
20	72.637	72.855	72.640
25	74.799	71.483	73.896
30	72.243	73.708	72.428
35	70.472	70.464	67.365
40	71.658	67.357	71.117
45	72.845	70.899	70.766

When the blade angles in all blade configurations were between 20° and 30°, the power output obtained was at a maximum (Fig. 8). The 15-blade configuration had its highest power at 25°, at 74.799 kW; the 25-blade configuration had its highest power at 30°, at 73.708 kW; and the 30-blade configuration had its highest power at 25°, at 73.896 kW. The results of the 15-blade runner at 25° showed the highest overall power value (74.799 kW), and it was also found that the optimum value of power did not correspond to the maximum number of blades, which is the key to solving the torque–speed compromise; the 15-blade runner at 25° was optimum. This result aligns with a study by Wu et al. (2022) that showed a higher power coefficient of a 2-blade hydrokinetic crossflow rotor compared to a 4-blade rotor, which was attributed to the lower blade–blade interaction and smaller losses from interference. In the broader context, Lillahulhaq et al. (2022) and Alfeuz et al. (2024) have shown the importance of effective momentum extraction at moderate blade angles rather than only having a high number of blades or a high rotational speed.

These current CFD results align with the experimental results of Hunt et al. (2024) and agree that turbines with more blades show a decrease in efficiency, although there is a more even distribution of power. In line with this, Bendeş et al. (2023) also validated the optimal number of blades as 30, where the Banki turbine efficiency of 74.91% was obtained through experimentation, which is close to the trends of power output obtained in the present study.

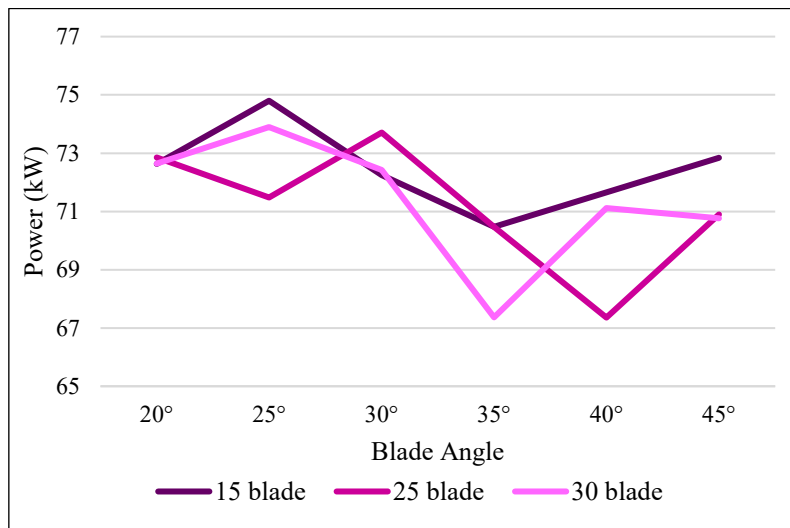


Fig. 8. Power versus angles for 3 different blade numbers.

The main reason for the loss of performance above a 30° blade angle is flow separation (Fig 8). However, for moderate angles (20° to 30°), the boundary layer is mostly attached to the pressure side of the blade, which allows for good pressure recovery and momentum transfer. If the blade angle is increased beyond 30°, then the adverse pressure gradient created on the suction side is more severe and separation occurs. The results by Choi et al. (2008) through CFD analysis confirmed that unfavourable blade angles resulted in recirculation within runner passages, generating a significant hydraulic loss in both turbine stages, whereas the results of Dave et al. (2022) demonstrated how reducing the peak apparent angles can increase power generation and further mitigate dynamic stall effects, therefore solidifying the relationship between high apparent angles and power loss.

The 30-blade configuration at 35° showed the worst power reduction (67.365 kW, the lowest value among all configurations), such that a high number of blades leads to increased separation effects due to an increased number of surfaces in contact with the disrupted flow. On the other hand, the partial power recovery at 45° for the 15-blade configuration (72.845 kW) is due to the less cumulative drag of a low-blade-count runner, which has the ability to partially reorganise separated flow between widely spaced blades. In the transition between attached and separated flow, fluctuations in power for the adjacent blade angles were observed, especially for the 25-blade configuration between 25° (71.483 kW) and 30° (73.708 kW). This sensitivity has been identified by Assefa and Tesfay (2025b), who showed that a change from an optimised blade entry angle (15°) to a conventional set-up (30°) resulted in a loss of 5.8% in hydraulic efficiency as a result of jet dispersion and flow mismatch.

Conclusion

The results indicated that having blades has a significant non-monotonic influence on the pressure distribution. The 30-blade configuration always gave the most even and greatest pressure values on the blades, especially at low blade angles, which allowed momentum to be transferred more effectively, thus leading to greater torque production. The 15-blade arrangement resulted in more localised pressure distributions and lower pressure values in comparison to the 3-blade arrangement, and thus energy capture was reduced. The pressure concentration seen along the blade leading edge at 20° and 25° indicated optimal

flow–surface interaction, while higher blade angles above 25° experienced significant pressure reduction, highlighting the lower effect of energy transfer at higher angles.

Blade angle was shown to be a significant factor in determining the ability of the flow to attach itself to a blade and the effectiveness of energy transfer. For all set-ups, the angles that consistently achieved the best hydrodynamic conditions (20°–25°) were the ones that allowed attached flow, reduced drag and effective momentum transfer by pressure. As the angle increased, the performance decreased, and flow separation increased more and more when the angle exceeded 30°. The power reduction was the most severe with the 30-blade configuration at 35° (67.365 kW), showing that the more blades there are, the worse the impact of the flow separation, as it has more surfaces to interact with. In contrast, the 15-blade configuration showed some recovery of the power at 45° (72.845 kW), indicating that at low blade density, the separated flow could reattach between the blades, thus partly offsetting losses.

The connection between the investigated parameters is that the number of blades and the blade angle are dependent on each other and affect the performance of the turbine. Increasing the number of blades will improve the uniformity of the pressure and the torque created but will cause decreased rotational speed and will increase the likelihood of flow separation at higher angles. Under adverse conditions, lower blade counts favour angular velocity and flow reorganisation at the expense of reduced torque production by the pressure-created flow. The interdependence establishes that the optimal turbine configuration does not involve maximisation of any one parameter, but rather balancing the development of pressure, torque generation and rotational response to meet application needs. According to the simulated conditions, it is found that the 30-blade configuration at 20°–25° is the best design with the maximum power generation in the current comparative analysis of this study, while the 15-blade configuration may be best in terms of rotating speed characteristics. The results provide a foundation for further experimental testing and design enhancement. To further clarify the flow separation dynamics and structural integrity under peak pressure loading, future research efforts should concentrate on experimental verification of the torque and power predictions, as well as transient simulations and fluid–structure interaction analysis.

Acknowledgements/ Funding

This research was supported by Universiti Tun Hussein Onn Malaysia (UTHM) through Tier 1 (vot J115) and GPPS (vot J236).

Conflict of Interest Statement

The authors agree that this research was conducted in the absence of any self-benefits, commercial or financial conflicts, and declare the absence of conflicting interests with the funders.

Authors' Contributions

The authors confirm their contribution to the paper as follows: study conception and design: Sharvika Pasubathy, Zuliazura Mohd Salleh, Muhammad Alif Z. Yusri; data collection: Sharvika Pasubathy, Zuliazura Mohd Salleh, Mohd Fairuz Marian; analysis and interpretation of results: Sharvika Pasubathy, Zuliazura Mohd Salleh, Izuan Amin Ishak; draft manuscript preparation: Sharvika Pasubathy, Zuliazura

Mohd Salleh, Nik Normunira Mat Hassan. All authors reviewed the results and approved the final version of the manuscript.

Declaration of Generative AI in the Writing Process

During the preparation of this manuscript, the authors used AI tools to enhance the readability and language quality of the work. The authors reviewed, edited and verified all AI- assisted content and take full responsibility for the final content of the manuscript.

Data Availability/ Supplementary Materials

The datasets used and/or analysed during the current study are available from the corresponding author on reasonable request.

Ethics Statement

This study is based on computational simulations and does not involve human participants, animal subjects, clinical data, or any personally identifiable information. As such, no ethical approval from an institutional review board or ethics committee was required. All simulations were conducted in accordance with standard scientific computing practices.

Reference

- Alfeuz, A., Tamiri, F., Yan, F. Y., Muzammil, W. K., Hong, M. G. J., Abang Mahmud, D. S., Bohari, N., & Ismail, M. A. (2024). Performance analysis of a crossflow vortex turbine for a gravitational water vortex power plant. *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*, 116(2), 13–26. <https://doi.org/10.37934/arfmts.116.2.1326>
- Assefa, E. Y., & Tesfay, A. H. (2025a). Effect of blade number on internal flow and performance characteristics in low-head cross-flow turbines. *Energies*, 18(2), 318. <https://doi.org/10.3390/en18020318>
- Assefa, E. Y., & Tesfay, A. H. (2025b). Synergetic improvement of blade entry and water admission angles for high efficiency cross-flow turbines in micro-hydropower applications. *Energies*, 18(17), 4540. <https://doi.org/10.3390/en18174540>
- Assefa, E. Y., & Tesfay, A. H. (2025c). Effect of blade profile on flow characteristics and efficiency of cross-flow turbines. *Energies*, 18(12), 3203. <https://doi.org/10.3390/en18123203>
- Bendeş, O., Yılmaz, B., Koç, F., & Yıldız, A. (2023). Numerical and experimental investigation of design parameters for efficiency increase of Banki turbine. *Dicle University Journal of the Institute of Natural and Applied Science*, 12(1), 49-64. <https://doi.org/10.55007/dufed.1243168>
- Benzon, D., Židonis, A., Panagiotopoulos, A., Aggidis, G. A., Anagnostopoulos, J. S., & Papantonis, D. E. (2015). Impulse turbine injector design improvement using computational fluid dynamics. *Journal of*

Fluids Engineering, 137(4), 041106. <https://doi.org/10.1115/1.4029310>

Bowman, J., Bhushan, S., Burgreen, G., & Dettwiller, I. (2021). Hydrokinetic turbine performance and wake analysis using a data-driven actuator line model. *Proceedings of the ASME International Mechanical Engineering Congress and Exposition* (pp. 1-10). ASME Publisher. <https://doi.org/10.1115/imece2021-71957>

Center for Global Sustainability. (2024). Projects. University of Maryland. <https://cgs.umd.edu/research-impact/projects>

Chen, Z. M., & Choi, Y. D. (2013). Performance and internal flow characteristics of a crossflow turbine by guide vane angle. *IOP Conference Series: Materials Science and Engineering*, 52, 052031. <https://doi.org/10.1088/1757-899X/52/5/052031>

Choi, Y. D., Lim, J. I., Kim, Y. T., & Lee, Y. H. (2008). Performance and internal flow characteristics of a crossflow hydro turbine by the shapes of nozzle and runner blade. *Journal of Fluid Science and Technology*, 3(3), 398–409. <https://doi.org/10.1299/jfst.3.398>

Dave, M., Strom, B., & Snortland, A. (2022). Simulations of intracycle angular velocity control for a cross-flow turbine [Preprint]. *arXiv*. <https://doi.org/10.48550/arxiv.2009.06085>

Epple, P., Steppert, M., Wunder, L., & Steber, M. (2017). Verification of Torricelli's efflux equation with the analytical momentum equation and with numerical CFD computations. *Applied Mechanics and Materials*, 871, 220–229. <https://doi.org/10.4028/www.scientific.net/AMM.871.220>

Grid System Operator. (2025). System demand profile for peninsular Malaysia. <https://www.gso.org.my/>

Haryono, D. A., Fahrudin, A., Akbar, A., & Mulyadi, M. (2019). The effect of blade angle on two-stage water turbine against power and efficiency. *Journal of Physics: Conference Series*, 1402(4), 044044. <https://doi.org/10.1088/1742-6596/1402/4/044044>

Homa, M., Pałac, A., Żoładek, M., & Figaj, R. (2022). Small-scale hybrid and polygeneration renewable energy systems: energy generation and storage technologies, applications, and analysis methodology. *Energies*, 15(23), 9152. <https://doi.org/10.3390/en15239152>

Ho-Yan, B., & Lubitz, W. D. (2011). Performance evaluation of cross-flow turbine for low head application. *World Renewable Energy Congress* (pp. 1394–1399). Linköping University Electronic Press. <http://dx.doi.org/10.3384/ecp110571394>

Hunt, A., Strom, B., Talpey, G., Ross, H., Scherl, I., Brunton, S., Wosnik, M., & Polagye, B. (2024). An experimental evaluation of the interplay between geometry and scale on crossflow turbine performance. *Renewable and Sustainable Energy Reviews*, 206, 114848. <https://doi.org/10.1016/j.rser.2024.114848>

Kementerian Ekonomi Malaysia. (2023). National Energy Transition Roadmap (NETR). <https://www.ekonomi.gov.my/ms>

Kim, M. S., Edirisinghe, D. S., Yang, H. S., Gunawardane, S. D. G. S. P., & Lee, Y. H. (2021). Effects of blade number and draft tube in gravitational water vortex power plant determined using computational fluid dynamics simulations. *Journal of Advanced Marine Engineering and Technology*, 45(5), 252–262. <https://doi.org/10.5916/jamet.2021.45.5.252>

Kumar, A., Schei, T., Ahenkorah, A., Caceres Rodriguez, R., Devernay, J. M., Freitas, M., Hall, D., Killingtveit, Å., & Liu, Z. (2011). Hydropower. IPCC special report on renewable energy sources and climate change mitigation (pp. 437–496). Cambridge University Press.

<https://doi.org/10.24191/jmeche.v23i3.10844>

<https://doi.org/10.1017/CBO9781139151153.009>

- Lillahulhaq, Z., Sandy, F. D., Mahardika, B. P., Akbar, M. A., & Saragih, D. B. (2022). Experimental study of small hydro turbine propeller performance with a variety of blade angles of attack. *Sinergi*, 26(3), 273-278. <https://doi.org/10.22441/sinergi.2022.3.001>
- Lukeš, R., Honus, S., Blejchař, T., & Balco, M. (2025). Reaction crossflow turbine for small hydropower plants: flow profile design and CFD analysis. *Renewable Energy*, 252, 123560. <https://doi.org/10.1016/j.renene.2025.123560>
- Marsh, P., Ranmuthugala, D., Penesis, I., & Thomas, G. (2016). Numerical simulation of the loading characteristics of straight and helical-bladed vertical axis tidal turbines. *Renewable Energy*, 94, 418–428. <https://doi.org/10.1016/j.renene.2016.03.060>
- Maulana, A. P., Indrayani, & Arifin, F. (2023). CFD analysis in investigating the impact of turbine blade number on the performance of hydro turbine. *Jurnal Polimesin*, 21(3), 311-317. <http://dx.doi.org/10.30811/jpl.v21i3.3671>
- Mohsin, M., Orynassarov, D., Anser, M. K., & Oskembayev, Y. (2023). Does hydropower energy help to reduce CO2 emissions in European Union countries? Evidence from quantile estimation. *Environmental Development*, 45, 100794. <https://doi.org/10.1016/j.envdev.2022.100794>
- Monsalve, O., Velez-Garcia, S., & Revuelta-Acosta, J. D. (2024). Influence of blade number on the hydrodynamic performance of a propeller-type axial turbine for in-pipe installation. *CFD Letters*, 16(4), 134–158. <https://doi.org/10.37934/cfdl.16.4.134158>
- Naseem, M., Saleem, A., & Naseem, M. S. (2022). Investigation of blade design parameters for performance improvement of hydraulic cross flow turbine. *Ocean Engineering*, 257, 111663. <https://doi.org/10.1016/j.oceaneng.2022.111663>
- Nishi, Y., Hatano, K., & Inagaki, T. (2017). Study on performance and flow field of an undershot cross-flow water turbine comprising different number of blades. *Journal of Thermal Science*, 26(5), 431–438. <https://doi.org/10.1007/s11630-017-0956-1>
- Prabowoputra, D. M., Prabowo, A. R., Nubli, H., Harsito, C., Ubaidillah, Susilo, D. D., Wibowo, & Lenggana, B. W. (2023b). Forecasting effect of blade numbers to cross-flow hydro-type turbine with runner angle 30° using CFD and FDA approach. *Mathematical Modelling of Engineering Problems*, 10(2), 419–424. <https://doi.org/10.18280/mmep.100205>
- Prabowoputra, D. M., Prabowo, A. R., Yaningsih, I., Tjahjana, D. D. D. P., Laksono, F. B., Adiputra, R., & Suryanto, H. (2023a). Effect of blade angle and number on the performance of Bánki hydro-turbines: assessment using CFD and FDA approaches. *Evergreen*, 10(1), 519–530. <https://doi.org/10.5109/6782156>
- Quaranta, E., & Revelli, R. (2017). Hydraulic behaviour and performance of breastshot water wheels for different numbers of blades. *Journal of Hydraulic Engineering*, 143(1), 04016072. [https://doi.org/10.1061/\(asce\)hy.1943-7900.0001229](https://doi.org/10.1061/(asce)hy.1943-7900.0001229)
- Quaranta, E., Perrier, J. P., & Revelli, R. (2022). Optimal design process of crossflow Banki turbines: literature review and novel expeditious equations. *Ocean Engineering*, 257, 111582. <https://doi.org/10.1016/j.oceaneng.2022.111582>
- Reis, M. G. A., & Carvalho, S. R. (2018). Archimedes turbine numerical simulation using the OpenFOAM software. *Proceedings of the 17th Brazilian Congress of Thermal Sciences and Engineering* (pp. 1-11).

<https://doi.org/10.24191/jmeche.v23i3.10844>

ABCM Publisher. <https://doi.org/10.26678/abcm.encit2018.cit18-0413>

Sahu, K. K., Prasad, V., & Dhurvey, P. (2025). Cross-flow turbine's CFD analysis for hydropower generation. *ISH Journal of Hydraulic Engineering*. <https://doi.org/10.1080/09715010.2025.2521623>

Sammartano, V., Aricò, C., Sinagra, M., & Tucciarelli, T. (2015). Cross-flow turbine design for energy production and discharge regulation. *Journal of Hydraulic Engineering*, 141(3), 04014083. [https://doi.org/10.1061/\(asce\)hy.1943-7900.0000977](https://doi.org/10.1061/(asce)hy.1943-7900.0000977)

Sammartano, V., Morreale, G., Sinagra, M., & Tucciarelli, T. (2016). Numerical and experimental investigation of a crossflow water turbine. *Journal of Hydraulic Research*, 54(3), 321–331. <https://doi.org/10.1080/00221686.2016.1147500>

Sinagra, M., Sammartano, V., Aricò, C., & Collura, A. (2016). Experimental and numerical analysis of a cross-flow turbine. *Journal of Hydraulic Engineering*, 142(1), 04015040. [https://doi.org/10.1061/\(asce\)hy.1943-7900.0001061](https://doi.org/10.1061/(asce)hy.1943-7900.0001061)

Tiwari, G., Kumar, J., Prasad, V., & Patel, V. K. (2024). Multi-fidelity CFD for obtaining the runner blade profile parameters of a Francis turbine for optimum hydrodynamic performance. *Ocean Engineering*, 305, 117994. <https://doi.org/10.1016/j.oceaneng.2024.117994>

Tuapetel, J. V., Priyawan, P., Rasyid, M. K., Ichsan, D. K., & Pangestu, R. (2025). Innovative study on utilising drainage channels for renewable energy with gravitational vortex turbines. *Journal of Mechanical Engineering*, 22(2), 111–131. <https://doi.org/10.24191/jmeche.v22i2.3750>

Velichkova, R., Iliev, R., Pichurov, G., Markov, D., Simova, I., Pushkarov, M., & Tsalov, T. (2023). Numerical study of the torque and power of a hydraulic turbine with oscillating blades. *Energies*, 16(18), 6744. <https://doi.org/10.3390/en16186744>

Woldemariam, E. T., & Lemu, H. G. (2019). Numerical simulation-based effect characterisation and design optimisation of a micro cross-flow turbine. *Strojniški Vestnik – Journal of Mechanical Engineering*, 65(6), 386–397. <https://doi.org/10.5545/sv-jme.2018.5901>

Wu, X., Wu, H., & Zuo, L. (2022). The effect of the blade number on a cross-flow hydrokinetic turbine. *IFAC-PapersOnLine*, 55(37), 234–239. <https://doi.org/10.1016/j.ifacol.2022.10.489>



© 2026 by the authors. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY-NC-SA) license (<https://creativecommons.org/licenses/by-nc-sa/4.0/>)