Indonesian Journal of Computer Science
ISSN 2549-7286 (*online*)
Jln. Khatib Sulaiman Dalam No. 1, Padang, Indonesia
Website: ijcs.stmikindonesia.ac.id | E-mail: ijcs@stmikindonesia.ac.id

Numerical Analysis of Flow Velocity and Performance Test of Cross-Flow Turbine (500 W)

Sandar Kyaw[1], War War Min Swe[2], Aung Kyaw Soe[3], Aung Ko Latt[3]
sanndar.1988@gmail.com, warwarminswe288@gmail.com, aungkyawsoe.tu@gmail.com, dr.aungkolat@gmail.com
[1, 2, 3]Department of Mechanical Engineering, Mandalay Technological University, Mandalay, Myanmar

Article Information

Submitted : 28 Jun 2024
Reviewed : 29 Jun 2024
Accepted : 30 Jun 2024

Keywords

Cross-flow turbine, numerical analysis, velocity, efficiency, turbine performance test

Abstract

The purpose of this study is to analyze the flow velocity and assess the performance of a cross-flow turbine through numerical simulations and performance tests. Using computational fluid dynamics (CFD), the flow patterns and velocity distributions around the turbine blades to gain insights into the turbine's hydrodynamic behavior. In the experimental phase, constructed a prototype of the cross-flow turbine and tested it in a water flume, measuring key performance metrics such as power output and efficiency.The test were performed under design flow condition. The numerical results were validated with performance tests. The numerical analysis revealed critical areas of flow separation and turbulence, which were consistent with the observed experimental data.

A. Introduction

Cross-flow turbines, also known as Banki-Michell or Ossberger turbines, are a type of water turbine used primarily in small-scale hydroelectric power generation. Unlike traditional axial or radial turbines, cross-flow turbines allow water to flow transversely across the blades, making multiple passes through the runner. This design is particularly effective for sites with low head and variable flow conditions, as it can efficiently harness energy from relatively slow-moving water.

Preious studies have attempted to determine the cross-flow turbine distribution. Nirmal Acharya , Oblique Shrestha., (2016) examined performance analysis of 5 kW cross-flow turbine with insertion of air layer effect. The result obtained by justifying with the fact that air layer formation in the runner passage with the supply of air from air suction hole imparts some effect on the improvement of output torques at stage 1 and stage 2 along with the suppression of some recirculating areas that help to minimize the negative power in region 1.

R.C. Adhikari (2018) created computational analysis of part-load flow control for crossflow hydro-turbines. The results showed that by using the slider, maximum efficiency was maintained at 80% down to 40% of Q_{max}, for a turbine with η = 87% at Q_{max}. However, the investigator calculated that the efficiency was based on shaft power divided by the power lost between the inlet and outlet of the runner which slightly overestimates the turbine efficiency.

Use of successive numerical simulations to design and optimize the performance of high-efficiency hydraulic cross-flow turbines were studied by Goodarz Mehr (2019). The authors arrived at the conclusion that, used 2D approximations to simulate flow inside the turbine, but further 3D simulations can provide insight into the effect of side walls on turbine performance (especially for high-head, low flow cases where rotor length is relatively small). Future work will also analyze the interaction between the water jet and rotor from a structural standpoint to help improve rotor life and reduce service needs.

Ali Abbas (2020) review analytical analysis of combined effect of interior guide tube and draft tube on cross flow turbine performance. The author calculated that by reducing the flow angle at second stage by inserting the guide vane, performance at second stage may be improved. Based on the theoretical analysis, it is found that efficiency of cross flow turbine can be improved by providing a guide tube on proper position and with a draft tube.

Jusuf Haurissa's research study from (2020) review the analysis of cross flow turbine performance with guide passage gate vane (gg) at runner turbine by using a triangle velocity method. It was discovered that, with a guide passage gate vane addition attached on the cross flow turbine runner, there was an turbine power increase.

The design techniques performance and efficiency of turbine were the primary focus of all of the previous studies. Several studies and numerical simulation techniques have been tried by numerous researchers in an effort to forecast and enhance turbine performance. Futhermore, the velocity distribution of cross-flow turbine have been detected by this research effort. This idea can be used by the new researchers to approach the turbine's design techniques.

This research aims to examine the flow distribution of cross-flow turbine and performance test. Computational fluid dynamics is also the foundation of flow analysis, which provides finding for the estimation of the velocity distribution. The main aim of this research were compared by turbine efficiency and torque according to the results of simulation and performance test. Understanding the flow behavior of cross-flow turbine and increasing turbine efficiency are the goals of this effort.

B. Research Method

For the numerical analysis, utilized ANSYS CFX, a widely used computational fluid dynamics (CFD) software, to simulate the flow around the cross-flow turbine. The governing equations for fluid flow, including the Navier-Stokes equations, were solved using a finite volume approach. The SST turbulence model was employed to account for the effects of turbulence in the flow.

1. Simulation Setup

The simulation domain was designed to replicate the experimental test setup, ensuring consistency between numerical and physical experiments. The turbine was placed within a rectangular computational domain with inlet and outlet boundaries set at a sufficient distance from the turbine to minimize boundary effects. The location and boundary condition details are shown in Table 1 and boundary condition of cross-flow turbine is shown in Figure 1.

Table 1. Boundary Physics CFX for Fluid Domains

Location	**Boundary condition details**	
Control Volume	Material	Water
	Domain Motion Option	Stationary
	Turbulence Option	Shear Stress Transport
Inlet	Domain Motion Option	Stationary
	Relative Pressure	0 Pa
Channel	Domain Motion Option	Stationary
	Turbulence Option	Shear Stress Transport
Outlet	Mass And Momentum Option	Mass Flow Rate
	Mass Flow Rate	45 kg s^{-1}
Rotating	Domain Motion Option	Rotating
	Angular Velocity	141 rev min^{-1}
	Turbulence Option	Shear Stress Transport

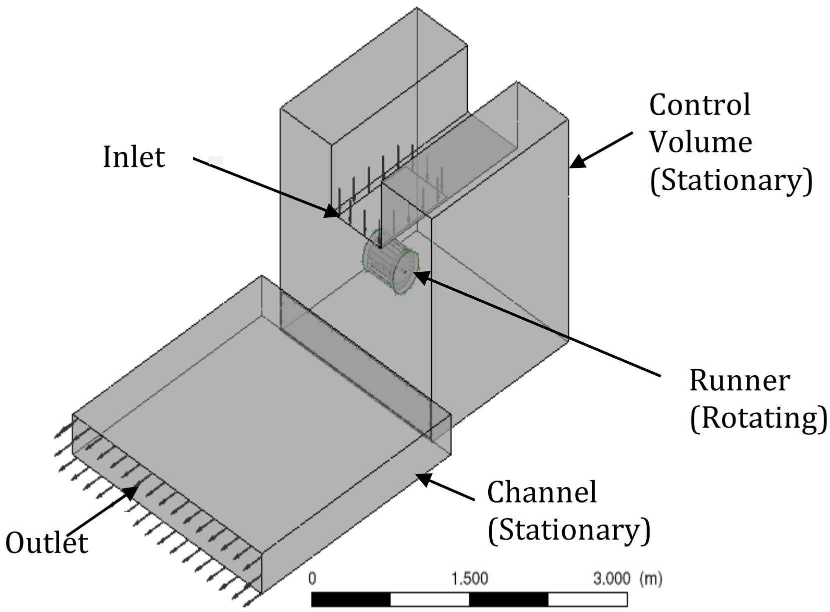

Figure 1. Boundary condition of Cross Flow Turbine

- **Inlet Boundary:** A uniform velocity profile was applied at the inlet based on the desired flow rate.
- **Outlet Boundary:** A constant pressure condition was set at the outlet.
- **Turbine Walls:** No-slip boundary conditions were applied to the turbine blades and other solid surfaces.
- **Assumptions:** The fluid was assumed to be incompressible and Newtonian. Steady-state simulations were conducted.

Table 2. Mesh properties used in the Ansys CFX simulation study

Mesh Property	Value/Action
Sizing	
Span Angle Center	Fine
-Min Size	1.e-002 m
-Max Face Size	0.50 m
-Max Tet Size	0.50 m
Statistics	
-Nodes	266453
-Elements	907392
Mesh Metric	Skewness
Min	3.48e-007
Max	0.99994

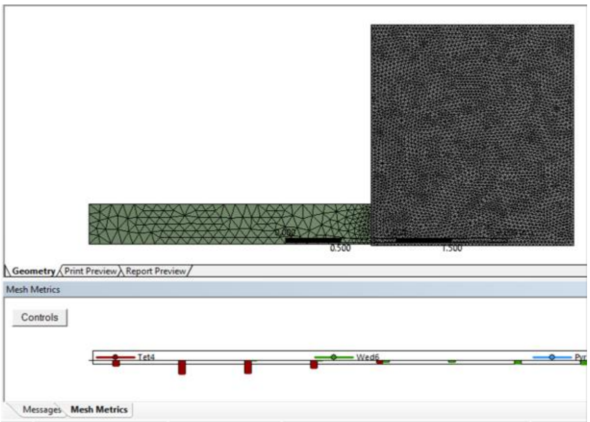

Figure 2. Meshing of Cross Flow Turbine

A structured mesh was generated using ANSYS Meshing, with tetrahedron mesh elements around the turbine blades to accurately capture flow details. A grid independence study was performed to ensure that the results were not dependent on the mesh size. The mesh properties are given in Table 2 and meshing of cross-flow turbine shows in Figure 2.

2. Experimental Setup

The experimental setup consisted of a water flume with adjustable flow rates, for operating conditions of the cross-flow turbine. The turbine prototype was securely mounted within the flume to allow for accurate measurement of performance metrics. Figure 3 shows the layout diagram of site location.

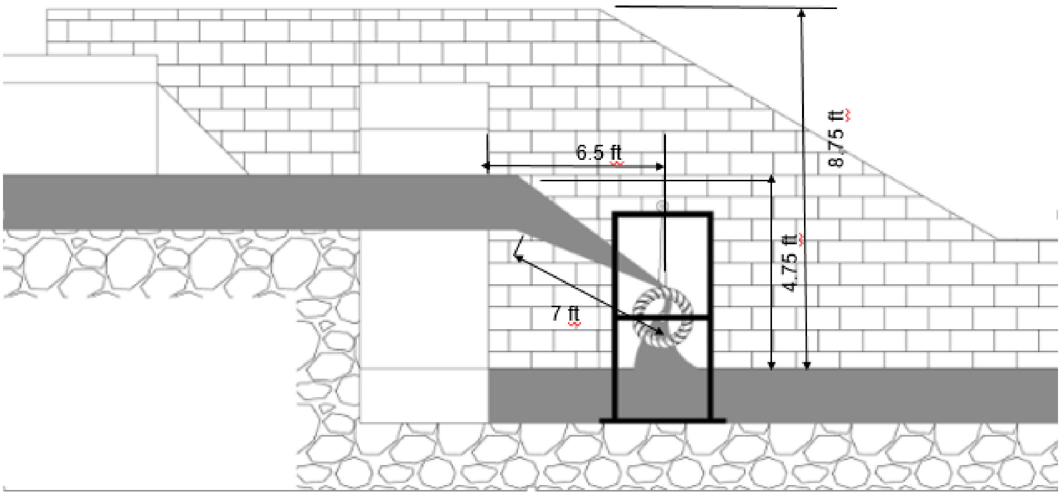

Figure 3. Layout Diagram of Site Location

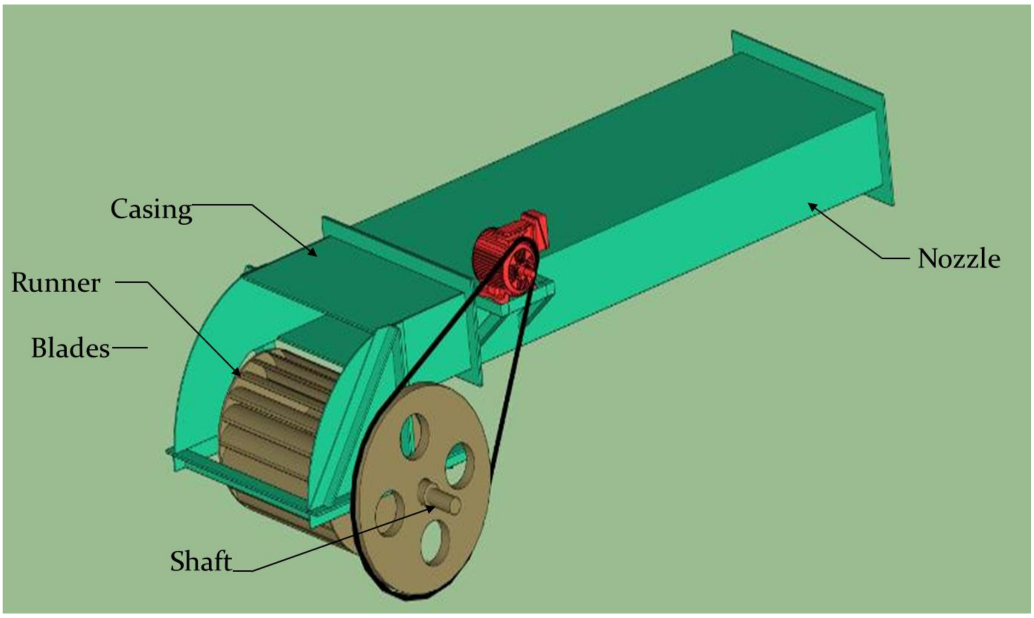

Figure 4. Cross-Flow Turbine

The components of this turbine are runner, blades, shaft, bearing, casing, draft tube, guide vane and nozzle which are described in Figure 4. Generally, the boundaries for a water turbine installation will depend on the specific type and size of the turbine, as well as the characteristics of the water flow at the site. Some general points about boundaries for water turbines:

- The boundaries will be established by constructing a weir or dam upstream of the turbine to create a head of water, and a tailrace downstream of the turbine to channel the water away.
- The boundaries may also include structures to control the flow of water through the turbine, such as intake gates and wicket gates.

Test Conditions:

- **Flow Rates:** A range of flow rates was tested to assess turbine performance under design flow condition.
- **Head Height:** The water head was kept constant to isolate the effects of flow rate on performance.

Equipment Used:

- **Water Flume:** Equipped with flow control and measurement devices.
- **Turbine Prototype:** Constructed based on design specifications used in the numerical simulations.
- **Measurement Instruments:** Tachometer, and clamp meters to measure rotational speed, voltage and current are shown in Figure 5 and Figure 6.

Figure 5. Tachometer

Figure 6. Clamp Meter

C. Previous Work

The normalized design parameters are given in Table 3 and result datas are shown in Table 4. These value are calculated according to theory[1].

Table 3. Design specification Data[1]

Parameter	Symbol	Dimension	Units
Net Head	H	1.5	m
Generator Power	PG	500	W
Generator Efficiency	ηG	95	%
Overall Efficiency	ηo	80	%
Specific Speed	Ns	60	rpm
Coefficient of Velocity	Cv	0.98	-
Angle between absolute and peripheral velocity	α	16	degree

Table 4. Result Data for 500W Cross-Flow Turbine [1]

No	Calculated parameter of parts	Symbol	Result	Unit
1	Outer diameter of runner	D1	330	mm
2	Inner diameter of runner	D2	220	mm
3	Pitch diameter of runner	D0	244	mm
4	Length of the runner	B	295	mm
5	Radius of curvature	R	53	mm

6	Radial rim width	A	55	mm
7	Blade angle	β1	30	deg
8	Central angle of blade	δ	73	deg
9	Blade spacing	P1P2	49.5	mm
10	Nozzle throat width	S1	280	mm
11	Shaft diameter	ds	25	mm
12	Number of blades	n	21	No.

Figure 7 shows the geometry of runner, shaft and blades by using calculated theoretical data.

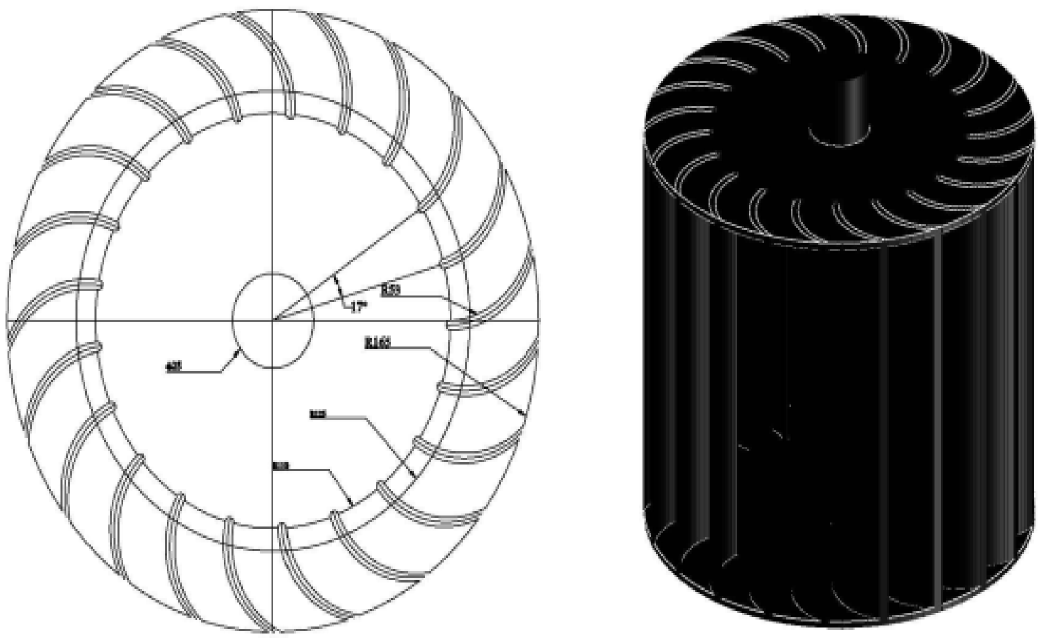

Figure 7. Runner

D. Results

Numerical Results: Flow Velocity Analysis

Flow Velocity around the Turbine:

The numerical simulations provided detailed insights into the flow velocity around the cross-flow turbine. The flow parameter; velocities were analyzed. The velocity vectors were analyzed between the blade passages to investigate the flow behavior and its impact on turbine performance.

Velocity Vectors:

The velocity vector plot (Figure 8) shows the distribution of relative velocity in the vicinity of the turbine. High-velocity regions are observed at the leading edges of the blades in stage 1, where the water impacts the turbine, while low-velocity regions are present stage 2 trailing edge of the blades due to flow separation and recirculation. Significant flow separation was observed at the trailing edges of the blades, leading to the formation of vortices. This phenomenon results in energy loss and reduced turbine efficiency. The flow enters the turbine, interacts with the blades stage 3, and exits with a reduced velocity stage 4. Recirculation zones were detected near the turbine exit, where water re-enters the flow stream, causing turbulent mixing and energy dissipation. Recirculation zones and vortices are visible, indicating areas of flow separation that can affect turbine efficiency. It can be seen from Table 5 that the numerical results of inlet flow velocity and outlet flow velocity.

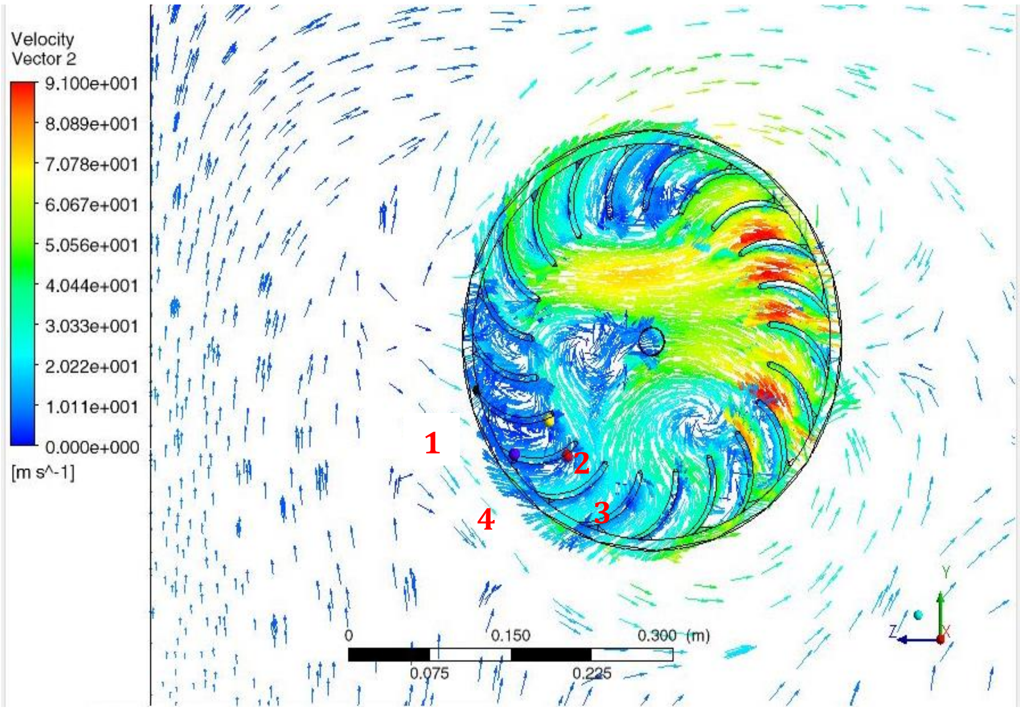

Figure 8. Velocity Distribution at inlet and outlet velocity

Table 5. Numerical Results of Flow Velocity at Inlet and Outlet

Parameter	Symbol	Value	Unit
Inlet velocity at stage 1	v_1	5.62871	m/s
Outlet velocity at stage 2	v_2	2.80136	m/s
Inlet velocity at stage 3	v_3	2.79927	m/s
Outlet velocity at stage 4	v_4	1.87332	m/s

Performance Evaluation

The performance of the cross-flow turbine was evaluated based on key metrics such as efficiency and power output. Both numerical simulations and experimental tests were conducted to determine these metrics under designed flow condition.

Experimental Results:

The performance test of cross-flow turbine is shown in Figure 9 and the experimental results of cross-flow turbine performance are given in Table 6.

Table 6. Experimental Results of Cross-Flow Turbine Performance

No	Turbine Speed(rpm)	Turbine Power, P_T(W)	Flow rate (m^3/s)	Efficiency(%)
1	141	526	0.045	79.32

Figure 9. Performance Test of Cross-Flow Turbine

Comparison of Numerical and Experimental Results:

$$\eta = \frac{P_{Shaft}}{P_{Hydraulic}} = \frac{T_s \omega}{\rho g Q H_e} \qquad (1)$$

where P_{Shaft} shaft power (W), $P_{Hydraulic}$ is the hydraulic power (W), T_s is shaft torque (Nm); ω is runner angular velocity (rad/s), ρ is the density of water (kg/m^3), g is the acceleration due to gravity (m/s^2), Q is the water flow rate (m^3/s) and H_e effective head (m). The comparison of numerical and performance results are shown in Table 7 and Figure 10 shows comparison of numerical and performance of efficiency result at design flow condition.

Table 7. Comparison of Numerical and Performance Results

Parameter	Numerical Results	Performance Results	Deviation (%)
Efficiency(%)	82.22	79.32	3.5

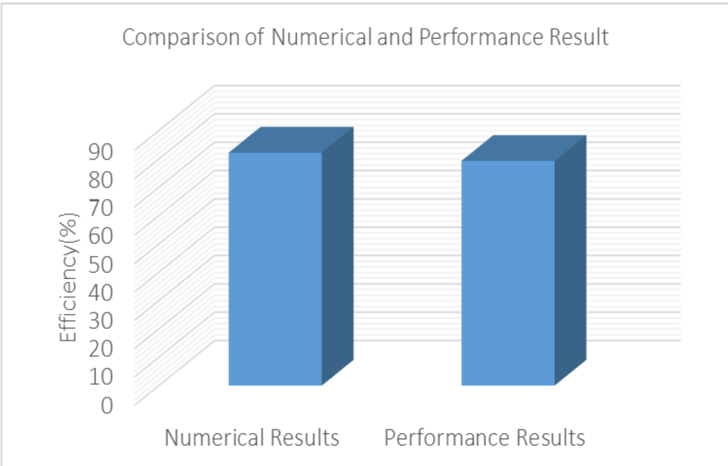

Figure 10. Comparison of Numerical and Performance of Efficiency Result at design flow condition

Conclusion

The numerical result of maximum efficiency is 82.22% and performance result of maximum efficiency is 79.32% and output power is 526W occurs at a design flow condition. The deviation of efficiency is 3.5%.

Discussion

The findings of this study contribute to the existing body of research on cross-flow turbines by providing a comprehensive analysis of flow dynamics and performance metrics. Previous studies have primarily focused on empirical testing and basic performance evaluations, often lacking detailed insights into flow behavior and efficiency under varying operating conditions. By integrating numerical simulations with experimental tests, this study offers a more thorough understanding of the turbine's performance characteristics. Design modifications, such as optimizing blade shape and angle, could minimize flow separation and improve performance. Adjusting the turbine housing and exit design could reduce recirculation and improve flow efficiency.

E. Acknowledgment

All the authors of the research paper are highly acknowledged to Mechanical Engineering Department, Mandalay Technological University, Myanmar for encouraging and supporting to do this research.

F. References

[1] Sandar Kyaw, War War Min Swe, Aung Kyaw Soe, Aung Ko Latt, "Design Optimization and Flow Analysis of Cross Flow Turbine (500 W)", ICRIMEC, March,pp. 20-26, 2024.

[2] Espresssif Systems., "ESP32." Espresssif Systems., 2024. [Online]. Available: https://www.espressif.com/sites/default/files/documentation/esp32_datasheet_en.pdfGaurav Saini, R.P. Saini & S.K. Signal Energy Sources, Part A: Recovery, Utilization, and Environmental Effects, Volume 44, 2022- Issue 1.

[3] Ali Abbas, Analytical analysis of combined effect of interior guide tube and draft tube on cross flow turbine performance., Material Today: Proceedings, 2020.

[4] Jusuf haurissa, Analysis of cross flow turbine performance with guide passage gate vane (gg) at runner turbine by using a triangle velocity, Volume 10, 2020- Issue 3.

[5] Goodarz Mehr, Use of Successive Numerical Simulations to Design and Optimize the Performance of High-Efficiency Hydraulic Cross-Flow Turbines, 2019.

[6] Vanessa Ruiz Gómez, Computational analysis of a cross flow turbine performance, ICCHMT 2018 MATEC Web of Conferences 240, 03011.

[7] Adhikari, R.C., Wood, D.H., 2018. Computational analysis of part-load flow control for crossflow hydro-turbines. Energy Sustain. Dev. 45, 38–45.

[8] Takeo, K.: Text Book of Micro-Hydropower for Rural Development. https://www.researchgate.net/publication/344227098

[9] Nebiyu Bogale Mereke.: Highly Efficient Cross-flow Turbine Runner Design for Upgrading Traditional Water Mill into Micro Power Plant, 2015.

[10] C.S.Kaunda, C.Z. Kimambo and T.K. Nielsen, "Experimental Study on a Simplified Cross-flow Turbine," IJEE, vol.5, pp.155-158,2014.

[11] Bila lAbdullah Nasir, "Design of High Efficiency Cross-flow Turbine for Hydropower Plant," IJEAT, vol.2,pp.308-310, Feb.2013.

[12] Miroslav Nechleba, Dr. Techn, "Hydraulic Turbines", 1957.

[13] C.A. Mockmore, "The Banki Water Turbine," Bulletin Series No.25,1949.

[14] R.L.Daugherty, "Hydraulic 3rd Edition," A Text on Practical Fluid Machines,1937.