Investigation of the draft tube variations against the first stage and the second stage flow of banki turbine

S Sirojuddin*, L K Wardhana and A Kholil
Mechanical Engineering, State University of Jakarta, Indonesia

*sirojuddin@unj.ac.id

Abstract. Cross Flow Turbine usually for micro hydro-electrical power plant application especially for isolated areas. Increasing power efficiency still the most interesting topic for researches. One of the method to increase it was to control the inflow of water from the first stage to the second stage of banki blades to be a good flow trajectory and did not strike the shaft using a draft tube. This research aimed to investigate the draft tube length variations against the flow from the first stage to the second stage of Banki turbine, so that good flow trajectory, uniform color velocity contour and avoid shaft struck could be obtained. There were 2 variants runner casing and 3 variants length of draft tube, 140 mm, 240 mm, and 340 mm. The 2D and 3D geometry were drawn in CAD software, while Flow Simulation test using Solidwork Flow Simulation with a discharge of water 2 m$^3$/min and head 5.5 m. From simulation results, shown that the second variant C2-2 was the highest efficiency and the best flow trajectory.

1. Introduction
Banki turbine is cross-flow turbine, the application usually used in micro-hydroelectric power plants. Generally, the microscale is below 1 MW [1]. The selection of the turbine is based on the low price, easy manufacture and transportation, simple assembly, and maintenance, it suitable for isolated areas that do not have an electricity line. The most important point in the Banki turbine is to obtain the maximum output power. The length and shape of the draft tube can increase power efficiency, therefore its must be analyzed and researched seriously to get a good flow trajectory and not struck the shaft so that increasing power can be obtained.

From issued Bulletin No. 25 about the translation of Donat Banki’s turbine paper “Neue Wasser-Turbine” according to the paper maximum efficiency would be 87.8 % but experimentally reached only 68% [2]. Investigated the influence of the turbine nozzle shape would affect the performance and internal characteristics of the turbine using CFD analysis [3]. Analyzed water flow and pressure in several rounds of the turbine runner. Inflows made in tapered form without guide vane and the results showed that the flow trajectory not inline and still struck the shaft runner [4]. Investigated the flow of water into four types of the casing with an opening width of the guide vane 100%. It was found that the highest efficiency was 4th modeled [5]. Researched of turbine efficiency based on the number of blades and water angle of attack. The result has shown that the 35 blades and 22˚ water angle of attack got the efficiency of 86% but the inflow discharge regulated only by the width of the nozzle (not used a guide vane) [6]. Investigated the pattern of the outflow of the turbine by adding a draft tube to eliminate vortices shown that it could be removed [7]. A numerical investigation on the inlet and outlet water of the blade runner where inflows of water regulated by the nozzle. The flow was slightly struck the shaft
and the cavitation still appear [8]. Analyzed the flow of banki turbines with horizontal inflow. Showed that the water fully struck the shaft, which means that the turbine must be not efficient [9]. Studied about cavitation that occurs in banki turbines with horizontal inflow. Showed that there was cavitation on the blade [10].

Investigation of the performance of the constructed cross-flow turbine and verify that all parts and systems installed worked fine and performing their functions correctly. The designed cross-flow turbine was capable to produce up to 300 W AC power at the head of 6 m and flow rate of 0.0091 m³/s. Maximum efficiency was obtained 86% compared to 300 W design power, but if compared to theoretical power the efficiency was 48% only [11]. The investigation conducted for 3 variants of blade radius that were 0.334 r₁, 0.326 r₁, 0.323 r₁ and 3 variants attack of angle that were 15°, 16°, 17° [12].

Researched to get the optimum profile of the leading edges of the blade of banki turbine with four different profiles were modeled and simulated with ANSYS CFX and then compared their performance in terms of their efficiencies showed that the efficiency of the round tip blade profile was better than the other three profiles of the blade [13]. The investigation conducted against the effect of inlet nozzle shape on the performance and internal flow of a cross-flow turbine by CFD analysis. The results show that relatively narrow and converging inlet nozzle shape gives better power [14]. From investigation on the effects of the air layer in the turbine chamber on the performance and internal flow of the cross-flow turbine with a newly developed air supply method by CFD. The result has shown that with air suction flow rate 400 L/min could get maximum output Power 73 kW with the vertical water level in draft tube 0.72 m [15].

The focus of this research was to investigate the draft tube variations against the first stage and the second stage flow of banki turbine to get maximum drag force, a good flow trajectory, and did not strike the shaft so that the power generated was expected to be increased.

Main parts of Banki turbine can be seen in figure 1 below:

![Main Parts of Turbine](image)

**Figure 1.** Main parts of banki turbine.

2. Methods

In this research, there were 2 variants runner casing and 3 variants length of draft tube. Dimension of runner casing was 0.5 D₁ and D₁ shown in figure 2. Length of draft tube were 140 mm, 240 mm and 340 mm as shown in figure 2 and table 1. Parameter and dimension of turbine basically taken from discharge of water Q (m³/s), head of water H (m), angle of entrance velocity α (degree), angle of blade β, (degree) as shown in table 2. The 2D and 3D geometry were drawn in CAD software, while Flow Simulation test using CFD Solidwork Flow Simulation with a discharge of water 2 m³/min and head 5.5 m. The simulation result of the drag force and flow trajectories could be obtained after the process was convergent as shown in figure 4 and table 3.
Shape and length (L) of draft tube can be seen in figure 2 and the length of the draft tube and its ratio can be seen in Table 1 below.

![Figure 2](image)

**Figure 2.** Side view of variant C1 (a), Variant C2 (b).

| Variant of Draft Tube | L / D₁ | L (mm) |
|-----------------------|--------|--------|
| C1-1 and C2-1         | 0,7    | 140    |
| C1-2 and C2-2         | 1,2    | 240    |
| C1-3 and C2-3         | 1,7    | 340    |

Parameter turbine can be seen in Table 2 below:

| Design Parameter | Data Calculation | Design | Description                      |
|------------------|------------------|--------|----------------------------------|
| α₁               | 16°              | 16°    | Angle of Entrance Velocity       |
| β₁               | 30°              | 30°    | Angle of Blade                   |
| D₁               | 200 mm           | 200 mm | Outer Diameter of Runner         |
| D₂               | 131.97 mm        | 132 mm | Inner Diameter Runner            |
| N                | 18 pieces        | 18 pieces | Number of Blades               |
| V₁               | 10.18 m/s        | 10.18 m/s | Absolute Velocity of Water     |
| L                | 200 mm           | 200 mm | Width of Wheel/Runner            |
| H                | 5,5 m            | 5,5 m  | Head of Water                    |
| N                | 468.84 rpm       | 470 rpm | Runner Speed                     |
| Q                | 2 m³/min         | 2 m³/min | Water Discharge                 |
| Dₛ               | 30 mm            | 30 mm  | Shaft Diameter                   |
Theoretical power [2] calculated by:

\[ P_{th} = \rho \cdot g \cdot Q \cdot H = \frac{w \cdot Q \cdot V_1^2}{C^2 \cdot 2 \cdot g} \]  

Drag Force in the first stage and the second stage of the blades could be obtained from CFD SolidWorks Flow Simulation. Brake force power (BP) could be calculated according to [2]:

\[ BP = \frac{(wQ/g)(V_1 \cos \alpha_1 + V_2 \cos \alpha_2)u_1}{2} \]  

Using CFD SolidWorks Flow Simulation, the power of turbine can be approached by drag force:

\[ \begin{align*} 
F_d &= 0.5 \rho \cdot C_d \cdot A \cdot V^2 \\
P_d &= F_d \cdot \cos \alpha_1 \cdot U_1 
\end{align*} \]  

Fd is drag force, \( \rho \) is density, \( C_d \) is drag coefficient, \( A \) is the frontal area, \( V = V_1 \) is absolute velocity of water, \( P_d \) is the power of drag, and \( U_1 \) is the tangential speed of turbine runner.

From the velocity profile in figure 3:

\[ \begin{align*} 
U_1 &= \frac{V_1 \cos \alpha_1}{2} \\
V_1 &= C \sqrt{2 \cdot g \cdot H} 
\end{align*} \]  

C is constant = 0.98, \( g \) gravitational acceleration and \( H \) is Head of water.

Efficiency of Turbine:

\[ \eta = \frac{P_d}{P_{th}} \times 100 \% \]  

3. Results and discussion

3.1. Results

From the results of CFD flow simulation software, the velocity trajectory contours and drag forces from the first stage to the second stage could be seen in figure 4 and table 3, while power and efficiency from table 4 below:

![Figure 4](image-url)

**Figure 4.** Velocity trajectory of variants, (a) C1-1, (b) C1-2, (c) C1-3, (d) C2-1, (e) C2-2 and (f) C2-3.
Table 3. Drag force at the first and second stage of blades.

| Variant of Blade Profile | C1-1   | C1-2   | C1-3   | C2-1   | C2-2   | C2-3   |
|--------------------------|--------|--------|--------|--------|--------|--------|
| Drag Force Fd (N)        |        |        |        |        |        |        |
| First Stage              | 185.696| 200.715| 170.68 | 195.117| 243.736| 175.178|
| Upper Second Stage       | 52.541 | 9.898  | 4.233  | 71.538 | 49.605 | 22.141 |
| Lower Second Stage       | 22.579 | 56.185 | 12.692 | -      | 12.394 | -      |
| Total Fd (N)             | 260.8  | 266.8  | 187.6  | 266.65 | 293.3  | 210.7  |

Drag power and efficiency can be seen in table 4, below:

Table 4. Drag power of turbine.

| Variant | Pth (Watt) | Pd (Watt) | Efficiency (%) |
|---------|------------|-----------|----------------|
| C1-1    | 1226.7     |           | 68.2           |
| C1-2    | 1254.8     |           | 69.8           |
| C1-3    | 882.3      |           | 49.1           |
| C2-1    | 1798.5     | 1254.17   | 69.7           |
| C2-2    | 1379.5     | 991       | 54.8           |
| C2-3    |            |           |                |

3.2. Discussion

Variants of draft tube with throat opening 0.5 x D₁ for C₁-1 and 1xD₁ for C-2, diameter of runner D₁ = 200 mm width L = 200 mm, ratio L/D₁ = 0.7 for C₁-1 and C₂-1, L/D₁ = 1.2 for C₁-2 and C₂-2, L/D₁ = 1.7 for C₁-3 and C₂-3. From figure 4, it could be seen that variant C₂-2, the velocity still high and good trajectory from the first stage to the second stage of the blade otherwise other variants in the second stage drop. According to table 3, the highest total drag force was variant C₂-2 = 293.3 N, while others were lower. From table 4, the power of C₂-2 = 1379.5 W with efficiency 76.7%, the highest compare to others. There were significant effects between the throat opening of draft tube and ratio L/D₁ against power required. For L/D₁ < 1.2 the power decreased and also for L/D₁ > 1.2 the power decreased.

4. Conclusion

From simulation results, shown that variant C₂-2 with the length of draft tube L = 1.2 D₁ = 240 mm was the highest power with the best flow trajectory and velocity from the first stage to the second stage of the blade and did not strike the shaft, efficiency is 76.7 % was the highest compared to others.

Acknowledgments

Authors wishing to acknowledge to all my colleagues that already supported this research.

References

[1] Minister of Energy and Mineral Resources The Republik of Indonesia 2011 Regulation of Minister of Energy and Mineral Resources The Republic of Indonesia No.8
[2] MOCKMORE C A and MERRYFIELD F 1949 The Banki Water Turbine vol 2 Engineering Experiment Station (of Oregon State College)
[3] CHOI Y-D, LIM J-I, KIM Y-T and LEE Y-H 2008 Performance and Internal Flow Characteristics of a Cross-Flow Hydro Turbine by the Shapes of Nozzle and Blade Runner Journal of Fluid Science and Technology 3(3) pp 398-409
[4] De Andrade J, Curiel C, Kenyery F, Aguilln O, Vásquez A and Asuaje M 2011 Numerical investigation of the internal flow in a Banki turbine International Journal of Rotating Machinery pp 1-12
[5] Kokubu K, Kanemoto T, Son S-W and Choi Y-D 2012 Performance Improvement of a Cross-Flow Micro Eco Hydro Turbine. J. of the Korean Society of Marine Eng, 36 (7), 902-909
[6] Sammartano V, Arico C, Carrawetta A, Fecarotta O and Tucciarelli T 2013 Banki-Michell optimal design by computational fluid dynamics testing and hydrodynamic analysis Energies 6(5) pp 2362-2385
[7] Kaniecki 2014 Modernization of the outflow system of cross-flow turbines *TASK Quarterly: Scientific Bulletin S* **4(4)** pp 601-608
[8] Kaunda C S Z and Cuthbert K N K 2014 A numerical investigation of the flow profile and performance of a low cost Crossflow turbine *International J. of Energy and Environment* **5(3)** pp 275-296
[9] Girma M and Dribssa E 2014 Flow Simulation and Performance Prediction of Cross Flow Turbine Using CFD Tool *International J. of Eng. Research and General Science* **2 (6)** pp 747-757
[10] Adhikari R C, Vaz J and Wood D 2016 Cavitation inception in crossflow hydro turbines *Energies* **9** pp 1-12
[11] ZAR N T W, HTAY H W and MYINT T 2016 Design, Construction and Performance Test of Cross-Flow Turbine *International J. of Mech. And Prod. Eng.* **4(12)** pp 95–100
[12] Jasa L, Ardana I P, Priyadi A and Purnomo M H 2017 Investigate Curvature Angle of the Blade of Banki's Water Turbine Model for Improving Efficiency by Means Particle Swarm Optimization *International J. of Renewable Energy Research* **7(1)**
[13] Chattha J A, Zaffar A, Ibrahim B, Asif M and Sarwar M A 2018 Optimisation of blade profiles of cross flow turbine *International J. of Power and Energy Conversion* **9(4)** pp 311
[14] Choi Y-D and Son S-W 2012 Shape Effect of Inlet Nozzle and Draft Tube on the Performance and Internal Flow of Cross-Flow Hydro Turbine *J. of the Korean Society of Marine Eng.* **3(36)** pp 351-357
[15] Choi Y-D, Shin B-R and Lee Y-H 2010 Air Layer Effect on the Performance Improvement of a Cross-Flow Hydro Turbine *J. of Fluid Machinery* **13**