The Effect of Blades Gap on Propeller Open-flume Picohydro Turbine Performance

Warjito\textsuperscript{1}, Budiarso\textsuperscript{1}, Muhammad Farhan Syahputra\textsuperscript{1} and Sanjaya BS Nasution\textsuperscript{1}

\textsuperscript{1}Department of Mechanical Engineering, Universitas Indonesia
Beji, Depok, 16425, Indonesia, warjito@eng.ui.ac.id, budiarso@ui.ac.id
syahputramuhammadfarhan@gmail.com, sanjayabaroonasution@gmail.com

Abstract

To meet the target of using renewable energy and electrification in remote areas, Indonesia needs to develop pico hydropower plants. One of the picohydro turbines that can be used is the propeller openflume turbine. To obtain the optimum performance, a numerical study of the effect of the gap between blades was carried out. This study analyzes 3 different blades by using the computational fluid dynamics method. The difference between the three blades is the gap between the blades. The first blade is designed with gaps, the second blade is without gaps, and the third blade is designed with overlap gaps. Based on the numerical results, it is shown that the gap on blades of propeller openflume turbine affects the performance of turbine with the overlap turbine has a better performance than the gap and no-gap blade. The overlap blades generate 9.48 Watts power and 8.8% efficiency at 202 rpm.

Keywords: Renewable energy, Picohydro, propeller, openflume, Blades gap, CFD

1. Introduction

Indonesia has a target to use 23% of its total renewable energy sources by 2025 and 31% by 2050 [1]. In 2018, Indonesia only used 12.3% of its renewable energy sources, where the total renewable energy available in Indonesia was 443 GW [2][3]. One of the greatest potential sources of renewable energy in Indonesia is hydropower with a potential of 75 GW [2][4][5]. Therefore, hydropower can be used as one of the best options to achieve Indonesia's targets in terms of using renewable energy as previously described.

Besides being able to be used to achieve the target of using renewable energy, hydropower can also be used to electrify remote areas. One type of hydropower that is suitable for remote areas is the pico-scale hydropower, commonly called the Pico-hydro turbine. Pico-hydro turbines are suitable for remote areas, because the pico-hydro turbine has low manufacturing and maintenance cost [6].

The propeller turbine is a pico hydro turbine that is often used in remote areas. The advantage of a propeller turbine compared to the other types of the turbine is that it has a wide specific speed range so that the propeller turbine can work optimally in various head conditions and flow rates [7][8][9]. The geometry of the propeller turbine is similar to the Kaplan turbine. The difference between the two turbines is that the propeller turbine has fixed guide vanes; thus, the propeller turbine has lower manufacturing and maintenance costs compared to the Kaplan turbine. The pico scale propeller turbine itself works most effectively at heads less than 5 m.

Several studies have been carried out to improve the performance of propeller turbines. Bozic et al. propose a method to predict secondary losses on propeller turbines using Voytashevski’s formula, and the result is the secondary energy losses will be reduced along with increasing flow rate [10]. Singh et al. explained the effect of the hub to tip ratio and the number of blades on the propeller turbine. They concluded that the number of blades significantly affects turbine performance compared to the hub to tip ratio [11]. Byeon and Kim explained that the propeller turbine at head conditions below 5 m, the number of effective blades are 4 [12]. A Podnar et al. suggest a method to determine cavitation in propeller turbines using computer-aided visualization. Besides, Podnar et al. proposed to be selective in determining the profile of blades to use because this affects the formation of cavitation (reduces efficiency) [13]. Simpson and Williamson were developing methods for pico scale propeller turbine manufacturing for remote areas. That study found that an incorrect match between turbine rotor design and available flow rates can significantly affect the turbine operation [14]. K.V. Alexander et al. compared four different propeller turbine runners with four different Ns 176, 242, 355, and 544. The study found that the turbine with Ns of 242 works most efficiently compared to other turbines [15]. Nasution et al. propose that the propeller turbine design based on the specific speed Ns with the power function is better than using the specific speed with the flow rate function based on numerical analysis [16]. Adanta et al. explained that the distance between the blades could change the flow velocity at the outlet where this makes the torque decrease, which results in decreased efficiency [8].

Received September 14 2020; revised October 26 2020; accepted for publication February 17 2021: Review conducted by Xuelin Tang, Ph.D. (Paper number O20046C)
Corresponding author: Muhammad Farhan Syahputra, Research assistant, syahputramuhammadfarhan@gmail.com
This study was conducted to follow up on the study by Adanta et al., which investigated the gap between blades. A study conducted by Adanta et al. used a different number of blades. Thus, the difference in blade performance in the study could be due to the difference in the number of blades, not only the difference in the gap between the blades. Therefore, this study will use the number of blades four from Ho-Yan’s study [17] with three variations of the distance between the blades, namely: gap blade, no gap blade, and the overlap blade. The method used in this study uses computational fluid dynamics (CFD) because the CFD method can produce accurate calculations compared to the analytical approach and does not require high time and cost like the experimental method [18][19].

2. Methodology

2.1 Turbine Geometry

This study compares 3 blades with the same boundary condition, which are discharge (Q) and Head (H). The 3 blades also have the same initial parameters such as the hub to tip ratio, the number of blades (z) and the velocity triangle which are adapted from Ho-Yan’s Study [17]. The variation in this study is the distance between the blades where there are three variations of the distance between the blades, namely: gap blade, no gap blade, and overlap blade. Fig. 1 is a front view of each blade variation and Fig. 2 is a top view of each blade variation. The following is Table 1 which is the main parameters of blade and draft-tube geometry.

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of blades, z</td>
<td>4</td>
</tr>
<tr>
<td>Outer diameter, D_o</td>
<td>130 mm</td>
</tr>
<tr>
<td>Inner diameter, D_i</td>
<td>70 mm</td>
</tr>
<tr>
<td>Stagger angle, β</td>
<td>73.7°</td>
</tr>
<tr>
<td>Absolute flow angle-inlet, α_1</td>
<td>34.8°</td>
</tr>
<tr>
<td>Relative flow angle-inlet, β_1</td>
<td>72.4°</td>
</tr>
<tr>
<td>Absolute flow angle-exit, α_2</td>
<td>7.5°</td>
</tr>
<tr>
<td>Relative flow angle-exit, β_2</td>
<td>74.9°</td>
</tr>
<tr>
<td>Inlet Diameter, D_in</td>
<td>141.3 mm</td>
</tr>
<tr>
<td>Outlet Diameter, D_out</td>
<td>298.06 mm</td>
</tr>
<tr>
<td>Draft tube height, T</td>
<td>1700 mm</td>
</tr>
</tbody>
</table>

2.2 Simulation Set-up

Computational calculations in this study are carried out with the ANSYS FLUENT® 18.1 computational fluid dynamics (CFD) software. This study was conducted with a three-dimensional (3D) analysis and used the six-degrees of freedom (6-DoF) feature. The 6-DoF feature was chosen because this feature makes the rotation as an independent variable. The rotation depends on the loading fluid on the blade. The 6-DoF function is different from MRF (multi-reference frame) and Mesh motion, where
rotation is the dependent variable [20][23]. With six DoF the rotation will change each time step according to the external force obtained by the blade from the fluid flow through it. The numerical method in this study used the governing equations which are the mass conservation and momentum conservation equation. Thus equations are shown below.

$$\frac{\partial v_j}{\partial x_j} = 0$$  \hspace{1cm} (1)

$$\rho \left( \frac{\partial v_i}{\partial t} + \frac{\partial}{\partial x_j} (v_i v_j) \right) = - \frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \left( \frac{\partial v_i}{\partial x_j} + \frac{\partial v_j}{\partial x_i} \right) \right) + f$$  \hspace{1cm} (2)

The stages in this computational method are divided into three parts, which are: 3D geometry creation, meshing, and simulation setup. At the stage of making 3D geometry, this study uses the 3D CAD application [24]. Then, the geometry that has been created in the CAD application is then imported into fluent. The next step is to determine the number of elements. However, before summing the elements, each part of the geometry is named so that the part is defined. This step is called the create name selection stage. Then, on the meshing stage, to determine the correct number of elements, a mesh independency test process is needed. As with meshes, the appropriate timestep size also needs to be determined, this is called timestep independency. Mesh and timestep independency tests are done because the difference in the size of the timestep can affect the accuracy of the simulation [25][26].

After finding the number of meshes and the appropriate timestep, the final step in the computational method is the simulation setup. At this stage, boundary conditions are defined. The simulation in this study used the mass flow rate of 40 kg/s water with zero static pressure at the inlet boundary condition and used zero pressure at the outlet boundary condition. The turbulent flow type was also chosen to determine the turbulent model that is closer to the original result. The turbulent model used in this study is the k-ε standard. Adanta et al suggested using the standard k-ε turbulent model in a hydro turbine to reduce simulation time [27]. After carrying out these stages, the next step is calculation. After completing the simulation stage, the results of the simulation can be seen in the results option. In the results section, data such as torque, speed contour, pressure, flow contour, and so on can be displayed. The following Fig. 3 is a 3D geometry visualization of propeller turbines and mesh visualization of propeller turbine geometry.

![Boundary condition](image1.png)
![Visualization of mesh](image2.png)

**Fig. 3** Simulation set-up and mesh visualization

### 2.3 Independency test

In this study, the number of mesh that was varied was three. The variable observed was the torque generated by each of the variations in the number of meshes. The mesh elements used in this variation are 318306, 691996, and 1460865. To determine the
To determine the number of time steps, the GCI analysis was adopted to determine the optimum discrete time called the timestep convergence index (TCI). The various values of the frequencies are 1000 Hz, 2500 Hz, and 5000 Hz. On Fig. 4, there is the result of the TCI calculation. Based on Fig. 4, the time step with a frequency of 2500 Hz is considered to be optimum because it has an error of ± 1%.

### 3.1 Independence test results

The number of mesh elements used in study is varied into three variations which are 318306, 691996, and 1460865 elements. Torque at each number of elements is calculated to analyze the appropriate number of elements with GCI calculation method. The GCI calculation results are shown in Table 4 where the best number of elements is 1460865.

![Fig. 4 Time step independency](image)

**Table 2 GCI results**

<table>
<thead>
<tr>
<th>Normalized Grid Spacing</th>
<th>Number of Elements</th>
<th>Torque, τ</th>
<th>GCI</th>
</tr>
</thead>
<tbody>
<tr>
<td>2.14</td>
<td>318306</td>
<td>0.806 N·m</td>
<td>-</td>
</tr>
<tr>
<td>1.45</td>
<td>691996</td>
<td>0.835 N·m</td>
<td>0.718%</td>
</tr>
<tr>
<td>1</td>
<td>1460865</td>
<td>0.839 N·m</td>
<td>0.109%</td>
</tr>
<tr>
<td>0</td>
<td>Δx→0</td>
<td>0.839 N·m</td>
<td>-</td>
</tr>
</tbody>
</table>

3.2 Turbine performance

Since the simulation was conducted with Six-DoF methods, the blade rotations and torque were continuously changed in every time step. Figure 5 shows the blade rotation and torque at each time steps which are started from 0 s to 2.5 s. It is shown that the overlap blade has higher blade rotation and torque than others. The higher rotation and torque indicates that the overlap blade adsorbs more energy than two others blade.
Fig. 5 Blade rotation and torque

Figure 6 is the comparison of power and efficiency of blades. Power and efficiency are calculated based on equation 1 and 2 below. $P_b$ is power generated by blade turbine, $P_i$ is inlet or potential power, $\tau$ and $\omega$ are torque and angular velocity, $Q$ and $H$ is water discharge and Head, and $\eta$ is turbine efficiency.

$$P_b = \tau \times \omega$$  \hspace{1cm} (3)

$$P_i = \rho g Q H$$  \hspace{1cm} (4)

$$\eta = \frac{P_b}{P_i} \times 100\%$$  \hspace{1cm} (5)

Based on Fig. 6, the overlap blades produce the greatest power and efficiency and no-gap blade produce the lowest power and efficiency. The overlap blades generate 9.48 Watts power and 8.8% efficiency at 202 rpm. Then the blade with gap blade produces power and efficiency of 9.1 Watt with an efficiency of 8.48% at 206.5 rpm. At the same time, the no gap blade distance produces the lowest power and efficiency which is 8.16 Watt power and 7.58% efficiency at 192 rpm.

Fig. 6 Power and efficiency of turbine

Figure 6 shows that the turbine performance based on simulation and experiment has a similar pattern even though at each rotation there is a difference in the power generated. The power generated by the simulation calculations is greater than the power generated by the experimental results. This error between simulation and experiment occurs due to additional losses in the experimental study such as mechanical losses, surface roughness which was not taken into account in the simulation, imperfect turbine component manufacturing results, etc. As for the error between simulation and experiment at the maximum turbine efficiency is 0.16. Based on the errors and the pattern of power in Fig. 6, it shows that the CFD results are in accordance with experiment analysis. Additionally, the experiment data in Fig. 6 is conducted from Ho-Yan’s study. Ho-Yan’s study has the same geometry and boundary condition with the no gap blade [17].
6.3 Discussions

The propeller turbine is a turbine that converts the hydraulic energy of the water to mechanical power. The kinetic and potential energy from the water flow source is converted into pressure energy at the propeller turbine blade, then the pressure energy on the blade surface converts to mechanical power. Propeller blade has two sides which are pressure side (upper side) and suction side (lower side). The pressure difference between upper and lower side generates mechanical power. Figure 7 shows the pressure distribution on the upper side. The higher the pressure on the upper side, the greater the mechanical power produced. Based on Fig. 7, it can be seen the pressure distribution is higher at the leading edge of the blade. This indicates that the stagnation point is close to the leading edge. The stagnation point is the point where the velocity of the fluid passing through a surface is zero. Stagnation Point is also the point where static pressure is maximum. Other studies show the same phenomena which the highest pressure on the blade surface is near the leading edge and then the pressure reduces [16]. Besides that, it can also be seen at the leading edge near the hub, there is an area with low pressure. The low pressure area at gap and no-gap blades is wider than at overlap blade. The wider the low pressure area, the lower the mechanical power generated by the turbine. In addition, the pressure distribution on the hub does not affect turbine performance.

![Pressure Distribution](image)

**Fig. 7** Pressure contour on the top surface of blades at 200 rpm

Figure 8 is pressure coefficient (Cp) on blade surfaces. Pressure coefficient is a variable that gives important information of blade loading. Cp is calculated based on eq. (4) which is the ratio between static pressure at the blade and dynamic pressure of free stream. P is static pressure on the blade surface, \( P_\infty \) is static pressure of free stream, and \( V_\infty \) is velocity of free stream. Cp is plotted in mid span as shown in Fig. 9.

\[
C_p = \frac{P - P_\infty}{1/2 \rho V_\infty^2}
\]  

![Pressure Coefficient](image)

**Fig. 8** Pressure coefficient of mid-span section at 200 rpm
The difference between \( C_p \) on the upper side and the lower side generates blade loading or mechanical power. The higher the difference of \( C_p \) on blade sides is, the greater the power output. The pressure coefficient distribution in Fig. 8 shows that the overlap blade has higher pressure on upper side than others. It is also in accordance with Fig. 7 where the pressure on the gap and no gap blade decreases significantly after through the leading edge (leading edge is in \( x/C_0 \) and trailing edge is \( x/c_1 \)). On the other side, there is no significant difference of pressure coefficient on the lower side. This condition explains the reason why the overlap blade generates higher power than gap and no gap blades.

![Pressure contour](image)

**Fig. 9** Pressure contour on mid-span of blades at 200 rpm

In order to show the differences of pressure distribution on the upper and lower surface, the contour of pressure is plotted on the mid span section plane. Mid span position is in the middle of hub and tip blade. Figure 9 shows the pressure distribution related to blade surface and gap between blades. Based on Fig. 9, it is shown that the pressure on the upper side of the overlap blade is more distributed well. Figure 9 shows that there is a drop in pressure in the area between the blades. Thus, pressure indicates the losses or no uniform flow. The extended explanations are shown in Fig. 10. Furthermore, based on Fig. 9, the smallest pressure drop is on the overlap blades. The larger pressure drop in the blade gap and the no-gap causes the performance of those blades to be lower than the overlap blade.

![Velocity contour](image)

**Fig. 10** Velocity contour of mid-span section at 200 rpm
Figure 10 is a visualization of the velocity contour in the mid span of blades. Based on Fig. 10, it is shown that velocity distribution in gap and no gap blades are more random than overlap blade (see fig.10). Fig. 10 also shows how the velocity is decreased in the red circle on the gap and no gap blade. On the other side, there is no significant decrease in Overlap blade. This explains the reason for the low pressure in the area (red circle) as shown in Fig. 9. The decrease in velocity in Fig. 10 indicates the change in the direction of the flow velocity. This change causes the velocity magnitude to decrease (see Fig. 10). Then, the decrease in the velocity magnitude indicates an increase in losses at the blade gap and no gap. The change in flow direction can be seen more clearly in Fig. 11 and Fig. 12.

Figure 12 is a stream-line velocity. Figure 12 shows that at the blade gap and no gap the water flow through the gaps between the blades is disturbed in the gap between the blades. The causes of disruption of water flow at the blade gap and no gap are due to
the high flow velocity at the leading edge of the upper blade and relatively different flow directions (see Fig. 10.) This causes the shear stress in the area to be high. The high shear stress can cause swirling flow in the area. The swirling flow phenomenon can be seen based on the high vorticity in the area. Figure 11 shows the vorticity contours of the three blades. Based on Fig. 11, the vorticity in the area between the blades is higher at the gap and the no-gap blades compared to the overlap blades. The high vorticity in the area between blades indicates the flow unable to through the blade properly. This is why the vorticity presence in hydrodynamic blade is unexpected.

4. Conclusion

In this study, the effect of blade gap on propeller hydro turbine was investigated using computational fluid dynamics methods. The unsteady simulation was conducted in this study with six degree of freedom method. The performance and flow characteristics of three blades which are gap, no gap, and overlap blade has been calculated and analyzed. Based on the numerical results, it is shown that the gap on blades of propeller openflume turbine affects the performance of turbine. The overlap turbine has a better performance than gap and no-gap blade with the overlap blades generate 9.48 Watts power and 8.8% efficiency at 202 rpm. Then the blade with gap blade produces power and efficiency of 9.1 Watt with an efficiency of 8.48% at 206.5 rpm. At the same time, the no gap blade distance produce power and efficiency which is 8.16 Watt power and 7.58% efficiency at 192 rpm.

Acknowledgments

This work was supported by the Ministry of Research, Technology, and Higher Education (KEMENRISTEK DIKTI) of Republic of Indonesia with grant No: NKBI-2870/UN2.RST/HKP.05.00/2020

Nomenclature

<table>
<thead>
<tr>
<th>CFD</th>
<th>Computational Fluid Dynamics</th>
</tr>
</thead>
<tbody>
<tr>
<td>$P_i$</td>
<td>Potential Power (Watt)</td>
</tr>
<tr>
<td>$C_p$</td>
<td>Pressure Coefficient</td>
</tr>
<tr>
<td>$g$</td>
<td>Gravitation (m/s$^2$)</td>
</tr>
<tr>
<td>$H$</td>
<td>Head (m)</td>
</tr>
<tr>
<td>$P$</td>
<td>Static Pressure (Pa)</td>
</tr>
<tr>
<td>$Q$</td>
<td>Discharge (m$^3$/s)</td>
</tr>
<tr>
<td>$P_h$</td>
<td>Hydrolic power (Watt)</td>
</tr>
<tr>
<td>$P_{\infty}$</td>
<td>Static pressure of freestream (Pa)</td>
</tr>
<tr>
<td>$V_{\infty}$</td>
<td>Velocity of free stream (m/s)</td>
</tr>
<tr>
<td>$\rho$</td>
<td>Fluid Density (kg/m$^3$)</td>
</tr>
<tr>
<td>$\tau$</td>
<td>Torque (N.m)</td>
</tr>
<tr>
<td>$\omega$</td>
<td>Blade rotation (rad/s)</td>
</tr>
<tr>
<td>$\eta$</td>
<td>Efficiency (%)</td>
</tr>
</tbody>
</table>

References