Effect of tangential absolute velocity at outlet on open flume turbine performance

Sanjaya BS Nasution¹, Dendy Adanta¹, Warjito and Budiarto¹

1 Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Indonesia
E-mail: warjito@eng.ui.ac.id

Abstract. To address the electricity crisis in Indonesia, the use of open flume pico hydro turbines is a potential solution because Indonesia has 19 GW of hydropower resources at mini, medium, and pico scales. There are two concepts for the design of an open flume pico hydro turbine runner: Euler and Nechleba. Euler’s concept recommends that the absolute tangential velocity at the outlet of the blade be zero to optimise power absorption. By contrast, Nechleba explains that to reduce flow separation at the outlet of the blade (separation can reduce efficiency), tangential absolute velocity at the outlet of the blade must not be zero. Accordingly, this study compares the performance of blades designed with these two concepts using a computational fluid dynamics method. The results showed that the maximum efficiency of the blade using the Euler concept was 70.08%, while the blade using the Nechleba concept was 74.39%. Based on the results, a runner design concept that assumes a non-zero tangential absolute velocity at the outlet of the blade is recommended for use in designing an open flume pico hydro turbine runner.

1. Introduction
Indonesia has 19 GW of hydropower resources at mini, medium, and pico scales [1][2][3]. However, Indonesia still faces a lack of electricity, especially in rural areas [4]. To overcome this problem, Indonesia is expected to be able to utilise its hydropower resources. One possible option is the use of open flume pico hydro turbines.

An open flume turbine is a pico-scale hydropower turbine which can generate high rotation with stable efficiency [5][6]. An open flume turbine is an axial turbine whose performance depends on the shape of its blade. In designing a blade shape, a velocity triangle is used. A velocity triangle is a triangle that represents absolute, relative, and rotational velocity, and can be represented by a Euler equation [7]. Based on Euler’s equation, to generate the highest power, the absolute velocity tangential at the outlet \(C_{ax2}\) of the blade should be zero [7][8].

Contradicting Euler’s equation, Hothersall [9] explained that to reduce the separation of flow at the outlet of the blade, where separation could reduce efficiency, the absolute velocity tangential should not be zero. As an example, Singh and Nestmann [10] investigated the exit tip angle effect on runner performance. Their experimental analysis showed that increasing the exit tip blade angle could increase the efficiency of the runner [10]. In addition, Nechleba [9] and Williamson [11] revealed that to receive the highest power, the absolute velocity tangential at the outlet should be the same in magnitude as the relative velocity.

In order to prove the Nechleba explanation and to determine the most appropriate method to calculate the velocity triangle, this study conducts a computational fluid dynamics simulation to compare the performance of two blades, one based on Nechleba’s suggestion and the other on a Euler equation.
Furthermore, to describe the comprehensive fluid phenomena at the blade runner, this study conducts a 3D unsteady calculation. The results of the study are expected to act as a future reference for the design of open-flume turbine blades.

2. Methodology

2.1. Geometry of the turbine

This study compares two blades with the same initial parameters. The initial parameters in this case are a discharge \( Q \) of 0.040 \( \text{m}^3/\text{s} \), a diameter hub \( (D_h) \) of 70 mm, a diameter tip \( (D_t) \) of 125 mm, and the number of blades \( (z) \), 5. The blades are differentiated only by blade angle. Blade angles are calculated based on the velocity triangle method. The first blade, which we call the Euler blade, is designed with a \( C_{x2} \) magnitude of zero \( (C_{x2} = 0) \). The second, the Nechleba blade, has a \( C_{x2} \) magnitude of not zero \( (C_{x2} \neq 0) \) (see Figure 1).

![Figure 1. Velocity triangle.](image)

The equation below is a power equation based Euler’s equation. This equation explains that to receive maximum power, \( C_{x2} \) should be zero [7]:

\[
P = \dot{m}U(C_{x1} - C_{x2})
\]

where \( \dot{m} \) is the mass flowrate, \( U \) is the turbine rotational velocity, and \( C_{x1} \) is the absolute velocity tangential at inlet. On the other hand, Dixon [12] and Gorla et al. [13] revealed that for axial flow turbo machinery, free vortex theory is used. Free vortex theory explains that the product of tangential flow velocity and radius vector should be constant [14][15]. Therefore, velocity triangles are differentiated from hub to tip [16]. Furthermore, in this case, the blade is divided into seven sections for which the velocity triangle at each section is calculated based on these equations:

\[
C = U + W
\]

\[
C = C_x + C_r
\]

\[
C_r = W + W_x
\]

\[
\text{blade angle} = \frac{\beta_1 + \beta_2}{2}
\]

where \( C \) is absolute velocity, \( W \) is relative velocity, \( C_r \) is absolute velocity radial, \( W_x \) is relative velocity tangential, and \( \beta_1 \) is the relative velocity angle at inlet and \( \beta_2 \) at outlet.

Figure 1 shows the geometry of the two blades. Figure 1a is a blade with its blade angle calculated based on Euler and Figure 1b is calculated based on Nechleba’s recommendation. Based on the velocity triangle, the blade angle was also obtained.

2.2. Numerical calculation
Numerical calculation is performed with computational fluid dynamics (CFD) software. This study is performed with three-dimensional (3D) analysis. The simulation contains three steps: pre-processing, processing, and post-processing. Pre-processing is performed before running the simulation. It involves defining geometry, boundary condition, and meshing. Processing is the running process of the simulation. In this case, the simulation was run with unsteady calculation. Post-processing displays the results of the simulation.

Figure 2 shows the boundary condition of the study. The inlet is defined as a mass flow inlet with a discharge of 40 l/s, and the outlet is defined as a pressure inlet. In addition, the simulation also contains a stator and rotor interface. This is because the simulation set-up was conducted with a six-degrees of freedom set-up (6-DoF). The 6-DoF is used based on the recommendations of other researchers to define rotational velocity as the output of a simulation [17][18][19]. Therefore, in this study, blade rotation was not defined in the initial parameter. Further, as this is a 6-DoF simulation, it is also necessary to define the moment of inertia of the blade runner. The moment of inertia of the blade in this case is 0.05 N.m. Finally, the k-ԑ model is used as the turbulent model. Dendy et al. [20] recommend the k-ԑ model in turbo machinery simulation to reduce time consumption. Simpson et al. [11] also used the k-ԑ model to predict the performance of a propeller turbine.

2.3. Mesh and time-step independency test
After defining the boundary condition, mesh and time-step independency tests were used. The mesh independency test is a method to determine the appropriate mesh size and number of elements [21]. This study used three different numbers of elements: 1508868, 3062380, and 6006250. To determine the appropriate mesh size, the torque at each mesh number was calculated. Furthermore, the error of simulation was calculated based on the Roache method called the grid convergence index [22]. Figure 2b shows the geometry mesh of all domains; to improve the mesh quality, mesh size near the wall of the blade was refined. The same method was used in the time-step convergence test (TCI). Time step size differs from 0.002 s to 0.001 s and 0.0005 s.

3. Results and discussion
3.1. Runner blade geometry
From hub to tip, the blade was divided into seven sections. This condition induced the angle and chord line at each section to be different. Table 1 below describes the results of calculations according to blade geometry. It shows that the blade angles were different at each section for both blades: the blade angles of the two blades decreased from hub to tip. There was no significant difference at the tip, but at the hub, the blade angle of the Euler runner was lower than that of the Nechleba runner. The chord lines (L) of the two blades in this study were also the same.
Table 1. Comparison of Euler and Nechleba blades.

| $D_b/D_t$ | Blade Angle (°) | $L$ (m) |
|---------|----------------|---------|
|         | Euler | Nechleba |         |
| 1       | 12.51 | 28.79    | 0.022   |
| 0.8     | 24.97 | 26.41    | 0.023   |
| 0.67    | 40.77 | 31.95    | 0.033   |
| 0.57    | 50.6  | 45.57    | 0.045   |
| 0.5     | 57.09 | 54.11    | 0.060   |
| 0.44    | 61.68 | 59.79    | 0.076   |
| 0.4     | 65.09 | 63.82    | 0.094   |

3.2. Mesh independence test result

Figure 3 shows the results of the calculation of convergence index for mesh and time step. Normalisation of space in Figure 3 relates to the comparison of the independent variable. For CGI, 6006250 elements normalised to 1, 3062380 normalised to 1.96, and 1508868 normalised to 3.98. From Figure 3, it can be seen that increasing the number of elements induces a increasing of blade torque. And in this study, based on CGI analysis, 4402703 elements are used with an error of 0.36 % compared to the exact value.

![Figure 3. Mesh and time step independency.](image)

Furthermore, the same method is used for the TCI. Time step size 0.0005 normalised as 1, 0.001 normalised as 2, and 0.002 normalised as 4. Based on the TCI calculation, the appropriate time-step size was 0.01 with error of 1.2 % compared to exact value.

Figure 4. Torque and efficiency of turbine.

![Figure 4. Torque and efficiency of turbine.](image)

3.3. Runner performance

The results of the numerical calculation are shown in Figure 4. Figure 4 is a graph comparison of blade performance between Euler’s and Nechleba’s blades. The graph compares the torque and efficiency of two turbines to velocity ratio (U/V). The U/V is the comparison between rotational velocity and inlet flow velocity. As shown in Figure 4, increasing rotational velocity (U), in which the inlet flow velocity (V) is constant, decreases the torque of both blades. This is because at constant power, rotational velocity is inversely proportional to torque.

Figure 4 also shows the efficiency of the blades. At low rotational velocity, both blades have similar efficiency; however, there is a significant difference in efficiency at high rotational velocity. The Euler blade has a peak efficiency 70.08 % while the Nechleba blade has a peak efficiency 74.39 %. On the other hand, as seen in Figure 4, to generate maximum efficiency it is necessary to consider the optimum U/V ratio. The optimum U/V ratio for a Euler blade is 1.4 and for a Nechleba blade 1.7.
Thus, based on the results shown on Figure 4, it is recommended that an open flume turbine blade be designed using the Nechleba concept where $C_{x2} \neq 0$, because it generates the higher performance.

4. Conclusion

The results of numerical studies using a computational fluid dynamics method show that there are differences in performance between a Euler blade and a Nechleba blade. The study found that the optimum efficiency of a Euler blade is 70.08% at $U/W 1.4$, while a Nechleba blade has an optimum efficiency of 74.39% at $U/W 1.7$. Thus, based on this study, the Nechleba concept is recommended for the design of an open flume turbine blade.

5. References

[1] P. Gokhale et al., “A review on micro hydropower in Indonesia,” *Energy Procedia*, vol. 110, pp. 316–321, 2017.
[2] N. A. Wibowo, V. Dermawan, and D. Harisuseno, “Studi Perencanaan Pembangkit Listrik Tenaga Mikro Hidro (PLTMH) Wamena di kabupaten Jayawijaya Provinsi Papua,” *Malang Tek. Pengair. Fak. Tek. Univ. Brawijaya*, 2013.
[3] R. A. Subekti, “Survey Potensi Pembangkit Listrik Tenaga Mikro Hidro di Kuta Malaka Kabupaten Aceh Besar Propinsi Nanggroe Aceh Darussalam,” *J. Mechatronics, Electr. Power, Veh. Technol.*, vol. 1, no. 1, pp. 5–12, 2012.
[4] ESDM, “Statistik Ketenagalistrikan,” 2015.
[5] S. B. Nasution, Budiario, Warjito, and D. Adanta, “A Comparison of Openflume Turbine Designs with Specific Speeds (Ns) Based on Power and Discharge Functions,” *J. Adv. Res. Fluid Mech. Therm. Sci.*, vol. 45, pp. 53–60, 2018.
[6] S. J. Williamson, B. H. Stark, and J. D. Booker, “Low head pico hydro turbine selection using a multi-criteria analysis,” *Renew. Energy*, vol. 61, pp. 43–50, 2014.
[7] Harinaldi, Budiario, *Sistem Fluida: Prinsip Dasar dan Penerapan Mesin Fluida, Sistem Hidrolik, dan Sistem Pneumatik*. Jakarta: Erlangga, 2015.
[8] G. F. Round, *Incompressible flow turbomachines: design, selection, applications, and theory*. Elsevier, 2004.
[9] M. Nechleba, “Hydraulic turbines, their design and equipment,” 1957.
[10] P. Singh and F. Nestmann, “Exit blade geometry and part-load performance of small axial flow propeller turbines: An experimental investigation,” *Exp. Therm. Fluid Sci.*, vol. 34, no. 6, pp. 798–811, 2010.
[11] R. G. Simpson and A. A. Williams, “Application of computational fluid dynamics to the design of pico propeller turbines,” in *Proceedings of the international conference on renewable energy for developing countries*, 2006.
[12] C. Hall and S. L. Dixon, *Fluid mechanics and thermodynamics of turbomachinery*. Butterworth-Heinemann, 2013.
[13] R. S. R. Gorla and A. A. Khan, *Turbomachinery: design and theory*. CRC Press, 2003.
[14] K. V Alexander, E. P. Giddens, and A. M. Fuller, “Axial-flow turbines for low head microhydro systems,” *Renew. Energy*, vol. 34, no. 1, pp. 35–47, 2009.
[15] P. Singh and F. Nestmann, “Experimental optimization of a free vortex propeller runner for micro hydro application,” *Exp. Therm. Fluid Sci.*, vol. 33, no. 6, pp. 991–1002, 2009.
[16] S.-S. Byeon and Y.-J. Kim, “Influence of blade number on the flow characteristics in the vertical axis propeller hydro turbine,” *Int. J. Fluid Mach. Syst.*, vol. 6, no. 3, pp. 144–151, 2013.
[17] D. Adanta, S. A. Arifianto, and S. B. S. Nasution, “Effect of Blades Number on Undershot Waterwheel Performance with Variable Inlet Velocity,” in *2018 4th International Conference on Science and Technology (ICST)*, 2018, pp. 1–6.
[18] D. Adanta, R. Hindami, Budiario, Warjito, and A. I. Siswantara, “Blade Depth Investigation on Cross-flow Turbine by Numerical Method,” in *2018 4th International Conference on Science and Technology (ICST)*, 2018, pp. 1–6.
[20] D. Adanta, W. Budiarso, and A. I. Siswantara, “Assessment of turbulence modelling for numerical simulations into pico hydro turbine,” J. Adv. Res. Fluid Mech. Therm. Sci., vol. 46, pp. 21–31, 2018.

[21] S. A. Richards, “Completed Richardson extrapolation in space and time,” Commun. Numer. Methods Eng., vol. 13, no. 7, pp. 573–582, 1997.

[22] P. J. Roache, “Quantification of uncertainty in computational fluid dynamics,” Annu. Rev. Fluid Mech., vol. 29, no. 1, pp. 123–160, 1997.

Acknowledgments
This work supported by Directorate of Research and Service Community (DRPM) Universitas Indonesia with grant No: NKB-0050/UN2.R3.1/HKP.05.00/2019.