SAKARYA ÜNİVERSİTESİ
FEN BİLİMLERİ ENSTİTÜSÜ DERGİSİ

Sakarya University Journal of Science
SAUJS

e-ISSN 2147-835X Period Bimonthly Founded 1997 Publisher Sakarya University
http://www.saujs.sakarya.edu.tr/

Title: A Numerical Study of a Pico Hydro Turbine

Authors: Ümit BEYAZGÜL, Ufuk DURMAZ, Orhan YALÇINKAYA, Mehmet Berkant ÖZEL, Ümit PEKPARLAK

Recieved: 2022-05-30 00:00:00

Accepted: 2022-07-04 00:00:00

Article Type: Research Article

Volume: 26
Issue: 4
Month: August
Year: 2022
Pages: 842-849

How to cite
Ümit BEYAZGÜL, Ufuk DURMAZ, Orhan YALÇINKAYA, Mehmet Berkant ÖZEL, Ümit PEKPARLAK; (2022), A Numerical Study of a Pico Hydro Turbine. Sakarya University Journal of Science, 26(4), 842-849, DOI: 10.16984/saufenbilder.1123442
Access link
http://www.saujs.sakarya.edu.tr/en/pub/issue/72361/1123442

Sakarya University Journal of Science 26(4), 842-849, 2022

A Numerical Study of a Pico Hydro Turbine

Ümit BEYAZGÜL[1], Ufuk DURMAZ*[1], Orhan YALÇINKAYA[1],
Mehmet Berkant ÖZEL[1], Ümit PEKPARLAK[2]

Abstract

Pico hydro turbines are suitable for low head applications in power plants since their efficiency is more stable than other turbine types. In some situations, computational fluid dynamics (CFD) has also been utilized as well as experimental studies for the performance prediction of water turbines at a pico scale. Also, CFD methods are getting much closer to real conditions in terms of steady-state with moving references and transient domains with rotor movements. For this purpose, electricity production related to the flow in a PHT was investigated numerically. This study presents six degrees of freedom (6-DOF) and moving reference frame (MRF) methods to predict the maximum conditions of a pico scale two-dimensional turbine by comparing the torque and angular velocities on the runner based on the turbine output power of 1 W determined by an experimental study. Besides, the effect of the torque, angular velocity, tip speed ratio, and turbine body profile was investigated comprehensively. In this regard, MRF and 6-DOF methods were performed to validate and compare the numerical model with the experimental results. Also, the results obtained from 6-DOF and MRF methods were compared to experimental study. It is concluded that PHT is generating 0.3 W power under 6.47 rad/s angular velocity with 6-DOF method, however; this value corresponds to 31.4 rad/s angular velocity against 1 W with the MRF method. Also, the maximum velocity of the turbine was 6.1 m/s according to the simulation result. It is accepted that the turbine maximum velocity inlet was 0.53 m/s based on the experimental study. As a conclusion, numerical results for the pico hydro turbine were reasonable taking the experimental study into account. It is also concluded that there is a tip speed ratio of 2.36 with the MRF method and 0.48 with the 6-DOF method between water tangential velocity and runner velocity for the turbine model.

Keywords: Pico hydro turbine, moving reference frame, 6-DOF, turbine power output, tip speed ratio

---

* Corresponding author: udurmaz@sakarya.edu.tr
[1] Sakarya University
E-mail: umitbyzgl1@hotmail.com, orhanyalcinkaya@sakarya.edu.tr, mozel@sakarya.edu.tr
ORCID: https://orcid.org/0000-0001-5534-8117, https://orcid.org/0000-0002-1471-793X, https://orcid.org/0000-0003-2380-1727, https://orcid.org/0000-0002-2439-1494
[2] TBM Industry
E-mail: u.pekparlak@porte.com.tr
ORCID: https://orcid.org/0000-0002-5554-6989

1. INTRODUCTION

A pico hydro turbine (PHT) is an energy harvesting device that uses the energy of a water pressure drop. This energy is captured by the pico turbine and converted into electricity. The availability of a continuous energy source enables optimal pressure management that is continuously maintained at a lower value to reduce water leaks, improve network efficiency, and extend pipeline lifetime. PHTs are frequently used in water distribution pipelines. It is widely used for opening and closing valves, which regulate water flow rate [1, 2]. A previous study has conducted an experimental study to predict maximum water turbine performance.

Thakur et al. [3] evaluated the performance of the modified turbine using the CFD software. They compared it with some of the leading literature designs of the Savonius water turbine with a simple two-blade Savonius turbine. The ready-made design provided better performance than the selected Savonius turbine designs. To illustrate the increased performance of the Savonius turbine, several velocity vector plots were used utilizing the impinging jet duct design. Chichkhede et al. [4] examined the design parameters of the cross-flow turbine for different nozzle opening and water inlet angles. A full 3-dimensional steady-state flow simulation of the cross-flow turbine was performed considering the body, runner, and nozzle assembly. With the increasing inlet angle, the optimum blade angle value was observed to increase for the proposed geometry. Jiyun et al. [5] developed an inline vertical cross-flow turbine to collect the potential hydropower inside water supply pipes for providing power to the water monitoring systems. Besides, the effects of tip clearance and block shape on turbine performance were investigated. The proposed model turbine can supply enough power without influencing the regular water supply for any general water leakage monitoring system. Kim et al. [6] investigated the effect of blade thickness on the efficiency of a Francis turbine. The blockage effects were studied with numerical analysis at best yield and off-design conditions. Since the blockage ratio increased, the power and yield of the hydro turbine gradually decreased. Therefore, hydraulic performance characteristics should be considered a significant factor in designing the runner blades. As a result of their studies, they verified that the blockage effect considerably affects the design of Francis turbine models. Ji-Feng Wang et al. [7] estimated the flow characteristics from the water turbine with nozzle, wheel, and diffuser based on three-dimensional numerical flow analysis. They calculated and analyzed the extracted power and torque of a composite water turbine at different rotation speeds for a specific flow rate. They showed with the simulation results that the nozzle and diffuser could increase the pressure drop on the turbine and obtain more power from the available water energy. Acharya et al. [8] numerically conducted the characteristics and fluid flow in a cross-flow hydro turbine and optimized its performance by changing various geometric parameters. They determined a base model during the process, changing the nozzle's shape, the guide vane's angle, the number of runner blades, and simultaneously the design by making simulations separately. Oliy et al. [9] designed a suitable turbine using all turbine parameters to achieve a maximum efficiency cross-flow turbine and performed computational fluid dynamic simulation as an important tool for the turbine's performance. Ranjan et al. [10] aimed to increase the efficiency of a cross-flow hydro turbine with geometric modification by conducting numerical research of the cross-flow hydro turbine in Ansys Fluent using multiple physics finite volume solver. Makarim et al. [11] examined the effect of the blade depth ratio and the number of blades on the power coefficient of cross-flow water turbines. They carried it out in a 2-D method by using Ansys Fluent software. Prakoso et al. [12] compared the six-degree of freedom method by a using user-defined function (UDF) and the moving mesh method for the simulation of small-scale cross-flow turbines. They found that 6-DOF had a smaller deviation of about 6.8% from experimental results. Wang et al. [13] discussed the hydrodynamic performance of the vertical axis water turbine model both experimentally and numerically. In line with the current research, they proposed the 6-DOF method with CFD, which is suitable for performing analysis of rotor motion and predicting hydraulic performance for a vertical axis water turbine.

In this study, a two-dimensional water turbine model was designed and simulated in the numerical study with computational fluid dynamics moving reference frame (MRF) and six-degree of freedom (6-DOF) methods. On the one hand, it obtained 31.4 rad/s angular velocity of the turbine under given 1 W with MRF method. On the other hand, it is determined that the turbine with the 6-DOF method generates almost the same power as in the experimental result. As a result of the study, the maximum turbine output power obtained from experimental data is in good agreement with the numerical simulation results.

2. METHODS

The two-dimensional geometry of the water turbine in this study was created in computer-aided design (CAD) software. PHT has a 13 mm inlet diameter, 32 mm inner diameter, and 40 mm outer diameter. Also, while the volumetric water flow rate at the inlet was 0.25 L/s, it was measured as 0.167 L/s at the outlet.

The boundary conditions of the mentioned geometry are illustrated in Figure 1. While the ratio between turbine inner diameter and outer diameter (D_o/D_i or R_o/R_i) was found to be 0.8. Also, the turbine blade inlet angle (β_1) and turbine blade outlet angle (β_2) was measured 48° and 118°, respectively.

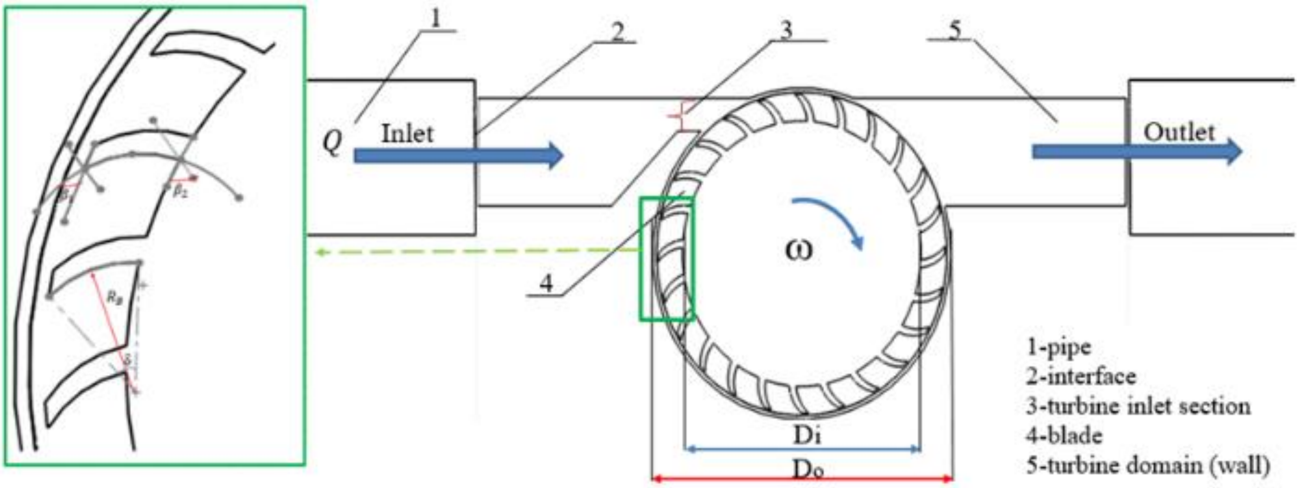

Figure 1 Schematic of geometry and boundary named selections

R_B represents the blade curve radius given relevance with β_1 in Eq. (1).

$$R_B = \frac{D_O^2 - D_i^2}{2R.cos(\beta_1)} \quad (1)$$

δ represents the blade curve angle, which is also given relevance with β_1 in Eq. (2).

$$tan\left(\frac{\delta}{2}\right) = \frac{cos(\beta_1)}{sin(\beta_1) + D_o/D_i} \quad (2)$$

Eq. (1) and Eq. (2) are simplified when the substituted value of blade inlet angle, it is obtained that R_B equals to 0.135D$_o$, and δ corresponds to 37°.

To sum up, the main parameters as a result of equation sequences can be found in Table 1.

Table 1 PHT design parameters

Design Parameter	Value (mm)
Outlet velocity, V_{outlet}	0.53
Number of blades, z	24 pcs
Outer diameter, D_0	40

Inner diameter, D_i	32
Blade's length, L	3.15
Blade's width, B	1
The angle of attack, α	23°
Blade's inlet angle, β_1	48°
Blade's outlet angle, β_2	118°
Blade curve radius, R_B	5.38
Blade curve angle, δ	37°
Turbine discharge angle, λ	90°
Turbine inlet height, S_h	4

2.1. MRF and 6-DOF Simulation Setup

2-D steady domain was used to perform the analyses in this study. The simulation was run under steady CFD simulation with enabled gravitational acceleration, which was turned on in the y-axis direction, which corresponds to $g_y = -9.81\ \mathrm{m}/s^2$. The standard wall function k-ε turbulence modeling was employed in this simulation because this provides sufficient precision and has a high accuracy of backflow and rotation. Water-liquid was determined as the main phase. The velocity inlet is 0.53 m/s given from the experimental study. The turbulence intensity was 5%, and its turbulent viscosity ratio was 10 for the velocity inlet. In addition to the MRF simulation setup, a dynamic mesh setting with 6-DOF user-defined function was used in this study. The dynamic mesh option is turned on for the settings, and the 6-DOF properties name was written as one-DOF rotation with active mode. The moment of inertia in this study was given 0.005 kg.m^2 based on the mass properties simulation in computer-aided drawing software. The coupled pressure-velocity coupling discretization scheme was used for the calculations to obtain a robust and efficient single-phase implementation for steady-state flows, with superior performance compared to the segregated solution schemes. As for spatial discretization, the standard least-squares cell-based method was employed for gradient discretization and second order for pressure discretization as the default method in the simulation program. Other variable's discretization schemes were set as first-order upwind. The first-order discretization method is a promising and stable calculation process, yet, a finer mesh is supposed to obtain a good result. The autosave during calculation was turned on every five timesteps. The computations were run for a one-second simulation divided into 300 timesteps for MRF and 6-DOF methods.

2.2. *Procedure*

2-D model of original turbine components (runner blades, runner, domain, etc.) was modeled, and numeric analysis was carried out for the standard turbine model shown in Figure 2. Required parameters such as torque and angular velocity were acquired in both 6-DOF and MRF methods, and thus turbine efficiency was calculated between numerical and experimental results.

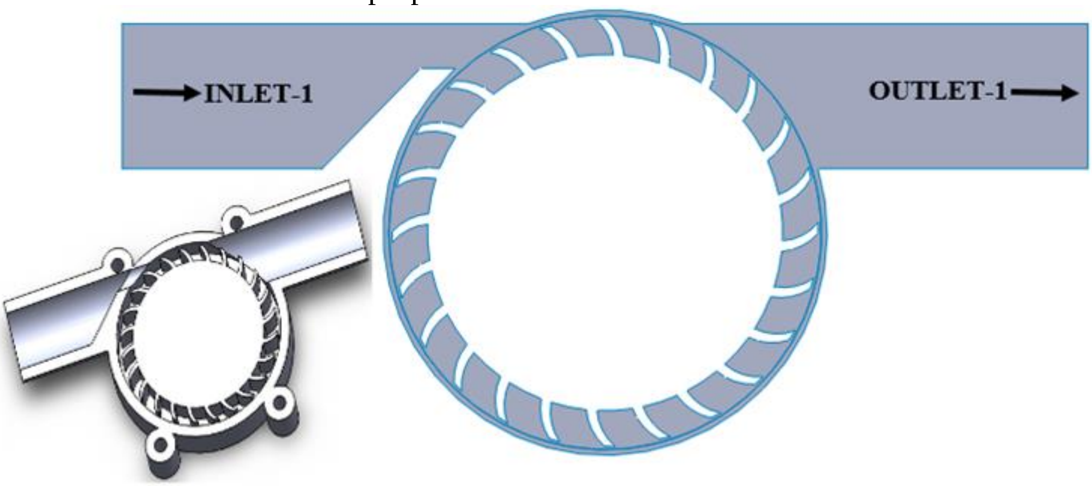

Figure 2 Original turbine body modification

The power output of PHT was determined using the basic water turbine equation between torque (T) and angular velocity (w) given in Eq. (3).

$$P_{out} = T.w \quad (3)$$

While P_{out} symbolizes the shaft power of the turbine, T is the torque at the shaft, and w is the angular velocity. Angular velocity coupled with torque will be directly calculated depending on the power in the named expressions section in CFD-Post simulation by using Eq. (8).

$$w = \frac{2.\pi.n}{60} \quad (4)$$

Where n is ascribed to the revolution per minute of the turbine.

In accordance with the analysis result, the tip speed ratio (TSR) is introduced along with the torque of blades, angular velocity, and the water velocity from the inlet to the outlet of the turbine that was recorded based on the volume flow rate in the experimental study. TSR means the ratio of the tangential velocity (V_t) of the turbine to the turbine runner velocity (V), and it can be calculated using Eq. (5), where r is the runner radius.

$$TSR = \frac{wr}{V} \quad (5)$$

3. RESULTS AND DISCUSSIONS

Original turbine MRF simulation was implemented to predict torque and angular velocity in a given power output defined by the experimental study. Firstly, to initiate the numerical study, it was given lower and upper angular velocity ranges to run the simulation automatically for the original turbine under the velocity of 0.53 m/s. It was found that angular velocity of 31.4 rad/s and the torque of 0.03 Nm were respectively determined to run the simulation under 1 W. The turbine was also run in the 6-DOF method, and 0.3 W was produced under the angular velocity of 6.47 rad/s with 0.05 Nm torque. Moreover, from simulation results, while the TSR value was obtained 2.36 in the MRF method, it was obtained 0.48 in the 6-DOF method.

Figure 3 shows the velocity contours in the internal flow field of the original model. It can be concluded that the water velocity has a uniform profile at the inlet, and the flow is distributed evenly. The velocity in the internal flow field is dominated directly by the runner shape. The water flow velocity field reaches the maximum velocities at the turbine outlet.

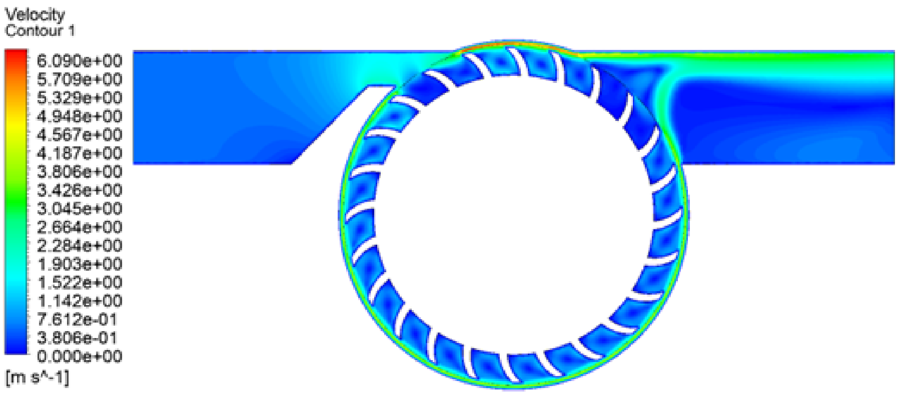

Figure 3 Velocity contour of original model

The maximum velocity of the original model is determined as 6.1 m/s. Figure 4 illustrates the pressure contours in the original model, where the turbine inlet pressure decreases along the runner passage, but the pressure at the turbine outlet is almost uniform.

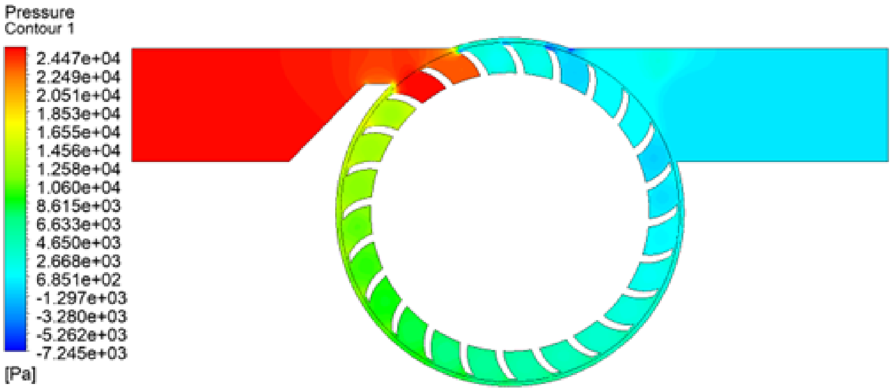

Figure 4 Pressure contour of original turbine model

The fluid pressure passing through the blades decreases rapidly. The maximum pressure of the original model was obtained 25 kPa.

Moreover, the mesh independency test was also executed using the steady-state method. The Richardson extrapolation method was used to obtain a suitable mesh for the assessment process in this study. Analysis results for four different mesh structures, Grid1, Grid2, Grid3, and, Grid4, were given in the table, and these structures consisted of 14175, 57764, 228176, and 904901 elements, respectively. The result of Richardson extrapolation is called Grid Convergence Index (GCI). The GCI refers to the error-index of the tested value from the extrapolated value in the extrapolation process. The number of the mesh was converted to a Normalised Grid Spacing (NGS): 1, 0.5, 0.25, and 0.125, respectively.

Table 2 Mesh independency test result

Grid	Number of elements	Torque (Nm)	NGS	GCI (%)
Grid1	14175	0.032858	1	-
Grid2	57764	0.034157	0.5	1.83%
Grid3	228176	0.03789	0.25	5.06%
Grid4	904901	0.041360	0.125	4.24%
	~	0.04786	0	-

Table 2 shows the mesh independency test result. Torque output value was variable in the mesh independency test, and since time is a one-dimensional unit, an equation for ratio modification was necessary for the Richardson extrapolation method. The acceptance criteria for a good simulation in terms of GCI index is below 2%. It can be concluded that Grid2 and below have the least torque error. Therefore, Grid2 or below is good enough to use for the simulation. As a result of the independency result, this study used 26191 mesh numbers with a skewness of 0.45.

4. CONCLUSION

This study provides a fundamental understanding of the pico water turbine, and this design and analysis methods are used to determine the turbine's performance significantly. The effect of the torque, angular velocity, TSR, and turbine body profile was investigated numerically on a pico hydro turbine in detail. As a result, numerical results show a good agreement with experimental data. When the MRF method was preferred, a deviation of about 2% from the experimental results was observed. In contrast, this rate reached nearly 30% with the 6-DOF method. According to the numerical results obtained, the MRF method for predicting water turbine performance at pico scale is more reasonable than the 6-DOF method. However, it could be taken advantage of calculating transient physics phenomena like torque and angular velocity rather than experimental devices. MRF or 6-DOF method results can be a preliminary reference the turbine model's performance can be captured effectively. Thus these methods shall be the right choice for water turbine simulations. In conclusion, there are three possible ways to optimize water turbine performance, firstly standard turbine inlet height (S_h) should be minimized, or recirculation or vortex should be minimized. With this study, it is aimed to provide a better understanding of hydro turbines in terms of power performance.

NOMENCLATURE

α	Angle of attack [°]
β_1	Blade inlet angle [°]
β_2	Blade outlet angle [°]
δ	Blade curve angle [°]
λ	Turbine discharge angle [°]
w	Angular velocity [rad/s]
6-DOF	Six degrees of freedom [-]
B	Blade's width [mm]
CAD	Computer-aided design [-]
D_i	Inner diameter [mm]
D_o	Outer diameter [mm]
GCI	Grid Convergence Index [-]
L	Blade’s length [mm]
n	Revolution per minute [rpm]
NGS	Normalized Grid Spacing [-]
PHT	Pico hydro turbine [-]
P_{out}	Shaft power [W]
R_B	Blade curve radius [°]
T	Torque [Nm]
TSR	Tip speed ratio [-]
S_h	Turbine inlet height [mm]
V	Runner velocity [m/s]
V_{outlet}	Velocity outlet [m/s]
V_t	Tangential velocity [m/s]
UDF	User-defined function [-]
MRF	Moving reference frame [-]
z	Number of blades [-]

REFERENCES

[1] B. R. Cobb, K. V. Sharp, "Impulse (Turgo and Pelton) turbine performance characteristics and their impact on pico-hydro installations," Renewable Energy, vol. 50, pp. 959–964, 2013.

[2] L. Alveyro, F. Jose, S. Aida, "Performance improvement of a 500-kW Francis turbine based on CFD," vol. 96, 2016.

[3] N. Thakur, A. Biswas, Y. Kumar, M. Basumatary, "CFD analysis of performance improvement of the Savonius water turbine by using an impinging jet duct design," Chinese Journal of Chemical Engineering., vol. 27, no. 4, pp. 794–801, 2019.

[4] S. Chichkhede, V. Verma, V. K. Gaba, S. Bhowmick, "A Simulation Based Study of Flow Velocities across Cross Flow Turbine at Different Nozzle Openings," Procedia Technology., vol. 25, pp. 974–981, 2016.

[5] D. Jiyun, S. Zhicheng, Y. Hongxing, "Performance enhancement of an inline cross-ow hydro turbine for power supply to water leakage monitoring system," Energy Procedia, vol. 145, pp. 363–367, 2018.

[6] S. J. Kim, Y. S. Choi, Y. Cho, J. W. Choi, J. H. Kim, "Effect of blade thickness on the hydraulic performance of a Francis hydro turbine model," Renewable. Energy, vol. 134, pp. 807–817, 2019.

[7] J. F. Wang, J. Piechna, N. Müller, "A novel design of composite water turbine using CFD," Journal of Hydrodynamics, vol. 24, no. 1, pp. 11–16, 2012.

[8] N. Acharya, C. G. Kim, B. Thapa, Y. H. Lee, "Numerical analysis and performance enhancement of a cross-flow hydro turbine," Renewable Energy, vol. 80, pp. 819–826, 2015.

[9] G. Birhanu Oliy, "Design and Computational Fluid Dynamic Simulation Study of High Efficiency Cross Flow Hydro-power Turbine," International Journal of Science, Technology and Society., vol. 5, no. 4, p. 120, 2017.

[10] R. K. Ranjan, N. Alom, J. Singh, B. K. Sarkar, "Performance investigations of cross flow hydro turbine with the variation of blade and nozzle entry arc angle," Energy Conversion and Management vol. 182, no. January, pp. 41–50, 2019.

[11] D. A. Makarim, D. D. D. P. Tjahjana, S. I. Cahyono, S. A. Mazlan, "Performance investigation of the crossflow water turbine by using CFD," AIP Conference Proceedings, vol. 2097, 2019.

[12] A. P. Prakoso, Warjito, A. I. Siswantara, Budiarso, D. Adanta, "Comparison between 6-DOF UDF and moving mesh approaches in CFD methods for predicting cross-flow pico-hydro turbine performance," CFD Letters., vol. 11, no. 6, pp. 86–96, 2019.

[13] X. Wang, X. Luo, B. Zhuang, W. Yu, H. Xu, "6-DOF numerical simulation of the vertical-axis water turbine," ASME-JSME-KSME 2011 Joint Fluids Engineering. Conferenes. AJK 2011, vol. 1, no. PARTS A, B, C, D, pp. 673–678, 2011.