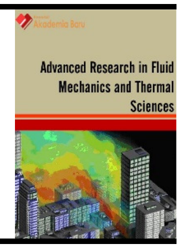
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Performance Comparison of NACA 6509 and 6712 on Pico Hydro Type Cross-Flow Turbine by Numerical Method

Open Access

Dendy Adanta[1,*], Budiarso[1], Warjito[1], Ahmad Indra Siswantara[1], Aji Putro Prakoso[1]

[1] Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, 16424, Depok, Indonesia

ARTICLE INFO

Article history:
Received 28 February 2018
Received in revised form 22 March 2018
Accepted 5 May 2018
Available online 17 May 2018

ABSTRACT

Many Indonesian villages do not have access to electricity due to remote location and low income. Using pico hydro cross-flow turbines can provide them with energy. Cross-flow turbines are economical because their simple shape and construction allow for manufacturing in remote areas. This study examines the use of airfoils for cross-flow turbine blades to determine whether lift force can improve performance. Using numerical methods, this study compares two blade shapes: National Advisory Committee for Aeronautics (NACA) blades 6509 and 6712. The numerical results are compared with previous studies. The boundary conditions are two-dimensional transient domain, model turbulent using SST k-ω and 6-degree of freedom (6-DOF) function. From the results obtained, the maximum mechanical efficiencies of cross-flow turbine in NACA blades 6509 (47.6%) and 6712 (46.9%) are less efficient than standard blades at 77.8%. There are three possible reasons for this: adequate lift force is not produced by airfoil blades because the blade on the impeller resembles a straight shape, rotation and torque decrease in stage 2 as a result of pressure decrease at the bottom of the blade and energy absorption in stage 2 is not optimal due to the internal impeller occurrence of flow recirculation or vortex. There are three possible ways to optimise cross-flow turbine performance: standard blade shape should be used, flow recirculation or vortex should be minimised and the runner should be designed based on the ratio between turbine tangential velocity and water tangential velocity of 1.8 or the ratio of turbine velocity and inlet velocity of 0.53.

Keywords:
Pico hydro, cross-flow turbine, numeric, airfoil, recirculation flow

1. Introduction

Currently, there are 2519 villages in Indonesia that do not have access to electricity due to their remote location [1] and the villager's low income [2]. To overcome the electrical energy crisis in remote areas, pico hydro is considered suitable because it has a higher life cycle cost (LCC) value than wind turbines and solar panels [3]. In addition, pico hydro is suitable for use as an independent power plant for remote areas in Indonesia because it has a potential water energy of 19 GW [4]. Pico hydro is a hydroelectric power plant that generates less than 5 kW [5]. Cross-flow turbines utilise the flow of water directly from rivers and streams and can work on high debit deviation [6,7]. In addition,

* *Corresponding author.*
E-mail address: dendyadanta@ymail.com (Dendy Adanta)

cross-flow turbines are economical because their simple shape and construction allow them to be manufactured in remote areas [6].

There have been many studies conducted to improve cross-flow turbine performance. Mockmore and Merryfield [8] analysed cross-flow turbine performance using non-dimensional analysis. This study achieved the best performance at the specific speed of 14 n_s. Durgin and Fay [9] characterised the flow pattern in the centre of the cross-flow turbine in a comprehensive manner and determined that the turbine contributes 17% to cross-flow performance during stage 2. Using a numerical method, Kaniecki [10] modernised the cross-flow turbine into a reaction turbine by using a draft tube at outflow. Draft tubes in cross-flow turbines can reduce adverse flow, such as backflow and separations, thus increasing efficiency. Andrade *et al.,* [11] analysed the cross-velocity and pressure fields in runners to characterise their performance with different runner speeds. The study showed that recirculation flow in the runner inter-blade passages and shocks of the internal cross-flow cause considerable hydraulic losses, decreasing the efficiency of the turbine significantly. To minimise the hydraulic losses, the blade must be designed properly. Sammartano *et al.,* [12] designed cross-flow turbines to convert water energy more efficiently. Chen and Choi [13] investigated the effect of guide vane angle change on performance of cross-flow turbines. The study showed that guide vane angle considerably influences the internal flow and performance of the cross-flow turbine, and the optimum guide vane angle is 20°. Sammartano *et al.,* [14] developed a suitable equation to determine velocity inlet on cross-flow turbines, making the prediction of performance using computational methods more precise. Sinagra *et al.,* [15] analysed cross-flow turbine performance by varying the discharge of water obtained from rainfall for one year to determine turbine reliability at fluctuating discharge. They found that fluctuation discharge only affects turbine performance in certain months, therefore no discharge regulation system is needed. Sinagra *et al.,* [16] validated a new approximate formula relating main inlet velocity and inlet pressure and obtained the velocity coefficient (C_v) of 1. Sammartano *et al.* [6] analysed the inlet angle (β) with blade position and determined that the best attack angle is 42°.

Previous research has provided many methods to improve cross-flow turbine performance such as using n_s, considering energy conversion in stages 1 and 2, minimising hydraulic loss and adding a guide vane. However, there are still important physical phenomena that have not received enough attention, namely the role of lift force in the process of energy transfer from water to blade cross-flow turbine. Cross-flow turbine performance can be optimised through the design of the blade to convert the water's kinetic energy more efficiently. Because cross-flow turbines are a type of impulse turbine in which water's potential energy is converted into kinetic energy through the nozzle, then some kinetic energy will consequently be absorbed by the blade.

This study examines the contribution of lift force using airfoil National Advisory Committee for Aeronautics (NACA) blades 6509 and 6712 on a cross-flow turbine. This work is a continuation of previous work [17] in which the method used was not verified, causing the results obtained to be inaccurate. In addition, the results of this study can add to the understanding of energy conversion in the blade to assist in designing the optimal cross-flow turbine.

2. Methodology

2.1 Theoretical Background

In designing a cross-flow turbine, a specific speed value should first be determined. Specific speed is the basic selection of the turbine type because it affects which mechanical transmission system will be used. The optimum specific speed for a cross-flow turbine is 14 n_s. However, a cross-flow turbine can also work efficiently in the range of 6–16 n_s [8]. In addition, Mockmore and Merryfield [8] found that cross-flow turbines can work efficiently in a wide range of water discharges. Before determining turbine dimension parameters, the inlet velocity must be known. By analysing head losses and assuming non-zero pressure phenomenon, the following equation can be used to determine the inlet velocity [10]:

$$V = C_V \sqrt{2g\left(H - \frac{\omega^2 (D/2)^2}{2g}\right)} \tag{1}$$

where V is inlet velocity, C_V is velocity coefficient, g is gravity, H is head, D is diameter, and ω is rotational speed. C_V is considered to be 1 if the turbine works at ideal condition (steady-state, radial symmetry, etc.). However, Sammartano *et al.* [6] found the value of C_V to be about 0.95. The next step is to determine the angle of attack (α). The angle of attack that results in optimum power is 22° [18]. Water tangential velocity is a function of attack angle and C_V, therefore:

$$V_T = V \cdot cos(\alpha) \tag{2}$$

where V_T is tangential velocity and α is angle of attack. In addition, maximum efficiency can be attained if the relation between angle of attack and blade outer angle satisfies Eq. 3 [8]:

$$2\, tan(\alpha) = tan(\beta_1) \tag{3}$$

where β_1 is angle of turbine blades. The outer diameter of the turbine can be determined using:

$$U = \omega \cdot D/2 \tag{4}$$

where U is turbine velocity. The maximum efficiency can be attained when relation between turbine tangential velocity and water tangential velocity adheres to Eq. 5 and if the ratio between inner diameter and outer diameter is 0.65 [12].

$$V_T/U = 1.8 \tag{5}$$

Mockmore and Merryfield [8], Aziz and Desai [18] and Sammartano *et al.,* [12] agree that the optimum inner blade angle for a cross-flow turbine is 90°. Used a geometrical approach, Mockmore and Merryfield [8] found blade curve details using:

$$R_B = \frac{R^2 - r^2}{2R \cdot cos(\beta_1)} \tag{6}$$

where R_B is curve of the blade, R is outer radius, r is inner radius and equals $0.65R$, and $\beta_1 = 39^o$.

$$R_B = 0.3712R \tag{7}$$

Then, blade curve angle can be calculated using [8]

$$tan\left(\frac{\delta}{2}\right) = \frac{cos(\beta_1)}{sin(\beta_1) + D/d} \tag{8}$$

where δ is angle of blade curvature and *d* is inner diameter. The nozzle dimension of a cross-flow turbine can be found using [12]

$$q = V\, sin(\alpha) \cdot \lambda \frac{D}{2} \tag{9}$$

where q is nozzle dimension and λ is wide angle of inlet. From mass conservation law, nozzle height and width can be calculated using Eqs. 10 and 11 [12]:

$$S_0 = q/V \tag{10}$$

$$B = Q/q \tag{11}$$

where S_0 is nozzle height, B is nozzle width and Q is discharge. For more details, Figure 1 describes the steps to design a cross-flow turbine.

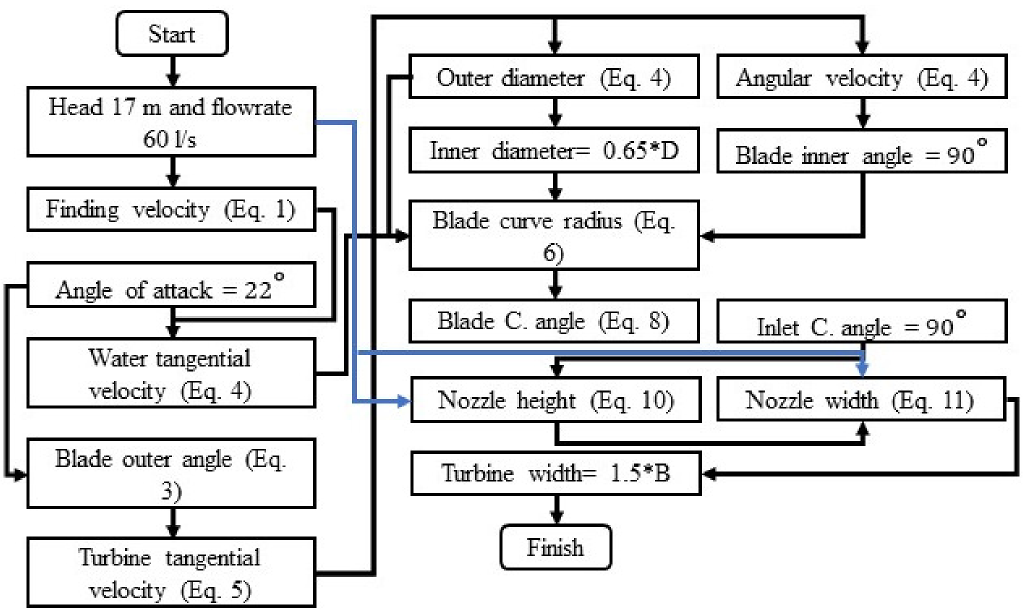


Fig. 1. Flow chart of cross-flow turbine design

2.2 Design Procedure of Airfoil Blade Cross-Flow Turbine

The difference between runner curvature with runner airfoil is only in blade calculation. However, there are some parameters that replace blade parameters without airfoil concept, including relative water inlet angle (β), blade angle of attack (α_b), blade chord angle (β_c) and blade chord length (L_c). Figure 2 describes the design of β, α_b, β_c and L_c.

As discussed, the best blade outlet angle is 90°. The airfoil which would be adapted to cross-flow turbines should be placed in such a way as to make the trailing edge face 90° towards the turbine inner tangential velocity. To make this happen, the airfoil trailing edge angle should be known beforehand so that the airfoil chord can be placed correctly in the turbine. Eqs. 12 and 13 can be used to determine blade chord angle [19]:

$$\frac{dy}{dx} = \frac{2M}{(1-P)^2}(P-1) \tag{12}$$

$$\beta_C = 90^0 - \arctan(\frac{dy}{dx}) \tag{13}$$

where, dy/dx is airfoil chamber gradient, M is airfoil maximum camber in percent of chord and P is position of maximum camber at 10% of chord. After β_c has been determined, the airfoil chord length (β_{CX}) is calculated using Eqs. 14 and 15:

$$\beta_{CX} = 90^0 - \arcsin(0.65 * \sin(90^o - \beta_C)) \tag{14}$$

$$L_c = \frac{\sqrt{1.4225 - 1.3 * cos(\beta_{CX} - \beta_C)}}{2} D \tag{15}$$

The angle of relative velocity is determined using Eq. 16:

$$\tan\beta = 1.8 \tan\alpha \tag{16}$$

Furthermore, the blade angle of attack is the deviation between blade chord angle and water inlet angle, or $\alpha_b = \beta_C - \beta$. The blade angle of attack is also used to find lift and drag coefficients in the airfoil polar diagram. Lift and drag coefficients are used to calculate the power output by the analytical calculation.

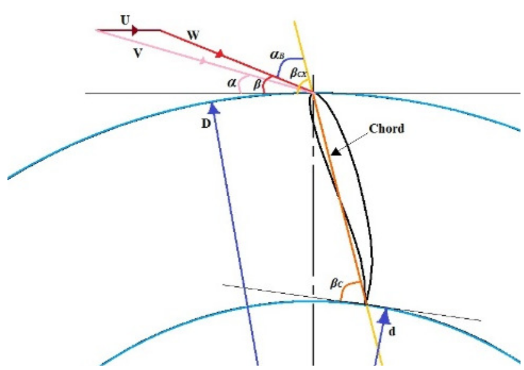


Fig. 2. Schematic of cross-flow turbine blade using airfoil

2.3 Simulation Procedure

The study by Sammartano *et al.,* [14] was used as a verifier in this study because they presented the results more comprehensively than others. In addition, Sammartano *et al.,* [14] compared numerical and experimental studies so that the studies performed can be said to be valid. Figures 3 and 4 show the comparison between this study and the study by Sammartano *et al.,* [14].

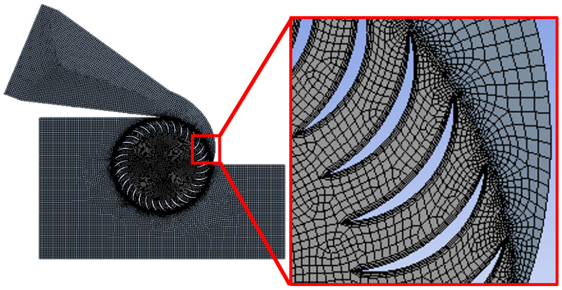


Fig. 3. Meshing of this study

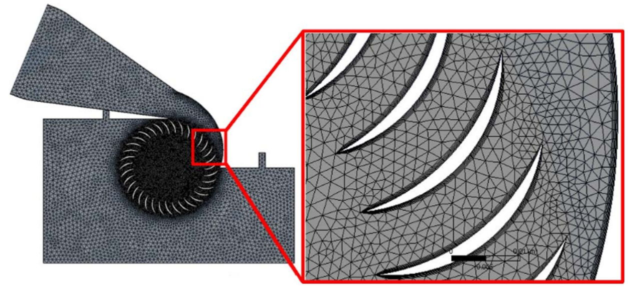


Fig. 4. Meshing of study by Sammartano *et al.* [14]

This study uses a two-dimensional transient domain to save computing power. In addition, the Volume of Fluid (VoF) model was used because the fluid phase is two phases with a surface tension of 0.0728 N/m. The turbulent flow is predicted using Shear Stress transport (SST) k-ω because the numerical results are closer to the actual conditions. This study uses the new six-Degree of Freedom (6-DOF) feature in ANSYS® FLUENT 18.2™. This feature makes it possible to investigate the fluid dynamics phenomena of the runner movement. There are several steps performed in the numerical process. Figure 5 describes in detail the process of numerical methods carried out in this study.

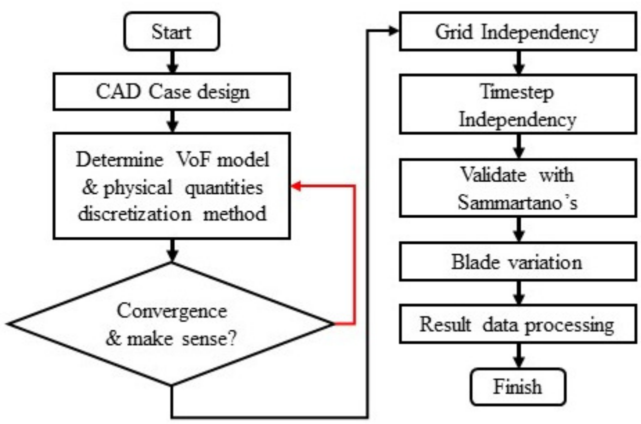


Fig. 5. Flow chart of numerical procedure

The numerical simulation has been done with several grid and timestep independency tests. In numerical calculation, the smaller the grid size, the more precise the results of the calculation obtained, but calculation runs slower. This is also the case for the size of timestep: the smaller timestep size in the calculation makes the condition change between timestep become smaller, helping the solver to converge easier. However, it also makes the calculation load heavier. The goals of grid and timestep independency were to determine the conditions which have precise enough results with the lightest calculation load possible.

3. Results and Discussions

3.1 Analytical Results

The turbine design calculation results are reported in Table 1, which is a summary of the analytic result using Eqs. 1–11.

Table 1

Cross-flow turbine design geometric parameters

Parameter	Value	Parameter	Value
D	161 mm	d	104 mm
N_B	35	λ	90^0
α	22^0	W	139 mm
β_1	39^0	β_2	90^0
R_b	29.8 mm	δ	62.6^0
S_0	47 mm	B	93 mm

Table 2 shows the power output of cross-flow turbine stage 1 with airfoil concept using Eqs. 12–16.

Table 2
Analytical Calculation Results

Blade Type	α	β_1 or β	β_2 or β_c	α_b	C_L	C_D	P	η
NACA-6509 Airfoil	22^0	36^0	76.5^0	45^0	1.23	0.32	3961.3	~75%
NACA-6712 Airfoil	22^0	36^0	68.2^0	40^0	1.5	0.6	5666.7	>85%

Representations of Tables 1 and 2 can be seen in Figures 7 and 8, which are compared with a standard impeller in Figure 6:

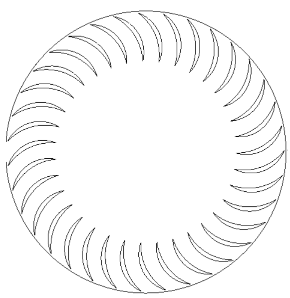


Fig. 6. Standard impeller for simulation validation

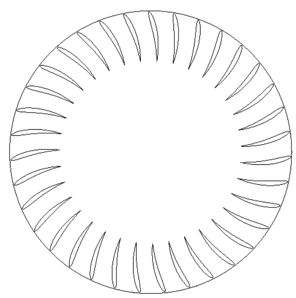


Fig. 7. NACA Impeller 6509

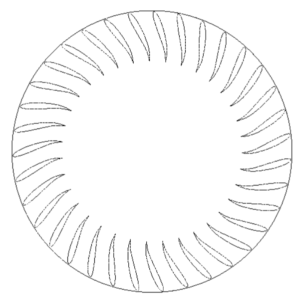


Fig. 8. NACA Impeller 6712

A series of numerical simulations were run to get more comprehensive results by using ANSYS® FUENT 18.2™ with 6-DOF function. The results of the simulations were then compared to the analytical results.

3.2 Numerical Verification and Validation

The coarsest grid size investigated in this study was 0.002 m, which generated 4438 elements, increased to 8645 elements, 22,272 elements, 26,172 elements, 33,216 elements and 63,267 elements. Furthermore, the largest timestep size investigated in this study was 0.002 second, or 500 Hz of frequency, increased to 1000 Hz, 1250 Hz, 2000 Hz and 2500 Hz. Figures 9 and 10 show the results of the grid independency and the timestep independency processes. From the results, the data gathering step in this study was run in the minimum grid size of 0.0004 m with three inflation layers, which generated about 33,000 elements, and the timestep frequency used in this study was 2000 Hz or 0.0005 s length per timestep. The simulation was run for about 650 timesteps.

Validation of the numerical results was compared between impeller standard shape and the study by Sammartano *et al.* [14]. From the results, this study showed similarities with the study by

Sammartano *et al.* [14] (see Figure 11). First, all of the results had the maximum efficiency of cross-flow turbine at 1.8 V_T/U condition. Furthermore, the maximum efficiencies in this study and that of Sammartano *et al.* [14] had similar results, about 81%. Finally, all of the results had a high slope between V_T/U of 1.2 and 1.4. From these similarities, this study was considered to have good validation. Figure 12 shows the comparison between blades NACA 6509 and NACA 6712.

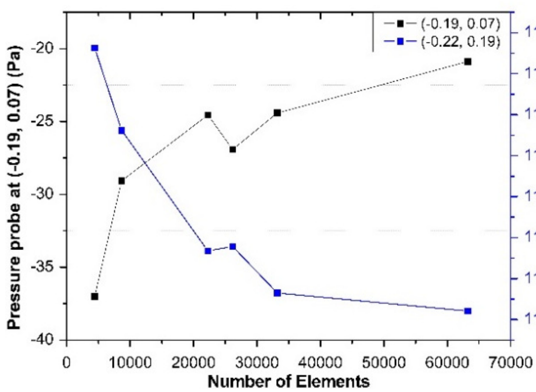


Fig. 9. Grid independency

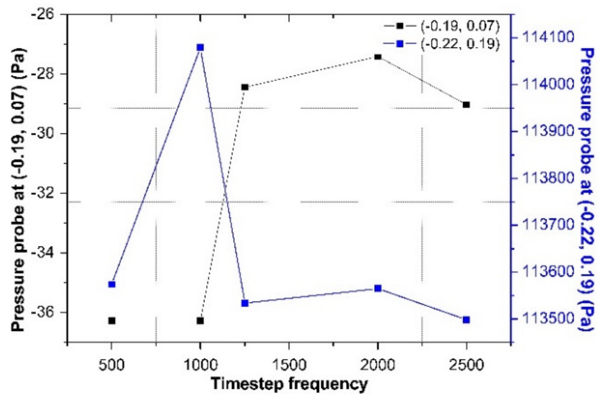


Fig. 10. Timestep independency

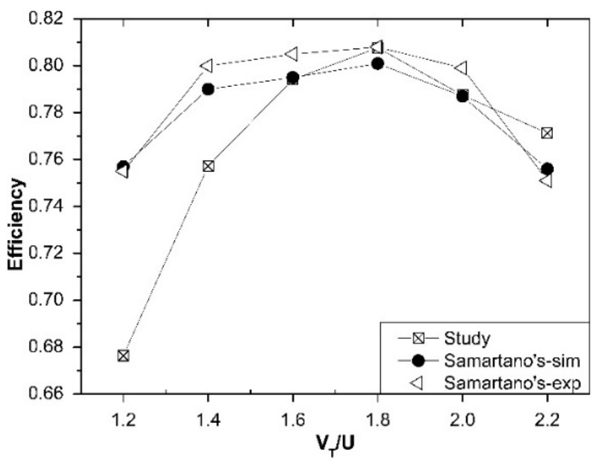


Fig. 11. Validation simulation of study by Sammartano *et al,.* [14]

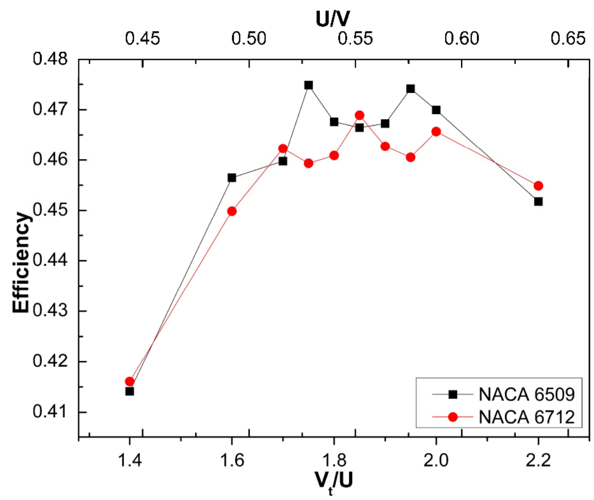


Fig. 12. Efficiency of NACA6509 and 6712

3.2 Numerical Results

The simulation was completed with ten variations of V_t/U. From the results obtained, NACA blade 6509 has a mechanical efficiency of 47.5%, which is higher than NACA blade 6712 at 46.9% (Figure 12). Theoretically, the maximum efficiency of the impulse turbine is obtained when $U/V = 0.5$, but the actual maximum efficiency is achieved if $U/V = 0.42$ to 0.47 [20]. However, from the study by Sammartano *et al.,* [14], the maximum mechanical efficiency of the cross-flow turbine is in the range of $U/V = 0.5$ to 0.53. In this study, the maximum efficiency obtained was $U/V = 0.53$, similar to the study by Sinagra *et al.,* [16]. There are two possible ways in which water energy can be absorbed when the U/V ratio of turbine performance is reduced: occurrence of recirculation flow or vortex and the water energy degradation or dissipation in the internal impeller.

As seen in the comparison graphs in Figures 11 and 12, the maximum mechanical efficiency of cross-flow turbine in NACA blades 6509 and 6712 was less efficient than a standard blade at 77.8%. This indicates that the reaction turbine concept cannot be used in a cross-flow turbine.

3.3 Discussions

When viewed from the pressure contours (see Figures 13 and 14), the NACA blades 6509 and 6712 did not produce lift force in stage 1, which was caused by the difference of water velocity at the top was not dominant with at the bottom because the blade resembles a straight shape. In addition, in stage 2, the more dominant downward pressure occurred at the bottom of the blade. Indirectly, this can cause rotation and torque to decrease, having a negative impact on the power being generated.

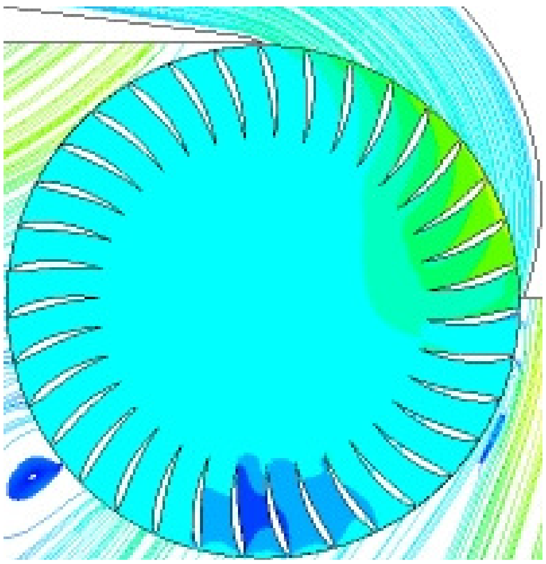


Fig. 13. Contour pressure on NACA blade 6509

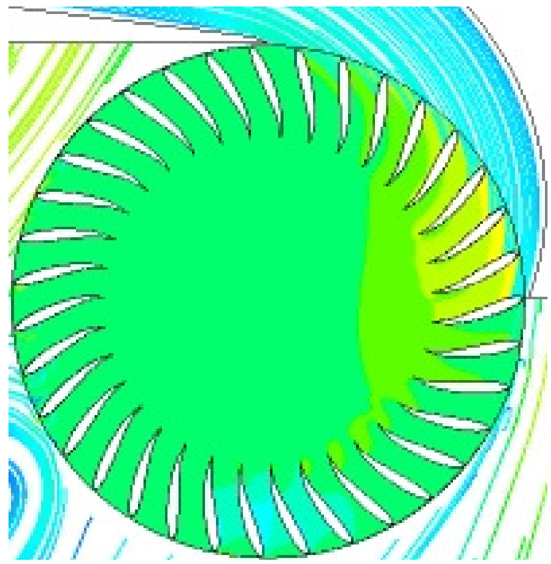


Fig. 14. Contour pressure on NACA blade 6712

Investigation of the internal flow of NACA blades 6509 and 6712 showed recirculation flow (see Figures 15 and 16). Recirculation flow that occurs in internal impeller cross-flow was also shown by Gebrehiwot *et al.,* [21], where the recirculation flow happened clockwise. Recirculation flow is caused by force imbalance in the formation of rotation vortex or force vortex [22]. When the velocity stream is greater that the centre point of the vortex, it is categorised as rotational vortex [23]. Recirculation flow or vortex on internal impeller cross-flow turbines should be avoided because they can reduce energy absorption in the second stage (impeller cross-flow turbine is effective if the water energy absorption in the second stage is 25–35% of the total power generated) [11,24]. Visualisation of recirculation flow or vortex is usually done by turbulent kinetic energy [25–28]. Theoretically and actually, the higher the turbulent kinetic energy, the decrease in power engine performance [20]. Therefore, recirculation flow or vortex on the internal impeller cross-flow turbine is also suspected to be one cause of decreased performance. Furthermore, the study by Sun *et al.,* [30] explained that any recirculation flow or vortex that occurs on the internal impeller cross-flow (turbine or fan) reduces the impeller's performance because it is working abnormally. Figures 15, 16 and 17 validate this conclusion by Sun *et al.,* [30]. For cross-flow impellers, the recirculation flow or vortex is one of the causes of reduced performance.

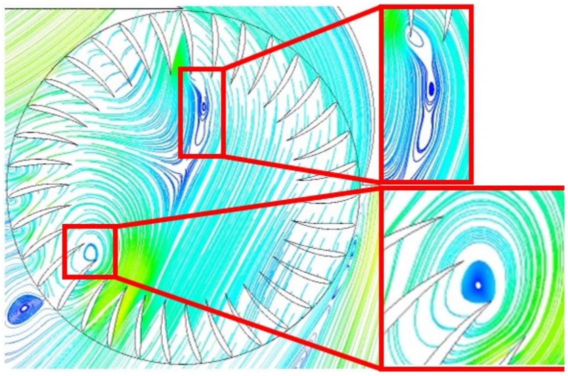


Fig. 15. Recirculation flow on internal impeller NACA blade 6509

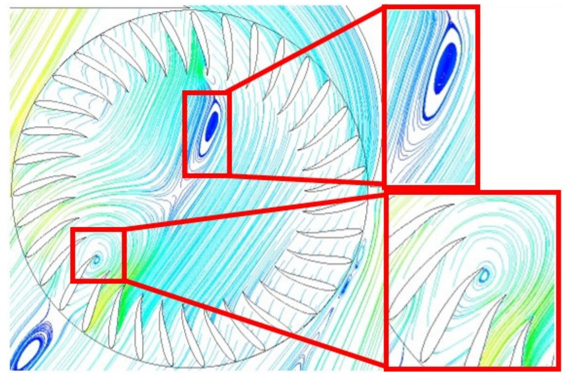


Fig. 16. Recirculation flow on internal impeller NACA blade 6712

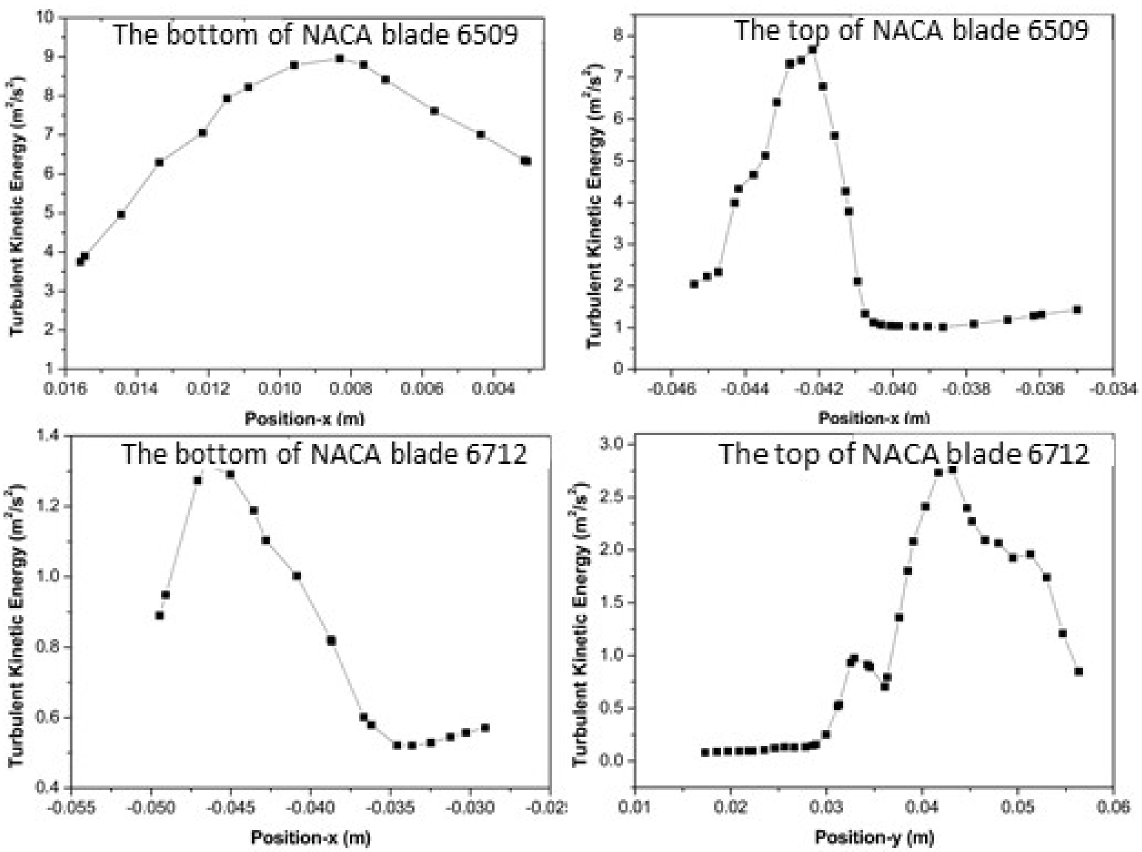


Fig. 17. Turbulent kinetic energy distribution on recirculation flow

4. Conclusions

From the results of the study, a cross-flow turbine using airfoil NACA blades 6509 or 6712 produces lower power than the standard blades, with a maximum efficiency of 47.5% for NACA 6509, 46.9% for NACA 6712 and 77.8% for a standard blade. There are three reasons for this poor performance. First, adequate lift force is not being produced by the airfoil blade because the blade is straight. Second, in stage 2 pressure decreases occur at the bottom of the blade so that rotation and torque decrease. Third, energy absorption in stage 2 is not optimal and functions abnormally due to internal impeller occurrence of recirculation flow or vortex. From this study, the reaction turbine concept (used in airfoil NACA 6509 and 6712 blades) cannot be used in cross-flow turbines.

Thus there are three considerations for optimising cross-flow turbine performance: the blade should have a curved shape, recirculation flow or vortex should be minimised and the runner should be designed based on a ratio between turbine tangential velocity and water tangential velocity of 1.8 ($V_T/U = 1.8$ or $U/V = 0.53$) because a smaller U/V ratio leads to water energy degradation or dissipation on the internal impeller, and a higher ratio causes the water energy to not fully convert.

Acknowledgement
The authors would like to express they're thanks to KEMENRISTEK DIKTI, which has funded this research with grant No. SP. DIPA-041.06.1.401516/2016.

References

[1] Budiarso, Warjito, Dendy Adanta, N.S. Putra, H. Vohra, "Type Of Cut-Out Bucket Selection For Pico Hydro Pelton Turbine," in *The 2nd International Conference on Engineering and Technology for Sustainable Development (ICET4SD)*, 2017.
[2] Maritje Hutapea, "Solusi Listrik Off-grid Berbasis Energi terbarukan di Indonesia: Kerangka dan Regulasi," Jakarta, 2016.
[3] Dendy Adanta, "Aplikasi Model Turbulen pada Turbin Piko Hidro," Universitas Indonesia, 2017.
[4] KESDM, "Rencana Umum Energi Nasional," Jakarta, 2017.
[5] Ridzuan, Mohd Jamir Mohd, S. M. Hafis, K. Azduwin, K. M. Firdaus, and Zawawi Zarina. "Development of pico-hydro turbine for domestic use." *Appl Mech Mater* 695 (2014): 408-12.
[6] Sammartano, Vincenzo, Gabriele Morreale, Marco Sinagra, and Tullio Tucciarelli. "Numerical and experimental investigation of a cross-flow water turbine." *Journal of Hydraulic Research* 54, no. 3 (2016): 321-331.
[7] Montanari, R. "Criteria for the economic planning of a low power hydroelectric plant." *Renewable energy* 28, no. 13 (2003): 2129-2145.
[8] Mockmore, Charles Arthur, and Fred Merryfield. "The Banki water turbine." (1949).
[9] Durgin, W. W., and W. K. Fay. "Some fluid flow characteristics of a cross-flow type hydraulic turbine." *Small Hydro Power Fluid Machinery* (1984): p77-83.
[10] Kaniecki, Maciej. "Modernization of the outflow system of cross-flow turbines." *Task Quarterly* 6, no. 4 (2002): 601-608.
[11] De Andrade, Jesús, Christian Curiel, Frank Kenyery, Orlando Aguillón, Auristela Vásquez, and Miguel Asuaje. "Numerical investigation of the internal flow in a Banki turbine." *International Journal of Rotating Machinery* 2011 (2011).
[12] Sammartano, Vincenzo, Costanza Aricò, Armando Carravetta, Oreste Fecarotta, and Tullio Tucciarelli. "Banki-Michell optimal design by computational fluid dynamics testing and hydrodynamic analysis." *Energies* 6, no. 5 (2013): 2362-2385.
[13] Chen, Z. M., and Young-Do Choi. "Performance and internal flow characteristics of a cross-flow turbine by guide vane angle." In *IOP Conference Series: Materials Science and Engineering*, vol. 52, no. 5, p. 052031. IOP Publishing, 2013.
[14] Sammartano, V., G. Morreale, M. Sinagra, A. Collura, and T. Tucciarelli. "Experimental study of cross-flow micro-turbines for aqueduct energy recovery." *Procedia Engineering* 89 (2014): 540-547.
[15] Sinagra, Marco, Vincenzo Sammartano, Costanza Aricò, Alfonso Collura, and Tullio Tucciarelli. "Cross-flow turbine design for variable operating conditions." *Procedia Engineering* 70 (2014): 1539-1548.
[16] Sinagra, M., V. Sammartano, C. Aricò, and A. Collura. "Experimental and numerical analysis of a cross-flow turbine." *Journal of Hydraulic Engineering* 142, no. 1 (2015): 04015040
[17] Warjito, Ahmad Indra Siswantara, Dendy Adanta, Aji Putro Prakoso, Reza Dianofitra, "Comparison Between Airfoil Profiled Blade and Ordinary Blade in Cross-Flow Turbine Using Numerical Simulation," in *15th International Conference on Quality in Research*, 2017.
[18] Aziz, Nadim M., and Venkappayya R. Desai. "A laboratory study to improve the efficiency of cross-flow turbines." *Engineering Report* (1993).
[19] Mason, W. H. "Appendix A: Geometry for Aerodynamicist." *Applied Computational Aerodynamics Text/Notes, http://www. aoe. vt. edu/~ mason/Mason_f/CAtxtTop. html*.
[20] Harinaldi, Budiarso, *Sistem Fluida (Prinsip Dasar dan Penerapan Mesin Fluida, Sistem hidrolik dan Sistem*

Pnuematik). Jakarta: Erlangga, 2015.

[21] Gebrehiwot, Mekonnen Gebreslasie, Josse De Baerdemaeker, and Martine Baelmans. "Numerical analysis of a cross-flow fan with two outlets." *HEFAT 2007* (2007).

[22] S. Darmawan, "Pengembangan model turbulen RNG k- ε untuk aplikasi CFD pada runner cross-flow dalam komponen turbin gas mikro bioenergy proto x-2a," Universitas Indonesia, Depok, 2015.

[23] P. G. Saffman, *Vortex dynamics*. Cambridge university press, 1992.

[24] Khosrowpanah, S., M. L. Albertson, and A. A. Fiuzat. "Historical overview of cross-flow turbine." *International Water Power and Dam Construction* 36, no. 10 (1984).

[25] Siswantara, Ahmad Indra, and Steven Darmawan. "Investigation of Inverse-Turbulent-Prandtl Number with Four RNG ke Turbulence Models on Compressor Discharge Pipe of Bioenergy Micro Gas Turbine." *Applied Mechanics & Materials*819 (2016).

[26] Steven Darmawan, Ahmad Indra Siswantara, Budiarso, Asyari Daryus, Agus Tri Gunawan, Achmad Bayu Wijayanto, Harto Tanujaya, "TURBULENT FLOW ANALYSIS IN AUXILIARY CROSS-FLOW RUNNER OF A PROTO X-3 BIOENERGY MICRO GAS TURBINE USING RNG k-ε TURBULENCE MODEL," *ARPN Journal of Engineering and Applied Sciences*, vol. 10, pp. 7086–7091, 2015.

[27] M. Lesieur, *Turbulence in fluids*, vol. 40. Springer Science & Business Media, 2012.

[28] Tennekes, Hendrik, and John Leask Lumley. *A first course in turbulence*. MIT press, 1972.

[29] Sun, Kangjie, Hua Ouyang, Jie Tian, Yadong Wu, and Zhaohui Du. "Experimental and numerical investigations on the eccentric vortex of the cross flow fan." *International Journal of Refrigeration* 50 (2015): 146-155.