CFD Letters

Journal homepage: www.akademiabaru.com/cfdl.html
ISSN: 2180-1363

Assessment of Turbulence Model for Cross-Flow Pico Hydro Turbine Numerical Simulation

Open Access

Ahmad Indra Siswantara[1], Budiarso[1], Aji Putro Prakoso[1,3,*], Gun Gun R Gunadi[1,2], Warjito[1], Dendy Adanta[1]

[1] Department of Mechanical Engineering, Faculty of Engineering, Universitas Indonesia, Depok 16424, Indonesia
[2] Department of Mechanical Engineering, Politeknik Negeri Jakarta, Depok 16424, Indonesia
[3] Department of Mechanical Engineering, Sekolah Tinggi Teknologi Texmaco, Subang 41262, West Java, Indonesia

ARTICLE INFO

Article history:
Received 29 September 2018
Received in revised form 16 December 2018
Accepted 24 December 2018
Available online 28 December 2018

Keywords:
cross-flow turbine, pico hydro, CFD, turbulence model, 6-DoF

ABSTRACT

This study will be assessing some turbulence model for numerical simulation in Pico hydro cross-flow turbine with three variables like errors, time per iteration and average iterations to converged. The computational fluid dynamics (CFD) methods in this study using ANSYS™ FLUENT® 18.2 academic licensed with two-dimensional (2D) and feature six-degrees of freedom (6-DoF). Then, validation and verification were done by comparing with previous study. There are six turbulence model that compared: standard wall function k-ε, scalable wall function k-ε, standard wall function RNG, scalable wall function RNG, standard transitional SST, and transitional SST with curvature corrections. From the results, all the turbulent models give the almost similar results. However, if by error assessment, the RNG turbulence model with scalable wall function was the most accurate than others. By number of timestep iterations assessment, the transitional SST with curvature correction resulting more quickly to convergence than others. By timestep calculations times, the standard transitional SST has relative average calculation timestep smaller than others. Thus, based on assessment the turbulence model suitable for numerical simulation in pico hydro cross-flow turbine is standard transitional SST.

1. Introduction

Indonesia recently has 95% electrification ratio than targeted to be 100% in 2025 [1]. To achieve the target, Indonesia must electrify more than 2500 of their rural areas [2]. However, development of on-grid electricity to the rural area is very expensive [3]. Hydropower, especially pico-hydro which scaled below than 5 kW is suitable for the off-grid electricity in rural area in Indonesia who has great hydropower potential [4]. Pico hydro is hydroelectric power generation that produces power less than 5 kW [5]. This is reason pico-hydro power plant is suitable to overcome of electricity crisis in rural area[6].

[*] *Corresponding author.*
E-mail address: ajippp13@gmail.com (Aji Putro Prakoso)

Cross-flow turbine is well known to be used in pico-hydro power plants[7]. This type of turbine is classified as an impulse turbine with low head and medium water discharge[8][9][10]. Cross-flow turbine was introduced by Donat Banki (Austrian), Anthony Michell (Australian), and Fritz Ossberger (German). From previous study, cross-flow turbine is concluded as the most suitable turbine for pico-hydro application in rural area because of its simple shape, portable and performance stability [3][6][7][9][11][12].

Previous studies have been carried out to improve the performance of cross-flow turbines such as design using n_s[13], energy conversion considerations on stage 1 and 2 [14], minimize hydraulic losses [15], inlet angle (β) optimum[16], and blade shape [6]. Currently, computational fluid dynamics (CFD) method considered so important to predict the performance of designed turbine before applied in real condition and more accurate than analytical method [7][17]. In the other hand, CFD method analysis results could be the initial reference before doing the experiment [7]. From previous studies, in studying the flow physics phenomenon that occurs in the internal impeller turbine cross-flow can be done using CFD method [6]. Moreover, it makes the engineer easier to design and apply cross-flow turbine in any place such as rural area. However, the quality of the results conducted by CFD method were influenced by some causes such as: existence and quality of independency test [18], definition of the boundary [19], turbulence model choosing [16], and discretisation method choosing [19]. This study will be focus on turbulence model choosing.

The comparison of some turbulence model has conducted by Sammartano *et al.* in 2016 [16]. That study was comparing k-ε, RNG k-ε, and transitional SST turbulence model. However, the study only compares the error of these turbulence model without enough assessment method. The assessment of turbulence model for cross-flow turbine is become important in this era since the increased usage of the numerical methods to build a hydropower plant. The assessment should not only compare the error of the calculation results from some turbulence models, but also compare the other variables. This study was assessing some turbulence model from some variables like errors, time per iteration and average iterations to converged. Furthermore, the results of this study can add to understanding of CFD method especially turbulence model is suitable on cross-flow turbine.

2. Methodology

2.1 Theory of Cross-flow Turbine

Sammartano *et al.* [9] introduced the more accurate equation to find the inlet velocity of water in this turbine. The equation is expressed in Eq. 1. Then, it was validated in their next study in 2016 [16]. After inlet velocity was found, the turbine construction main parameter could be found based on Table 1.

$$V = C_V\sqrt{2gH - \omega^2 R^2} \quad (1)$$

where V is inlet velocity, C_V is coefficient of velocity, H is head, ω is turbine rotating speed, and R is the turbine outer radius.

The cross-flow turbine's runner outer diameter and the rotation speed could be defined with classic Eq.

$$U = \omega \cdot D/2 \quad (2)$$

Then, the runner inner diameter could be determined. The runner's blade curve radius and angle could be determined in Eqs. 3 and 4 [8].

$$R_B = \frac{R^2 - r^2}{2R \cdot cos(\beta_1)} \tag{3}$$

$$\tan\left(\frac{\delta}{2}\right) = \frac{\cos(\beta_1)}{\sin(\beta_1) + D/d} \tag{4}$$

while d/D and r/R is 0.75, and β_1 is 39°, Eq. 3 and 4 was simplified in Eq. 5 and 6.

$$R_B = 0.14\,D \tag{5}$$

$$\delta = 59^{\circ} \tag{6}$$

Table 1
Main design parameters

Parameter	Value
Angle of Attack (α)	22° [20] [21]
Diameter Ratio (d/D)	0.75 [16]
Blade's inlet angle (β_1)	39° [9]
Blade's outlet angle (β_2)	90° [8]
Optimum speed ratio (V_T/U)	1.8 [9][16]
Nozzle discharge angle (λ)	90° [9]
Runner-Nozzle Width Ratio (W/B)	1.5 [9]

Nozzle design parameter could be defined by Eq. 7 and 8 [9]:

$$S_0 = 0.29\,D \tag{7}$$

$$B = Q/(S_0 V) \tag{8}$$

where S_0 is nozzle initial height, B is nozzle width, and Q is expected water discharge.

2.2 CFD Procedure, Case and Models

The CFD methods in this study was run on academic licensed ANSYS™ FLUENT® 18.2. Two-dimensional (2D) domain was chosen because previous studies reported results representing three-dimensional (3D) conditions [6]. The simulations were run using standard Volume of Fluid (VoF) multiphases modelling with constant interfacial surface tension. First order UPWIND discretization scheme also used in this study. This method provides stable iterating calculation results, but, it needs a fine enough mesh to be more accurate.

The case in this study uses the similar design of cross-flow turbine inside study by Sammartano *et al.* [16]. The specific design was summarized in Table 2. Then, as validation and verification of this results of this study could be compared with Sammartano's [16] results. The simulations case was shown in Figure 1 and 2.

Table 2
Specific design parameters

Design Parameter	Value	Design Parameter	Value
Outer diameter	161 mm	Inner diameter	121 mm
Number of blades	35	Angle of attack	22^o
Blade's inlet angle	39^o	Blade's outlet angle	90^o
Blade's curve radius	22.5 mm	Blade's curve angle	59^o
Nozzle discharge angle	90^o	Nozzle initial height	47 mm

To ensure the quality of the simulations, mesh and timestep independency test was conducted before data gathering using Richardson Interpolation method [18]. This method could determine lowest number of mesh number or timestep frequency which suitable with desirable calculation quality. The first step of the extrapolation is to determine the convergence coefficient which is notated as p, that shown in Eq. 9.

$$p = \ln\left(\frac{f_3 - f_2}{f_2 - f_1}\right)/\ln(r) \quad (9)$$

The extrapolation of the exact value by Richardson extrapolation could be determined by Eq. 10.

$$f_{h=0} = f_1 + \frac{(f_1 - f_2)}{r^p - 1} \quad (10)$$

which r is the ratio of refinement. Then the error calculated in Richardson extrapolation is called as Grid Convergence Index (GCI). The GCI for the refinement process is calculated based on Eq. 11.

$$GCI_{ab} = \frac{(f_a - f_b)/f_a}{r^p - 1} x100\% \quad (11)$$

which is the acceptable GCI should less than 1%. After independency test conducted, this study was use 39,928 elements of mesh and 2,500 Hz of timestep frequency. The meshing results was shown in Figure 2.

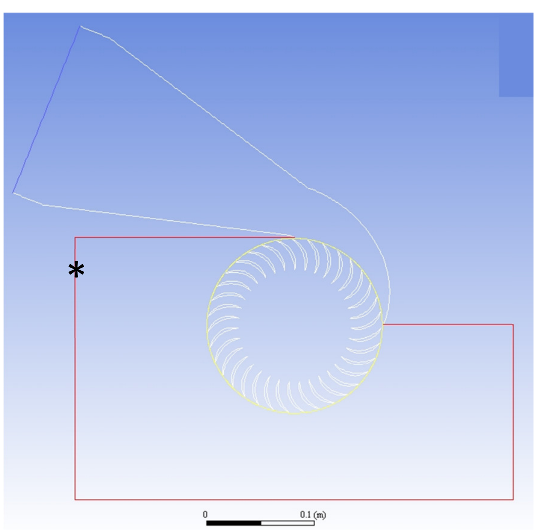


Fig. 1. Boundary conditions of simulations case

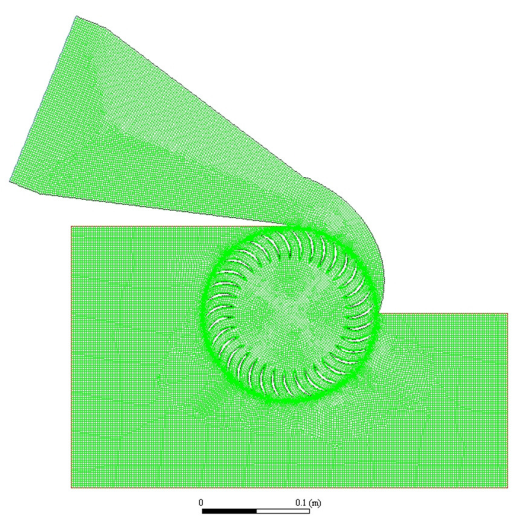


Fig. 2. Mesh density

This study varies some type of turbulence models with ANSYS™ native constants value. Tested turbulence models were standard wall function *k*-ε model, scalable wall function *k*-ε model, standard wall function RNG model, scalable wall function RNG model, standard transitional SST model, and curvature correlated transitional SST model. All the turbulence models were compared with Sammartano's Experimental results to see the error. Time of calculation and total number of iterations was also recorded for consideration. Furthermore, this study uses feature six-degrees of freedom (6-DoF). The 6-DoF was chosen can represent phenomenon physic more precisely because the rotation of the wheel is the results of a simulation not a boundary condition . However, this feature requires higher computing power than moving mesh because post processing data can be taken if steady conditions have been obtained.

There are several pre and post processing simulation processes such as: first the imported geometry is named as inlet, outlet, interior interface, impeller and wall. Then, the interface of impeller and interior interface are joined to avoid error calculations because if this is not defined the software will assume the interface as a wall. Note that simulation failures occur very often because they fail or do not define the interface. Next step is meshing process. Furthermore, giving the boundary conditions used such as: velocity or pressure inlet with solver types is pressure based because the fluid used is incompressible flow; gravity of 9.81 m/s^2; activating the turbulence model to capture vortex that occur become precision; the interface that has been joined is set up using rotational and re-meshing because when the impeller rotating, the mesh that changes in size readjusts to the initial conditions so that errors due to calculation or rounding do not increase; smoothing is selected to repair-improve size of mesh; and then, 6-DoF feature activation. Feature 6-DoF requires using preload and moment of inertia. The preload value can be knowing from the experimental test, but in this study the preload used is 1 N·m. The moment of inertia can be known from computer-aided design (CAD) software or from results of calculations using the Eq. of moment inertia this circular ring.

2.3 Turbulence Models Assessment Procedure

This study compared three type of turbulence models, which are k-ε, RNG, and Transitional SST. Each turbulence models have divided into standard and advance featured models. So that this study was comparing six different case, which are Standard k-ε model, k-ε model with Scalable Wall Function, Standard RNG model, RNG model with Scalable Wall Function, Standard Transitional SST model, and transitional SST with Curvature Correction model. The k-ε turbulence model is model is using two equations which are predicting the kinetic energy and the dissipation of the turbulence flow[22]. The first equation is predicting the kinetic energy of turbulence flow, and the second one is predicting the dissipation of turbulence flow. The RNG turbulence model is the improvement of the k-ε turbulence model which has developed by Yakhot *et.al* [23]. This turbulence model use renormalization group theory in statistical technique to derive the standard k-ε model. The transitional SST turbulence model is the modification of SST k-ω turbulence model by adding two more equations, which are called gamma and Retha equation[24]. The SST k-ω itself is the modification of k-ω model which use two equations like k-ε, and RNG k-ε model.

All mentioned turbulence model (k-ε, k-ε plus scalable W/F, RNG, RNG plus scalable W/F, Transitional SST, and Tr.-SST with curvature correction) was compared in three variables, which are simulation results error from Sammartano, *et al.* [16] results, average timestep iteration, and average timestep calculation time. All these three variables are representing the quality of the turbulence models. The simulation error was the root mean square of the deviation between simulation results and Sammartano, *et al.* [16] results. The average timestep iteration was the ratio between total number of iterations in the simulation and the number of timesteps, which is 2000. The average timestep calculation time is the dividing of total calculation time with 2000, then compared each other.

To summarize all the assessment variable, the Response Surface Methodology (RSM) process was conducted in this study. The response surface methodology is a method to determine the function of the result response to the change of some different input variable [25]. In this study, the RSM was used to normalize and summarize the output value of these three assessment variables. Due to all these assessment variables is better when they are smaller, the normalization function of these variables was similar which is linear $Ax + B$ function. The functions were transforming the results value of each assessment variables to be in range between 1 and 6. The normalization function for simulation results error variable is written in Eq. 12.

$$y_e = f(x_e) = 7400x_e - 107 \quad (12)$$

Then, the normalization function for the average timestep iteration variable and the average timestep calculation time variable was based on Eq. 13 and Eq. 14.

$$y_i = g(x_i) = 0.080x_i - 1.5 \quad (13)$$

$$y_t = h(x_t) = 14.64x_t - 13.6 \quad (14)$$

The summary of the assessment is the sum of y_e, y_i, and y_t which is the turbulence model with smallest value is the most suitable model for cross-flow turbine simulation.

3. Results

3.1 Independency Test Results

This study used Richardson extrapolation method to find the suitable mesh and timestep size for the assessment process. The variation of mesh in this study was about 62k, 40k, and 26k nodes. The number of mesh was translated as normalized grid spacing value of 1.295, 1 and 0.772. For the timestep independency test, the timestep frequency was varied for 500 Hz, 1250 Hz and 2500 Hz, then translated to 4.84, 2.2, and 1 of normalized timestep spacing value. Pressure probe at point (-0.19, 0.07) m was used for the testing variable. The location of the point in this study case is marked as (*) in figure 1. The Richardson extrapolation results for mesh and timestep independency is shown at the chart in figure 3 and figure 4.

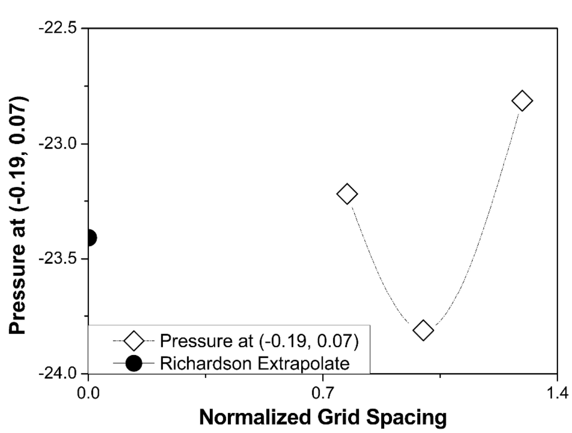


Fig. 3. Mesh Independency Result

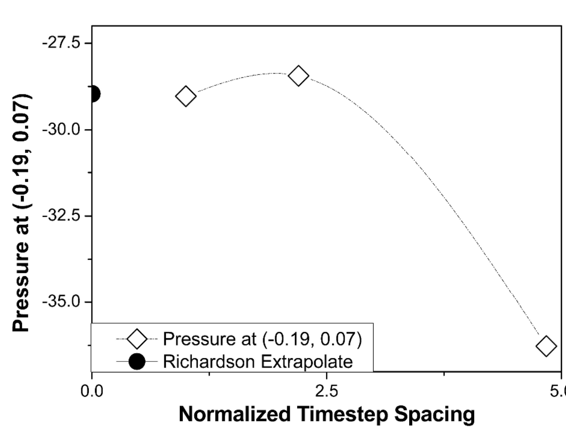


Fig. 4. Timestep Independency Result

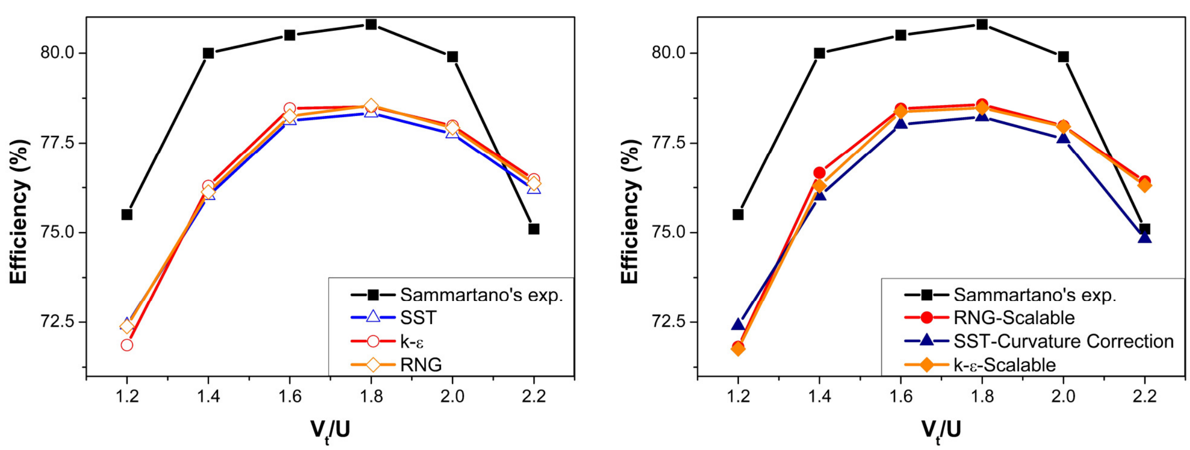
Figure 3 shows that the pressure was alternating up and down with increasing amplitude respectively to the normalized grid spacing. It was expected that the exact pressure at point (-0.19, 0.007) m was -23.41 Pa by the Richardson extrapolation. From Figure 3 it can be implied that the deviation between calculated pressure in 0.772 and 1 of normalized gird spacing value is smaller than the same results between 1 and 1.295 of normalized grid spacing value. The reduction of deviation is expected always happened at the smaller grid spacing than converged at -23.4 Pa. The calculated error between calculated pressure in 0.772 and 1 of normalized gird spacing value was 0.79% which is lower than appointed threshold, which is 1%. It can be concluded that for the assessment phase in this study can use meshing with density 40k nodes or 62k nodes. As the results of the mesh independency test, this study is using 40k nodes meshing due to computer processor and memory limitation.

The similar condition was also happened in timestep independency testing process. The pressure at the same point was alternating with amplitude increment respectively to the normalized timestep spacing. The Richardson extrapolation forecasted the exact pressure in the timestep independency test at the same point with the mesh independency test was -28.96 Pa. From the calculation, the error of calculated pressure between normalized timestep spacing of 1 and 2.2 was 0.24%. It was implied that timestep frequency 1250 Hz and 2500 Hz was good enough. This study was use 2500 Hz of timestep frequency with consideration that timestep size or frequency only affect to storage usage and total time of calculation.

The estimated exact value result from Richardson extrapolation is different in mesh independency test process and timestep independency process because of the test sequence. The mesh independency test was run before the timestep independency with steady state calculation mode for resource efficiency. The timestep independency test run afterwards and must be in the transient mode with rotating runner. The rotation of the runner induced the surrounding air then make the surrounding air kinetic energy was risen. The rising of the kinetic energy made the pressure of the surrounding air, including the tested point, decreased.

3.2 Turbulence Model's Errors Assessment

The main criteria for a good turbulence model was the error with the experimental results. Figure 5 is presenting the efficiency curve comparation in this study.



a. Turbulence model non-wall function b. Turbulence models with wall function

Fig. 5. Comparison of CFD results with Sammartano's study

Graphics in Figure 5 show that all the turbulent models give the almost similar results. But, in detail, it was implied that each turbulent model has different error characteristic for each V_T/U. Deviation at all point was summed for each turbulent model to summarize the results and make the ranking list. The ranking of turbulence model's accuracy is displayed in table 3.

Table 3
The numerical results error comparison (smaller is better)

Turbulence Model	Average Error	y_e
Standard k-ε	0.01508	4.576
k-ε with Scalable Wall Function	0.01525	5.847
Standard RNG	0.01473	1.979
RNG with Scalable Wall Function	0.01462	1.211
Standard Transitional SST	0.01513	4.989
Trans. SST with Curvature Correction	0.01523	5.731

This study error assessment results show that RNG turbulence model with scalable wall function was the most accurate model to predict the efficiency of cross-flow turbine, followed by Standard RNG model. First remark of comparation between numerical and experimental results is the numerical simulations results is lower than the experimental one. This phenomena was similar with some other studies before [16][26][27]. This can be implied that the loss of the cross-flow turbine performance caused by turbulent and velocity randomness which is could not be calculated in analytical approach was calculated too much in numerical simulations. The RNG model could renormalize the overcalculated turbulent to match the real condition. This caused the RNG family could resulting the higher efficiency than others and make them closer to the experiment results.

3.3 Average Timestep Iterations Number Assessment

As the CFD simulations use the numerical approach, the simulation is using iteration process to determine the correct value of all variables included in fluid dynamics phenomena. In ANSYS™ FLUENT® transient simulations, the maximum iteration number per timestep can be set to keep the calculation run efficiently. However, the iteration process can be converged before the maximum iteration number reached. A good turbulence models should have a good prediction of the exact value of variables in faced case, so the iteration process could be minimized. The ranking of the lowest iteration needs of turbulence models is shown in table 4.

Table 4
The average timestep iteration comparison (smaller is better)

Turbulence Model	Average Iterations Timestep	y_i
Standard k-ε	75.429	4.534
k-ε with Scalable Wall Function	48.468	2.377
Standard RNG	87.099	5.468
RNG with Scalable Wall Function	89.611	5.669
Standard Transitional SST	53.872	2.810
Trans. SST with Curvature Correction	32.389	1.091

Table 4 displayed that Transitional SST need fewer iterations to be convergence. With four equations, this model can determine the suitable calculation method at different point of the case. This was resulting this model more quickly to convergence than other models. The curvature correction feature also gives a better prediction of exact value of fluid dynamic variable at different place in the case. However, the scalable wall function RNG needs more iterations to be convergence although the calculation results were more precise.

3.4 Average Timestep Calculation Time Assessment

This study was run in 2000 timestep with 2500 Hz of timestep frequency. All the turbulence models were used in the same treated calculation process which is, 2 process of parallel calculation, same computer, and run without disturbance of other applications. The average timestep calculation time each turbulence model then normalized so that the shortest period was become 1. Table 5 is showing ranking of average timestep calculation time for each turbulence model.

Table 5
The relative average timestep calculation time comparison (smaller is better)

Turbulence Model	Relative Average Calculation timestep	y_t
Standard k-ε	1.138	3.060
k-ε with Scalable Wall Function	1.068	2.036
Standard RNG	1.321	5.735
RNG with Scalable Wall Function	1.288	5.256
Standard Transitional SST	1.000	1.040
Trans. SST with Curvature Correction	1.132	2.972

It can be implied from table 4 and 5 that the average needed time per timesteps has a high relation with the needed iterations per timesteps. Otherwise, the rumor about Transitional SST which has four equations is heavier and needs more times to calculate a CFD case was not proven. The Curvature Correction feature even give a high impact to the needed time instead of the number of equations in the turbulence models itself. This can be proven from the switching position between standard SST model and SST model with Curvature Correction feature in table 4 and 5.

3.5 Summary of The Assessments

After the assessment of errors, average iteration per timestep, and average time per timestep, the point which obtained by the turbulence model was summed as the final point. The final point of each turbulence model then compared and ranked to nominate the most suitable turbulence model for pico hydro cross-flow turbine CFD simulations. Figure 6 displaying the final point ranking of turbulence models.

From Figure 6, it can be concluded that the most suitable turbulence model for pico hydro cross-flow turbine is standard transitional SST turbulence model. Although this turbulence model was not very good in for the accuracy, this model was very good for the calculation process performance, especially for time spending. The remarkable result was pointed to k-ε model with scalable wall function which placed in third position. With still acceptable error (about 1.5%), this simple model could give the balance performance between errors, time, and iteration needs. Some advanced modification of this model could increase the performance of this model to give better prediction. The modification of k-ε model will be conducted in the next study.

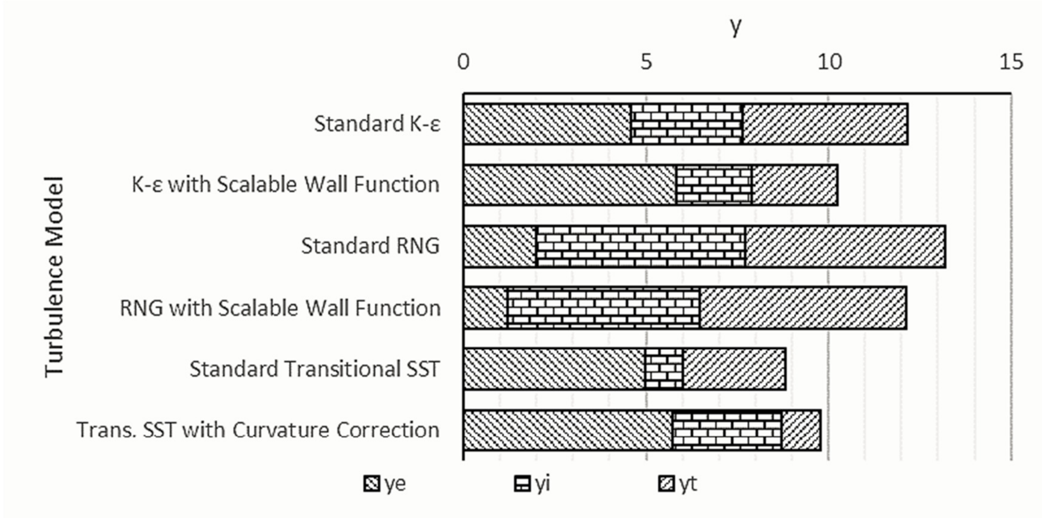


Fig. 6. Turbulence Model Assessment Results (smaller is better).

4. Conclusions

The assessment of six types of turbulence models with three criteria variables has been done in this study. The most suitable turbulence model for pico hydro cross-flow turbine is standard transitional SST turbulence model. This turbulence model has a good mark in time usage and rate of convergence which only has y total about 8.8. However, this conclusion is still not final because the validation data uses secondary data, but these results can be a preliminary reference that the turbulent model has a significant effect on the results will be obtained on the pico hydro cross-flow turbine object.

Acknowledgement
The authors would like to express they're thanks to KEMENRISTEK DIKTI, which has funded this research with grant No. SP. DIPA-042.06.1.401516/2018.

References
[1] "Rencana Umum Penyediaan Tenaga Lilstrik PT PLN (PERSERO) 2018-2027." Mar-2018.
[2] Budiarso, Warjito, Dendy Adanta, N. S. Putra, and 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.
[3] Kaunda, Chiyembekezo S., Cuthbert Z. Kimambo, and Torbjorn K. Nielsen. "A technical discussion on microhydropower technology and its turbines." *Renewable and Sustainable Energy Reviews* 35 (2014): 445-459.
[4] Adanta, Dendy, and Aji P. Prakoso. "The effect of bucketnumber on breastshot waterwheel performance." In IOP Conference Series: Earth and Environmental Science, vol. 105, no. 1, p. 012031. *IOP Publishing*, 2018.
[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] Dendy Adanta, Budiarso, Warjito, Ahmad Indra Siswantara, Aji Putro Prakoso, "Performance comparison of NACA 6509 and 6712 on pico hydro type cross-flow turbine by numerical method," *J. Adv. Res. Fluid Mech. Therm. Sci.*, vol. 45, no. 1, pp. 116–127, 2018.
[7] 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).
[8] C. A. Mockmore and F. Merryfield, *The Banki water-turbine*, vol. 25. Engineering Experiment Station, Oregon State System of Higher Education, Oregon State College Corvallis, Ore, USA, 1949.

[9] 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.

[10] Williamson, S. J., B. H. Stark, and J. D. Booker. "Low head pico hydro turbine selection using a multi-criteria analysis." *Renewable Energy* 61 (2014): 43-50.

[11] Choi, Young-Do, Jae-Ik Lim, You-Taek Kim, and Young-Ho Lee. "Performance and internal flow characteristics of a cross-flow hydro turbine by the shapes of nozzle and runner blade." *Journal of fluid science and technology* 3, no. 3 (2008): 398-409.

[12] Chichkhede, Shashi, Vinay Verma, Vivek Kumar Gaba, and Shubhankar Bhowmick. "A simulation based study of flow velocities across cross flow turbine at different nozzle openings." *Procedia Technology* 25 (2016): 974-981.

[13] Mockmore, Charles Arthur, and Fred Merryfield. The Banki water-turbine. Vol. 25. Corvallis, Ore, USA: Engineering Experiment Station, Oregon State System of Higher Education, Oregon State College, 1949.

[14] 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.

[15] Kaniecki, Maciej. "Modernization of the outflow system of cross-flow turbines." *Task Quarterly* 6, no. 4 (2002): 601-608.

[16] 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.

[17] Acharya, Nirmal, Chang-Gu Kim, Bhola Thapa, and Young-Ho Lee. "Numerical analysis and performance enhancement of a cross-flow hydro turbine." *Renewable energy* 80 (2015): 819-826.

[18] RICHARDS, SHANE A. "Completed Richardson extrapolation in space and time." *Communications in numerical methods in engineering* 13, no. 7 (1997): 573-582.

[19] Versteeg, Henk Kaarle, and Weeratunge Malalasekera. An introduction to computational fluid dynamics: the finite volume method. Pearson Education, 2007.

[20] N. M. Aziz and V. R. Desai, “A laboratory study to improve the efficiency of cross flow turbines,” pp. 1–52, 1993.

[21] Aziz, N. M., and H. G. S. Totapally. "Design Parameter refinement for improved Cross-Flow turbine performance." Engineering Report (1994).

[22] Gerald Recktenwald, “The k - ε Turbulence Model.” Portland State University, Portland, Oregon, 2009.

[23] Yakhot, V. S. A. S. T. B. C. G., S. A. Orszag, S. Thangam, T. B. Gatski, and C. G. Speziale. "Development of turbulence models for shear flows by a double expansion technique." *Physics of Fluids A: Fluid Dynamics* 4, no. 7 (1992): 1510-1520.

[24] Langtry, Robin B., and Florian R. Menter. "Correlation-based transition modeling for unstructured parallelized computational fluid dynamics codes." *AIAA journal* 47, no. 12 (2009): 2894-2906.

[25] Box, George EP, and Norman R. Draper. Empirical model-building and response surfaces. John Wiley & Sons, 1987.

[26] Elbatran, A. H., O. B. Yaakob, Yasser M. Ahmed, and Ahmed S. Shehata. "Numerical and experimental investigations on efficient design and performance of hydrokinetic Banki cross flow turbine for rural areas." *Ocean Engineering* 159 (2018): 437-456.

[27] Kaniecki, Maciej, and Janusz Steller. "Flow Analysis through a Reaction Cross-Flow Turbine." In Proceedings of Conference on modelling fluid flow CMFF, vol. 3, pp. 2003-06. 2003.