Application of Computational Fluid Dynamics Method for Cross-flow Turbine in Pico Scale

Imam Syofii[a], Dendy Adanta[b], Aji Putro Prakoso[c], Dewi Puspita Sari[a]
[a]Study Program of Mechanical Engineering Education, Universitas Sriwijaya
Jalan Palembang-Prabumulih km. 32 Indralaya, 30662, South Sumatera, Indonesia

[b]Department of Mechanical Engineering, Faculty of Engineering, Universitas Sriwijaya
Jalan Palembang-Prabumulih km. 32 Indralaya, 30662, South Sumatera, Indonesia

[c]Department of Mechanical Engineering, Faculty of Manufacturing Technology, Universitas Jendral Achmad Yani
Cimahi, 40531, West Java, Indonesia
e-mail: dendyadanta@ymail.com

Abstract

Crisis electricity was a crucial issue in the rural area. Crossflow turbine (CFT) in pico in pico scale is the best option for electricity provider for rural areas. Due to its usefulness and development of computer technology, computational fluid dynamics method application for CFT study becomes increasingly frequent. This paper compiles the implementation of the computational fluid dynamic (CFD) approach for CFT on a pico scale. Based on the literature, the Renormalization Group (RNG) $k - \varepsilon$ turbulence model is recommended to predict the flow field that occurs in CFT because its error is lower than others turbulence models, the RNG $k - \varepsilon$ error of 3.08%, standard $k - \varepsilon$ of 3.19%, and transitional SST of 3.10%. Furthermore, six-degrees of freedom (6-DoF) is recommended because it has an error of 3.1% than a moving mesh of 9.5% for the unsteady approach. Thus, based on the review, the RNG $k - \varepsilon$ turbulence model and 6-DoF are recommended for the CFT on the pico scale.

Keywords: *Pico hydro; cross-flow turbine; CFD; RANS; RNG $k - \varepsilon$; 6-DoF approach*

1. INTRODUCTION

The CFD has categorized an impulse turbine because the blade dominantly absorbs the water kinetic energy to generate power [1][2]. The CFT absorbs energy in two stages [3]. The two-stage energy transfer makes flow field (turbulence phenomena) that occurs is rotating flow [4][5]. The rotating flow in CFT is interesting to be studied because by understanding the flow field, the losses can be minimized (efficiency increases) [6][3] [7][8][9][10][11].

Improving the CFT can be used analytically, numerically, experimentally, or a combination of those methods. The development of computer technology makes the CFD method increasingly frequent [12][5]. In 1985, the CFD successfully characterized the fluid flow characteristic inside the CFT's nozzle, and the optimum design of CFT's nozzle was recommended [8]. Furthermore, by CFD results, the optimum angle of attack was proposed [7][13]. Investigation of the best pressure distribution for promising nozzle designs using CFD analysis. [14].

In the early twenty-first century, CFD software was allowed to investigate more complex fluid dynamics [15] with two or three dimensional (2 or 3D), steady or transient analysis about CFT at a more affordable price [9][10][11][16][17]. In 2008, A quasi 2D steady-state approach was carried out to find the optimal guide vane angle and

characterize its internal flow [16]. In 2011, a thorough investigation of the internal flow of CFT was undertaken, and the influence of shaft inside CFT was discovered [17]. Some losses were found in CFT's hydrodynamic flow using 2D transient CFD simulations [17]. Several research on CFT was undertaken in 2013, beginning with an exhaustive literature review and some quasi 2D and 3D transient simulations to determine the optimal design of CFT [9]; then, the results are validated and explored in subsequent studies [18][19]. The CFT performance was successfully improved by altering the nozzle curvature by a 3D steady-state numerical analysis. [10]. Furthermore, another study of CFT nozzle enhancement has discovered a 90% efficiency utilizing numerical simulations validated by experimental results [11][20]. Their research was currently focused on increasing CFT efficiency by doubling the nozzle size [21].

The CFT numerical simulations are becoming more complicated to obtain more detailed findings. The turbine motion can be a numerical calculation result employing rigid body alternatives inside a 6-DoF approach. [22]. This option has been used in some recent studies to acquire a deeper investigation or to produce some breakthrough enhancements to CFT design [1][6][23]. By analyzing three cases of CFT in 2018, the 6-DoF technique was utilized to find the effect of airfoil profile on the internal flow characteristic and the turbine's performance. [6]. The findings revealed that the airfoil profile might positively impact the flow field; it does not affect its performance because the impulse effect is more powerful than the reaction [6]. Furthermore, the effect of blade curvature depth on its performance was conducted in 2018 [1]. Then, the calculation formula for the outer diameter ratio with blade curvature depth was proposed [23].

The current study attempted to summarize CFT progress, particularly utilizing a numerical approach. This paper summarizes some investigations on the quality of CFD results for simulating pico hydro CFT. Its goal is to offer the best unsteady technique and turbulence model for CFD numerical simulation on a pico scale.

2. STUDY OF THE CFT IN 20TH CENTURY

A. Michell menemukan CFT pada tahun 1903, dan D. Banki menemukan pendekatan teoretisnya yang kemudian disempurnakan oleh Sonnek pada tahun 1923 [24]. The CFT's optimum specific speed, according to Mockmore and Merryfield (1994) [3], was 14, which was higher than other impulse turbines. The results also revealed that the CFT performs consistently under varying water discharge [3]. The pressure at the tip of the CFT's nozzle was not zero, according to Haimerl (1960) [25], so this turbine isn't strictly an impulse turbine.

The influence of CFT's nozzle shape and size on its performance was examined by Nakase (1982) [26]. Nakase (1982) [26] recommended a height-to-width ratio for CFT nozzles based on Equation 1.

$$\frac{S_o}{R\lambda} \cong 0.26 \tag{1}$$

where S_0 is the nozzle's initial height, R is the CFT outer radius, and λ is the nozzle's discharge angle. The suggestion was put to the test, and it improved CFT performance to the point where it could attain 82% efficiency [26]. Then, according to Durgin and Fay (1984) [28], the first stage energy transfer contributed roughly 83% of overall energy transfer to CFT performance; this discovery is still relevant [16]. According to Fukutomi et al. (1985) [8], the numerical approach is an appropriate alternative for improving the CFT's nozzle. Then in 1991, Fukutomi et al. [27] examined the CFT flow field and concluded the effect of a diameter ratio on the CFT performance.

Khosrowpanah et al. (1988) [24] discovered that the 90º λ is optimum for CFT. Then, Khosrowpanah et al. (1993) [7] and (1994) [13] defined that the 22º angle of attack (α) for CFT is optimum; this result is verified [28]. Finally, Aziz and Totapally (1994) [13] proved the Mockmore and Merryfield [3] results and concluded that propose of the 0.68 diameter ratio (d/D) is optimum conditions.

Before the 21st century, the finding of CFT studies is it's the primary design parameter of CFT. Table 1 summarizes investigations conducted before the twenty-first century.

Table 1. The CFT studies before 21st century

	The angle of attack (α)	Nozzle design	Nozzle discharge angle (λ)	Diameter ratio (d/D)	Blade number (N_b)
Aziz (1994) [13]	■		■		■
Fukutomi et al. (1992) [27]				■	
Khosrowpanah et al. (1988) [24]			■		
Fukutomi et al. (1985) [8]		■			
Durgin and Fay (1984) [29]				■	
Nakase (1982) [26]		■	■		
Haimerl (1960) [25]		■			■
Mockmore and Merryfield (1949) [3]	■	■		■	

3. THE CFT STUDY IN 21ST CENTURY

The CFD simulation was employed in practically all CFT investigations in the twenty-first century. The CFD is frequently utilized because it is inexpensive, quick, and allows for detailed visualization of the flow field [30]. Kaniecki (2002) [25] attempted to increase CFT performance by added a draft tube into the outflow and evaluating the flow characteristics through the draft tube using the CFD simulation. The CFD simulations were used by Kaniecki and Steller (2003) [4] to study the flow pattern of CFT and categorize it as a reaction turbine. Choi et al. (2006) [31] characterized the effect of blade angle on the internal flow and found that the best outlet angle of the blade (β_2) is 90° by CFD simulation. Then, Choi et al. (2007) [32] studied internal flow with nozzle shape change by CFD simulation and discovered that the CFT possesses both impulse and reaction turbine characteristics. Next, according to Choi et al. (2008) [16], the internal flow feature significantly impacts the CFT performance. Andrade et al. (2011) [17] characterized internal flow as a function of turbine angular velocity. They concluded that the recirculation flow that happens as a shock should be minimized so that the turbine can function effectively [17].

Sammartano et al. investigated CFT optimization using the CFD simulation. In 2014, Sammartano et al. [33] developed CFT nozzles that can be modified for changing discharge circumstances. In 2015, Sammartano et al. [34] attempted to use CFT to create energy and flow controls for the water conveyance system outright. In 2016, Sammartano et al. [19] verified the optimization done in 2013 [9], which also used the velocity corrected inlet velocity formula. In addition, Sammartano et al. (2016) [19] examined different turbulence models to find the best turbulent model for the CFT.

The CFD approach has been used in some research to increase CFT performance. The CFT nozzle was revised by Acharya (2015) [10] to improve its performance. The CFT nozzle has been redesigned, increasing its efficiency from 63.7 to 76.6% [10].

Adhikari and Wood (2017) [11] did several tests to develop high-efficiency CFT and improved the nozzle tip. Furthermore, Adhikari and Wood (2018) [20] studied flow patterns and turbine performance when CFTs were operated at part-load, resulting in more efficient water discharge regulation of CFTs. Then, to increase efficiency, Adhikari

and Wood (2018) [21] examined CFT with a twin nozzle. Finally, a review paper related to the effort of various investigations to achieve the high efficiency of CFT was published at the end of 2018 as a summary of earlier [28]. Table 2 summarizes much research conducted in the twenty-first century.

Table 2. The CFT studies in 21st century

Authors	Experiment efficiency	Numerical efficiency
Adhikari, et.al. (2017-2018) [11][20][21]	84%	91%
Chichkhede, et.al. (2016) [2]	-	88%
Acharya, et.al. (2015) [10]	-	76,6%
Sammartano, et.al. (2013-2016) [9][33][34][19]	80,6%	79,4%
De Andrade, et.al. (2011) [17]	72%	76%
Choi, et.al. (2006-2008) [31][32][16]	76,2%	65,7%
Kaniecky, et.al. (2002-2003) [25][4]	78,6%	74,3%

The 3D domain was used for all of the CFD simulation studies in Table 2. In 2018 [6], 2D CFD analysis was done to increase CFT performance by altering the turbine blade with an airfoil profile. It was discovered that there are some turbulence features in the internal flow of the CFT that affect its performance. In addition, a 2D CFD study [1] was conducted to explore the effect of the curve of the blade on its performance.

4. TURBULENCE MODEL STUDY FOR THE CFT IN PICO SCALE

Three turbulent models (k-ε model, RNG k-ε model, and Transitional SST) were compared to establish the effects on CFD simulation (prediction) [19]. Table 3 summarizes the outcomes of the comparison. The k-ε and RNG k-ε model have higher average relative error (ḡ) than Transitional SST [19], as seen in Table 3. Equation 2 defines the relative error [19].

$$\bar{g} = \frac{\eta_{sim} - \eta_{exp}}{\eta_{exp}} \qquad (2)$$

and Equation 3 is used to define the absolute relative error (g_{RE}).

$$g_{RE} = \frac{|\eta_{sim} - \eta_{exp}|}{\eta_{exp}} \qquad (3)$$

where η_{exp} is turbine efficiency by experimental and η_{sim} is efficiency by CFD simulation. This research was carried out in a 3D domain, which means there are more walls than in a 2D simulation, and the turbulence model of k-ε with the near-wall treatment has a minor weakness compared to the k-ε standard.

Table 3. In various turbulence models, relative error, according to Sammartano et al. [19]

V_t/U	Transitional SST	k-ε	RNG k-ε
2,2	0,58%	-2,07%	0,07%
2,0	-1,55%	-5,93%	-3,37%
1,8	-0,96%	-5,45%	-3,20%
1,6	-1,16%	-3,95%	-2,83%
1,4	-1,15%	-3,56%	-3,07%
ḡ	-0,70%	-3,91%	-2,39%
g_{RE}	0,91%	3,91%	2,41%

Siswantara et al. [5] verified Sammartano et al. [19] study. Siswantara et al. [5] utilize a 2D model, whereas Sammartano et al. [19] employ a 3D model. Furthermore, Siswantara et al. [5] proposed a 2D representation of the performance and flow field for CFT in pico scale. Therefore, the 2D analysis able is used to determine the effect of turbulence model utilization. Table 4 summarizes the findings of Siswantara et al. [5].

*Table 4. Relative error CFD simulation by Siswantara et.al. [5]*

Turbulence model	$\bar{g}$	$\overline{g_{RE}}$
Transitional SST	-3,10%	3,10%
k-ε	-2.65%	3,19%
RNG k-ε	-2,50%	3,08%

From Table 4, all turbulence model has $\overline{g_{RE}}$ of 3%. The deviation error of each model of below 0.11%. In CFT in pico scale CFD simulation, the turbulence model effect appears to be ignorable. Moreover, the transitional SST has a greater $\overline{g_{RE}}$ than RNG k-ε, this is contradicting to previous findings [5][19]. These investigations found that the maximum $\overline{g_{RE}}$ of CFT CFD simulation using difference turbulence model is less than 4%. Indicated that thrid turbulence model are close enough to predict the CFT performance in pico scale.

Table 3 and 4, for CFT simulation in pico scale, the k-ε standard is recommended since it requires less computational power (simpler equation) than Transitional SST and RNG k-ε. However, to confirm the validity of findings (feasibility k-ε standard turbulence model) the y^+ in near-wall should be of 30 ≤y⁺≤300 range [35].

5. UNSTEADY APPROACH FOR THE CFT CFD SIMULATION IN PICO SCALE

The CFT CFD simulation was carried out with ANSYS Fluent software utilizing two unsteady approaches: moving mesh and 6-DoF [36]. Figure 1 shows a comparison of the two approaches.
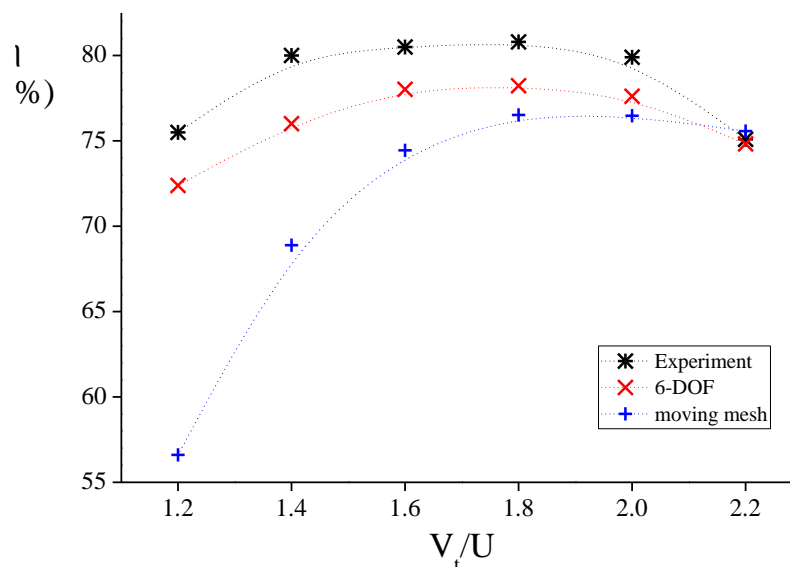
Graphic 1. Comparison 6-DoF with moving mesh, and experimental [36]

Graphic 1 shows that the 6-DoF has closer to experimental than the moving mesh. The $\overline{g_{RE}}$ with the 6-DoF to experimental data of 3.1%, while the $\overline{\delta_{RA}}$ of the moving mesh of 9.5% [36]. As a result, it can be stated that the 6-DoF approach is appropriate for CFT CFD simulations on the pico scale.

6. CONCLUSION

A review of CFT studies was undertaken, focusing on the unsteady numerical approach for CFD simulation. Based on results, $\overline{g_{RE}}$ of 3.08% for RNG k-ε, 3.19% for standard k-ε, and 3.10% for transitional SST. As a result, it is recommended RNG $k - \varepsilon$ turbulence model for CFT 2D numerical simulation. The 6-DoF is recommended for the unsteady approach because it has $\overline{g_{RE}}$ of 3.1% lower than the moving mesh of 9.5%.

REFERENCES

1. Adanta D, Hindami R, Budiarso, Warjito, Siswantara AI. Blade Depth Investigation on Cross-flow Turbine by Numerical Method. In: 2018 4th International Conference on Science and Technology (ICST). Yogyakarta: IEEE; 2018. p. 1–6. doi: https://doi.org/10.1109/ICSTC.2018.8528291
2. Chichkhede S, Verma V, Gaba VK, Bhowmick S. A simulation based study of flow velocities across cross flow turbine at different nozzle openings. Procedia Technology. 2016;25:974–81. DOI: https://doi.org/10.1016/j.protcy.2016.08.190
3. Mockmore CA, Merryfield F. The Banki water-turbine. Vol. 25, Engineering Experiment Station Bulletin Series. Engineering Experiment Station, Oregon State System of Higher Education, Oregon State College Corvallis, Ore, USA; 1949. 1–28 p.
4. Kaniecki M, Steller J. Flow Analysis through a Reaction Cross-Flow Turbine. In: Proceedings of Conference on modelling fluid flow CMFF. 2003. p. 2003–6.
5. Siswantara AI, Budiarso, Prakoso AP, Gunadi GGR, Warjito, Adanta D. Assessment of Turbulence Model for Cross-Flow Pico Hydro Turbine Numerical Simulation. CFD Letters [Internet]. 2018;10:38–48. Available from: akademiabaru.com
6. Adanta D, Budiarso, Warjito, Siswantara AI, Prakoso AP. Performance Comparison of NACA 6509 and 6712 on Pico Hydro Type Cross-Flow Turbine by Numerical Method. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences. 2018;45:116–27.
7. Aziz NM, Desai VR. A Laboratory study to improve the efficiency of cross-flow turbines. South Caroline; 1993.
8. FUKUTOMI J, NAKASE Y, Watanabe T. A numerical method of free jet from a cross-flow turbine nozzle. Bulletin of JSME. 1985;28(241):1436–40. DOI: https://doi.org/10.1299/jsme1958.28.1436
9. Sammartano V, Aricò C, Carravetta A, Fecarotta O, Tucciarelli T. Banki-Michell optimal design by computational fluid dynamics testing and hydrodynamic analysis. Energies. 2013;6(5):2362–85. DOI: https://doi.org/10.3390/en6052362
10. Acharya N, Kim C-G, Thapa B, Lee Y-H. Numerical analysis and performance enhancement of a cross-flow hydro turbine. Renewable energy. 2015;80:819–26. DOI: https://doi.org/10.1016/j.renene.2015.01.064
11. Adhikari RC, Wood DH. A new nozzle design methodology for high efficiency crossflow hydro turbines. Energy for Sustainable Development. 2017;41. DOI: https://doi.org/10.1016/j.esd.2017.09.004
12. Adanta D, Budiarso, Warjito, Siswantara AI. Assessment of Turbulence Modelling for

Numerical Simulations into Pico Hydro Turbine. Journal of Advanced Research in Fluid Mechanics and Thermal Sciences. 2018;46:21–31.

13. Aziz NM, Totapally HGS. Design Parameter refinement for improved Cross-Flow turbine performance. Engineering Report. 1994;

14. Pereira NHC, Borges JE. Study of the nozzle flow in a cross-flow turbine. International journal of mechanical sciences. 1996;38(3):283–302. DOI: https://doi.org/10.1016/0020-7403(95)00055-0

15. Ansys I. ANSYS FLUENT theory guide. Canonsburg, Pa. 2011;794.

16. Choi Y-D, Lim J-I, Kim Y-T, Lee Y-H. 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. 2008;3(3):398–409. DOI: https://doi.org/10.1299/jfst.3.398

17. De Andrade J, Curiel C, Kenyery F, Aguillón O, Vásquez A, Asuaje M, Aguilln O, Vásquez A, Asuaje M, Aguillón O, Vásquez A, Asuaje M. Numerical investigation of the internal flow in a Banki turbine. International Journal of Rotating Machinery. 2011;2011. DOI: https://doi.org/10.1155/2011/841214

18. Sammartano V, Morreale G, Sinagra M, Collura A, Tucciarelli T. Experimental study of cross-flow micro-turbines for aqueduct energy recovery. Procedia Engineering. 2014;89:540–7. DOI: https://doi.org/10.1016/j.proeng.2014.11.476

19. Sammartano V, Morreale G, Sinagra M, Tucciarelli T. Numerical and experimental investigation of a cross-flow water turbine. Journal of Hydraulic Research. 2016;54(3):321–31. DOI: https://doi.org/10.1080/00221686.2016.1147500

20. Adhikari RC, Wood DH. Computational analysis of part-load flow control for crossflow hydro-turbines. Energy for Sustainable Development. 2018;45:38–45. DOI: https://doi.org/10.1016/j.esd.2018.04.003

21. Adhikari R, Wood D. Computational Analysis of a Double-Nozzle Crossflow Hydroturbine. Energies. 2018;11(12):3380.

22. ANSYS FLUENT UDF Manual. Canonsburg, PA: ANSYS, Inc; 2011.

23. Adanta D, Prakoso AP, Siswantara AI, Warjito, Budiarso. Simplification Design of Nozzle and Blade of Pico Hydro Turbine type Cross-flow. In: 17th Annual National Ceminar on Mechanical Engineering (SNTTM XVII). Kupang: BKSTM; 2018. p. 212–7.

24. Khosrowpanah S, Fiuzat AA, Albertson ML. Experimental study of cross-flow turbine. Journal of Hydraulic Engineering. 1988;114(3):299–314. DOI: https://doi.org/10.1061/(ASCE)0733-9429(1988)114:3(299)

25. Maciej Kaniecki. Modernization of the outflow system of cross-flow turbines. Task Quarterly. 2002;6(4):601–8.

26. Nakase Y. A study of cross-flow turbine (effects of nozzle shape on its performance). In: ASME 103rd Winter Annual Meeting. 1982.

27. FUKUTOMI J, SENOO Y, NAKASE Y. A numerical method of flow through a cross-flow runner. JSME international journal Ser 2, Fluids engineering, heat transfer, power, combustion, thermophysical properties. 1991;34(1):44–51. DOI: https://doi.org/10.1299/jsmeb1988.34.1_44

28. Adhikari R, Wood D. The Design of High Efficiency Crossflow Hydro Turbines: A Review and Extension. Energies. 2018;11(2):267. DOI: https://doi.org/10.3390/en11020267

29. Durgin WW, Fay WK. Some fluid flow characteristics of a cross-flow type hydraulic turbine. Small Hydro Power Fluid Machinery. 1984;p77-83.

30. Versteeg HKHK, Malalasekera W. An introduction to computational fluid dynamics: the finite volume method. Fluid flow handbook. McGraw-Hill. Pearson Education; 2007. 267 p.

31. Choi YD, Lim JI, Kim CG, Kim YT, Lee YH. CFD analysis for the performance of cross-flow hydraulic turbine with the variation of blade angle. In: New Trends in Fluid Mechanics Research. Springer; 2007. p. 428–31. DOI: https://doi.org/10.1007/978-3-

540-75995-9_140

32. Choi Y-D, Lim J-I, Kim Y-T, Lee Y-H. Internal Flow Characteristics of Cross-Flow Hydraulic Turbine with the Variation of Nozzle Shape. In: ASME/JSME 2007 5th Joint Fluids Engineering Conference. American Society of Mechanical Engineers Digital Collection; 2007. p. 1089–94. DOI: https://doi.org/10.1115/FEDSM2007-37541

33. Sinagra M, Sammartano V, Aricò C, Collura A, Tucciarelli T. Cross-Flow turbine design for variable operating conditions. Procedia Engineering. 2014;70:1539–48. DOI: https://doi.org/10.1016/j.proeng.2014.02.170

34. Sammartano V, Aricò C, Sinagra M, Tucciarelli T. Cross-Flow Turbine Design for Energy Production and Discharge Regulation. Journal of Hydraulic Engineering. 2015;141(3). DOI: https://doi.org/10.1061/(ASCE)HY.1943-7900.0000977

35. Fluent A. ANSYS fluent theory guide 15.0. ANSYS, Canonsburg, PA. 2013;

36. Prakoso AP, Warjito, Siswantara AI, Budiarso, Adanta D. Comparison Between 6-DOF UDF and Moving Mesh Approaches in CFD Methods for Predicting Cross-Flow Pico- Hydro Turbine Performance. CFD Letters. 2019;11(6):86–96.