

Review Article

A Review on Computational Fluid Dynamics Applications in the Design and Optimization of Crossflow Hydro Turbines

Hamisi Ally Mrope , Yusufu Abeid Chande Jande , and Thomas T. Kivevele 

*Nelson Mandela African Institution of Science and Technology (NM-AIST),
The School of Materials, Energy, Water, and Environmental Sciences, P. O. Box 447, Arusha, Tanzania*

Correspondence should be addressed to Hamisi Ally Mrope; mropeh@nm-aist.ac.tz

Received 10 February 2021; Revised 6 July 2021; Accepted 27 July 2021; Published 15 October 2021

Academic Editor: Jing Shi

Copyright © 2021 Hamisi Ally Mrope et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

In recent years, advances in using computational fluid dynamics (CFD) software have greatly increased due to its great potential to save time in the design process compared to experimental testing for data acquisition. Additionally, in real-life tests, a limited number of quantities are measured at a time, while in a CFD analysis all desired quantities can be measured at once, and with a high resolution in space and time. This article reviews the advances made regarding CFD modeling and simulation for the design and optimization of crossflow hydro turbines (CFTs). The performance of these turbines depends on various parameters like the number of blades, tip speed ratio, type of airfoil, blade pitch, chord length and twist, and its distribution along the blade span. Technical aspects of the model design, which include boundary conditions, solution of the governing equations of the water flow through CFTs, and the assumptions made during the simulations are thoroughly described. From the review, a clear idea on the suitability of the accuracy CFD applications in the design and optimization of crossflow hydro turbines has been provided. Therefore, this gives an insight that CFD is a useful and effective tool suitable for the design and optimization of CFTs.

1. Introduction

Computational fluid dynamics is the discipline of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by resolving a set of governing mathematical equations numerically: conservation of mass, momentum, energy, and species mass. It is an important tool used in fluid mechanics to solve and analyze problems that involve fluid flow through numerical methods and algorithms. The CFD has been used to predict the behavior of fluid flow by approximations that solve partial differential equations (PDEs) governing flows. For its efficacy, the procedure for determining the degree to which CFD model is an accurate representation of the real world from the viewpoint of the projected use is essential. CFD solutions must indicate the error bands and uncertainties incurred in the results to have confidence in the predicted results [1].

Practical hydraulic turbine method has become a compelling technique to capture minor details of the flow

which are unbearable in the physical model testing and overcome the limitations of the physical laboratory setup [2]. Complementary to experimental investigation, the numerical simulation of flows is an auspicious way to investigate flows at real operating conditions [3].

Although the key geometric features and their effects on turbine efficiency have been experimentally studied, this knowledge does not readily help to design high-efficiency turbines, partly because of the knowledge about the details of the runner flow. As it is difficult and expensive to measure and visualize the flow fields in the runner, the alternative is computational simulation [4]. There are various CFD simulation codes available in the industry and designed for the application of computer-aided engineering. However, in crossflow turbine modeling, there is a necessity to select an accurate CFD code, which will save time and cost of simulation. CFD solvers are based on the finite volume method, whereby the fluid region is decomposed.

Into Finite Set of Control Volumes. General conservation equations of mass, momentum, energy, and species are solved on this set of control volumes.

Crossflow turbines were conceived by Michell [5] who referred to them as Michell-Banki turbines or impulse turbines in which water strikes the turbine transversely across its blades. The maximum efficiency of crossflow turbines tends to be 70–86%, which is weaker than that of more frequently used advanced turbines such as Pelton, Francis, and Kaplan, which have typical maximum efficiencies above 90% [6, 7].

The crossflow turbine operates with ambient air pressure on the free surface [8]. However, since the crossflow hydro turbines had low price and were mostly suitable for microhydropower units less than 2 MW and heads less than 200 m, they are mostly appropriate for the production of electricity particularly in rural communities of developing countries. Despite being enriched with perennial rivers, most of these communities lack electricity.

In demand to improve the efficiency of CFT, a commercial CFD simulation code has been used to study the turbine operation and determine the parameters and phenomena that affect its performance [2]. CFT has been successfully used [1, 4, 9–11] because, among others, its horizontal efficiency curves yield better annual performance in small river flows than other turbine systems, which are lesser in some months. Moreover, CFTs have a relatively small efficiency and can maintain efficiency under varying load and water flow conditions, the primary inspiration to improve its existing design.

Both numeric and experimental tests have been used for the design and optimization of CFTs. However, the trends of simulated and experimental efficiency curves are similar even if the experimental efficiency values are always a bit higher than the simulated ones [4, 9, 12–22]. Moreover, many of the authors argue that experimental tests provide a good validation of the results obtained by means of CFD analysis.

In the process of designing and optimizing CFTs, similar parameters are used in both experimental and numerical methods. Parameters that are used to influence the improvement of the performance of crossflow turbines are shown in Table 1 below [23].

Vital geometrical parameters of crossflow turbine as described in its cross section are as shown in Figure 1.

2. CFD Modeling Overview

CFD analysis can be categorized into four main steps (Figure 2): problem identification, preprocessing, solver/solution, and postprocessing. Figure 1 shows the main steps of performing a CFD analysis.

2.1. CFD. This is the first step in the numerical simulation whereby at least four activities are done: the formation of a robust model to characterize the domain mesh generation, set up fundamental physics, and define solver settings.

TABLE 1: Important parameters in crossflow turbine.

S/No	Design parameter	Unit
01	Outer diameter	mm
02	Inner diameter	mm
03	The angle of attack	Degree
04	Outer blade angle	Degree
05	Inner blade angle	Degree
06	Blade radius	mm
07	Blade thickness	mm
08	Number of blades	—
09	Impeller and nozzle width	mm
10	Nozzle thickness or throat	mm
11	Nozzle entry arc angle	Degree

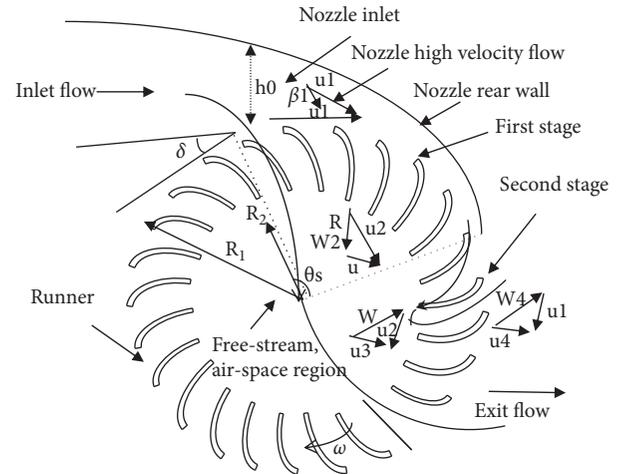


FIGURE 1: Critical geometrical parameters of crossflow turbine. Respondent [4] (copyright (2018), with permission under Creative Commons Attribution 4.0 International License).



FIGURE 2: CFD modeling overview.

2.2. Solver. The second step in the numerical simulation analysis is the solver. The CFD solver is used to set the numerical schemes and convergence control and perform computation. To produce a good result, the fundamental understanding of preprocessing is essential.

2.3. Postprocessing. The last stage in conducting CFD analysis is the postprocessing. In this stage, the results are examined. The CFD postprocessing can perform flow field visualization and quantitative data analysis. CFD simulation codes can present and plot flow variables on a point, line, and any plane of interest. Statistical reporting tools can be used to compute quantitative results such as forces and moments, average heat transfer coefficients, and flux balance.

3. Validation of CFD Codes with Experiments

There was a need to review validations on CFD codes with experiments in order to demonstrate their accuracy and

suitability for design and optimization activities. Moreover, this provides a clear insight on the prospects including cons and pros of CFDs.

Modeling and uncertainties are due to assumptions and approximations in the mathematical representations of the physical problem and incorporation of previous data into the model (such as fluid properties), and the physical problem in modeling includes geometry, mathematical equation, coordinate transformation, boundary conditions, and turbulence models. Numerical errors and uncertainties are due to the numerical solution of the mathematical equations (such as discretization, artificial dissipation, incomplete iterative and grid convergence, lack of conservation of mass, momentum and energy, internal and external boundary noncontinuity, and computer round-off [22]).

The procedure of determining the degree to which a model is an accurate representation of the real world from the viewpoint of the projected uses of the model is essential. CFD solutions must indicate the error bands and uncertainties incurred in the results in order to have confidence in the predicted results [1]. Experimental hydraulic turbine method has become a compelling technique to capture minor details of the flow which are unbearable in the physical model testing and overcome the limitations of the physical laboratory setup [2]. Complementary to experimental investigation, the numerical simulation of flows, commonly referred to as CFD, is an auspicious way to investigate flows at real operating conditions [16].

[24] studied the effect of the nozzle in the performance of crossflow hydro turbines. It concluded that nozzle rear-wall shape for both circular and logarithmic spiral rear-wall gave the same maximum efficiency of the crossflow hydro turbine.

[21, 22] focused on developing a suitable numerical model such that the results have a close agreement with experiments. In their studies, the authors made a comparison of three numerical models (k - ϵ , RNG, and SST) in which one of the studies showed that SST turbulence model has an efficiency of 67.3%–67.7% for CFT and 71% for the experiment. From these results, the efficiencies of the CFD and experiment had a close match with the experimental results, even if the experimental efficiency values were higher than the simulated ones.

Commercial CFD ANSYS codes have been used to compare RANS/URANS transition and scale resolving simulation (SRS) turbulence methods. Studies showed that CFD simulation applications are computationally practicable and suitable for most of the turbo machinery design analysis [25–27]. Moreover, flow field simulations for predicting the action of turbine performance on free-surface flow situations have appeared due to the complex nature of this physic spectacle [10]. This authenticates its role in improving efficacy and reducing the effort and cost required for experimentation thus facilitating better designs of CFTs. This suggests that the advancement in CFD simulation codes has allowed researchers and engineers to confront the complex nature of the physic spectacle by modeling to obtain three-dimensional flow field simulations in turbines.

Results in Figure 3 and Table 2 show a comparison of CFD and experimental results for the power output of 0.268–12.54 KW. For the power output of 7 kW (Figure 4), there is a satisfactory agreement between CFD and experimental results with a relative error of less than 6%. Similarly, CFD and experimental results for 0.53 kW turbine differed by less than 3.8%. The results elucidate that CDF has facilitated the conversion of the head into kinetic energy and matching of nozzle flow with runner design. This points out that CDF is crucial in turbine design.

[32] conducted a study on the performance improvement of CFT by air layer effect. CFD analysis on the performance and internal flow of the turbine was conducted in the unsteady state using a two-phase flow model to embody the air layer effect on the turbine performance effectively. From the results, the air layer effect on the performance of the turbine is considerable. The air layer located in the turbine runner passage plays a role of preventing shock loss at the runner axis and suppressing a recirculation flow in the runner. Moreover, the ratio of air from the suction pipe to water from the turbine inlet is also a significant factor in the turbine performance. The difference of efficiency ratio between experiment and CFD was 1.4%. This disparity was due to the difference in mechanical loss because the amount of mechanical loss predicted by CFD analysis is usually smaller than that obtained by experiment.

Figure 5 shows the use of CFD tools in the design of an internal deflector for CFT efficiency improvement [33]. Commercial CFD code ANSYS CFX v.13 was used to carry out all simulations. A transient regime, two-dimensional, two-phase (air-water) homogeneous model with a Eulerian-Eulerian approach was selected, and the SST turbulence model was preferred. The results obtained concluded that the performance of a CFD model was in line with the experimental data. Maximum efficiency deviation from the experiment was around 5% at the three evaluated specific speeds.

From the investigation results in Figure 5, it was also found that the overall use of the internal deflector increases CFT performance and output power. Prasad et al. [37] studied the flow control in Banki turbines by using ANSYS Fluent code with k - ω SST turbulence model for numerical analysis. The study showed that the experimental and numerical simulation results were reasonably good. Based on the CFD fluent code analysis, the velocity fields obtained by numerical simulations, which are responsible for the decrease in turbine efficiency, were identified.

Numerical investigation of a turbine in a transient state reaction determination was conducted by [38]. The investigation was conducted purposely to understand the flow inside CFTs using CFD tools.

Furthermore, steady and transient state simulations were performed for a CFT at a specific speed $N_s = 45$. The results obtained from SST and k - ϵ turbulence models were compared in terms of simulation requirements. In addition, the proposed runner-nozzle interface by considering a real CFT existent gap between these two components was compared with the available experimental data. The study revealed that the maximum numerically calculated efficiency deviated

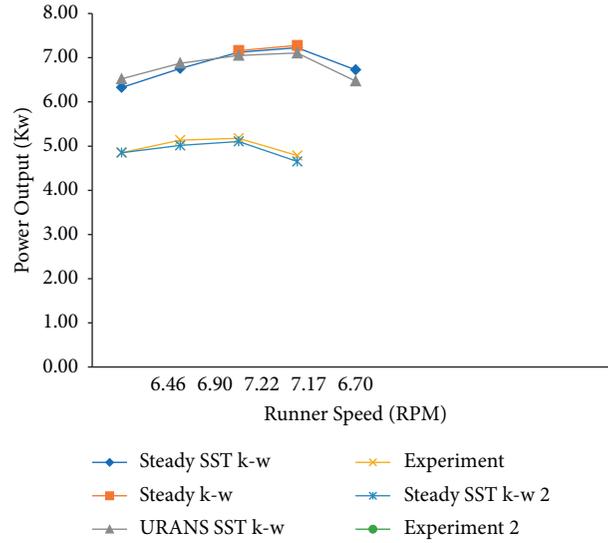


FIGURE 3: Comparison of CFD and experimental outcomes [4]. Experimental data were taken from [28] (copyright (1982), with permission under Creative Commons Attribution 4.0 International License).

TABLE 2: Performance comparison.

S/No	Author	Targeted power	Modal	Head	Flow rate	Flow control mechanism	Efficiency
01	[29]	7.20 kW	SST	10.0 m	0.10m ³ /s	Guide vane	76.6%
02	[30]	—	<i>k-ε</i>	—	—	Guide vane	Less than 80%
03	[1]	7.00 kW	(SST) <i>k</i>	10.0 m	0.11 m ³ /s	—	—
04	[15]	—	<i>k-ωT</i>	1.2 m	0.23 m ³ /s	—	58%
05	[31]	7.00 kW	(SST) <i>k</i>	10 .0 m	0.11 m ³ /s	Control device	87%
06	[10]	—	<i>k-ε</i>	—	—	—	71%
07	[32]	6.50 kW	SST	4.3 m	0.25 m ³ /s	Guide vane	76.04%
08	[9]	—	<i>k-ε</i>	—	—	—	82%
09	[33]	0.26 kW	SST	—	—	Internal deflector	81%
10	[34]	0.51 kW	SST	2.9 m	—	—	65.7%
11	[35]	—	<i>k-ωT</i>	1.2 m	0.23 m ³ /s	—	58%
12	[12]	—	RNG	15.0 m	0.95 m ³ /s	—	80%
13	[36]	—	SST	20.0 m	0.53 m ³ /s	Guide vane	80%
14	[13]	12.54 kW	<i>k-ε</i>	10.0 m	0.15 m ³ /s	Guide vane	86%

from the reported experimental global efficiency by 15%. In this investigation, a two-dimensional approach was selected. The obtained results, which deviated from the experimental values, agree with those reported for three-dimensional computation for the same CFT.

Hence, it is recommended that the two-dimensional approach could be used whenever a three-dimensional computation is not feasible due to hardware and time requirements. Moreover, the analysis suggests that steady-state conditions are not recommended for CFT studies as the results in terms of flow behavior are not as accurate and representative of real CFT like those obtained from transient state computations.

Crossflow turbine design for variable operating conditions studies was conducted by CFD code. A 2D simulation of the CFX-ANSYS code was used to get the maximum efficiency very close to the design flow. In the study, the flow rate can be maintained when Q is reduced to 20% of its maximum by using the flow control mechanism [6].

A study of flow and performance features of a direct drive crossflow turbine for wave power generation using commercial CFD code ANSYS-CFX for simulation showed good agreement with the experimental data at the wave period of 2 s and a 3% difference in the results was noted [38]. Moreover, the power of the turbine was high at $T = 3$ s for all turbine speeds. From the study, it is noted that the efficiency increases with an increased turbine speed, and then it reaches a maximum and then decreases (Figure 4).

Experimental and numerical investigations (using CFD) of a crossflow turbine [17] concluded that the experimental efficiencies showed a good agreement with their numerical approximation. Furthermore, the impeller shaft provides a significant reduction in turbine efficiency, for a velocity ratio close to the optimal value ($Vt/U = 1.8-2.0$), and an impeller with a shaft has 5% lower efficiency than that without a shaft. The efficiency curves tend to assume the same value for the two cases by reducing the velocity ratio. The efficiency is about 75% for a velocity ratio Vt/U close to 1.2. In addition,

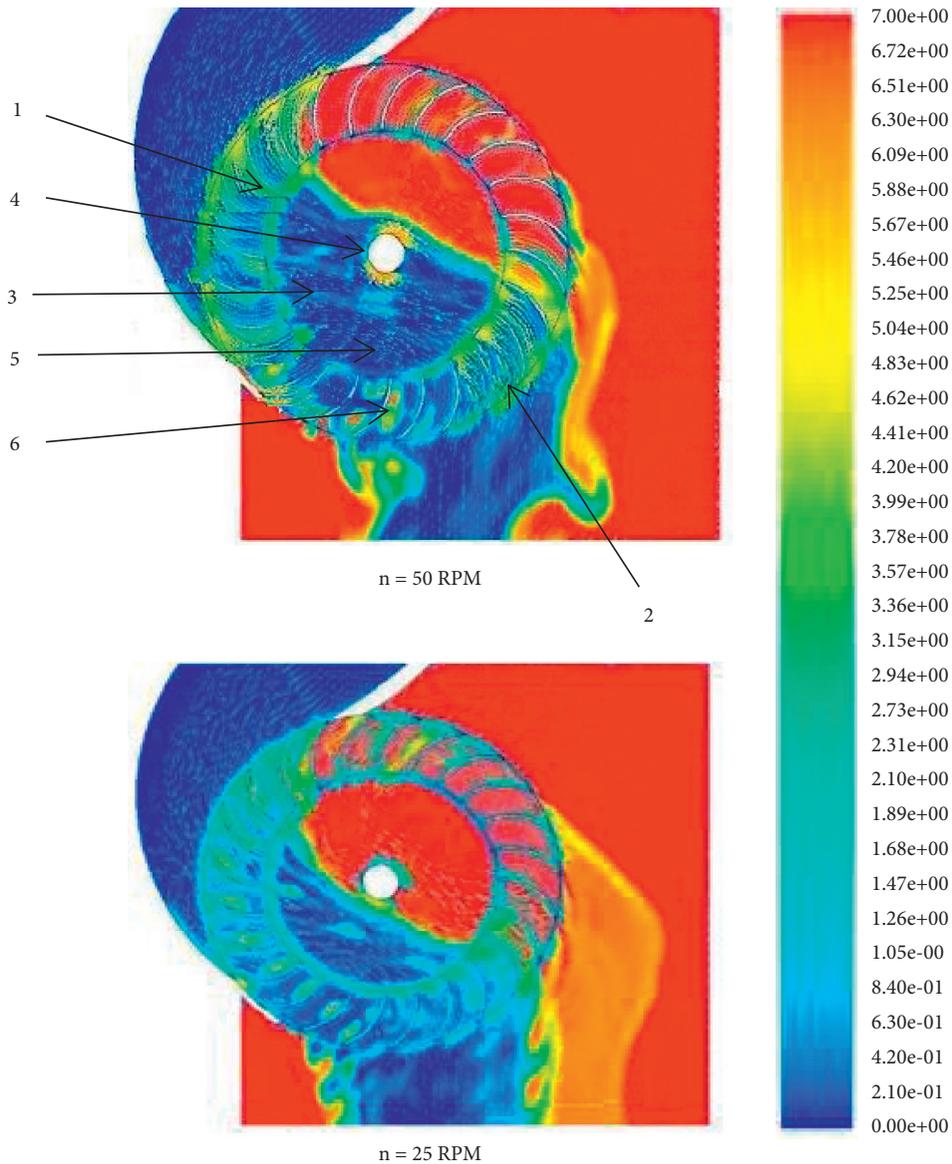


FIGURE 4: Comparison between numerical hydraulic efficiency and experimental global efficiency with runner speed [10] (copyright (2011), with permission under Creative Commons Attribution 4.0 International License).

from the study it was observed that a maximum efficiency very close to the design flow rate was attained when Q was reduced to 20% of its maximum. 2D numerical simulations were carried out employing ANSYS CFX code. The $k-\epsilon$ turbulence model was selected using ANSYS CFX code.

[39] conducted a numerical investigation of the flow profile and performance of a low-cost crossflow turbine. ANSYS CFX with $k-\epsilon$ turbulence model was employed in the simulation process. It was observed that the numerical study obtained the flow profiles that had a favorable correlation with the actual flow pictures. Additionally, the study concluded that the numerically determined performance results compare favorably with the experimental results.

Generally, these reviews have not only demonstrated the validity of CFD models but also pointed out the prospects of using CFD for designing and optimizing a CFT. Therefore,

the findings suggest that it is plausible that numerical modeling with CFD is a powerful alternative to experimental designs because it can avoid some limitations and provide detailed information on the relevant flow variables in the field data, under well controlled conditions and without similarity constraints. However, the accuracy of CFD is an important matter of concern. Therefore, adequate care is required in the implementation of the model, in grid generation, and in selecting proper solution strategies.

CFD simulations were used to analyze the internal flow in a hydraulic crossflow turbine using a nozzle, runner, shaft, and casing. Analysis of the velocity and pressure fields of the crossflow within the runner and characterization of its performance for different runner speeds was conducted. In the study, ANSYS CFX code with water-air free surface and $k-\epsilon$ turbulence model was used. The simulation results were

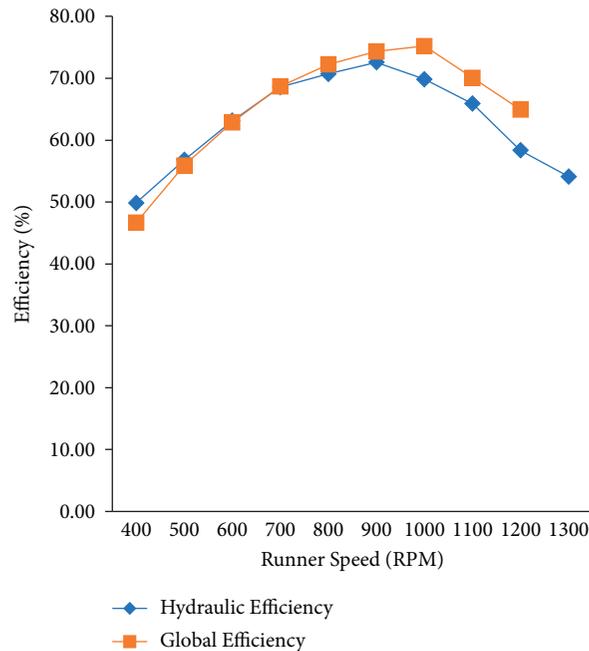


FIGURE 5: Air-water volume fraction and velocity fields inside the Banki turbine at two speeds: (a) $n = 50$ rpm and (b) $n = 125$ rpm. The background color indicates the air-water volume fraction (blue for water, red for air). The legend shows velocity values [15] (copyright (2011), with permission under Creative Commons Attribution 4.0 International License).

related to the experimental data and showed that they were consistent with the global performance parameters. However, a typical maximum relative error of 10% between the computational fluid dynamics (CFD) and experimental results has been reported [10]. These results suggest that, despite the observed error of 10%, there are high chances that due to its efficacy in terms of time and resources it might be more feasible.

An experiment and numerical investigation of a CFT, operating far away from design point, was studied by [15]. The experimental results suggested that even for operating at heads and discharges that go below 5% and 30% the turbine could still reach a peak efficiency of more than 55%, although at the expense of a significant reduction in speed. To develop an understanding of the flow through the turbine, arithmetical simulations were carried out by commercial code ANSYS Fluent. The two-phase flow of water and air was considered unsteady and turbulent. The study showed that, above 175 rpm, there is a good agreement between the moment values obtained experimentally and numerically. The study concluded that experimental results and those of numerical simulations are in good agreement, but it was not stated to what extent.

4. Advances in the Optimization of CFTs

Aeroplane industry was the first to use CFD applications which gave results relevant for conceptual studies of new designs, full product development, troubleshooting, and redesign by, among others, reducing the total effort and cost required for experimentation [40]. Since then, as a result of the continuous advancement of CFD tools and exponentially

increased computation capabilities along with better understandings of the underlying physics, CFD simulations have increasingly been applied widely in diverse areas including process safety and loss prevention in processing industries, and improved efficiency of turbines. CFD were extensively used [41] for power plants simulations such as thermal patterns of boiler furnaces and turbine blading performance as well as heat exchangers designs. Moreover, it can be inferred from the research papers that CFD, among others, reduces the total effort and cost required for experimentation and thus is of great benefit in improving efficiency, rapid development, and enabling broad applications.

Being a computer-based mathematical modeling tool, the application of CFD incorporates the solution of the fundamental equations of fluid flow, the Reynolds-Averaged Navier-Stokes equations, using computerized software and turbulence models to work out the averaged turbulence stresses. Commonly used CFD software for design and optimization of CFTs includes ANSYS CFX, ANSYS Fluent, CFD2000, STAR CCM+, FLUENT, STAR-CD, and PHOENICS. Moreover, in recent studies [8, 9, 18–22], ANSYS CFX software has been used to evaluate the small efficiency of CFTs and has shown to be a primary inspiration to improve its existing design since these turbines operate with ambient air pressure on the free surface [8]. Nowadays, CFD simulation codes are regarded as an industry standard for this process [42].

Application of CFD simulations in the optimization of CFTs provides detailed information on manipulation of the geometry characteristics, and other parameters in optimization of CFTs. It also offers a relatively economical and safe

approach that enables design modification of CFTs and is of great benefit to improving efficiency of CFTs. Additionally, CFD simulations help to evaluate and optimize the effectiveness of various mitigation methods, especially those which are too costly or dangerous for which to conduct experiments. This review has identified geometry optimization and flow features as two important categorical attributes in the design and optimization of CFTs.

4.1. Analysis of Nozzle Parameters. Various studies [1, 9, 10, 12, 13, 15, 29, 32, 35, 36, 38, 43] have used CFD for optimizing these parameters in order to influence the improvement of the performance of crossflow turbines. According to the studies, ANSYS CFX software has been highly used to perform simulations and analysis of turbines compared to other CFD software. This is probably due to its high-level potential for general purpose in CFD analysis. Similarly, a large number of turbulence models, which are available in the ANSYS CFX software and have been used with specific applications to the broader class of flows for a reasonable degree of confidence, include standard $k-\epsilon$, RNG $k-\epsilon$, standard $k-\omega$, SMC- ω , and SST $k-\omega$. In CFTs, CFD is found to have been used for optimizing the shapes of the nozzles in many studies [18–20]. Global performance parameters were presented for different operating conditions. [44], and [45] have investigated through 2D numerical calculations the unsteady water flow inside the runner, paying attention to the flow along the runner entrance and unsteady forces on the blades. On the other hand, Choi et al. performed an entire 2D-CFD steady-state crossflow turbine simulation, considering water and water-air flow conditions. With this approach, the authors studied the influence of nozzle shape, runner blade angle, and runner blade number on the turbine performance. Moreover, the important role of the air layer on the numerical calculation was verified. It was also studied [18] that the presence of an air layer in the runner passage improves the turbine's performance by preventing the collision loss between the flow and shaft, and eliminating the recirculation flow in the passage. One of the studies [9] has also implemented a Multiobjective Genetic Algorithm and a Metamodel-Assisted Optimization to optimize the shape of the valve in crossflow turbines and found improvement in the output power by 4.73% and 5.33%, respectively.

The effects of nozzle inclination on the performance of crossflow hydro turbine were investigated by [14, 24, 46–48]. The inclination angle of 16° was found to provide maximum efficiency between 80 and 82% ([24, 46]). Nozzle inclination of 24° gave a maximum efficiency of 89% by [47]. [14] found efficiency of 88% at an inclination of 22° . Also, [48] reported an efficiency to be maximum at an inclination of 22° and got 90%.

Using a base model, numerical analysis and performance improvement of a crossflow hydro turbine enabled design modification by optimizing the nozzle shape and guide vane angle, and changing the number of blades. A CFD code ANSYS CFX 13.0, two-phase air and water at 25°C with steady state and SST turbulence model was selected. From

the study, the best efficiency found from the base nozzle was 63.67%, which was geometrically altered and improved the turbine performance and efficiency up to 76.60 [29]. This elucidates that the efficiency increased by 12.93%. Velocity vectors of the base model and the modified model are shown in Figure 6.

Similarly, a numerical study on the effect of guide nozzle shape on the performance improvement of low head crossflow turbine showed an increase in efficiency by 12.5% to 76.04% [13]. From this study, it is depicted that the appropriate guide nozzle radius and nozzle angles play an essential role on the relative angle that should be close to the inlet angle of the blade. However, the influence of the guide nozzle on the perfect angle at the outlet of stage 2 (Figure 1) decreased. Moreover, it is observed that the guide nozzle plays a role of suppressing the negative torque by reducing the pressure difference on the blades.

Design modification was done by optimizing the nozzle shape and guide vane angle and changing the number of blades using CFD simulation code ANSYS CFX 13.0, whereby two-phase air and water at 25°C with steady state and SST turbulence model was selected. From the study, the best efficiency found from the base nozzle was 63.67%, which was geometrically altered and improved the turbine performance and efficiency up to 76.60 as shown in the velocity vectors of the base model and the modified model in Figure 6. These findings elucidate that it is plausible that CFD facilitates efficient design for improved performance and gains of CFT.

Pressure and velocity profiles along with nozzle outlet, energy transfer stage location, and CFT reaction values were addressed (Figure 7). The attained results were compared in terms of the runner-nozzle interface (gap vs no gap), turbulence model (SST vs $k-\epsilon$), and calculation regime (steady vs transient simulation). The study revealed that the calculation state (steady vs transient simulation) was found to have a significant influence over the results. Transient state calculations presented a better complex flow inside the CFT. The 3D simulations carried out for the designed optimal configuration suggest an optimal value of the ratio between the width of the impeller and the nozzle equal to one; this validates the adopted 2D approach but disagrees with previous literature results [14]. Figures 7(a) and 7(b) show water volume fraction and pressure contours. According to [49], the average efficiency of the turbine was greater than 80% for a value of V_t/U varying between 1.2 and 3.0 (corresponding to a water discharge varying between 35 l/s and 90 l/s). The distributions of the water volume fraction, the velocity field, and the pressure in the simulated sub-domains for the optimum configuration are shown.

A new methodology for designing nozzles without a guide vane based on the conversion of H at the nozzle inlet into kinetic energy at the runner was presented [41]. The same principle governs Pelton nozzle design for high-efficiency CFTs. 3D Reynolds-Averaged Navier-Stokes simulations uses SST $k-\omega$ turbulence model and a 2-phase homogeneous free-surface flow model. Using numerical simulations, an analytical model was formulated to convert the head into kinetic energy at the entry to obtain an

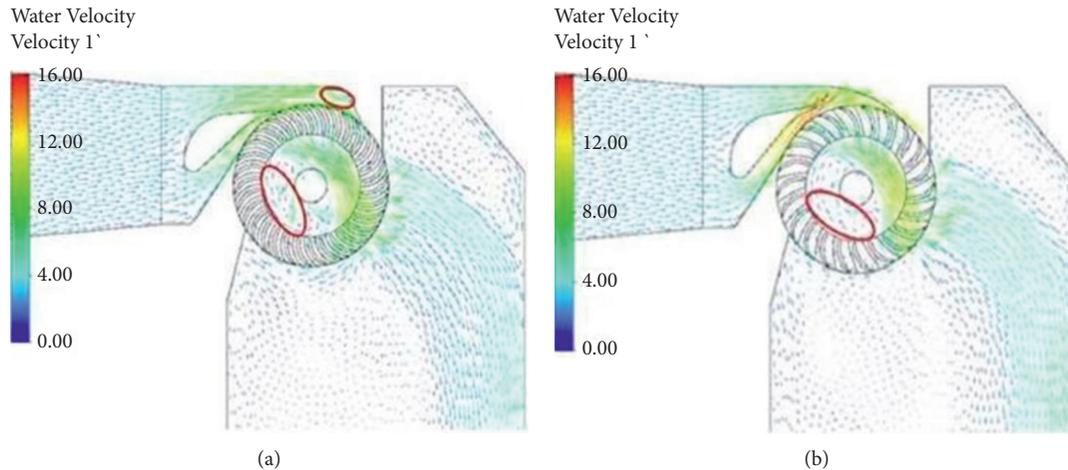


FIGURE 6: Velocity vectors of (a) base model and (b) modified model [29] (copyright (2014), with permission under Creative Commons Attribution 4.0 International License).

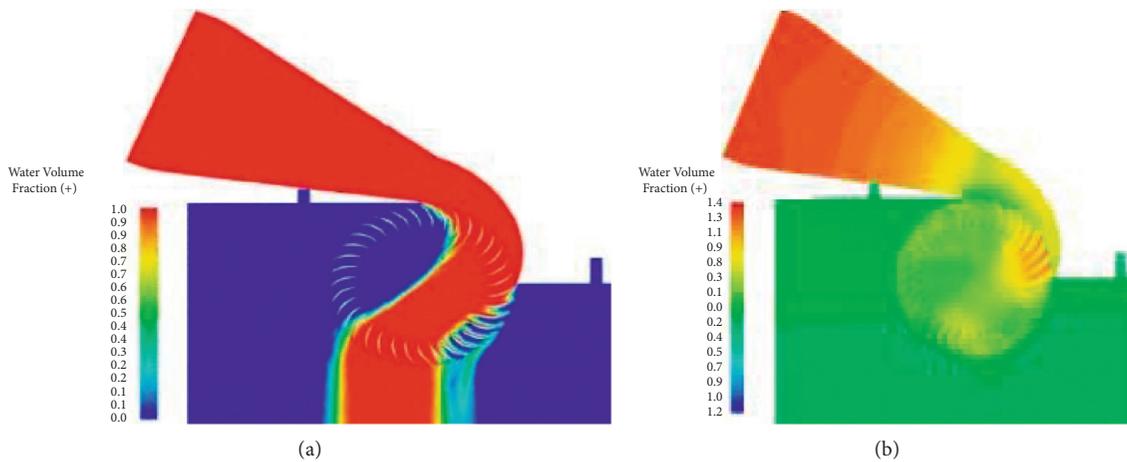


FIGURE 7: (a) Water volume fraction. (b) Pressure contours [49] (copyright (2013), with permission under Creative Commons Attribution 4.0, International License).

appropriate flow angle. Three-dimensional Reynolds-Averaged Navier-Stokes simulations were conducted on a 7 kW turbine with 69% efficiency and another 0.53 kW turbine with 88% efficiency. The efficiency increased up to 91%. This was done by redesigning the nozzle of 7 kW turbines through this methodology [4].

The authors of [31] presented a new methodology for designing nozzles without a guide vane based on the conversion of (H) at the nozzle inlet into kinetic energy at the runner, which is the same principle that governs Pelton nozzle design so that a high-efficiency crossflow turbine can be designed. The study was conducted using CFD with 3D Reynolds-Averaged Navier-Stokes simulations, SST $k-\omega$ turbulence model, and a two-phase homogeneous free-surface flow model. An analytical model was formulated to convert the head into kinetic energy at the entry and obtain an appropriate flow angle. Three-dimensional Reynolds-Averaged Navier-Stokes simulations conducted on a 7 kW turbine measured the efficiency of 69% and 0.53 kW turbine with a maximum efficiency of 88%. By only redesigning the

nozzle of the 7 kW turbine by the new methodology, the maximum efficiency increased from 69% to 91%.

A study conducted by [24, 46, 47] on the nozzle angle found that changing the nozzle configurations such as rear-wall shape and nozzle orientation by keeping the same runner design entry arc angle of 90 degrees was suitable to give maximum efficiency of 69%.

4.2. Analysis of Runner Parameters

4.2.1. Blades. The computational simulation that studied the variation of power depending on the number of blades in the runner was conducted, whereby ANSYS CFX V17.0 homogeneous multiphase fluid (water-air) with $k-\epsilon$ turbulence model was applied. The study discovered that there was an increase in power generated by 3.2% when the number of blades in the runner increased from 16 to 28. The maximum power output attained was 12.54 kW, with an efficiency of 86% [9]. From the study, it was concluded that CFD analysis

applied in the investigation showed results with high accuracy compared to real experiments, thus indicating the results of fluid computational analysis to be reliable.

The impact of the number of blades on the power generated by a Michell-Banki turbine was studied by [13]. Computational fluid dynamics ANSYS CFX v17.0 code was employed as a solver. Homogeneous turbulence model was used for the multiphase fluid (water-air) with $k-\varepsilon$ turbulence model for both phases implemented. The authors found that there was an increment of 3.2% in the power generated by the turbine when the number of blades in the runner changed from 16 to 28. The maximum output power of 12.54 kW with an efficiency of 86% was obtained. From the study, it was concluded that the CFD analysis applied in the investigation showed results with high accuracy compared with real experiments, which allows the results of the fluids computational analysis to be reliable.

Crossflow turbine is a possible choice for the exploitation of small hydropower sources because of their simplicity and their excellent efficiency, but a complete step-by-step procedure for their design was missing to the authors knowledge [49]. A CFD ANSYS CFX code was used to analyze some design parameters on the influence of inefficiency. A turbine with 35 blades and an attack angle equal to 22° exhibited at the design point a high efficiency equal to 86%. 2D CFD analyses showed that the linear nozzle ensures an almost constant value of the angle of attack and confirms the simplifying hypothesis of negligible energy dissipation inside the impeller, which is a cornerstone of the theoretical approach [49]. From the study, the number of blades and diameter ratio had little influence on the peak efficiency, while the presence of the shaft did not affect the characteristics curve but led to a considerable reduction in efficiency.

Investigation of the effects of several blades on the performance of a crossflow turbine using Star CCM+ was done by [2]. The study used CFD Star CCM+ simulation code combined with CATIA V5 computer-aided design software to improve the operation of the turbine, according to the analysis result to establish the optimal number of the runner blades for the chosen turbine. The internal flow simulation is used to characterize the turbine performance for a different number of blades. Six different turbine runners of 15, 20, 25, 30, 35, and 40 blades were investigated, and the study showed that 30-blade runner was the most effective. The prediction results revealed key and exciting details of the structure of the flow in the runner of the crossflow turbine by using the different number of blades. Such details revelations are not possible with conventional flow visualization or other techniques.

4.2.2. Rotor. Numerical simulations have been carried out imposing a constant rotor rotational speed, $n = 245$ rpm, in all the simulations [50]. Hence, in order to guarantee the desired U_{tip}/V^* ratio, the head difference between the up- and downstream ponds, H_t , has been opportunely assigned. Four simulations have been performed considering the following flow coefficients: $U^* = 0.165$, $U^* = 0.200$, $U^* = 0.25$,

and $U^* = 0.293$, or in terms of U_{tip}/V^* ratios: 5.6, 4.6, 3.7, and 3.2. The solutions have been initialized by imposing uniform flow conditions computed from the inlet boundary. Three blades were observed to behave the same way and the rotor had not only a geometric but also a fluid dynamic periodicity reported in terms of pressure drop coefficient, Δp^* , nondimensional torque*, and efficiency, η vs. flow coefficient, U^* .

4.2.3. Flow Features. CFD techniques have proven to be versatile tools in evaluating the hydraulic performance of hydro turbines [50]. In demand to improve their efficiency, commercial CFD simulation code has continued to be used to study and determine parameters and phenomena that affect their performance [50].

A study was conducted on flow features in crossflow turbine T15 300 by CFD deployed Fluent 14.5 with the standard $k-\varepsilon$ turbulence model as observed by Legonda [30] and a comparison of the values of efficiency extracted from an experimental performance test reported by Entec [51] on the guide vane angle against the total pressure shown in Figure 8. According to the comparison, there exists sensitivity to the guide vane angle and shaft diameter on the overall performance of the turbine.

The study highlights a strong correlation between efficiency and pressure in the crossflow turbine. From the study, it is suggested that the model prediction is in good agreement with the experimental data. However, the result does not mention to what percentage.

The results in Table 2 show the use of CFD in the design of CFT targeting variable power and using diverse head, flow rates, and flow control mechanisms. In the results, the efficiencies obtained are a minimum of 58% and a maximum of 86%. The mean efficiency is 75.5% and considering the frequency of the efficiencies at the range of 58.0–68.0% three researchers obtained this efficiency [15, 34, 35], three researchers obtained 68–78.0% [10], and [29, 32] and seven researchers obtained $78 - \geq 88\%$ [1, 12, 13, 30, 31, 36]. Moreover, the results show that the highest efficiencies of ≥ 82 are during use of CFD software and ANSYS CFX code with turbulence model $k-\varepsilon$, SST. These results explain that the application of CFD software and associated turbulence models is effective in designing improved CFTs for development of renewable energy technologies. This is supported by [42], that CFD simulation codes are an industry standard for design of turbines.

Numerical computation of flow through SSH-300/150 and TP-300/300 turbines was done to determine the fields of pressure and velocity, and the final specification of their efficiency in different points of operation. The analysis was performed using the CFD fluent 5.0 code based on the finite volume method with RHG $k-\varepsilon$ model. The analysis of the flow A through SSH-300/150 model turbine showed that the runner operates in a pumping regime on the part of its perimeter [12]. This result is confirmed by an experimental test which was conducted in a laboratory test rig. Furthermore, the computation result showed the deadly flow field in the internal part of the runner. To increase the

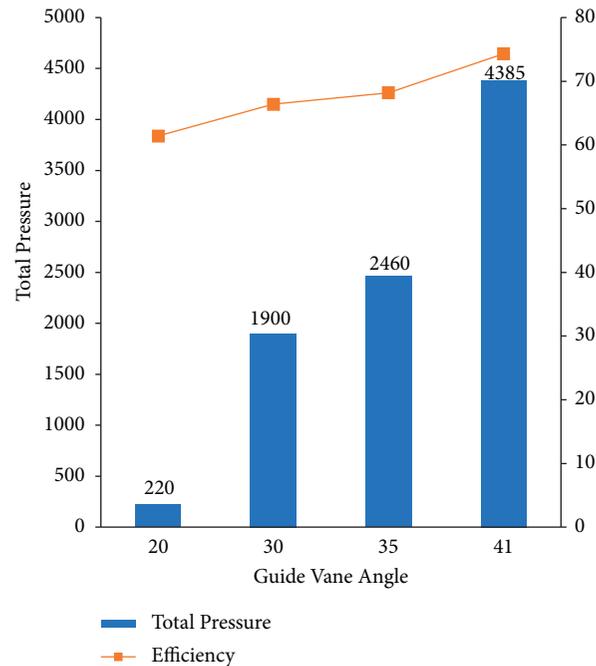


FIGURE 8: Comparison between model and experimental data [30] (copyright (2016), with permission under Creative Commons Attribution 4.0 International License).

significance of the numerical results, the computations conducted on efficiency showed that the experimental efficiency was more than 80% while that of CFD was less than 80%. The study concluded that the newly designed TPP-300/300 crossflow turbine with a draft tube fulfills an essential function of utilizing the difference in specific energy between the runner bottom edge and the tail water surface. Therefore, the numerical computations performed on the new design show that the properly designed draft tube reduces some undesirable phenomena like backflows and separation.

Figure 9 shows the results of a study on cavitation inception in CFT using 3D Reynolds-Averaged Navier-Stokes (RANS) computations with a homogeneous free-surface two-phase flow model [31]. Pressure distribution on the blades was examined for different rates, heads, and impeller speed. The aim was to assess cavitation inception. In this study, cavitation occurred in the second stage of the turbine and was observed on the suction side near the inner edge of the blades [1]. The study suggests that cavitation may be an essential consideration for CFTs but only if they are poorly designed to operate past the maximum efficiency point in terms of Q .

Furthermore, the study suggests that further experimental and numerical investigations are needed to test the generality of the specific conclusion reached from simulations done on the low-efficiency turbine. Figure 10 shows the velocity contour in the crossflow hydro turbine.

[34] considered performance improvement of crossflow hydro turbine by air layer effect. CFD analysis on the performance and internal flow of the turbine was conducted with the unsteady state using a two-phase flow model in order to embody the air layer effect on the turbine performance effectively. The result showed that

the air layer effect on the performance of the turbine is considerable. The air layer located in the turbine runner passage plays a role of preventing a shock loss at the runner axis and suppressing a recirculation flow in the runner.

Moreover, the ratio between air from suction pipe and water from the turbine inlet is also a significant factor in the turbine performance. Commercial CFD code of ANSYS-CFX was adopted, and the study showed that difference of efficiency ratio between experiment and CFD was 1.4%. This difference between results by experiment and CFD analysis was due to the difference in mechanical loss because the amount of mechanical loss predicted by CFD analysis is usually smaller than that obtained by experiment.

CFD tools were used in designing internal deflector for CFT efficiency improvement by [33]. Commercial CFD code ANSYS CFX v.13 was used to carry out all simulations. A transient regime, two-dimensional, two-phase (air-water) homogeneous model with a Eulerian-Eulerian approach was selected, and the SST turbulence model was preferred. The results obtained concluded that performance of a CFD model agreed with experimental data. Maximum efficiency deviation from the experiment was around 5% at three evaluated specific speeds. The investigation also reveals that the overall use of internal deflector increases CFT performance and output power. [35] studied the flow control in Banki turbines by using ANSYS Fluent code with $k-\omega$ SST turbulence model for numerical analysis. The study showed that experimental and numerical simulation results were reasonably good. Based on the CFD fluent code analysis, the velocity fields obtained by numerical simulations which are responsible for the decrease in turbine efficiency were identified.

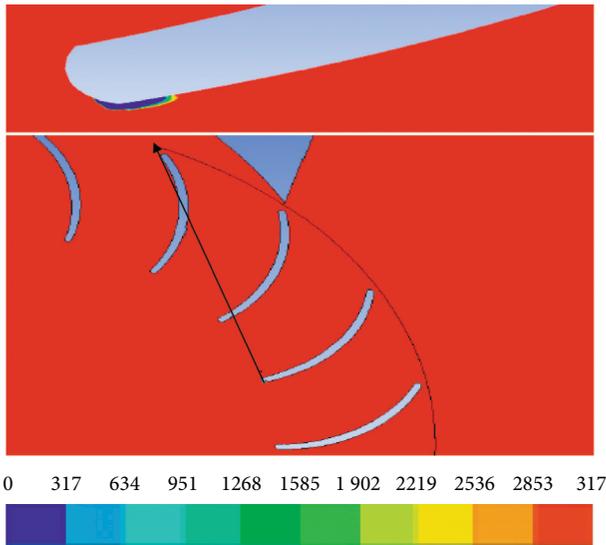


FIGURE 9: Contours of pressure disseminations on the impeller blades at the second stage ($Q = 105$ lps and $H = 10$ m) at the impeller speed 550 RPM, presenting the indication of cavitation inception on the suction side near the inner edge of the blade [31] (copyright (2017), with permission under Creative Commons Attribution 4.0 International License).

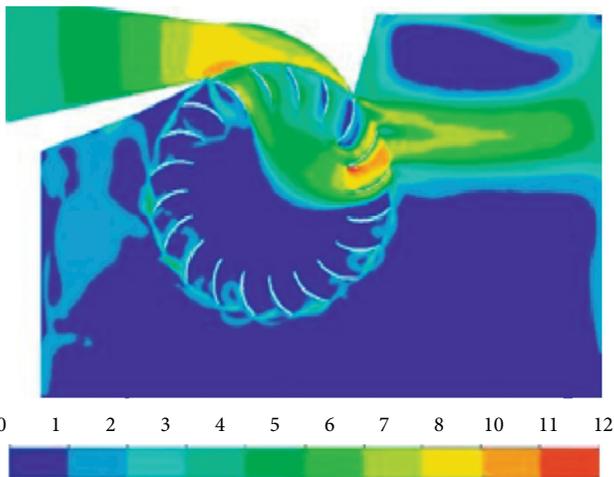


FIGURE 10: A typical velocity contour showing the main flow path of water in the turbine [1] (copyright (2016), with permission under Creative Commons Attribution 4.0 International License).

5. Conclusion

A review of the computational fluid dynamics applications in the design and optimization of crossflow hydro turbines has demonstrated possible success in nonconventional solutions. CFD tools have shown to reduce cost and provide quick solutions to the design of, among others, nozzle and runner parameters of CFTs. In particular, the turbulence model SST $k-\omega$ and standard $k-\epsilon$ have widely been used because of low cost and short run. Other models such as DNS and LES are not useful due to the high cost and extended run time. Moreover, the results from CFD are in good agreement with the experimental studies ranging from

2% to 10% but in some studies vary up to 15%. For a case of significant deviation, which is very rare, it has been envisaged that a specific design checkup may become necessary.

This study is still ongoing, but it has provided a possible orientation that CFDs are essential tools for both designing CFTs and validating experimental results. They make different turbine runners and flow conditions more understandable. However, the use of CFDs requires a more sensitive and detailed analysis of the interaction between the flow and the boundary for different flow conditions using innovative tools associated with real setup limitations. Since CFD commercial codes are easily accessible and can fulfill the requirements of the design of crossflow hydro turbines, this code will be used to complement the ongoing studies on specific analyses of flow behavior in the design of CFT for rural electrification in Tanzania.

Abbreviations

CFD:	Computational fluid dynamics
CFHT:	Crossflow hydro turbine
$k-\epsilon$:	Kapa epsilon
$k-\omega$:	Kappa omega
RANS:	Reynolds-Averaged Navier-Stokes
RNG:	Renormalization group
RNG $k-\epsilon$	Normalization group (RNG) k -epsilon
Re:	
RSM:	Reynolds stress model
SIMPLEC:	Semi-implicit method for pressure linked equations-consistent
SMC- ω :	Second moment closure
SST $k-\omega$:	Shear stress transport URANS unsteady Reynolds
H :	Head
ω :	Speed
Q :	Flow rate
R1:	Outer radius
R2:	Inner radius
N :	Speed (rev/min).

Data Availability

This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

Conflicts of Interest

The authors declare that they have no conflicts of interest.

References

- [1] R. Adhikari, *Design Improvement of Crossflow Hydro Turbine*, University of Calgary, Calgary, Canada, 2016.
- [2] S. R. Yassen, "Investigation of the effects of number of blades on the performance of cross-flow turbine using STAR CCM+," *Polytechnic Journal*, vol. 7, no. 4, 2017.
- [3] N. Gourdain, "Large eddy simulation of flows in industrial compressors: a path from 2015 to 2035," *Philosophical*

- Transactions of the Royal society A: Mathematical, Physical and Engineering Sciences* 2014, vol. 372, p. 20130323, 2022.
- [4] R. Adhikari and D. Wood, "The design of high efficiency crossflow hydro turbines: a review and extension," *Energies*, vol. 11, no. 2, p. 267, 2018.
 - [5] A. G. M. Michell, *Impulse-Turbine*, Google Patents, Alexandria, VA, USA, 1904.
 - [6] M. Sinagra, "Cross-Flow turbine design for variable operating conditions," *Procedia Engineering*, vol. 70, pp. 1539–1548, 2014.
 - [7] A. Elbatran, "Operation, performance and economic analysis of low head micro-hydropower turbines for rural and remote areas: a review," *Renewable and Sustainable Energy Reviews*, vol. 43, pp. 40–50, 2015.
 - [8] F. M. White and I. Corfield, *Viscous Fluid Flow*, McGraw-Hill, New York, NY, USA, 2006.
 - [9] V. Sammartano, "Numerical and experimental investigation of a cross-flow water turbine," *Journal of Hydraulic Research*, vol. 54, no. 3, pp. 321–331, 2016.
 - [10] J. De Andrade, "Numerical investigation of the internal flow in a Banki turbine," *International Journal of Rotating Machinery*, vol. 2011, 2011.
 - [11] V. Brunet, "Comparison of various CFD codes for LES simulations of turbomachinery: from inviscid vortex convection to multi-stage compressor," in *Proceedings of the ASME Turbo Expo 2018: Turbomachinery Technical Conference and Exposition*, American Society of Mechanical Engineers Digital Collection, Oslo, Norway, June 2018.
 - [12] M. Kaniecki, "Modernization of the outflow system of cross-flow turbines," *TASK Quarterly*, vol. 6, no. 4, pp. 601–608, 2002.
 - [13] Y. C. Ceballos, "Influence of the number of blades in the power generated by a Michell Banki turbine," *International Journal Of Renewable Energy Research IJRER*, vol. 7, no. 4, 2017.
 - [14] V. Desai and N. Aziz, "Parametric evaluation of cross-flow turbine performance," *Journal of Energy Engineering*, vol. 120, no. 1, pp. 17–34, 1994.
 - [15] A. Dragomirescu and M. Schiaua, "Experimental and numerical investigation of a Bánki turbine operating far away from design point," *Energy Procedia*, vol. 112, pp. 43–50, 2017.
 - [16] N. Gourdain, "Prediction of the unsteady turbulent flow in an axial compressor stage. Part 1: comparison of unsteady RANS and LES with experiments," *Computers & Fluids*, vol. 106, pp. 119–129, 2015.
 - [17] W. Johnson, "Design and Testing of an Inexpensive Crossflow Turbine," *ASME Small Hydro-Power Fluid Machinery*, p. 129, 1982.
 - [18] W. L. Oberkampf and T. G. Trucano, "Verification and validation in computational fluid dynamics," *Progress in aerospace sciences* 2002, vol. 38, no. 3, pp. 209–272.
 - [19] H. Olgun, "Investigation of the performance of a cross-flow turbine," *International journal of energy research* 1998, vol. 22, no. 11, pp. 953–964.
 - [20] R. F. Ott and J. R. Chappell, "Design and efficiency testing of a cross-flow turbine," *Waterpower '89*, 1989, ASCE, Reston, VA, USA.
 - [21] M. Tiwari and R. Shrestha, "Effect of variation of design parameters on cross flow turbine efficiency using ANSYS," *Journal of the Institute of Engineering*, vol. 13, no. 1, pp. 1–9, 2013.
 - [22] F. Stern, "Comprehensive approaches to verification and validation of CFD simulations—part 1: methodology and procedures," *Journal of Fluids Engineering*, vol. 123, no. 4, pp. 793–802, 2001.
 - [23] V. R. Desai and N. M. Aziz, "An experimental investigation of cross-flow turbine efficiency," *Energy Procedia*, vol. 141, 1994.
 - [24] Y. Nakase, "A study of cross-flow turbine (effects of nozzle shape on its performance)," in *Proceedings of the ASME 103rd Winter Annual Meeting*, New York, NY, USA, May 1982.
 - [25] J. Denton and W. Dawes, "Computational fluid dynamics for turbomachinery design," *Proceedings of the Institution of Mechanical Engineers - Part C: Journal of Mechanical Engineering Science*, vol. 213, no. 2, pp. 107–124.
 - [26] P. Tucker, "Trends in turbomachinery turbulence treatments," *Progress in Aerospace Sciences*, vol. 63, pp. 1–32, 2013.
 - [27] C. Cornelius, "Experimental and computational analysis of a multistage axial compressor including stall prediction by steady and transient CFD methods," *Journal of Turbomachinery*, vol. 136, no. 6, 2014.
 - [28] A. Dakers and G. Martin, "Development of a simple cross-flow water turbine for rural use," in *Agricultural Engineering Conference 1982: Resources, Efficient Use and Conservation; Preprints of Papers* Institution of Engineers, Australia, 1982.
 - [29] N. Acharya, "Numerical analysis and performance enhancement of a cross-flow hydro turbine," *Renewable Energy*, vol. 80, pp. 819–826, 2015.
 - [30] I. A. Legonda, "An investigation on the flow characteristics in the cross-flow turbine-T15 300," *Journal of Power Energy Engineering* 2016, vol. 4, no. 9, pp. 52–60.
 - [31] R. Adhikari and D. Wood, "A new nozzle design methodology for high efficiency crossflow hydro turbines," *Energy for Sustainable Development*, vol. 41, pp. 139–148, 2017.
 - [32] Z. Chen, P. M. Singh, and Y.-D. Choi, "Effect of guide nozzle shape on the performance improvement of a very low head cross flow turbine," *한국유체기계학회논문집*, vol. 17, no. 5, pp. 19–26, 2014.
 - [33] S. D. Croquer, "Numerical investigation of a Banki turbine in transient state: reaction ratio determination," in *Proceedings of the ASME Turbo Expo 2018: Turbomachinery Technical Conference and Exposition*, Oslo, Norway, June 2012.
 - [34] Y.-D. Choi, "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*, vol. 3, no. 3, pp. 398–409, 2008.
 - [35] D. Popescu, C. Popescu, and A. Dragomirescu, "Flow control in Banki turbines," *Energy Procedia*, vol. 136, pp. 424–429, 2017.
 - [36] Z. Chen and Y.-D. Choi, "Performance and internal flow characteristics of a cross-flow turbine by guide vane angle," in *Proceedings of the IOP Conference Series: Materials Science and Engineering*, IOP Publishing, 2013.
 - [37] D. D. Prasad, M. R. Ahmed, and Y.-H. Lee, "Flow and performance characteristics of a direct drive turbine for wave power generation," *Energies*, vol. 81, pp. 39–49, 2014.
 - [38] S. D. Croquer, "Use of CFD tools in internal deflector design for cross flow turbine efficiency improvement," in *Proceedings of the ASME 2012 Fluids Engineering Division Summer Meeting Collocated with the ASME 2012 Heat Transfer Summer Conference and the ASME 2012 10th International Conference on Nanochannels, Microchannels, and Minichannels*, American Society of Mechanical Engineers Digital Collection, Rio Grande, PR, USA, July 2012.
 - [39] C. S. Kaunda, C. Z. Kimambo, and T. K. Nielsen, "A numerical investigation of flow profile and performance of a low cost Crossflow turbine," *International Journal of Energy and Environment*, vol. 5, no. 3, 2014.

- [40] C. Ansys, *ANSYS CFX-Solver Theory Guide*, vol. 15317, pp. 724–746, ANSYS CFX Release, Canonsburg, PA, USA, 2009.
- [41] E. E. Khalil, “CFD history and applications,” *CFD Letters*, vol. 4, no. 2, pp. 43–46, 2012.
- [42] J. Carregal-Ferreira, *Advanced CFD Analysis of Aerodynamics Using CFX*, pp. 1–14, Technology GmbH, Otterfing, Germany, 2002.
- [43] Y.-D. Choi, “Performance and internal flow characteristics of a cross-flow hydro turbine by the shapes of nozzle and runner blade,” *Journal of fluid science*, vol. 3, no. 3, pp. 398–409, 2008.
- [44] J. FUKUTOMI, Y. SENOO, and Y. NAKASE, “A numerical method of flow through a cross-flow runner,” *JSME international journal. Ser. 2, Fluids engineering, heat transfer, power, combustion, thermophysical properties*, vol. 34, no. 1, pp. 44–51, 1991.
- [45] J. Fukutomi, “Unsteady fluid forces on a blade in a cross-flow turbine,” *JSME International Journal - Series B: Fluids and Thermal Engineering*, vol. 38, no. 3, pp. 404–410, 1995.
- [46] S. Khosrowpanah, M. Albertson, and A. Fiuzat, “Historical overview of cross-flow turbine,” *Water PowerDam Construction*, vol. 36, no. 10, pp. 38–43, 1984.
- [47] A. A. Fiuzat and B. Akerkar, “The use of interior guide tube in cross flow turbines,” *Waterpower*, vol. 89, 1989.
- [48] H. G. Totapally and N. M. Aziz, “Refinement of cross-flow turbine design parameters,” *Journal of Energy Engineering*, vol. 120, no. 3, pp. 133–147, 1994.
- [49] V. Sammartano, “Banki-Michell optimal design by computational fluid dynamics testing and hydrodynamic analysis,” *Energies*, vol. 6, no. 5, pp. 2362–2385, 2013.
- [50] M. Torresi, B. Fortunato, and S. M. Camporeale, “Numerical investigation of a Darrieus rotor for low-head hydropower generation,” *Procedia Computer Science*, vol. 19, pp. 728–735, 2013.
- [51] Entec, *Technical Report on Characteristics of Cross- Flow Turbine T15*, Entec, Bangalore, Karnataka, India, 2014.