IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)
e-ISSN: 2278-1684,p-ISSN: 2320-334X, Volume 17, Issue 2 Ser. IV (Mar - Apr 2020), PP 01-06
www.iosrjournals.org

Optimization of Horizontal Axis Hydrokinetic Turbine Performances using Computational Fluid Dynamics (CFD)

Solomon Eriamiatoe[1],Augustine Udo[2]
[1]*Mechanical department, Faculty of Engineering, University of Port Harcourt, Rivers state, Nigeria*
[2]*Mechanical department, Faculty of Engineering, University of Port Harcourt, Rivers state, Nigeria*

Abstract:

Background:*The study of the optimization of horizontal axis hydrokinetic turbineperformances using computational fluid dynamics (CFD), creates awareness on the recent development in renewable energy industries and enhanced the output power of the Hydrokinetic power plant. In this work. the effect of TSR and chord length was investigated using CFD approach and. the results shows that power coefficient (*C_p*) depends on TSR and slightly affected by chord length.*

Materials and Method:*A three dimensional CFD analysis performed using ANSYS CFX 15.0 and SolidWorks as Preprocessor to draw the rotor, boundary conditions were created using the pre-processing tool solid-works, Hydrofoil SG-6043 was chosen for the simulation and mesh was created using structured quadrilateral cells around the hydrofoils. The computational domain was assumed to be sufficiently large compared to the chord length to enable larger area of flow visualization around the hydrofoil. A finer mesh was applied on the vicinity of the hydrofoil to obtain better flow characteristics and flow orientation very near to surface. Quadrilateral elements were used to mesh the entire geometry to ensure uniform aspect ratios of cells across the domain.*

Results*: As the rotational speed increases from root to tip of a blade, the flow angle decreases andas solidity increase from 0.084 to 0.127 there was a corresponding increase of *C_p*from 0.112 to 0.284 implying strong influence of solidity on horizontal axis hydrokinetic turbine performances. From analysis the results of the optimization performed shows that a *C_p* value of 0.45 achievable for a variable chord rotor of 1.0m at appropriate combination of turbine parameters.*

Conclusion*: Hydrodynamic analysis and optimization shows that the performance of hydrokinetic turbine can be maximized by choosing the right combination of design variables. Secondly, the three-dimensional results for optimum design suggested a strong dependence of maximum *C_P* on TSR when different turbine geometries are being considered. It was also observed that, Increase in turbine solidity results in increased *C_P* under the entire operating range of TSR studied with maximum *C_P* observed in lower TSR*

Key Words*: ANSYS CFX 15.0, Chord Length, Power Coefficient (Cp),Solidity (σ), Tip speed ratio(TSR}*

Date of Submission: 22-03-2020 Date of Acceptance: 09-04-2020

I. Introduction

This work focuses on maximizing the performance of a horizontal axis hydrokinetic turbine (HAHKT) operating in a cavitation free environment that is sufficiently away from the free-surface using a CFD analyses. There is a continually developmental journey for optional techniques of power which can be generated in the world today due the unfavorable environmental change impact coming around because of utilization of fossil fuels and the consumption of fossil energy exuding from overdependence on this type of energy. This mindfulness has prompted research and ensuing improvement of different types of renewable energy [1],[2]and this has come about into developing capital interests in green and clean energy innovation everywhere throughout the world particularly in the non-rustic areas of the world. In Nigeria it has been predicted that by 2025 our energy demand will rise to about 29,000 (MW) with renewable energy accounting for 10% of the source of energy [3]. As at 2014, Nigeria was only capable of generating about 6000(MW) of electricity[4] and its actual electricity generation alternates between 2000(MW) – 3700(MW) on a daily basis [5].Nigerian Ministry of Power has identified renewable energy as a way of curbing the crisis in the power sector.

The research into hydrodynamics of HKT was carried out using computational fluid dynamics (CFD) tool and laboratory scale experiments [6], Navier-Stokes equations such as the panel method and vortex lattice method forms part of series of CFD tools which was utilized for the analysis of hydrodynamics of these devices. These affordable computational techniques integrate solution of Reynolds-averaged-Navier- Stokes equations generally called RANS with turbulence models and has been utilized for the analysis of hydrodynamics of HKT which yielded a huge success.

An experiments and 2-D numerical analysis to study the influence of total number of blades and their tensile strength on the operation of horizontal axis wind turbine was performed by [7],[8]. A numerical analysis involving the use of BEM and lifting line which is based on the wake theory was carried out on the wind turbine. From the result of the analysis, it was concluded that the range of TSR for maximum Cp was strongly dependent on tensile strength and poorly on total number of turbine blades. This actually implies that the optimum TSR range of the turbine is determined by the length of chord used in the design of the turbine. According to the observation from the laboratory experimental results, it was found out that use of micro-turbines with flat plate blades resulted in a significant decrease in the optimum TSR range with a small change in maximum Cp for larger blade pitch angles when compared to smaller blade pitch angles [2] experimentally investigated the flow field and wake recovery behind tidal turbine using mesh disk simulators and found that recovery is dependent on closeness to surface of water, roughness of the sea bed and minimally to a certain extent on the thrust of the motor. Advance studies can be carried out for design optimization of the turbine based of the wealth of knowledge amassed from research and analysis wind/hydrokinetic turbines hydro-dynamically. Many of wind/hydrokinetic turbines optimization studies are centered on maximization of coefficient of performance and energy production annually of such systems. A genetic algorithm (GA) was utilized for optimizing annual energy production (AEP) and cost of energy of low-lift airfoils [9] for regulating wind turbines that are stalled. A multi-disciplinary optimization on stall regulated horizontal axis wind turbine considering fatigue, maximum load and annual energy production.

Hydrokinetic turbines (HKTs) are grouped into horizontal axis turbines (where the rotational axis of rotor is parallel to incoming water stream) and vertical or cross-flow turbine where rotational axis is perpendicular to the incoming water stream [10]based on the nature of flow. It was observed that Horizontal Axis Hydrokinetic Turbine (HAHKT) has proven to be more efficient than its vertical axis counterpart owing to lower incidence losses, less vibration and more uniform lift forces. The blades in HAHKT move perpendicular to the fluid motion receiving power through whole rotation. In other words, horizontal axis turbine’s swept area always faces the fluid as contrary to vertical axis turbines where swept area is perpendicular to the fluid motion.

According to [6], a non-dimensional similarity analysis suggests that overall performance of such turbines is primarily governed by four quantities, Reynolds number (Re), Tip speed ratio (TSR), Solidity (σ) which is the ratio of total blade chord to turbine circumference and Number of blades (N)

II. Material and Methods

The optimized geometry is modeled in appropriate software and a three-dimensional computational fluid dynamics (CFD) analysis is performed to calculate the fluid forces and torque developed by turbine [11] A grid independence study was carried out to study effect of number of elements on the CFD analysis. Mesh size was varied from a coarser mesh of 3.5 million to a finer mesh of 10 million elements and flow the variables were monitored.

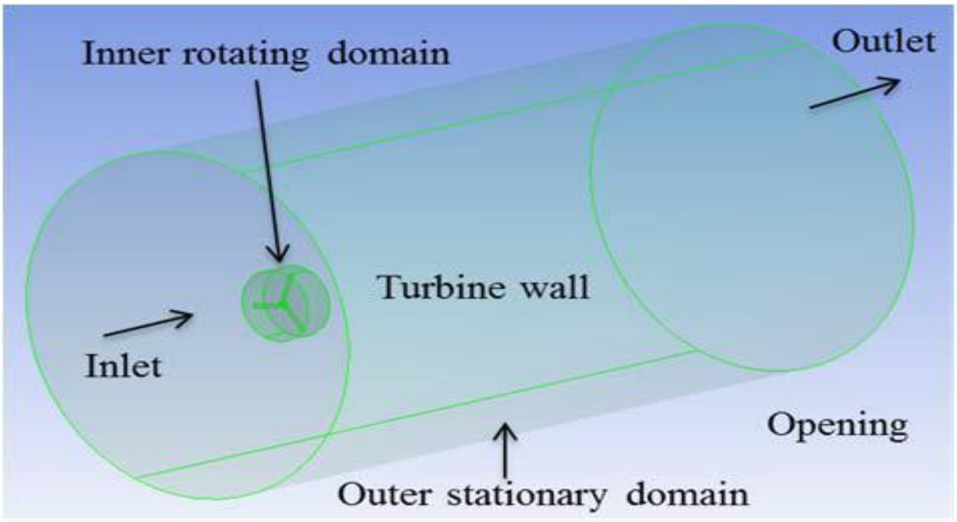

Fig. 1. Computation domain and mesh used for CFD analysis

This study assumes steady, incompressible flow where numerical solutions were carried out for both two-dimensional and three-dimensional flow geometries using ANSYS CFX 15.0. The geometrical models for three-dimensional (rotating) boundary conditions were created using the pre-processing tool solid-works. The choice of hydrofoil for HAHKT was primarily governed by the geometry that produces maximum lift coefficient (CL) as well as maximum lift to drag ratio (CL/ CD) under the operating range of Re. Hydrofoil SG-6043 was chosen for the simulation. The hydrofoil coordinates were imported from the UIUC hydrofoil geometry database[12] and the mesh was created using structured quadrilateral cells around the hydrofoils. The computational domain was assumed to be sufficiently large compared to the chord length to enable larger area of flow visualization around the hydrofoil. A finer mesh was applied on the vicinity of the hydrofoil to obtain better flow characteristics and flow orientation very near to surface. Quadrilateral elements were used to mesh the entire geometry to ensure uniform aspect ratios of cells across the domain. The computational domain consists of two cylinders; the inner one and outer one extending 10 rotor diameters and 11 rotor diameters respectively in the axial direction. The turbine blade has SG-6043 hydrofoil section and is placed inside the inner cylinder. Multiple reference frames have been adapted with a stationary outer cylinder and rotating inner cylinder and an interior boundary between the two. Second order up-winding discretization schemes and SIMPLE (Semi-implicit method for pressure linked equation) algorithm was chosen for solving pressure-velocity coupling.

Furthermore, the PRESTO (pressure staggering options) scheme was also adopted. In the iteration, the convergence criteria have been pegged to be lower than 10^{-4} for the residuals obtained from the continuity, x-momentum, y-momentum, z-momentum equations, k and ε computations.

Table no.1: Input Data for CFD Analysis

S/N	QUANTITY	SYMBOL	VALUE
1	Water density	ρ	998.2Kg/m^3
2	Pressure	P	101.3KPa
3	Number of blade	N	3.0
4	Free stream velocity of river	V	2.0m/s
5	Chord length	C	0.2m
6	Rotor Radius	R	1.0m
7	Blade pitch angle	θ	10.0^0
8	Rotor speed	Ω	6.0rad/s
9	Interpolation scheme		2nd order upwing
10	Pressure Scheme		PRESTO
11	Hydrofoil		SG6043
12	Residual error		10^{-4}
13	Turbulence Model		K- ω SST

Hydrodynamic Optimization Methodology

As a starting point for the hydrodynamic optimization, a three-bladed HAHKT turbine rotor was modelled as a single blade entity with four radial stations. These radial stations were selected along the blade radius: (1) at 25% radius, (2) at 50% radius, (3) at 75% radius and (4) 95% radius. The turbine rotor model was simplified by choosing some geometric characteristics of the rotor as constants and others as design variables. A constant rotor radius of 1m and inlet flow speed of 2 m/s was chosen as a reference design. The design rotational speed is selected between 3-10 rad/s based on the minimum and maximum RPM that the turbine blades will be rotating under given river current speed. The chord length and pitch angle distributions along the blade become the design variables for rotor optimization. These rotor design variables are modulated to achieve the peak hydrodynamic performance possible for the rotor in the design rotational speed range. Once the initial values, chord and pitch distributions are provided, the angle of attack (α) is calculated. Table4.3 provides the summary of the design variables for rotor optimization. As the rotational speed increases from root to tip of a blade, the flow angle decreases. Knowing the extents of the river water speed, rotational speed range and a particular hydrofoil, it is possible to estimate the range of flow angles and thus angles of attack to be encountered at each radial station.

Table no 2. Input Data for Optimization

S/N	VARIABLE	DESIGN VALUE	MINIMUN VALUE	MAXIMUM VALUE
1	Pitch angle (1)	20^0	16^0	25^0
2	Pitch angle (2)	8^0	4^0	12^0
3	Pitch angle (3)	1.5^0	-2^0	5^0
4	Pitch angle (4)	-2^0	-5^0	1^0
5	Chord (1)	0.25m	0.245m	0.255m
6	Chord (2)	0.26m	0.14	0.37
7	Chord (3)	0.17m	0.09	0.255
8	Chord (4)	0.12m	0.074	0.17
9	Rotor Radius	1.0m	-	-
10	Rotor rotational speed	-	3rad/s	10rad/s
11	River speed	2.0m	-	-

III. Result and Discussion

Effect of TSR and Blade Pitch Angle at different Chord Length

The effect of chord length, blade pitch and tip speed ratio, the chord length was varied from 0.03, 0.06 and 0.12m and three dimensional simulation executed. The result is shown in Fig 2.1 – 2.3 below.

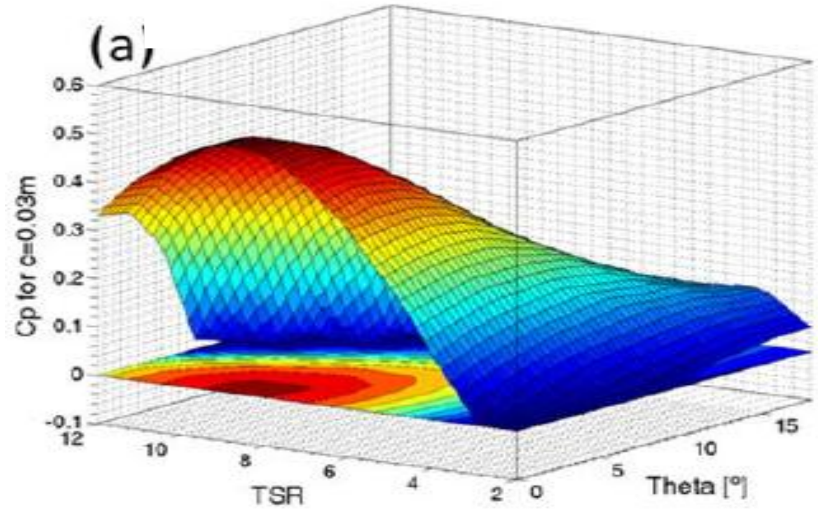

Fig.2(a). Power Coefficient at 0.03m Chord Length

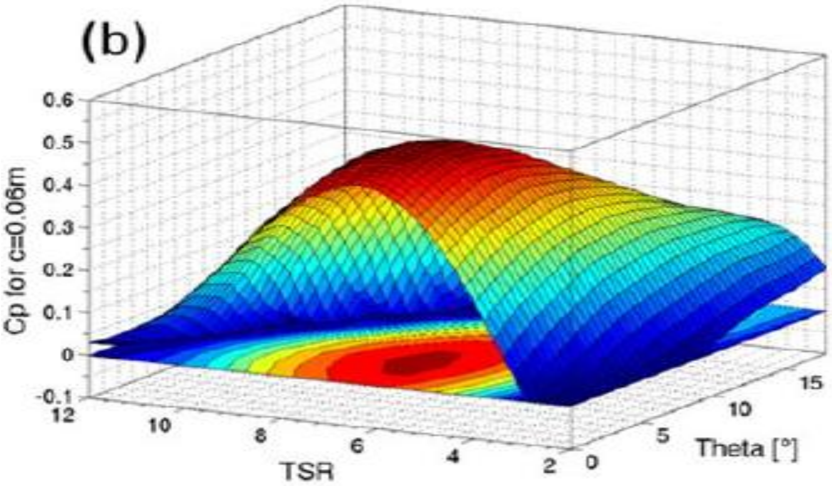

Fig.2(b). Power Coefficient at 0.06m Chord Length

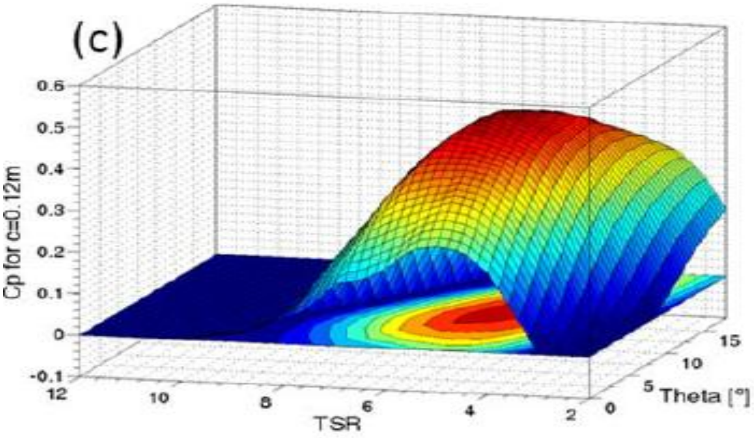

Fig.2(c). Power Coefficient at 0.12m Chord Length

Effect of Solidity on HAHKT Performance

An optimum design of HAHKT is associated with turbine solidity (σ) and TSR since these two variables primarily control the volume of fluid which can be utilized for power extraction. In order to examine the influence of solidity on turbine performance, three dimensional numerical simulations were performed using different values of solidity. As turbine solidity is approximately doubled from 0.084 to 0.127, the resulting C_P has also doubled from 0.112 to 0.284 implying strong influence of solidity on turbine performance. The results also indicate that the initial starting torque of a four bladed turbine is higher than that of the other two cases. This is expected since more blades will contribute more lift resulting in increased torque at the rotor hub.

Furthermore, the results obtained from Fig. 3gives more in-depth knowledge for choice of turbine solidity for user-specific applications. Utilization of higher solidity turbines is deemed necessary when higher initial starting torque and lower rotational speed is needed and vice versa. The later can be utilized for production of electricity.

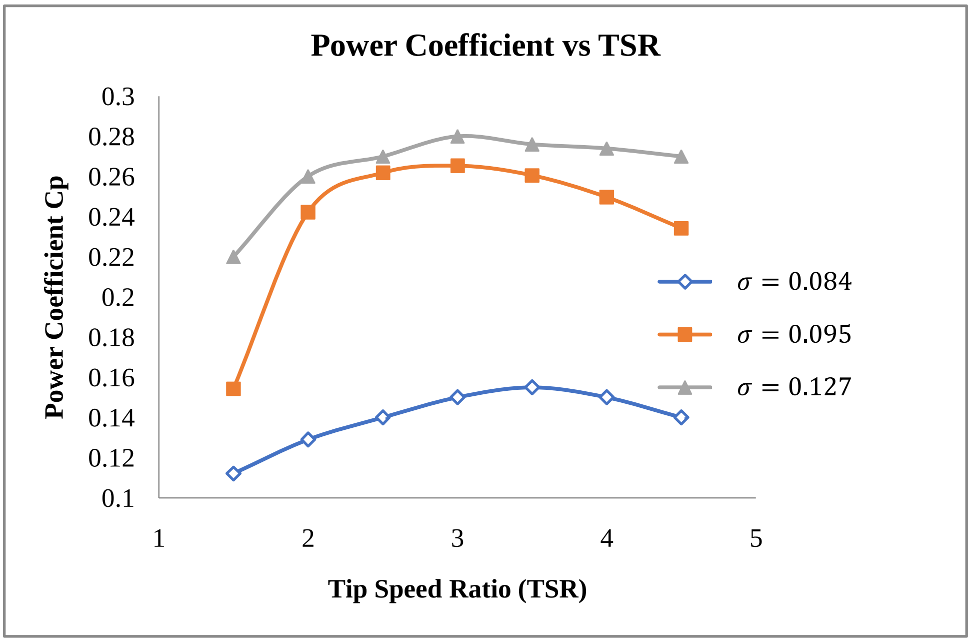

Fig.3. Power Coefficient versus Tip Speed Ratio under different Turbine Solidities for N = 3

Optimization Result

The power curve from the hydrodynamic optimization of the given turbine blades is shown in Fig.4. The maximum C_P of approximately 0.45 was observed for a corresponding TSR = 4.25 Showing significant improvement when compared with constant-chord and fixed pitch turbine blades.

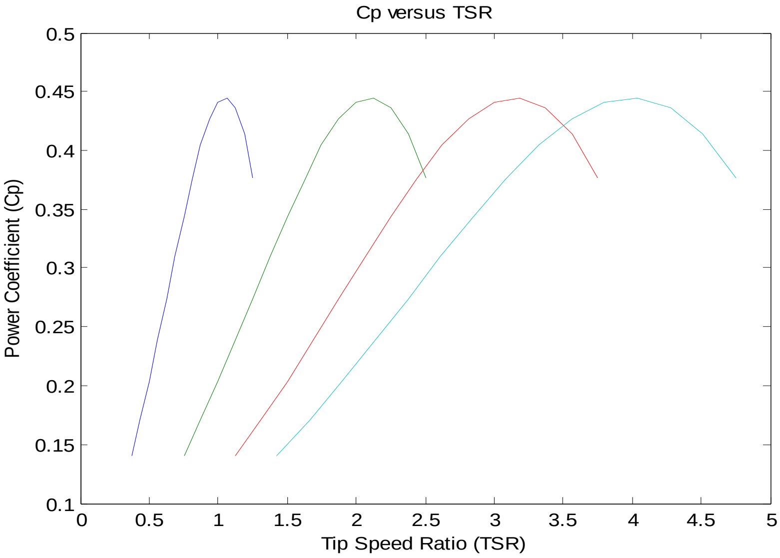

Fig.4. The Power Coefficient Plot for different TSR using Variable Chord and Variable Pitch Turbine Blade

IV. Conclusion

Optimization of the performances of Horizontal Axis hydrokinetic turbine using Computational fluid dynamics (CFD) was investigated.ANSYS CFX 15.0 was implemented in solving the flow conditions. The effect of several non-dimensional hydrodynamic parameters on the turbine performance wereanalyzed such as TSR, Solidity and Reynolds number which increase the efficiency of the hydrokinetic turbines. Numerical investigations were performed using three-dimensional (rotating) models to examine the performance of HAHKTs under different turbine solidities ranging 0.084 - 0.127. Secondly, the three-dimensional results for optimum design suggested a strong dependence of maximum C_P on TSR when different turbine geometries are being considered. It was observed that, Increase in turbine solidity results in increased C_P under the entire operating range of TSR studied with maximum C_P observed in lower TSR. Thirdly, hydrodynamic analysis and optimization shows that the performance of hydrokinetic turbine can be maximized by choosing the right combination of design variables. Coefficient of performance as high as 0.45 was achieved with a variable chord blade turbine in a non-cavitation environment.

The optimization of hydrokinetic turbine performance, maximizes the Annual Energy Production (AEP) and minimizes the Cost of Energy (COE) and this study will also aid the appropriate governmental organizations in deciding on whether or not to invest in Hydrokinetic power plant for power supply in areas closed to river and sea.

References

[1]. Myers, L and A.S. Bahaj,(2006): Power output performance characteristic of a horizontalAxis marine current turbine,' Renewable energy, vol. 31: pp. ; 197-208.

[2]. Myers, L and A.S. Bahaj,(2007): Wake studies of a 1/30th scale horizontalAxis marine current turbine,' ocean engineering, vol. 34: pp. ; 758-762.

[3]. Nkemjika M. A, (2014): Assessment of Offshore Renewable Energy Utilization Potentials and a Hybrid Power Plant Design for a Nigerian Coastal Area. A MEng Thesis Presented to the School of Graduate Studies, University of Port Harcourt

[4]. Nigerian Bulletin (2014): Nigeria Now Has Capacity to Generate 6000MW Electricity – Nebo. April 25, 2014. Retrieved: www.nigeriabulletin.com/thread/.

[5]. Vanguard, (2013): Power Supply: Nigerians generate over 600MW, FG 5482MW, Says UNILAG VC, Michael Eboh. Retrieved: www.vanguardngr.com/2013/03.

[6]. Mukherji, S.S.(2011). Numerical Investigation and Evaluation of Optimum Hydrodynamic Performance of a Horizontal Axis Hydrokinetic Turbine. Journal of Renewable andSustainable Energy, vol. 3: p. 063-105.

[7]. Sørensen, J.N.(2011): hydrodynamic Aspects of Wind Energy Conversion. Annual Review of FluidMechanics. 43(1): p. 427-448.

[8]. Fuglsang, P. and H.A. Madsen (1999): Optimization Method for Wind Turbine Rotors - A New Dual Method using Mixed Variables. Journal of WindEngineering and Industrial Aerodynamics. 80(1): p. 191-206.

[9]. Giguerre and M.S. Selig (1998), "new airfoils for small horizontal axis wind turbine,"Journal of Solar Energy Engineering, vol,120. Pp. 108-114

[10]. Khan M. J.(2009). "Hydrokinetic energy conversion systems and assessment of horizontal and vertical axis turbines for river and tidal applications: A technology status review," Applied Energy, vol. 86, pp. 1823-1835, 2009.

[11]. Hwang I. S.(2009). "Optimization of cycloidal water turbine and the performance improvement by individual blade control," Applied Energy, vol. 86, pp. 1532- 1540.

[12]. UIUC (2015). Airfoil Coordinate Database. University of Illinois at Urbana Champaign. Retrieved: http://www.ae.illinois.edu/m-selig/ads/coord_database.html

Solomon Eriamiatoe,etal. "Optimization of Horizontal Axis Hydrokinetic Turbine Performances using Computational Fluid Dynamics (CFD)." *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, 17(2), 2020, pp. 01-06.