Influences of yaw angle and turbulence intensity on the performance of a 20kW in-stream hydrokinetic turbine

Wenlong Tian^{1,2*}, James H. VanZwieten², Parakram Pyakurel², Yanjun Li³

¹School of Marine Science and Technology, Northwestern Polytechnical University, Xi'an, Shaanxi, China

²Southeast National Marine Renewable Energy Center, Florida Atlantic University, Boca Raton, FL, USA

³College of Engineering, Florida Atlantic University, Boca Raton, FL, USA

*Corresponding author: Wenlong Tian

Address: School of Marine Science and Technology, Northwestern Polytechnical University, 127 Youyi Xilu, Xi'an, Shaanxi Province 710072, China.

E-mail Address: tianwenlong@mail.nwpu.edu.cn

James H. VanZwieten

Address: 777 Glades Road, EE319, Boca Raton, FL 33431, USA.

E-mail Address: jvanzwi@fau.edu

Parakram Pyakurel

Address: 777 Glades Road, EE318, Boca Raton, FL 33431, USA.

E-mail Address: ppyakurel2014@fau.edu

Yanjun Li

Address: 777 Glades Road, EW210, Boca Raton, FL 33431, USA.

E-mail Address: yli2013@fau.edu

Influences of yaw angle and turbulence intensity on the performance of a 20kW in-stream hydrokinetic turbine

Wenlong Tian^{1,2*}, James H. VanZwieten², Parakram Pyakurel², Yanjun Li³

¹School of Marine Science and Technology, Northwestern Polytechnical University, Xi'an, Shaanxi, China

²Southeast National Marine Renewable Energy Center, Florida Atlantic University, Boca Raton, FL, USA

³College of Engineering, Florida Atlantic University, Boca Raton, FL, USA

*Corresponding author. Email address: tianwenlong@mail.nwpu.edu.cn

Abstract

Three-dimensional transient Computational Fluid Dynamics (CFD) simulations are performed to study the hydrodynamic performance of an ocean current turbine with a 3.0 m diameter 3-bladed rotor. Simulations are based on the Reynolds Averaged Navier-Stokes (RANS) equations and the shear stress transport $k-\omega$ turbulent model is utilized. The influence of yaw angle and upstream turbulence intensity (TI) on the turbine performance is studied. The CFD method is first validated using existing experimental data and good agreement is obtained. The performance of the turbine, including power, thrust and wake characteristics are then studied at different tip speed ratios (TSR). The turbine obtains a maximum coefficient of power (C_p) of 0.4642 at TSR=6 and the coefficient of thrust (C_t) increases over the entire evaluated TSR range to a value of 0.8788 at a TSR=10. Simulations are also performed at four different yaw angles, 0°, 5°, 10° and 15° which show that both C_p and C_t decrease as yaw angle increases. Finally simulations of three different TIs, 3%, 6% and 9%, are performed and analyzed. Results show that TI minimally affects C_p and C_t for the considered TI range, but greatly influences the downstream wake structure.

Key words: yaw; turbulence intensity; hydrokinetic turbine; computational fluid dynamics; CFD; turbine

Nomenclature

c	Blade chord length(m)	TI	Turbulence intensity
C_p	Coefficient of power	TSR	Tip speed ratio
C_{paxial}	Axial coefficient of power (W)	U	Upstream velocity (m/s)
C_{ppeak}	Peak coefficient of power (W)	v	Axial velocity (m/s)
C_{ptotal}	Coefficient of power at zero yaw position (W)	x	Downstream distance (m)
D	Diameter of the rotor (m)	y	Radial distance (m)
P	Power (W)	y^+	y-plus value
Q	The second invariant of the rate of strain tensor	θ	Yaw angle (°)
r	Spanwise location of the blade(m)	ω	Angular velocity (rad/s)
R	Radius of the rotor (m)	φ	Blade twist angle (°)
t	Blade Thickness(m)	ρ	Water density (kg/m ³)
T	Thrust (N)		

1. Introduction

The past few years have seen an increase in worldwide interest of using river, tidal, and ocean current hydrokinetic energy to generate electricity without the use of dams, in-stream hydrokinetic energy production. Each of these resource types are distributed globally with riverine energy primarily located in the world's larger river systems, tidal energy along the coast and primarily at higher latitudes, and ocean current energy primarily along the western boundaries of the world's oceans. Ocean currents with average energy densities greater than 0.5 kW/m² are located at numerous locations along the western boundaries of the world's ocean basins [1], with average energy densities measured in excess of 3.0 kW/m² [2]. The technically extractable average ocean current energy from US waters has been estimated at 19 GW [3],

with several other countries such as Japan, South Africa, Taiwan, the Philippines, and Brazil also having large scale ocean currents accessible off their coasts [1]. Extracting power from these renewable resources could impact the energy portfolios of these countries, providing a potential solution to the energy crisis and environment pollution.

Ocean current turbines (a subset of in-stream hydrokinetic turbines that are designed specifically for ocean currents) are being designed and tested for converting the kinetic energy contained in ocean current to electricity. These turbines can be generally classified into two main categories according to the angles between the upstream water and the rotating axis of the turbine, the horizontal axis ocean current turbines and the vertical axis ocean current turbines, a detailed description of these turbines can be found in [4, 5]. Currently ocean current turbine prototypes are being designed and tested to extract power from the ocean current resources in the above paragraph (See [6, 7] for example). If deployed in US waters, these systems will most likely be moored in water depths from 300 to 400 m (potentially up to 800 m) and will likely be operated within the top 100 m of the water column where the current is strongest. To help address many of the challenges associated with operating in-stream hydrokinetic turbines in ocean currents the Southeastern National Marine Renewable Energy Center (SNMREC) at Florida Atlantic University (FAU) designed, built, and tested at-sea a 20 kW horizontal axis experimental ocean current turbine (OCT) [8, 9]. This OCT (Fig. 1) consists of a single rotor with a radius of $R=1.5$ m, a torpedo shaped nacelle and two elliptically shaped cylindrical weights designed to make the turbine negatively buoyant and counteract the hydrodynamic rotor torque. The three bladed rotor utilized on this system was geometrically designed using the National Renewable Energy Laboratory's HarpOpt program using the approach described in [10], with the final blade design presented in [8]. The nacelle is designed to contain a generator, gear box, and mechanical brake to stop the rotor blades from turning. The nacelle diameter is 0.6 m and a length is 3.7 m.

Experimental trials have been carried out to evaluate the performance of several hydrokinetic rotor designs. Bahaj et al. carried out a power and thrust coefficient study on a 0.8 m-diameter turbine in a towing tank and in a cavitation tunnel, providing comprehensive high-quality data for the validation of numerical computations [11]. Coiro et al. conducted towing tank experiments of a scaled model of a hydrokinetic turbine and provided the power and thrust curves at different water velocities [12]. Galloway et al. studied the power and thrust performance of a 1/20th scale hydrokinetic turbine operating at yaw and in wave by performing towing tank experiments [13]. Later they studied the effect of yaw angle and wave on the loads of the turbine with both experimental and numerical method [14]. Tedds et al. provided many turbine performance curves for hydrokinetic rotor design variants with different numbers of blades, pitch angles, etc [15]. Recently, Mycek et al. studied the upstream turbulence intensity effect and the interaction between two turbines, with emphasis paid on the wake of the turbine [16, 17].

Numerical methods utilized to evaluate rotor designs include blade element momentum methods (BEM), which have been used widely for engineering design because of the low computational overhead associated with this approach. Over the past several years several correction factors have been added to the originally proposed BEM method that account for three-dimensional (3D) effects such as tip loss, rotational flow and dynamic stall [18, 19]. 3D inviscid models provide more physics of the turbine hydrodynamics than the BEM method. Current 3D inviscid models include lifting line, panel, and vortex-lattice [20-22]. However, these methods neglect the viscous effects, which need to be considered to achieve the most accurate turbine performance predictions possible.

Computational fluid dynamics (CFD) simulations of the Navier-Stokes equations model fluid flows starting from first principles, and therefore inherently capture viscous effects. Comprehensive CFD simulations of horizontal axis water/wind turbines have been created. Michael et al. numerically modeled a 20 m-tidal turbine at different flow velocities using the commercial CFD code STAR CCM+, including an analysis of the effect of grid density and time step on the calculated torque [23]. Monier et al. designed and optimized a winglet for the NREL Phase VI turbine using the Fine/Turbo of the commercial CFD code NUMECA [24]. They provided a detailed CFD validation study of the NREL Phase VI turbine [25] and the CFD results was in good agreement with the experiment results. Yuwei et al. carried out CFD simulations of the NREL Phase VI turbine utilizing both unsteady Reynolds-Averaged Navier-Stokes (RANS) and Detached Eddy Simulation (DES) methods [26]. Tongchiphakdee et al. studied the aerodynamic performance of the NREL Phase VI horizontal axis wind turbine under yawed flow conditions [27]. Schleicher et al. designed a novel portable micro-hydrokinetic turbine and characterized the performance of the turbine with a RANS method [28]. Nak et al. studied the effect of the distance between dual rotors on the performance and efficiency of a counter-rotating tidal turbine

using both CFD and experimental methods[29]. More recently, Liu et al. studied the wakes of a horizontal axis ocean current turbine with three different RANS methods, the results show that the Sliding Mesh Method is more accurate than the Moving Reference Frame and the Actuator Disk Model[30].

The present study aims at investigating the performance of the SNMREC OCT design [8] with a CFD approach. While mooring turbine will be designed to approximately align themselves with the current, it is likely that some misalignment will occur in some flow conditions and therefore simulations are performed at four yaw angles, $\theta = 0^\circ, 5^\circ, 10^\circ$ and 15° , to study the yaw effects. Turbulence intensity (TI) also impacts the fluid/structure interactions associated with in-stream hydrokinetic energy production and therefore dependence of these processes on turbulence intensity are evaluated.

2. Turbine Blade Design

2.1 Site specific flow measurements

The testing site for FAU's ocean current turbine will be located off the coast of South Florida where the Gulf Stream flows approximately northward. Ocean current measurements were taken off Southeast Florida (Lat: $26^\circ 04.3'$ N, Lon: $79^\circ 50.5'$ W) over a 13 month period from February 2009 – March 2010 [31]. These measurements show that the mean current speed at a depth of 25m was 1.6 m/s, with a range between 0.4 and 2.5 m/s (Figure 2) [31]. The average mean flow speed calculated using these measurements identify an approximately linear decrease in the mean horizontal flow speed with depth over the top 325 m of the water column.

2.2 Blade design

The rotor that is numerically modeled in this study is the three-bladed rotor, with a diameter of $D=3$ m and a hub diameter of 0.6 m, shown in Figure 1 and described in [8]. The airfoil shapes used by this rotor were created from the FF-77-W airfoil type [32]; more specifically they were interpolated from the fx77w121, fx77w153, and fx77w258 airfoil geometries provided available at [33]. The HarpOpt rotor design code was used to create optimize this rotor design for the Gulf Stream current using the process defined in [10], except that the rotor used in this study was optimized for a constant RPM of 50 [8]. The details of the utilized blade geometry are summarized described in Table 1, where r is the spanwise location, ϕ is the blade twists angle, c is the chord length, t is the thickness of the blade. Figure 3 shows the three views of the solid model of the blade.

3. Numerical method

The commercial CFD code FLUENT v15.0 was used to solve the incompressible RANS equations using a second-order-accurate finite-volume discretization scheme. The shear stress transport (SST) $k - \omega$ turbulence model was selected to model the turbulence terms of the RANS equations. The SST $k - \omega$ turbulence model is able to model the transport of turbulent shear stress and gives accurate predictions on the onset and amount of flow separation under adverse pressure gradients, and has been successfully used in the CFD simulation of wind/water turbines [23, 29, 30].

3.1 Computation domains and boundary conditions

In this simulation the computational domain was sized to allow for full development of the upstream flow and to decrease the blockage effect. The computation domain is a cylinder with a diameter of $5D$ and a length of $14D$. The turbine was placed in the centerline of the cylinder and at a distance of $4D$ from the left boundary. Figure 4 shows the dimension of the domain. The overall domain was split into two subdomains. The first domain contains the grid elements surrounding the rotor (referred to as the rotor domain). The second one contains the cells in the outer region.

A uniform and steady velocity profile of 1.6 m/s, which is the measured mean velocity of the Florida Current at a water depth of 25m [26], was applied at the inlet of the computation domain and a pressure outlet boundary was applied at the outlet of the domain. To improve the stability of the numerical simulations, symmetry boundary conditions were applied at the side wall of the cylinder. The symmetry boundary condition is useful because it allows the solver to consider the wall as part of a larger domain, avoiding the wall effects. No-slip boundary conditions were imposed at the surface of the blades and the hub. Two interfaces were imposed at the overlap faces between the adjacent subdomains, allowing the transport of the flow properties.

3.2 Grid generation

The fidelity of any CFD simulations depends on the type of grid utilized (e. g. structured or unstructured) with the

structured hexahedral grids being the most desirable because they inherently provide highly accurate numerical solutions [20]. In the present study, all structured grids were generated using the ICEM grid generation utility in ANSYS 15.0. The grids of the two subdomains were generated separately and then merged together. Figure 4 shows the surface grids of the nacelle and the rotor, with the grid node density being higher in the rotor domain than in the outer domain. The grids behind the rotor and around the nacelle, where high velocity gradients were expected, was refined with a higher grid resolution (Figure 5a).

Prism layer grid elements were extruded from the surfaces of the blade to improve the grid quality and to provide sufficient precision to describe the boundary layer flow (Figure 5b). The height of the first prism layer above the surfaces of the blades was set so that the y^+ value for the first layer of elements from the wall was below 1, depending on the rotation velocity of the rotor and the position of the elements on the blade. This y^+ value is suggested by the Fluent User's Guide [29] for the use of the SST $k-\omega$ turbulence model. A mesh growth rate of 1.2 was chosen for all subdomains.

Three grids were generated for the grid resolution verification. The main difference of these grids is the grid resolution on the blades and behind the rotor.

3.3 Solution sets

Six sets of simulations, as listed in Table 2, are performed, in which cases 1-4 differ in yaw angle and cases 1, 5 and 6 have different turbulence intensities. For each case listed in Table 1, simulations were carried out for several tip speed ratios (TSRs) varying from 1 to 10. Tip speed ratio represents the ratio of the blade tip speed to the upstream flow speed, and has the following expression,

$$TSR = \frac{\omega R}{U}, \quad (1)$$

where U is the flow speed at the inlet and is equal to 1.6 m/s and ω is the angular velocity of the rotor. A sliding mesh model was used to simulate the rotation of the rotor with each simulation lasting for six revolutions to allow for convergence. The mean performances of the rotor, including thrust and power, are averaged using the data in the last revolution. The time step for each simulation was set so that 1° of rotor rotation was achieved each time step. This time step was selected to be conservatively small relative to a previous study that used a time step of $2.5^\circ/\text{step}$ and obtained good results [26].

Convergence was determined by the order of magnitude of the residuals. The drop of all scaled residuals below 1×10^{-4} was utilized as the convergence criterion. The maximum number of iterations in a time step was set as 30, which enabled all the residuals to meet convergence in the simulation. In order to obtain a fast convergence, the pressure-velocity coupling method was used in all simulations. A Second Order Upwind spatial discretization algorithm was used for all the equations, including pressure, momentum and turbulence, and a Least Squares Cell Based algorithm was used for the gradients. The second order algorithms produce more accurate results than first order ones because they reduce interpolation errors and false numerical diffusion [34].

4. Numerical method verification and validation

4.1 Verification

A grid resolution study was performed to evaluate the influence of grid resolution on the torque and thrust of the rotor. The simulations were conducted at zero yaw condition, $TSR=6$ and $TI=3\%$. Table 3 summarizes the results with different grid resolutions. It can be seen that grids with approximately 8.0 million and 10.1 million elements gave approximately the same results. This indicates that further increasing the grid resolution would not significantly affect the rotor torque or thrust predicted by CFD simulations. Considering the economy of time in the simulation, the grid with 8.0 million elements was chosen for the following simulations.

The main settings for the final mesh include: 120 and 100 nodes in the spanwise and chordwise direction of the blade, respectively; the maximum size of the elements in the rotor domain is $1/30^{\text{th}} D$; the maximum size of the elements downstream of the rotor and in the outer domain is $1/20^{\text{th}} D$; about 3 million elements in the rotor domain and about 5 million elements in the outer domain.

4.2 Validation

In order to validate the accuracy of the method used in this paper, CFD simulations were performed on the rotor presented in [11]. Numerical results calculated using this design are compared with the cavitation tunnel experimental data, with the characteristics of the experimental model is listed in Table 4 [11]. Figure 6 shows a comparison of the coefficients of power and thrust calculated using this CFD approach and the experimental over a range of TSRs, with the coefficient of power defined as

$$C_p = \frac{P}{0.5\rho\pi R^2 U^3}, \quad (2)$$

and the coefficient of thrust defined as

$$C_t = \frac{T}{0.5\rho\pi R^2 U^2}, \quad (3)$$

where P is the generated power and T is the thrust on the rotor in the axial direction. It can be seen that the coefficients of power are in very good agreement with the experimental results, except for a slight underestimation at TSR=10, where the relative error is 8%. The coefficients of thrust show a similar trend to that of the experimental data, but are slightly lower at all TSRs. The reason for the relatively larger error in thrust possibly lies in the fact that the hub is not modeled in the validation study and the thrust near the root of the blade is reduced due to the miss of the hub. Overall, the numerical method in the present study has predicted results with acceptable accuracy.

5. Results and discussion

5.1 Performance of the OCT at different TSRs for zero yaw and TI=3%

To investigate the relationship between the performance of the OCT and TSR, rotational velocities varying from 2.13 rad/s to 10.67 rad/s were evaluated, corresponding to TSRs from 2 to 10. Particularly, simulations at a rotational velocity of 7.85 rad/s (which is the designed operating RPM for the rotor of 50 and corresponds to a TSR of 4.9) were also performed. All the following simulations are performed at an inflow velocity of 1.6 m/s, resulting with a Reynolds number of about 5×10^6 based on the diameter of the turbine. This turbine has not been fully tested experimentally and the only published performance data were calculated using the BEM method presented in [8]. Figure 7 presents the comparison of the rotor power and thrust coefficients calculated using the CFD simulations described in this document and the BEM approach. Both the power and thrust are calculated on the three blades with the forces on the nacelle not being considered. It should be noted that the hydrodynamic coefficients for the BEM were calculated for a Reynolds number of 1×10^6 which is based on the cord of the airfoil section. Figure 7a shows that the C_p predicted by CFD increases with TSR up to a TSR of 6 and then decreases with increasing TSR after this, with a peak C_p value of 0.4642. The C_p curve predicted by BEM method WT_Perf follows a similar trend to that of the CFD, but reaches its peak earlier at a TSR of 4.9. The CFD method predicts a slightly lower C_p value than the BEM approach for TSR<6 and a slightly higher C_p value above this TSR. In general, these two methods show close power curves and approximately the same peak coefficients of power, 0.4642 for the CFD method and 0.4640 for the BEM method. The similarities between the C_p values predicted by these two well documented approaches suggest that they both create reasonable estimates of power output. However, since no experimental data is available for this rotor blade design it is not possible to determine which approach creates more accurate results.

The total thrust coefficient of the three blades predicted by CFD (Figure 7b) increases with increasing TSR over the entire evaluated range of TSRs (1.0-10), with the slope of the curve decreasing with increasing TSR and a maximum value of 0.8788. The BEM method calculates a higher C_t value than CFD when TSR≤8. Near this TSR the BEM approach predicts that the C_t begins to decrease with increasing TSR. A similar finding in the decrease of C_t at high TSRs can be seen in the experimental data presented in Figure 6, where C_t begins to decrease at a TSR of 10. In general, both methods predict similar curves and peak thrusts.

For better understanding the flow characteristics at different TSRs, the vortical structures are shown in Figure 8. Four typical TSRs, including TSR=2 (the rotor starts to rotate), TSR=4 (before the maximum C_p point), TSR=6 (the maximum C_p point) and TSR=10 (after the maximum C_p point), are selected. The vortex structures are plotted using iso-surfaces of

the second invariant of the rate of strain tensor [34] at $Q=1$ and colored with the magnitude of velocity. Flow separation is observed at the nose of the nacelle, as indicated by the contours, where the shape of the nacelle transfers from a half of a sphere to a cylinder. Vortices also detach from the tail of the nacelle, forming a low-velocity zone just behind the tail. This low-velocity zone has a radial dimension which is slightly larger than the diameter of the nacelle. The streamwise length of this zone reduces with the increase of TSR and in the case of $TSR=10$, the low-velocity zone is hardly seen. At $TSR=2$, the blades are in deep stall condition and suffer strong unsteady flow separation along the whole span. At this TSR strong vortices are shed from the tips of the blades like spiral lines, with a diameter slightly larger than that of the rotor. It can also be seen that at a TSR of 2 vortices detach from the root of the blades and form small spiral vortex lines. However, in the other three cases where the $TSRs$ are higher ($TSR=4, 6$ and 10), the root vortices merge with the nacelle tail vortices and cannot be seen in these pictures. At $TSR=4$, attached flow is seen for most of the blade and the unsteady trailing edge vortices are weaker than $TSR=2$. The velocities of the tip vortices increase with TSR and more periods of the vortices are observed. At $TSR=6$ and $TSR=10$, the blades experience fully attached flow across the whole blade and the unsteady trailing edge vortices are even weaker, and the tip vortices merge into a cylindrical surface but dissipate quickly downstream of the rotor and at a distance of about $1R$.

To more clearly show the flow around the blade, the pressure contours and streamlines at four blade sections, 30%, 50%, 70% and 95% of the radial distance between the axis of rotation and the blade tips (span), are presented for the four evaluated $TSRs$ in Figure 9. At $TSR=2$, the blade is in a deep stalled condition along its entire length, suffering from flow separation at the four evaluated blade sections. The structure of the flow separation differs with radial location. At 30% and 50% span, the flow separates from both the leading edge and the trailing edge with two strong contra-rotating vortexes. However, at 70% and 95% span the flow separates from the leading edge and reattaches to the suction side of the blade, forming a large vortex which covers the whole suction surface. Significant pressure recovery is observed on the suction surface due to the shedding of the vortexes. At $TSR=4$ airfoil stall is seen over part of the blade with weak separation near the trail of the suction surface at 30% and 50% span. For 70% and 95% span, fully attached flow is seen. The blade is under fully attached flow conditions at all the sections for $TSR=6$ and $TSR=10$.

The wake generated by turbines will be an important consideration when turbines are located in arrays as the velocity deficits and increased turbulence levels created by systems will adversely affect the performance of downstream system. Turbines designed and operated such that their wakes recover to near ambient flow conditions will be able to be located closer together, enabling the production of greater electrical power from a smaller array footprint. For this reason the relationship between TSR and the turbine generated wake field is evaluated here. Figure 10 presents two-dimensional contours of axial velocity at the longitudinal cross-section planes for the four evaluated $TSRs$. It can be seen that the velocity deficit is primarily located in a limited area that increases in diameter smoothly along the flow direction. The diameter of this reduced flow region increases as TSR increases from 2 to 6, where the flow transits from stalled to attached (Figure 10) and the thrust coefficient increases rapidly (Figure 7). However, in the attached flow cases larger TSR ($TSR=10$ instead of 6) leads to a narrower wake, $0.98D$ compared to $1.08D$ for a downstream distance of $10D$. However, the magnitude of the axial wake reduction increases with TSR (and therefore C_t as seen in Figure 7) for the presented results with axial flow reductions of 10%, 14%, 18% and 21% seen for $TSRs$ of 2, 4, 6, and 10 respectively at a downstream distance of 10 rotor diameters. The upstream flow is slowed when it passes the nacelle and rotor and forms a low-velocity zone behind the tail of the nacelle. The length of this zone decreases with the increase of TSR and disappears at $TSR=10$. A small high-velocity zone lies at the nose of the nacelle, which is caused by the separation flow and reduces as TSR increases. High velocity zones are also observed just behind and outside of the rotor tips. These zones correspond to the vortices detached from the tip of the blade (Figure 7) and last for about $1R$.

5.2 Performance of the OCT at different yaw angles

While moored turbines will be designed to approximately align themselves with the current, it is likely that some misalignment will occur in some flow conditions. To study the yaw effect four yaw angles, $\theta=0^\circ, 5^\circ, 10^\circ$ and 15° , are considered and the associated turbine performance is compared. The first turbine performance characteristic evaluated is the how the power changes with yaw angles. Figure 11 presents the coefficients of power for the rotor at four different yaw angles. All four C_p vs TSR curves are very similar in shape, reach their peak values of 0.4642, 0.4600, 0.4494 and 0.4299 respectively at a TSR of 6. The C_p values calculated at all four yaw angles are nearly identical for $TSRs$ of 1 and 2

where the rotor is under deep stall conditions. However, for evaluated TSRs of 4 and above it can be clearly seen that C_p decreases with yaw angle. Three reasons contribute to this decrease in C_p . Firstly, as yaw angle increases, the axial projected area of the rotor decreases and the kinetic energy contained in the upstream flow decreases as well. Secondly, the separation at the nose of the nacelle becomes stronger as yaw angle increases (Figure 14), causing the flow in face of the rotor more turbulent. Thirdly, due to the yawed position, the upstream flow is not perpendicular to the rotor blade and can be divided into two components, a spanwise component and an axial component. The latter contributes to the lift and torque of the blade. At higher yaw angles, the axial component of the velocity gets smaller and leads to smaller lift and torque.

Figure 12 shows the relationship between the peak C_p and yaw angle, along with a quadratic function that is fit to these data. This quadratic function is defined as

$$C_{p_{peak}} = -0.000149\theta^2 + 0.464. \quad (4)$$

At a yaw angle of zero the rotor has a peak C_p of 0.4642; when it's at a yaw angle of 15° , the peak C_p reduced to 0.4299, and is 7.4% less than the zero yaw condition. This is significantly less than the 3.4% reduction in C_p that would be expected if yaw angle effects were only assumed to be a function of the decreased projected frontal area of the turbine caused by yaw,

$$C_{p_{axial}} = C_{p_{total}} \cos \theta, \quad (5)$$

where $C_{p_{total}}$ is the power at zero yaw condition.

To clearly show the impacts of yaw angles on the thrusts of the turbines, the coefficients of thrust for the rotor operating at four different yaw angles are presented in Figure 13. The effect of yaw angle on the C_t is similar to that on the C_p . Yaw angle minimally impacts C_t at low TSR conditions ($TSR \leq 2$), but noticeably decreases C_t at higher TSR conditions ($TSR \geq 4$). As explained above, the decrease of the axial projected area of the rotor is part of the reason. Besides, axial component of the inflow is reduced due to the yaw and the thrust depends mainly on the axial component rather than the spanwise component. The analysis conducted for zero yaw predicts a maximum C_t of 0.8788. However, when the yaw angle is 15° this is reduced to 0.8257, 6.0% less than for zero yaw. This is significantly less than the 3.4% reduction in C_t that would be expected if yaw angle effects were only a function of the decreased projected frontal area of the turbine caused by yaw.

To more clearly visualize the impact that yaw angle has on the near-rotor flows the vortical structures of the rotor at different yaw angles using iso-surfaces of the second invariant of the rate of strain tensor at $Q=1$ and $TSR=6$ are plotted in Figure 14. The tip vortices of the four cases have similar shapes - the tip vortices merge into a cylindrical surface but dissipate quickly downstream of the rotor and at a distance of about $1R$. The separation of the flow near the nacelle changes significantly with yaw angle. At zero yaw, separation flow occurs only at the nose of the nacelle where the shape of the nacelle transfers from spherical to a cylindrical. However, for $\theta = 10^\circ$ almost the entire surface of the nacelle suffers from flow separation and small vortices can be seen near the root of the blade. The same is true for $\theta = 15^\circ$, with the separation becoming more heavy and stronger vortices being shed from the top surface of the nacelle.

To better understand the effect of yaw angle on the component velocities of the flow near the blade the contours of pressure with streamlines near the blade tip when it is right above the nacelle are shown in Figure 15. The angle between the inflow and the rotor increases with yaw angle. As yaw angle increases, the axial component of the inflow decreases and causes a reduction of pressure on the pressure side of the blade; the spanwise component of the inflow increases and it is easier for the water to flow across the tip to the suction side of the blade, leading to a recovery of pressure.

Fig. 16 presents two-dimensional contours of axial velocity at the longitudinal cross-section planes for different yaw angles at a TSR of 6. The structures of the wake, including the size of the low-velocity zone, wake expansion rate and axial velocity rate, are almost the same four different yaw angles. The only difference is the asymmetry of the wake strip due to the yaw position. The wake strip is biased to the direction where the rotor is yawed.

5.3 Performance of the OCT at different upstream turbulence intensities

The upstream turbulence intensity is important for the performance of the OCTs operating in an array. Figure 16 presents the comparison of the coefficients of power and thrust of the rotor for three different upstream turbulence intensities, $TI=3\%$, 6% and 9% . These results show that for the relatively low turbulence intensities evaluated in this

analysis TI has very little influence on the mean performance, C_p and C_t , of the turbine over the evaluated TSRs. These relatively low TI values will likely be found in locations where the channel is relatively strait without significant features that will tend to increase turbulence levels or in the open ocean. Previous study has also shown that the mean power and thrust are only slightly dependent on the turbulence intensity [16].

While turbulence intensity has little effect on the power and thrust, it does play a decisive role in the wake of OCTs. Figure 18 presents two-dimensional contours of axial velocity at the longitudinal cross-section planes for upstream turbulence intensities of 3%, 6%, and 9% for a TSR of 6 and a flow speed of 1.6 m/s. These results show that the expansion rate of the flow reduction area increases with increased turbulence intensity. This means that turbine will have a wider wake when operating more turbulent environments, which is consistent with previous observations [16]. The three cases show similar features wake near the rotor, but for the far wake region (beyond 4D downstream) the axial velocity converges much more quickly towards the ambient velocity of 1.6 m/s for higher ambient turbulence intensities.

To more clearly present the convergence of the wake velocity towards ambient flow speeds the centerline wake deficit for each of the three evaluated turbulence intensities is presented in Figure 19 as a function of downstream distance. The axial velocity deficit used to define the recovery rate of the axial velocity and is defined as, $(U-v)/U$, where v is the axial velocity downstream. Distance downstream is normalized using the rotor diameter according to x/D , which is axial distance downstream, x , divided by rotor diameter, D . It is evident that the downstream axial velocity recovers faster at higher turbulence intensities. The axial velocity deficit is around 18% at 10D downstream of the turbine for TI=3%, whereas it has values near 10% and 6% for TIs of 6% and 9%, respectively. These results highlight the importance of understanding the ambient environment, and the impact that the ambient environment has on turbine interactions within an array, since kinetic power is proportional to velocity cubed.

To more clearly visualize the impact that turbulence intensity has on the mean flow field profiles of the normalized axial velocity are presented in Figure 20. These results are normalized according to v/U . The velocity profiles are approximately axisymmetric, with the axes of symmetry located near the turbine axis. The velocity profiles of the three turbulence intensities are nearly overlap near the turbine ($x/D \leq 3$). However, for $x/D \geq 4$ the axial velocity noticeably increases more quickly at higher ambient turbulence intensities towards their ambient values. This means that the turbulence intensity has little influence on the near wake of the turbine, but helps to recover the axial velocity in the wake. Another difference is the width of the wake, the distance between the longitudinal positions where the axial velocity increases to free-stream velocity. The width of wake increases with the turbulence intensity in the far wake. The width of wake is about 2D at 10D downstream from the turbine for TI=3%, and is about 2.5D and 2.8D for TI=6% and TI=9%, respectively.

6. Conclusion and prospects

In this study 3D transient RANS CFD simulations were performed to study the performance, including power, thrust and wake characteristics, of a 3m diameter turbine designed by SNMREC. The numerical method was validated with existing experimental data and good agreement was obtained. Simulations were performed to evaluate the impact of yaw angle and turbulence intensity on the performance of the turbine.

Important results of this research are as follows:

(1) The evaluated turbine design was predicted to obtained a maximum C_p of 0.4642 at TSR=6. The C_t of the turbine continuously increased with TSR over the evaluated range up to a value of 0.8788 at TSR=10.

(2) The blade is stalled at a TSR of 2, suffers partial separation at a TSR of 4 and is fully attached at and beyond a TSR of 6.

(3) Both C_p and C_t decrease with the increase of yaw angle at a greater rate than predicted by simply assuming these decrease as a function of projected area. The peak C_p has therefore been approximated using a quadratic function relationship with the yaw angle. The yaw angle was however shown to have minimal effect on the wake of the turbine.

(4) Changes in turbulence intensity between 3% and 9% were shown to have almost no effect on C_p and C_t , but significantly impact the wake structure behind a turbine at distances greater than 4 diameters downstream. Expansion rate of the wake width has been show to increase with the turbulence intensity, while the axial velocity recovers more quickly at higher turbulence intensity.

Comprehensive CFD studies of hydrokinetic turbines have been done by researchers, as reviewed in the introduction, however, most of the existing papers only gave a brief describing of the CFD modeling procedure. In this paper the CFD model of the horizontal hydrokinetic turbine is described in detail, including the size and generation method of the mesh, the step size and the solution settings. This will help enable readers to reproduce the CFD simulations in this study and to carry out new simulations of similar hydrokinetic turbines.

Compared with experiments, CFD methods have advantages of saving time and money. The only concern is its accuracy. The validation study in this paper illustrates that the RANS CFD method can predict the hydrodynamic forces of the turbine with good accuracy. Therefore the CFD method can be used to evaluate the performance of a hydrokinetic turbine at design stage and shorten the design period. The simulation results also suggest that commercial CFD softwares, which are much easier to be obtained than those professional programming, can give satisfactory results only if the model is properly built.

In this research, the turbine was modeled as rotating about a fixed axis and other turbine movements was ignored. Future work will consider the force of cable and focus on the interaction between the movements and the hydrodynamic forces of the turbine.

Acknowledgements

We would like to thank the National Science Foundation (NSF) and specifically the Energy, Power, Control and Networks (EPCN) program for their valuable ongoing support in this research within the framework of grant ECCS-1307889 ‘Collaborative Research: Optimized Harvesting of Hydrokinetic Power by Ocean Current Turbine Farms Using Integrated Control’. We would also like to thank the Doctorate Foundation of Northwestern Polytechnical University and the support of the High Performance Computing center at Florida Atlantic University.

Reference

1. VanZwieten JH, Duerr AES, Alsenas GM, and Hanson HP. Global ocean current energy assessment: an initial look. In Proc. of the 1st Marine Energy Technology Symposium (METS13) hosted by the 6th annual Global Marine Renewable Energy Conference, Washington DC, April 10-11, 2013. Available: <http://www.foroceanenergy.org/mets/2013-peer-reviewed-mets-papers/>
2. Machado MCPM, VanZwieten JH, Pinos I. A measurement based analyses of the hydrokinetic energy in the gulf stream. Accepted to the Journal of Ocean and Wind Energy, no. JOWE AS-R-06.
3. Georgia Tech Research Corp. Assessment of energy production potential from ocean currents along the united states coastline. Georgia Tech Research Corp, Atlanta, GA, 2013. Available: http://www1.eere.energy.gov/water/pdfs/energy_production_ocean_currents_us.pdf.
4. Khan MJ, Bhuyan G, Iqbal Mt, Quaioc JE. Hydrokinetic energy conversion systems and assessment of horizontal and vertical axis turbines for river and tidal applications: A technology status review. Applied Energy 2009; 86:1823-35.
5. Vermaak HJ, Kusakana K, Koko SP. Status of micro-hydrokinetic river technology in rural applications: A review of literature. Renewable and Sustainable Energy Reviews 2014; 29:625-33.
6. Swales H, Coakley D, Gupta S, Way S. Stability and loads validation of an ocean current turbine: a measurement based analyses of the hydrokinetic energy in the Gulf Stream. In Proc. of 2nd Marine Energy Tech. Symp, Seattle, WA, April 15-18, 2014
7. Bolin WD. Ocean stream power generation: unlocking a source of vast, continuous, renewable energy. In Proc. of 2nd Marine Energy Tech. Symp., Seattle, WA, April 15-18, 2014. Available: http://www.globalmarinerenewable.com/images/pdf/METS_PAPERS_VII/71-Bolin.pdf
8. Borghi M, Kolawole F, Gangadharan S, Engblom A, VanZwieten JH, Alsenas G, Baxley W, and Ravenna S. Design, fabrication and installation of a hydrodynamic rotor for a small-scale experimental ocean current turbine. In Proc. of the IEEE SoutheastCon, Orlando, FL, March 15-18, 2012.

- 441 9. Southeast Marine Renewable Energy Center. SNMREC turbine tow test 1 video posted. Florida Atlantic
442 University, 2014. Available: <http://snmrec.fau.edu/news/snmrec-turbine-tow-test-1-video-posted.html>.
443 Accessed: October 15, 2015.
- 444 10. VanZwieten JH, Oster CM, and Duerr AES. Design and analysis of a rotor blade optimized for extracting
445 energy from the Florida Current. In Proc. of the ASME 2011 International Conference on Ocean, Offshore, and
446 Arctic Engineering, Rotterdam, Netherlands, June 19-24, 2011.
- 447 11. Bahaj A, Molland A, Chaplin J, Batten W. Power and thrust measurements of marine current turbines under
448 various hydrodynamic flow conditions in a cavitation tunnel and a towing tank. *Renewable Energy* 2007; 32:
449 407-26.
- 450 12. Coiro P, Maisto U, Scherillo F, et al.. Horizontal axis tidal current turbine: numerical and experimental
451 investigations. In Proc. of Owemes 2006. Civitavecchia, Italy, April 20-22, 2006.
- 452 13. Galloway PW, Myers LE, Bahaj AS. Experimental and numerical results of rotor power and thrust of a tidal
453 turbine operating at yaw and in waves. In World Renewable Energy Congress 2011, Linköping, Sweden, May
454 8-13, 2011.
- 455 14. Galloway PW, Myers LE, Bahaj AS. Quantifying wave and yaw effects on a scale tidal stream turbine.
456 *Renewable Energy* 2014; 63: 297-307.
- 457 15. Tedds S, Poole R, Owen I, et al.. Experimental investigation of horizontal axis tidal stream turbines. In 9th
458 European wave and tidal energy conference, Southampton, UK, September 5-9, 2011.
- 459 16. Mycek P, Gaurier B, Germain G, et al.. Experimental study of the turbulence intensity effects on marine current
460 turbines behavior. Part I: one single turbine. *Renewable Energy* 2014; 66: 729-746.
- 461 17. Mycek P, Gaurier B, Germain G, et al.. Experimental study of the turbulence intensity effects on marine current
462 turbines behavior. Part II: two interacting turbines. *Renewable Energy* 2014; 68: 876-92.
- 463 18. Shen W, Mikkelsen R, Sørensen N, et al.. Tip loss corrections for wind turbine computations. *Wind Energy*
464 2005; 8:457-475.
- 465 19. Burton T, Sharpe D, Jenkins N, Bossanyi E. *Wind energy Handbook*. John Wiley & Sons Ltd. 2001.
- 466 20. Epps P, Stanway J, Kimball W. An open-source Design Tool for Propellers and Turbines. In Society of Naval
467 Architects and Marine Engineers Propeller/Shafting '09 Symposium, Williamsburg, VA, 2009
- 468 21. Liu, P. A computational hydrodynamics method for horizontal axis turbine-Panel method modeling migration
469 from propulsion to turbine energy. *Energy* 2010; 35: 2843-51.
- 470 22. Lei H, Spyros A, Kinnas A. Vortex-Lattice method for the prediction of unsteady performance of marine
471 propellers and current turbines, *International Journal of Offshore and Polar Engineering* 2013; 23:210-217.
- 472 23. Michael L, Ye L, Danny S. Development and verification of a computational fluid dynamics model of a
473 horizontal-axis tidal current turbine. In 30th International Conference on Ocean, Offshore, and Arctic
474 Engineering ,Rotterdam, Netherlands, June 19-24, 2011.
- 475 24. Monier E, Nilay S, Sinan A. NREL VI rotor blade: numerical investigation and winglet design and
476 optimization using CFD. *Wind Energy* 2013; 17: 605-626.
- 477 25. Hand M, Simms D, Fingersh L, et al.. Unsteady aerodynamics experiment Phase VI: wind tunnel test
478 configurations and available data campaigns. NREL/TP-500-29955, Colorado, 2001.
- 479 26. Yuwei L, Kwang-Jun P, Tao X, et al.. Dynamic overset CFD simulations of wind turbine aerodynamics,
480 *Renewable Energy* 2012;37: 285-298.
- 481 27. Tongchitpakdee C, Benjanirat S, Sankar, L. Numerical simulation of the aerodynamics of horizontal axis wind
482 turbines under yawed flow conditions. *Journal of solar energy engineering* 2005;127: 464-474.
- 483 28. Schleicher WC, Riglin JD, Oztekin A. Numerical characterization of a preliminary portable micro-hydrokinetic
484 turbine rotor design. *Renewable Energy* 2015; 76: 234-41.

- 485 29. Nak L, In K, Chang K, et al.. Performance study on a counter-rotating tidal current turbine by CFD and model
486 experimentation. Renewable Energy 2015;79: 122-26.
- 487 30. Liu J, Lin H, Purimitla SR. Wake field studies of tidal current turbines with different numerical methods.
488 Ocean Engineering 2016; 117: 383-97.
- 489 31. Duerr AES and Dhanak MR. Hydrokinetic power resource assessment of the Florida Current. In MTS/IEEE
490 Oceans Conference 2010. Seattle, September 20-23, 2010.
- 491 32. Althaus, D., Niedriggeschwindigkeitsprofile. Friedr Vieweg & Sohn Verlagsgesellschaft mbH
492 Braunschweig/Weisbaden, Germany, 1996.
- 493 33. UIUC Applied Aerodynamics Group. Airfoil Coordinates Database, University of Illinois at
494 Urbana-Champaign, 2011. Available: http://www.ae.illinois.edu/m-selig/ads/coord_database.html#F. Accessed:
495 October 15, 2015.
- 496 34. Fluent 15.0 user's guide. ANSYS Inc., 2011.
- 497

Figure 1. Photograph of the 20 kW experimental ocean current turbine.

Figure 2. The maximum, average, and minimum ocean current speed

Figure 3. The three views of the solid model of the blade

Figure 4. Domains, boundary conditions and mesh on the surface of the turbine

Figure 5. Detailed mesh information in the domain (a) and near the blade (b).

Figure 6. Results of the numerical method validation

Figure 7. Comparison of the results predicted by CFD and BEM: (a) C_p , (b) C_t

Figure 8. Vortical structures using iso-surfaces of $Q=1$ and colored with magnitude of velocities for different TSRs.

Figure 9. Pressure contours with streamlines at different TSRs and sections of the blade.

Figure 10. Contours of the axial velocity at the mid plane for different TSRs and a flow speed of 1.6 m/s

Figure 11. Comparison of coefficients of power at different yaw angles.

Figure 12. The peak coefficients of power with respect to blade fullness

Figure 13. Comparison of coefficients of thrust at different yaw angles.

Figure 14. Vortical structures using iso-surfaces of $Q=1$ and colored with magnitude of velocities for different yaw angles.

Figure 15. Pressure contours with streamlines near the blade tip when it is right above the nacelle and at different yaw angles.

Figure 16. Contours of the axial velocity at the mid plane for different yaw angles at TSR=6

Figure 17. Comparison of the results predicted by CFD and BEM: (a) C_p , (b) C_t

Figure 18. Contours of the axial velocity at the mid plane for different TIs at TSR=6

Figure 19. Axial velocity deficits for turbulence intensities of 3%, 6% and 9% at TSR=6.

Figure 20. Longitudinal profiles of the normalized downstream axial velocities for turbulence intensities of 3%, 6% and 9% at TSR=6

561

Table 1. Detailed blade description

r/R	φ (deg)	c/R	t/c (%)
0.204	24.069	0.204	23.103
0.248	20.224	0.193	22.227
0.313	16.128	0.178	20.916
0.395	12.469	0.159	19.252
0.490	9.504	0.140	17.340
0.592	7.223	0.122	15.300
0.693	5.486	0.106	14.423
0.788	4.104	0.093	13.601
0.871	2.947	0.085	12.886
0.936	1.992	0.079	12.323
0.979	1.303	0.076	12.100
1	0.992	0.074	12.100

562

563

Table 2 Simulation cases

case	θ (°)	TI (%)
1	0	3
2	5	3
3	10	3
4	15	3
5	0	6
6	0	9

564

565

Table 3 Results of torque and thrust for mesh verification

Approx. number of grid elements (million)	Torque / kNm	Thrust / kN
6.3	1.069	7.317
8.0	1.048	7.206
10.1	1.045	7.202

566

567

Table 4 Setting for the simulation validation

Parameters	Value
Number of blades	3
Blade section	NACA63-8XX
Diameter of the turbine	0.8m
Inflow velocity	1.73m/s
TSR	5~10

568