Characterization of the mean flow field in the far wake region behind ocean current turbines

Parakram Pyakurel[1], Wenlong Tian[2], James H. VanZwieten[1], Manhar Dhanak[3]

[1]*Southeast National Marine Renewable Energy Center, Florida Atlantic University, Boca Raton, FL, USA*

[2]*School of Marine Science and Technology, Northwestern Polytechnical University, Xi'an, Shaanxi, China*

[3]*Department of Ocean &Mechanical Engineering, Florida Atlantic University, Boca Raton, FL, USA*

Corresponding author: Parakram Pyakurel

Address: 777 Glades Road, EE 320, Boca Raton, FL 33431, USA

Email: ppyakurel2014@fau.edu

Characterization of the mean flow field in the far wake region behind ocean current turbines

This paper forms, optimizes, and evaluates three numerical approaches for characterizing mean velocities in far wake region behind ocean current turbines. These approaches are derived from wake models originally developed for wind turbines and are referred here as the Larsen/Larsen, Larsen/Ainslie, and Jensen/Ainslie approaches based on the researchers originally credited with developing the expressions for dependence of the mean wake velocity on centerline and/or radial locations. The numerical coefficients utilized by these approaches are optimized to best match Computational Fluid Dynamics (CFD) generated wake velocity data. After optimizing the coefficients, this study finds that the Larsen/Ainslie and Jensen/Ainslie approaches best match the CFD generated flow data, with Larsen/Ainslie being the best match for an ambient turbulence intensity (TI) of 3% and Jensen/Ainslie being the best match for TIs of 6% and 9%.

Keywords: marine renewable energy; ocean current turbine; in-stream hydrokinetic; wake models; velocity deficit; turbulence intensity

1. Introduction

Ocean currents with time averaged kinetic energy fluxes above 0.5 kW/m^2 can be found along the western boundaries of most of the world oceans (VanZwieten et al. 2013), with values exceeding 2.0 kW/m^2 in several areas (VanZwieten et al. 2015). Multiple research teams are actively pursuing the extraction of this renewable energy resource. It has been estimated that ocean current based electricity production can feasibly approach 163 TWh/year in the US (GeorgiaTech Research Corp. 2013), which is equivalent to 4% of 2014 US electricity production (US EIA, 2015). Furthermore, average kinetic energy fluxes higher than 3.0 kW/m^2 can be found in the ocean currents off SE Florida (Machado, VanZwieten, and Pinos 2016). Ocean current turbine (OCT) prototypes being developed to harness this resource are now approaching the offshore testing stage (VanZweiten et al. 2014). Off Florida, these systems will likely be moored to the sea floor using compliant mooring systems in 300+ meters of water, and will operate in the top 100 meters of the water column (VanZwieten et al. 2013). To enable rapid transition from single device testing to farm scale deployments, research directly related to turbine arrays is needed.

In a turbine array, the performance of downstream turbines are affected by the wake of upstream turbines because of the associated decreased flow speed and increased turbulence intensities. Several experimental studies (Mycek et al. 2014; Myers and Bahaj 2011) have been carried out to study the propagation of wake created by a single turbine, as well as in an array setting. Experimental studies have shown that velocity deficit due to wake are felt beyond 10 rotor diameters (D) downstream and even up to 20 D downstream (Mycek et al. 2014; Myers and Bahaj 2011; Maganga et al. 2010; Bahaj et al. 2007). Therefore, it is important to quantify velocity deficit in a turbine's wake to evaluate the dependence of downstream turbine performance on device spacing.

Experimental studies discussed above were conducted in flume tanks that represent conditions where tidal turbines are subjected to prominent boundary effects. However, OCTs will likely be attached to the sea bed using mooring cables and will operate in water column away from the boundaries (VanZwieten et al. 2013; IHI Corp. 2014). Therefore, boundary effects will not significantly impact these systems. Computational Fluid Dynamics (CFD) can be used as a surrogate for estimating flow downstream from OCTs that operate in deep water where boundary effects can be neglected.

CFD analysis using Reynolds-Averaged Navier–Stokes (RANS) equations have been accurately used to predict the time averaged velocities in the wake of a marine turbine (Batten, Harrison, and Bahaj 2013). Furthermore, the commercial CFD software package FLUENT has been used for simulating and analysing the fluid flow behind tidal and wind turbines (Sun, Chick, and Bryden 2008; Mandas, Cambuli, and Carcangiu 2006).

Apart from CFD, several analytic wake models for quantifying the wake behind wind turbines are also available (Jensen 1983; Larsen 1988; Frandsen et al. 2006). These wake models contain mathematical expressions with empirical coefficients that were originally estimated based on either wind turbine performance or experiments in wind tunnels. Using these expressions, the mean velocity deficit downstream can be calculated along the rotor centerline (Jensen 1983; Frandsen et al. 2006), as well as a function of radial location from centerline (Larsen 1988; Ainslie 1988).

This paper presents, optimizes and evaluates three analytic wake approaches, which are based on expressions originally developed to characterize wake behind wind turbines. These approaches are used to define the mean axial flow speed in the far wake region ($\geq 5\ D$) behind OCTs. First, CFD software ANSYS FLUENT is used to simulate wake flow behind a representative OCT model up to 10 D downstream. Then, empirical coefficients of the considered wind wake models are optimized to best fit the centerline velocity deficit from these CFD data. To predict wake velocity as a function of radial distance from rotor centerline, existing wind wake models are modified and combined to suit OCT wake profiles to form three analytic wake approaches. Finally, wake velocity data calculated using these approaches are compared with CFD results as a function of radial location from centerline.

2. Wake analysis approaches

Three approaches for quantifying mean axial wake velocity behind OCTs are formed in this section. These approaches are referred to as Larsen/Larsen, Larsen/Ainslie and Jensen/Ainslie; based on the names of the researchers who developed original wind turbine wake models that are modified in this paper to form OCT wake models. The Larsen model originally presented in Larsen (1988) is referred as the Larsen/Larsen approach in this paper for naming consistency, whereas expressions from the existing Larsen (1988), Ainslie (1988) and Jensen (1983) models are utilized here to create analytical expressions for wake velocity as a function of downstream and radial locations. These new expressions are referred to as the Jensen/Ainslie and Larsen/Ainslie approaches. It is noteworthy that the actual Ainslie model is a numerical scheme which solves RANS whereas the Larsen/Ainslie and Jensen/Ainslie approaches we have proposed are analytical expressions that do not solve RANS. These approaches are created in the present paper to utilize advantages of Jensen, Larsen and Ainslie models in order to calculate wake velocity at radial location from the centreline. These modified algorithms are presented below.

2.1. Larsen/Larsen approach

The numerical algorithm termed Larsen/Larsen in this paper bases both the wake dependence on downstream centreline distance and radial distance from the rotor's centreline on the work of Larsen (1988). The mean axial wake deficit behind wind turbines were estimated by Larsen (Larsen 1988) using the assumption that Prandtl's turbulent boundary layer equations apply and the mean wake flow is both incompressible and stationary. The equation for the mean axial velocity deficit is given by:

$$1-\frac{U_w}{U_o}=\frac{1}{9}\left(C_T A_d x^{-2}\right)^{\frac{1}{3}}\left(y^{\frac{3}{2}}\left(3c_1 C_T A_d x\right)^{-\frac{1}{2}}-\left(\frac{35}{2\pi}\right)^{\frac{3}{10}}\left(3c_1^2\right)^{-\frac{1}{5}}\right)^2, \qquad (1)$$

where U_o is free-stream velocity, C_T is thrust coefficient, A_d is rotor area, c_1 is non-dimensional mixing length (empirical coefficient), and U_w is wake velocity corresponding to centerline distance x and radial distance y from centerline.

Equation 1 is a function of both centerline distance and radial location, but can be reduced to represent centerline velocity deficit if y is set to zero to obtain the expression:

$$U_c^* = 1 - \frac{V_x}{U_o} = \frac{1}{9}(C_T A_d x^{-2})^{\frac{1}{3}}\left(\left(\frac{35}{2\pi}\right)^{\frac{3}{10}}(3c_1^2)^{-\frac{1}{5}}\right)^2, \qquad (2)$$

where V_x is the centerline velocity at a distance x downstream from the rotor and U_c^* is axial centerline velocity deficit.

If centerline velocity deficits at different downstream distances are available from CFD simulation for an OCT with a known/calculated thrust coefficient and rotor area, empirical coefficient c_1 can be optimized by using Equation 2 to best fit the CFD data. The optimized value of c_1 can then be substituted in Equation 1 to calculate velocity deficit at any radial location.

2.2. Larsen/Ainslie approach

The numerical algorithm termed Larsen/Ainslie approach in this paper utilizes the work of Larsen (Larsen 1988) to quantify the wake's dependence on centerline distance and the work of Ainslie (Ainslie 1988) to define its dependence on the radial distance from the rotor's centreline. Numerical solutions of time averaged Navier-Stokes equations were used to describe wake behind wind turbines by J.F. Ainslie (Ainslie 1988). This model is based on solving differential momentum equation using eddy viscosity turbulence model to calculate wake flowfield. The initial wake velocity deficit at radial distance r and centreline distance 2 D is defined as (Ainslie 1988):

$$1 - \frac{U_w}{U_o} = U_{deficit}^* e^{\left(-3.56\left(\frac{y^*}{b}\right)^2\right)}, \qquad (3)$$

where $U_{deficit}^*$ is the centerline velocity deficit at 2 diameter (D) downstream, y^* is radial distance coordinate (from wake centerline) that has been made non-dimensional by dividing radial distance by rotor diameter, b is wake width parameter, and U_w is wake velocity corresponding to non-dimensional radial coordinate y^*.

The wake width parameter, b, in Equation 3 was calculated by Ainslie (1988) using:

$$b = \left[\frac{3.56C_T}{8U^*_{deficit}\left(1-0.5U^*_{deficit}\right)}\right]^{1/2}. \quad (4)$$

Equations 3 and 4 were only used for 2 D downstream by Ainslie (1988).

Equation 3 makes wake velocity deficit follow a Gaussian profile as a function of radial location from centreline. Since wake velocity deficit at radial locations from centreline is known to follow Gaussian profile (Jensen 1983; Sanderse et al. 2011), this paper seeks to examine applicability of Equations 3 and 4 for all centreline distances, and not just 2 D downstream. Therefore, these equations are modified as follows to test their prediction capabilities at any centerline distance downstream:

$$1-\frac{U_w}{U_o} = U^*_c\, e^{\left(-3.56\left(\frac{y}{2r_o b}\right)^2\right)}, \quad (5)$$

$$b = \left[\frac{3.56C_T}{8U^*_c(1-0.5U^*_c)}\right]^{1/2}, \quad (6)$$

where y is the radial location from turbine centreline, r_o is radius of turbine and U^*_c is the centerline wake velocity deficits for any centreline distance x (not just 2 D). The approach to quantify velocity deficit as a function of radial location (Equation 5) utilized with the value of U^*_c obtained from Equation 2 is termed the Larsen/Ainslie approach.

2.3 Jensen/Ainslie approach

An expression for the centerline wake behind a wind turbine was also developed by Jensen (Jensen 1983). The near field was not modelled and the far field wake was treated as a negative jet. The mean centerline wake velocity was calculated by the Jensen model according to:

$$V_x = U_o\left(1 - 2a\left(\frac{r_o}{r_o+\alpha x}\right)^2\right), \tag{7}$$

where α is an empirical coefficient (generally taken as 0.01 for wind turbines), and a is the axial induction factor. For the velocity distribution in radial location, a Gaussian or bell shaped profile was suggested (Jensen 1983) without presenting any characterizing analytical expressions.

Equations 7 can be re-arranged to calculate centerline wake velocity deficit, U_c^*, by utilizing the relationship between axial induction factor and thrust coefficient (Hansen 2008) as:

$$U_c^* = 1 - \frac{V_x}{U_o} = \frac{(1-\sqrt{1-C_T})}{\left(1+\frac{\alpha x}{r_o}\right)^2}. \tag{8}$$

The Jensen model also defines the wake radius, r_x, as:

$$r_x = r_o + \alpha x. \tag{9}$$

If velocity deficits at different distances downstream behind OCT are generated through CFD simulation for a known thrust coefficient and rotor diameter, empirical coefficient α can be optimized to best fit the CFD simulated data. The determination of α enables calculation of U_c^* using Equation 8 to find centerline velocity. The calculated

U_c^* can be plugged into Equation 5 to find velocity deficits at radial location as a function of centerline distance. This algorithm is termed as Jensen/Ainslie approach.

Thus, both Larsen/Ainslie and Jensen/Ainslie approaches use Equation 5. The Larsen/Ainslie approach uses the U_c^* value obtained from Equation 2 whereas Jensen/Ainslie uses the U_c^* value obtained from Equation 8.

3. CFD data generation and model validation

To optimize the coefficients α (Jensen model) and c_1 (Larsen model), which are associated with wake propagation as a function of centerline distance, CFD data are utilized. CFD data are utilized because experimental wake data are not available for turbines operating at a distance from boundaries where their effects are negligible. To generate these data, the commercial CFD code FLUENT v15.0 is used to solve the incompressible RANS equations using a second-order-accurate finite-volume discretization scheme. The RANS method utilized is real 3-D. The shear stress transport (SST) k-ω turbulence model is selected to model the turbulence terms of the RANS equations. The SST k-ω turbulence model is able to model the transport of turbulent shear stress. It provides accurate predictions of the onset and amount of flow separation under adverse pressure gradients, and has been successfully used in the CFD simulation of wind/water turbines (Lawson, Li, and Sale 2011; Lee et al. 2015). The details of the CFD data generation methodology and validation based on experimental results available in Bahaj et al. (2007) are presented in Tian et al. (2016), and briefly described here for the completeness of this paper.

3.1. Data generation methodology

In this CFD simulation, the computational domain is sized to allow for full development of the upstream flow and to decrease the blockage ratio so that boundary effects are negligible. This is done to simulate flow past an OCT operating away from boundaries, as opposed to a tidal turbine where blockage ratios are high and boundary effects are important. The computation domain is cylindrical with a diameter of 5 D and a length of 14 D, where D is diameter of turbine rotor being simulated. The turbine is placed in the centerline of the cylinder, at a distance of 4 D from the inlet (Figure 1). This figure shows the dimensions of the domains, with the overall domain split into two subdomains. The first subdomain contains the grid elements surrounding the rotor and the second subdomain 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 25 m (Duerr and Dhanak 2013), is applied at the inlet of the computation domain. This results in a rotor diameter based Reynold's number of approximately 5 x 10^6. A sliding mesh model is used to simulate the rotation of the rotor. Each simulation required six revolutions to allow for convergence. The mean performances of the rotor, including thrust and power, are averaged from those calculated during the last revolution. The time step for each simulation is set so that 1° of rotor rotation is achieved at each time step.

Using the data generation methodology described above, simulations were carried out for a reference 20kW three bladed OCT model, with rotor diameter 3 m and hub diameter 0.6 m. The airfoil shapes used by this rotor are created from the FF-77-W airfoil type and other rotor details are available in Tian et al. (2016). In addition to the 3 m rotor utilized to create the primary results presented in this paper, a 0.8 m rotor is simulated using the same CFD approach to validate the utilized methodology. This simulated 0.8 m rotor is designed to match the one used in experimental setup in Mycek et al. (2014). Further validation of the CFD approach utilized here was carried by comparing the CFD approach with experimental setup in Bahaj et al. (2007) and is provided in detail in Tian et al. (2016).

3.2. CFD validation

The CFD methodology presented in this paper was validated based on experimental studies of Bahaj et al. (2007) and Mycek et al. (2014). The performance of turbine in terms of thrust and power coefficients were validated by simulating experimental setup of Bahaj et al. (2007), and this validation is presented in detail in Tian et al. (2016). Likewise, the validation of wake profile generated by the CFD is carried out by comparing the wake profile of the CFD with the experimental studies presented in Mycek et al. (2014). This validation study has not been previously published and is presented here. For this study the CFD simulation of the 0.8 m diameter turbine and experimental condition utilized in Mycek et al. (2014) is carried out to validate wake profile obtained through CFD simulation. It is noteworthy that the CFD simulation of this turbine is carried out only for the validation of CFD generated wake profile and is different than the 3 m diameter OCT CFD simulations described in Section 3.1. It is noted that the diameter based Reynold number used for the CFD simulation is 5×10^6, whereas the diameter based Reynolds number in the experiment range between 0.28×10^6 and 0.84×10^6. An experimental study has shown that for a crossflow hydrokinetic turbine device performance became essentially diameter based Reynolds number independent above 0.8×10^6, but that no significant wake velocity deficit Reynolds number dependence was found for diameter based Reynolds numbers between 0.3×10^6 and 1.3×10^6 (Bachant and Wosnik 2014). Therefore, it is likely that the difference between the experiment and CFD Reynolds numbers will have minimal effect on device mean wake propagation.

Figures 2 and 3 are presented to compare wake flow calculated using CFD with the experimental results published in Mycek et al. (2014) for an ambient turbulence intensity (TI) of 15%. Figure 2 shows variation of centerline velocity deficit as a function of downstream distance, x, which is normalized as x/D. An important observation in Figure 2 is that the CFD analysis predicted a faster centerline velocity decay than the experimental results. Figure 3 shows variation of wake velocity as a function of radial distance, y, for normalized centerline distances, x/D, ranging from 1-10. Here, wake velocity is normalized by dividing wake velocity by freestream velocity. The horizontal straight lines in Figure 3 are error bars as published in Mycek et al. (2014). It is seen that CFD results are capable of calculating the mean axial wake flow field with enough accuracy to appropriately optimize wake analysis approaches presented in Section 2, especially after the transition region from the near wake to far wake field (around 5 D).

4. Coefficient optimization approach

This section presents the optimization of the analytic wake model empirical coefficients using CFD generated data. The wake profile for ambient turbulence intensities of 3%, 6% and 9% are simulated using the CFD methodology presented in Section 3.1 and in detail in Tian et al. (2016). It is noteworthy that these CFD generated data are based on different turbine and boundary conditions than the ones used for validation in Section 3.2. The validation was conducted simulating the experimental setup of Mycek et al. (2014), where the boundary conditions were more prominent and turbulence intensity was 15%.

The CFD data used for coefficient optimization are based on an OCT model (Section 3.1) with the boundaries selected such that boundary effects are negligible. These centerline velocity deficit data are utilized to optimize the coefficients α and c_1 in the Jensen and Larsen models (Equations 2 and 8) respectively. For a given ambient turbulence intensity, the root mean square error ($rmse$) for downstream distances from 5 - 10 D, at increments of 1 D, is minimized by iteratively tuning α and c_1, resulting in values of α and c_1 that are considered to be optimal for each turbulence intensity. This $rmse$ is defined as:

$$rmse = \sqrt{\overline{(U_s^* - U_c^*)^2}}, \tag{10}$$

where U_s^* represents centerline velocity deficits simulated using CFD and U_c^* are the corresponding centerline velocity deficits calculated from Equations 2 and 8. An iterative search algorithm is utilized to vary the coefficients α and c_1 to minimize the $rmse$. Thus, optimal values of α and c_1 are found for ambient turbulence intensities of 3%, 6% and 9%.

5. Results

In order to generate wake flow field data, CFD simulations with ambient Turbulence Intensities (TIs) equal to 3%, 6% and 9%, are performed. The 3 m diameter OCT model briefly discussed in Section 3.1 and presented in detail in Tian et al. (2016) and Borghi et al. (2012) is used for the CFD simulation. Figure 4 presents two-dimensional contours of axial velocity at the longitudinal cross-section planes. It can be seen that the expansion rate of the width of wake increases with the increase in turbulence intensity. This means that a turbine will have a wider wake at higher turbulence intensities, which is consistent with previous observations (Bahaj et al. 2007).

The aim of this paper is to develop computationally inexpensive analytic expressions for calculating the wake velocity behind OCTs without having to use CFD simulation. The model coefficients (α and c_1) are first obtained using CFD, but these coefficients will be used independently without having to generate the values of these coefficients from CFD. CFD generated profiles with low blockage ratios were selected for this study since OCTs will operate in conditions that are minimally affected by boundary conditions and experimental data are not available for these low blockage ratio conditions. Hence, we present a generic setup that is mainly governed by ambient turbulence since there are not sufficient studies examining the effects of ambient turbulence on mean wake profile.

5.1. Calculation of empirical coefficients

Empirical coefficients (α and c_1) are optimized based on minimizing $rmse$ of centerline wake velocity deficits obtained using Equation 10 for each Turbulence Intensity (TI). The optimized coefficients are then plugged in Equations 2 and 8 to compare how well results from these equations match the CFD data. Since 3 sets of TIs (TI= 3%, 6% and 9%) are considered, 3 corresponding values of α and c_1 are obtained (Table 1).

It is seen that Larsen model (Equation 2) is the closest match to CFD results for TI of 3%, with the $rmse$ values for Jensen and Larsen models being 0.021 and 0.019 respectively. Likewise, Jensen model (Equation 8) is closest match to CFD results for TIs of 6% and 9%. The $rmse$ values for Jensen model for TI of 6% and 9% are 0.027 and 0.034 whereas the corresponding values for Larsen model are 0.032 and 0.043.

Our study shows that the empirical coefficients (α and c_1) are strongly affected by TI and non-linear relationships exist between the empirical coefficients and TI (Figure 5). Since the orders of polynomials that relate TI and the values of the coefficients are not clear yet, further experimental and/or numerical studies are required to characterize the dependence of wake profile on TI. This will provide additional data points that will enable the development of mathematical expressions that relate TI with the empirical coefficients.

Figure 6 presents centerline velocity deficit results found using CFD and the Jensen and Larsen models with the optimized coefficients (Table 1) for TIs of 3%, 6% and 9%. It is noted that both wake models predict a slower convergence towards free stream water velocities, and thus more persistent wake profiles, than the CFD generated data. Similar observation is made in Section 3.2 where CFD predicted faster convergence towards free stream than experimental results (Mycek et al. 2014). Therefore, the discrepancy between decay rate from CFD results and wake models do not necessarily indicate a limitation of the analytic models but may also indicate limitation of RANS based simulation of wake profiles. However, as no experimental data of OCT that operate in deep water where boundary conditions can be neglected exist as of now, the present study has utilized RANS simulation despite its potential limitation.

Overall, it is to be noted that Jensen and Larsen models were originally developed for wind turbines but the results in Figure 6 show that these models can also be utilized for hydrokinetic turbines after optimizing empirical coefficients of these models to suit for hydrokinetic turbines.

5.2. Evaluation of wake approaches

Comparison of wake velocity profiles at radial locations for six downstream centerline distances calculated using the wake approaches (see Section 2) and CFD are presented here. These wake profiles are quantified in terms of normalized velocity, U_w/U_o, for centerline distances from 5 D – 10 D. Figures 7-9 present the results for TI of 3%, 6% and 9% respectively. Radial location is presented as a normalized parameter y/D, where y is radial distance from the centerline. Similarly, axial downstream distance (centerline) is presented as x/D.

It can be noted from Figures 7-9 that Larsen/Ainslie and Jensen/Ainslie approach predict very similar velocity profiles as a function of radial locations because they both base their radial dependence on Equation 5. Both these approaches better match the CFD data than Larsen/Larsen approach suggesting that modified expressions from Ainslie model predict the dependence on radial location for ocean current turbines more accurately than the Larsen model. The horizontal straight lines shown in Figures 7-9 are wake radii defined by Jensen model using Equation 9. This wake radius is seen to provide a reasonable estimate of the radial cut-off point beyond which wake approaches cannot be used.

It is observed that the wake velocity is lowest at the turbine centerline and velocity gradually increases with radial distance (Figures 7-9), eventually reaching back to free stream value. Nearly axisymmetric Gaussian curves of wake velocity profiles are observed in Figures 7-9. As the downstream distance x/D increases, wake velocity recovers closer to free-stream and the Gaussian shape gets less prominent. For TI of 9% and $x/D = 10$ (Figure 9), it is seen that the wake velocities at radial locations are almost a constant as Gaussian shape has turned into a near straight line. This is due to the fact that wake velocity has nearly recovered to free-stream.

The $rmse$ averaged over 5 - 10 D for radial locations within radius defined by Jensen model (Equation 9) at TI of 3% are 0.34, 0.34 and 0.46 for Jensen/Ainslie, Larsen/Ainslie and Larsen/Larsen approaches respectively. It is noteworthy that $rmse$ in this section is mean error/differences at radial locations and not centreline locations as in Equation 10. These $rmse$ values are calculated using CFD results as baseline. For TI of 6%, average $rmse$ values for Jensen/Ainslie, Larsen/Ainslie and Larsen/Larsen approaches are 0.16, 0.17 and 0.28 whereas the corresponding values for TI of 9% are 0.16, 0.19 and 0.27.

6. Conclusions

Two new approaches, Larsen/Ainslie and Jensen/Ainslie, are presented in this paper which utilize analytical expression from Larsen, Jensen and Ainslie wind wake models. Our study indicates that these wind turbine based analytic wake models can be applied to ocean current turbines after tuning the empirical coefficients present in the wind wake models. The present study calculates the empirical coefficient values for three ambient turbulence intensities and shows that a non-linear relationships exists between these empirical coefficients and ambient turbulence intensity.

CFD analyses are used to generate wake velocity flow data behind an OCT. These data are used to optimize coefficients α and c_1 of Jensen and Larsen models. The optimized coefficients are then utilized in three presented approaches to characterize wake, both as a function of centerline and radial locations.

The Larsen/Ainslie and Jensen/Ainslie approaches are found to be the closest matches to the CFD generated data for TI of 3% with equal *rmse* values, whereas Jensen/Ainslie approach is found to be the best match of the CFD generated data for TI of 6% and 9%. Overall, Jensen/Ainslie approach is found to be most suitable to predict wake velocity behind OCT.

Acknowledgements

This work was supported by the National Science Foundation (NSF) and specifically the Energy, Power, Control and Networks (EPCN) program under Grant ECCS-1307889 ‘Collaborative Research: Optimized Harvesting of Hydrokinetic Power by Ocean Current Turbine Farms Using Integrated Control’.

References

Ainslie J (1988) Calculating the flowfield in the wake of wind turbines. J. Wind Eng.&Industrial Aerodynamics, 27, 213-224.

Bachant P, Wosnik M (2014) Reynolds number dependence of cross flow turbine performance and near-wake characteristics. Proc. 2nd Marine Energy Technology Symposium, METS 2014, Seattle, WA.

Bahaj A, Molland A, Chaplin J, Batten W (2007) Power and thrust measurements of marine current turbines under various hydrodynamic flow conditions in a cavitation tunnel and a towing tank. Renewable Energy, 32, 407-26.

Bahaj AS, Myers LE, Thomson MD, Jorge N (2007) Chararcterizing the wake of horizontal axis marine current turbines. Proc. 7th European Wave Tidal Energy Conference, Porto, Portugal.

Batten WMJ, Harrison ME, Bahaj AS (2013) Accuracy of the actuator disc-RANS approach for predicting the performance and wake of tidal turbines. Philosophical Transcations A, The Royal Society Publishing.

Borghi, M, Kolawole F, Gangadharan S, Engblom A, VanZwieten JH, Alsenas G, Baxley W, Ravenna S (2012) Design, fabrication and installation of a hydrodynamic rotor for a small-scale experimental ocean current turbine. Proc. IEEE SoutheastCon, Orlando, FL, 2012.

Duerr AES, Dhanak MR (2010) Hydrokinetic power resource assessment of the Florida Current. Proc. MTS/IEEE Oceans Conference, Seattle, September 20-23, 2010.

Frandsen S, Barthelmie R, Pryor S, Rathmann O, Larsen S, Højstrup J, Thøgersen M, (2006) Analytical Modelling of Wind Speed Deficit in Large Offshore Wind Farms. Wind Energy, 40-48.

GeorgiaTech Research Corp (2013) Assessment of Energy Production Potential from Ocean Currents along the United States Coastline. Atlanta, GA.

Hansen, MOL (2008) Aerodynamics of Wind Turbines. Earthscan, pp. 31-33.

IHI Corporation (2014) Power Generation Using the Kuroshio Current: Development of floating type ocean current turbine system. IHI Engineering Review, Vol. 46, No. 2.

Jensen N (1983) A note on wind generator interaction. Risø National Laboratory, Roskilde, Denmark.

Larsen G (1988) A Simple Wake Calculation Procedure. Risø National Laboratory, Roskilde, Denmark.

Lawson M, Li Y, Sale DC (2011) Development and verification of a computational fluid dynamics model of a horizontal-axis tidal current turbine. Proc 30th International

Conference Ocean, Offshore, Arctic Engineering, Rotterdam, Netherlands, June 19-24, 2011.

Lee NJ, Kim IC, Kim CG, Hyun BS, Lee YH (2015) Performance study on a counter-rotating tidal current turbine by CFD and model experimentation. Renewable Energy, 79,122-126.

Machado MCPM, VanZwieten JH, Pinos I (2016) A Measurement Based Analyses of the Hydrokinetic Energy in the Gulf Stream. J Ocean Wind Energy, ISOPE, 3(1), 25-30.

Maganga F, Germain G, King J, Pinon G, Rivoalen E (2010) Experimental characterisation of flow effects on marine current turbine behaviour and on its wake properties. IET Renewable Power Generation, vol. 4, no. 6.

Mandas N, Cambuli F, Carcangiu CE (2006) Numerical prediction of horizontal axis wind turbine flow. Proc European Wind Energy Conference & Exhibition, Athens.

Mycek P, Gaurier B, Germain G, Pinon G, Rivoalen E (2014) Experimental study of the turbulence intensity effects on marine current turbines behaviour, Part I: One single turbine. Reneweble Energy, 66, 729-746.

Myers LE, Bahaj AS (2011) An experimental investigation simulating flow effects in first generation marine current energy converter arrays. Renewable Energy, 37, 28-36.

Sanderse B, van der Pijl SP, Koren B (2011) Review of computational fluid dynamics for wind turbine wake aerodynamics. Wind Energ. 2011; 14:799–819.

Sun X, Chick JP, Bryden IG (2008) “Laboratory-scale simulation of energy extraction from tidal currents. Renewable Energy, Vol. 33, Issue 6, pp. 1267–1274.

Tian W, VanZwiten JH, Pyakurel P, Li Y (In Press) “Influences of yaw angle and turbulence intensity on the performance of a 20kW in-stream hydrokinetic turbine. Energy.

U.S. Energy Information Administration (2015) Annual Energy Review. Web. http://www.eia.gov/. Accessed 1 June 2016.

VanZwieten JH, Baxley WE, Alsenas GM, Meyer I, Muglia M, Lowcher C, Bane J, Gabr M, He R, Hudon T, Steven R, Duerr AES (2015) SS Marine Renewable Energy – Ocean Current Turbine Mooring Considerations. Proc Offshore Technology Conference, Texas, 2015.

VanZwieten JH, Duerr A, Alsenas G, Hanson H (2013) Global ocean current energy assessment: an initial look. Proc Marine Energy Technology Symposium, Washington D.C., April 10-11, 2013.

VanZwieten JH, McAnally W, Ahmad J, Davis T, Martin J, Bevelhimer M, Cribbs A, Lippert R, Hudon T, Trudeau M (2014) In-Stream Hydrokinetic Power – A Review and Appraisal. J Energy Engineering, no. 04014024.

Table 1. Empirical coefficients and $rmse$ variation with TI

	Equation 2		Equation 8	
TI (%)	α	$rmse$	c_1	$rmse$
3	0.0325	0.0216	0.1178	0.0194
6	0.0477	0.0274	0.1656	0.0328
9	0.0679	0.0343	0.2450	0.0430

Figure 1. Domains, boundary conditions and mesh on the surface of the turbine

Figure 2. Centerline velocity deficit.

Figure 3. Normalized axial wake velocity at radial locations for different downstream distances.

Figure 4. Contours of the axial velocity at the mid plane for TI=3% (top), 6% (middle) and 9% (bottom).

Figure 5. Variation of model coefficients with TI

Figure 6. Axial velocity deficits for CFD and analytical models for turbulence intensities of 3%, 6% and 9%.

Figure 7. Wake profiles at radial locations from CFD and wake approaches for TI=3%.

Figure 8. Wake profiles at radial locations from CFD and wake approaches for TI=6%.

Figure 9. Wake profiles at radial locations from CFD and wake approaches for TI=9%.