Optimization and initial testing of a model-scale horizontal axis hydrokinetic turbine

S. Pournazeri, P. Aghsaee, R. Mantilla, and C. Markfort
IIHR-Hydroscience and Engineering, The University of Iowa, Iowa City, Iowa, United States of America

ABSTRACT: This study is concerned with designing and testing a model-scale horizontal axis hydrokinetic turbine (HAHkT) for in-stream hydrokinetic power plant applications. The goal is to design a turbine for lab-scale experiment to investigate strategies for harvesting energy from natural river flows. Optimization of a rotor design was carried out using blade element momentum theory (BEM) so that tip speed ratio and thrust coefficient matches field scale hydrokinetic turbine. The Göttingen 804 foil geometry and tip speed ratio of five were selected to obtain the best performance of the turbine. Furthermore, we measured the velocity profiles at different distances downstream of the turbine to investigate the optimum distance between turbines for maximum power extraction in turbine arrays. This was done for two different depths. The results show that for shallower flow, the wake exhibits a faster recovery rate allowing for a shorter distance between two rows of turbines.

1 INTRODUCTION

Development and expansion of renewable energy continues to be a major focus due to the potential for lower costs and concerns associated with climate change. Hydropower remains the dominant renewable energy source among wind, biomass, solar and geothermal (Hadjerioua et al., 2012; Kao et al., 2014). However, due to the high cost of dam construction and associated environmental concerns, traditional hydroelectric projects are rarely considered. The total energy resource that can be recovered from river channels of the continental United States is estimated about 1,146 TWh/year (EPRI, 2012; VanZwieten, 2015). In-stream hydrokinetic power may offer an environmental friendly alternative to provide distributed power generation that adds to the overall renewable energy portfolio. As an example, the Mississippi River basin provides an ideal location for considering the viability of in-stream hydrokinetic energy generation at a range of stream and river scales. In order to harvest energy at industrial scales, arrays of turbines need to be deployed. However, our understanding about how to best design turbine arrays for in-stream installations is limited. To improve our understanding, a model-scale turbine is designed to mimic the dynamics of full-scale horizontal axis hydrokinetic turbines (HAHkT) for laboratory experiments. The model turbine is designed to optimize the performance based on three scaling criteria: geometry, rotation rate and hydrodynamics forces on the turbine. Blade Element Momentum theory (BEM) is used to analyze the performance of possible designs and the results are compared with measurements collected in a laboratory flume.

Previous studies to optimize the design of hydrokinetic and wind turbines used a variety of different approaches (e.g. Javaherchi et al., 2014; Mukherji et al., 2011; Perfilev, 2013; Kolekar et al., 2013; Stallard et al., 2015). Mukhreji et al (2011) employed different geometry, angle of attack, number of blades, tip speed ratio and flow conditions using computational fluid dynamic simulations to investigate the maximum power generated from the turbine and maximum momentum extracted from the flow. Perfilev (2013) used a differential evolution algorithm (DEM) to obtain better performance of the turbine.

The method utilizes user provided inputs, and a desirability function approach provides the acceptable multi-objective solution. Kolekar et al. (2013) used a coupled computational fluid dynamics-blade element momentum analysis to maximize the performance of a HAHkT. The results of BEM were in good agreement with those obtained from the CFD analyses. Stallard et al. (2015) designed a model-scale turbine and performed experiments to investigate the wake flow characteristics.

Others have focused on the blockage effects on the extracted power by turbines (Hoseyni Chime, 2013) or on interaction of a turbulent open channel flow with a bed-mounted axial-flow hydrokinetic turbine (Chamorro et al., 2013a).

Experimental and numerical modeling have been performed to predict the flow pattern around the hydrokinetic turbines as well as wind turbines (Chamorro et al., 2013b; Kang et al., 2012; Kang et al., 2014; Zhang et al., 2012). Chamorro et al. (2013b) measured three-dimensional velocity in the near wake region of a miniature 3-blade axial-flow turbine within a turbulent boundary layer. Kang et al. (2012, 2014) simulated three-dimensional, turbulent flow past hydrokinetic turbine using Large Eddy Simulation (LES) and verified the predicted wake profile through a comparison with the measurement. Zhang et al. (2012) measured the coherent tip vortices and variation of the mean flow and turbulence statistics in the near wake. From the literature review, it can be seen that there is no investigation has been performed for the effect of flow depth on the wake downstream of hydrokinetic turbines.

In the present study, the hydrodynamic analysis and optimization for a scaled model turbine geometry is developed through a combination of BEM analysis and systematic variation of chord and twist angle distribution. Furthermore, laboratory experiments have been conducted to measure the streamwise velocity profile and wakes at different distances downstream of the scaled turbine model.

2 ROTOR DESIGN METHODOLOGY

Using BEM theory to design a model turbine, the blade element and momentum theory are combined and coupled equations was solved iteratively to determine the fluid forces (thrust and torque). In order to optimize the blade geometry to obtain the maximum turbine power extraction and thrust force, a range of initial values for chord length (between 0.004 m to 0.02 m) was used at three different positions from the hub along the blade (r=0.15R, 0.31R and 0.95R). Here R is the rotor radius and r is the distance from center of the rotor along the blade. Chord length values at the specified locations were varied between the initial values until an optimized model design was identified. The chord distributions are selected to be piecewise-linear. The chord length increases from the tip of the blade to 0.31R and then linearly decreases to the blade root at the hub. The distribution function of twist angle uses an initial exponential function ($f_1 = 46.98e^{-2.93(r/R)}$) added to a linear function ($ar+b$) giving $F=f_1+ar+b$, and varied to find the optimal design geometry (following that of Perfilev, 2013). This results in varying of twist angle between ~-2º and 35º from the tip to the root of the blade. A range for the tip speed ratio (TSR) was considered from 2 to 7 and seven foil types (Göttingen804, NACA0008, NACA0010, S3002, SD7003, SDD8000, and SG6040) were considered.

To analyze the rotor performance, we employed the BEM method, dividing the blade into 21 elements. The values of drag and lift coefficient from (C_D and C_L) of Stallard's (2015) were used in this study (Table 1). By assuming an initial value for angle of attack, the flow angle were calculated. The coefficients Cr and Ca which present the tangential and axial force coefficients of the foil, respectively, were obtained for each element.

$$C_r = C_L \sin\varphi - C_d \cos\varphi \quad (1)$$

$$C_a = C_L \cos\varphi + C_D \sin\varphi \quad (2)$$

where φ is flow angle. Prandtl's tip loss correction factor was incorporated into the algorithm to account for losses due to fluid flow from pressure side to the suction side (Kolekar et al., 2013):

$$F_{tip} = \frac{2}{\pi} \arccos\left(\exp\left(-\frac{N}{2}\left(\frac{R-r}{r\sin\varphi} \right) \right) \right) \quad (3)$$

where N is number of blades, and F_{tip} is tip loss factor. The axial and tangential induction factors (a and a', respectively) are calculated using the flow angle, tip loss factor, thrust coefficient, momentum coefficient and solidity ($\sigma = N \cdot C/2\pi r$) where C is chord length at the position of interest.

$$a = \frac{1}{\frac{4F_{tip}\sin^2\varphi}{\sigma C_a} + 1} \tag{4}$$

$$a' = \frac{1}{\frac{4F_{tip}\sin\varphi\cos\varphi}{\sigma C_r} - 1} \tag{5}$$

Table 1. Values of Lift Coefficient (C_l), drag coefficient (C_D), and angle of attack (α) of Stallard et al. (2015)

α deg	C_l	C_D
-30	-0.7	0.48
-20	-0.5	0.25
-10	-0.3	0.13
-7	-0.4	0.12
-6	-0.51	0.11
0	0.24	0.06
1	0.37	0.06
2	0.49	0.06
4	0.74	0.06
8	1.24	0.1
9	1.19	0.12
10	1.11	0.14
12	0.97	0.23
14	0.91	0.29
16	0.89	0.35
18	0.91	0.41
20	0.94	0.46
25	1.06	0.59
30	1.16	0.74
35	1.19	0.89
40	1.16	1.01
45	1.11	1.07
50	1.1	1.07

One of the limitations of the BEM theory is that when axial induction factor is greater than a_c=0.2, the basic theory becomes invalid and the rotors enters the turbulent wake state. According to momentum theory, this operating state occurs when flow in the far wake starts to propagate upstream which is a violation of the basic assumptions of BEM theory. Glauert (1935) developed a correction to the rotor thrust coefficient

$$K = \frac{4F_{tip}\sin^2\varphi}{\sigma C_a} \tag{6}$$

The induction factor will be then corrected using following formulae

$$a = \frac{1}{2}\left(2 + K(1-2a) - \sqrt{\left(K(1-2a_c)+2\right)^2 + 4\left(Ka_c^{\,2}-1\right)}\right). \tag{7}$$

By applying the calculated induction factors, the new value of flow angle, lift and drag coefficient can be determined for each blade element.

The flow angle can be calculated as

$$\varphi = \arctan\left(\frac{1}{TSR \cdot \frac{r}{R}} \cdot \frac{(1-a)}{(1+a')}\right).$$

This procedure is applied iteratively until the difference between two consecutive flow angles becomes smaller than a given tolerance. This process is repeated for other elements to obtain the total thrust and power coefficients over the length of the blade. The maximum power and thrust coefficients are obtained using the best foil shape, TSR, chord length and twist angle distributions. This procedure was implemented for a turbine with diameter D = 0.1 m in a steady water flow with a mean velocity of U_0 = 0.45m/s.

In this study, two model rotors were considered. One of them was based on the developed design procedure and the other is a commercial model.

3 OPTIMIZATION OF TURBINE GEOMETRY

Among the seven foil shapes considered, the Göttingen 804 resulted in the best performance. For a TSR = 5, the optimized chord and twist angle distribution was found to produce a maximum power coefficient of C_P = 0.28 and thrust coefficient of C_T = 0.83. Figures 1 and 2 show the different distributions of twist angle and chord length considered for the optimization, respectively. A combination of 949 different distributions of chord length and 572 distribution of twist angle were considered in the optimization model to obtain the best value of thrust and power coefficients of the turbine. Among those, the blade with the highest value of both thrust and power coefficients were selected and considered as the optimal blade geometry.

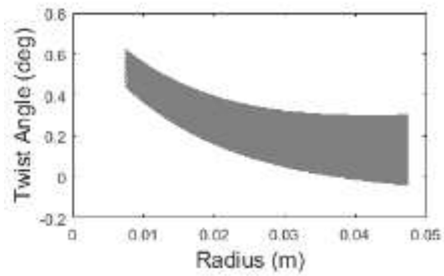

Figure 1. Twist angle distributions for optimization of the turbine model

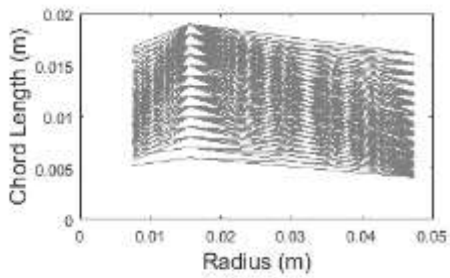

Figure 2. Chord length distributions for optimization of the turbine model

A 3D model of this turbine was manufactured using a high-resolution Fortus 250 3-D printer (Fig. 3)

Figure 3. 3D model of designed version of turbine

Figures 4 shows the distribution of chord length of the turbine model along the blade. In Figure 5, the change of twist angle of the turbine model versus distance from the turbine's center is shown.

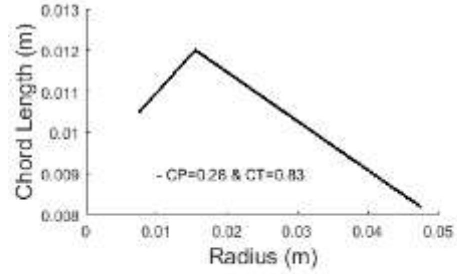

Figure 4. Chord length distribution of the designed turbine model

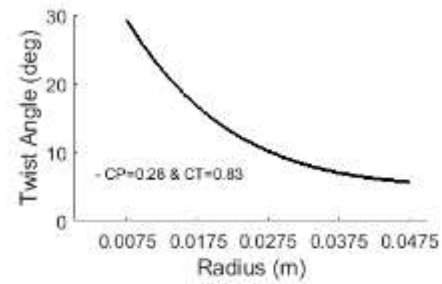

Figure 5. Twist angle distribution of the designed turbine model

Figure 6 shows the distribution of power coefficient and thrust coefficient with different values of tip speed ratio (TSR). The thrust coefficient (C_T) is 0.83 at the tip speed ratio of 5 while power coefficient (C_P) is 0.31 at TSR = 4 and decreases to almost zero as TSR increases to 6.

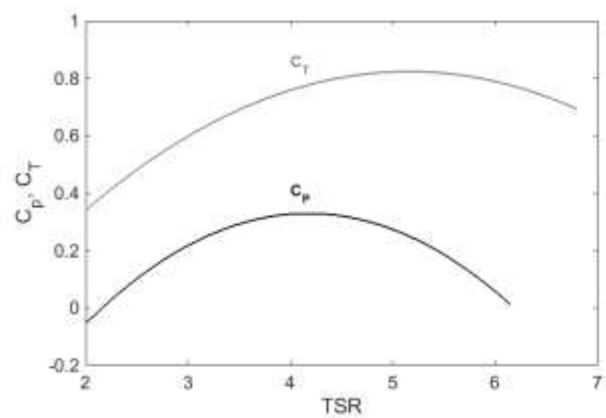

Figure 6. Variation of power and thrust Coefficient versus tip speed ratio for the designed turbine's model

As the thrust coefficient determines the nature of the wake flow (Sforza et al., 1981), it is important to verify the calculated thrust coefficient with the existing models. The calculated thrust coefficient in our turbine's model is close to that in model scale turbine of Stallard et al. (2015).

(b)

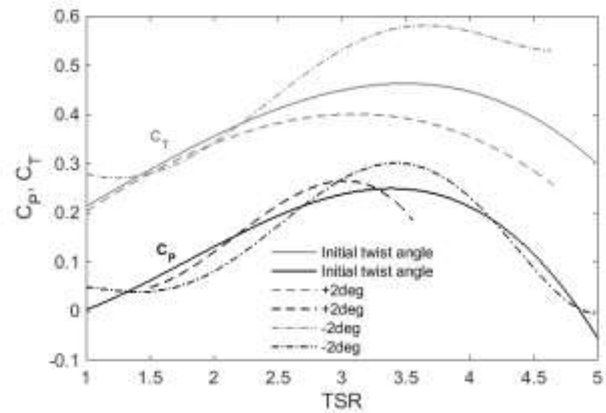

Figure 8. Sensitivity of C_P and C_T to the changes of twist angle for ±2° (commercial turbine)

4 EXPERIMENTAL RESULTS

The optimized model turbine was tested in flume experiments. As the available material (digital ABS) of the 3D printing was found not to have adequate strength to withstand the forces of the flow, we used a commercial turbine with similar scales as the optimized turbine with a more rigid material. The diameter of the commercial turbine is 0.11 m and the model of the rotor is E-flite 3.95*3.95 Ultra micro icon A5. The geometry of this turbine has been given in Table 2.

Table 2.Values of chord length and twist angle of the commercial turbine.

Chord (m)	0.013	0.013	0.0123	0.01111	0.00635
Twist (deg)	32	25	15.5	13	10

In order to ensure the accuracy of the commercial turbine's performance, a sensitivity analysis has been conducted for this turbine. A positive and negative increment of 1 mm for chord length and 2° for twist angle were added to the initial values of chord length and twist angle. It has been shown that the performance of the turbine is more sensitive to small changes in twist angle (Fig. 8) compared to that in chord length (Fig. 7).

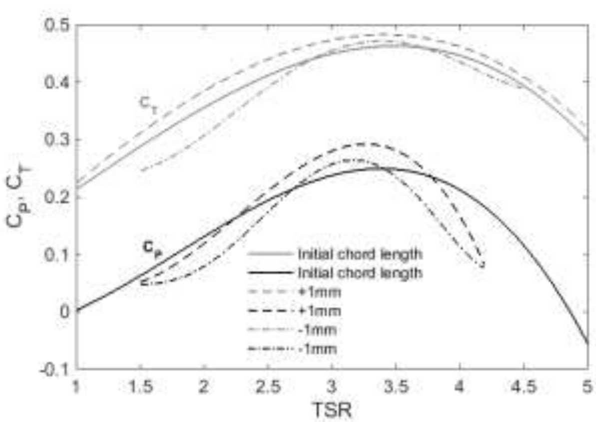

Figure 7. Sensitivity of C_P and C_T to changes of chord length for ±1 mm (Commercial turbine);

The maximum power and thrust coefficients of the commercial turbine considering the initial distributions of chord length and twist angle were calculated as C_P = 0.25 and C_T = 0.46, respectively. By changing the chord length with an increment of ±1 mm, the maximum thrust coefficient (C_T) ranges from 0.46 to 0.48, while the power coefficient (C_P) varies between 0.25 and 0.29 (Fig. 7). Figure 8 shows that by changing the twist angle with an increment of ±2°, the maximum thrust coefficient (C_T) ranges from 0.4 to 0.58, while the maximum power coefficient varies between 0.26 and 0.3. The results indicates that thrust coefficient is more sensitive to the changes of twist angle than power coefficient. This can be seen for tip speed ratios larger than 2.5 (TSR>2.5).

Experiments were conducted to assess the effects of confined channel flow on the behavior of the wake downstream of the model turbine. The goal was to investigate the effects of depth variations on wake recovery, which is an important factor when designing the layout and spacing between multiple turbines within a turbine array. We conducted experiments in a 30 m long, 0.9 m wide, and 0.5 m deep recirculating water channel. The turbine used in these experiments has a diameter of 0.11 m and hub height of 0.106 m. Two preliminary experiments, using different flow depths of 0.17 m and 0.23 m, were performed placing a single turbine 26 m downstream of the entrance section of the channel. This distance was long enough for the boundary layer to fully develop. This was evaluated by comparing mean streamwise flow velocity (U) profiles at different distances downstream of this location and making sure that the flow was self-similar.

A SonTek Acoustic Doppler Velocimeter (ADV) was used to measure vertical profiles of the three dimensional instantaneous velocity fields of the incoming flow as well as the wake downstream of the turbine. The wake measurements were made at multiple distances (3D, 5D, 10D and 13D) downstream the turbine, where D represents the turbine rotor diameter. As flow in most natural channels is uniform except near hydraulic structures such as dams, bridges piers, the entrance to large reservoirs or near junctions with larger rivers, the assumption of flow with constant velocity is an accurate approximation of flow condition in current channels. In this study, the flow rate was adjusted to achieve an incoming flow velocity of 0.45 m/s at the elevation of the turbine hub. This condition was held constant for both experiments.

The evolution of the mean velocity downstream of the turbine for both experiments is shown in Figure 9, the minimum stream-wise velocity occurs near the hub height level for both experiments.

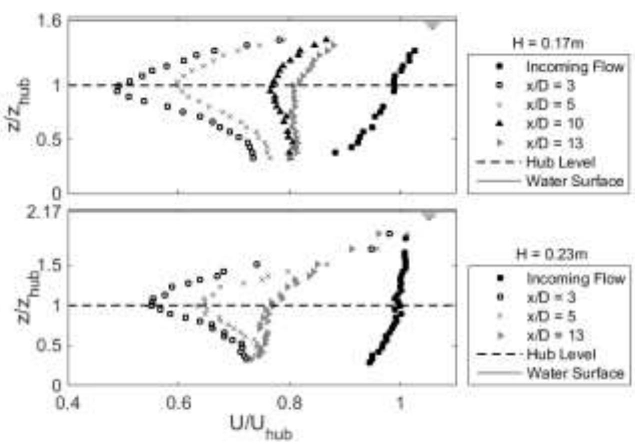

Figure 9. Mean streamwise velocity profiles in the wake of a commercial model turbine for 0.17 m and 0.23 m flow depths.

Figure 10 shows the maximum velocity deficit at the hub-height level versus the downstream distance from the turbine. The wake associated with the shallower flow exhibits a faster recovery rate. In experiment with water depth of 17 cm, the velocity at the hub height is 80% of the incoming flow at 13D downstream of the turbine. The velocity at the hub height in the experiment with water depth of 25 cm is 76% of the incoming flow at 13D downstream of the turbine.

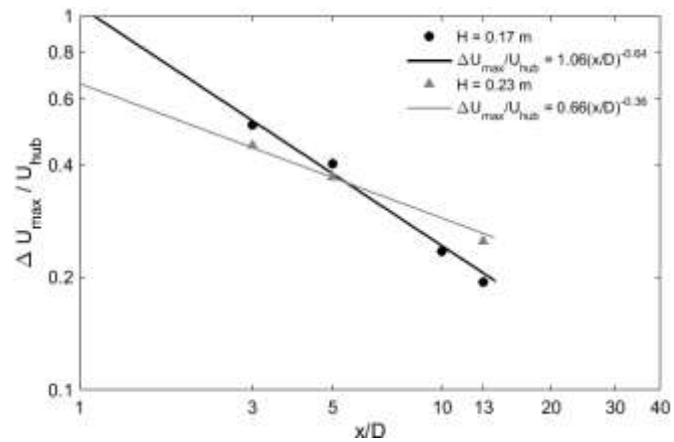

Figure 10. Normalized velocity deficit versus downstream distance

The effect of depth on the wake recovery is important when considering the optimum spacing between turbines for in-stream hydrokinetic turbine (InSHK) energy applications. These preliminary results are the first experiments to consider the wake behavior for a single turbine with different flow depths. Preliminary results show that changes in water level affect the dynamics of the flow and there is a need to investigate a range of flow depths and wake interactions with neighboring turbines.

5 CONCLUSIONS

In this study, we investigated the optimum design for the a scaled model turbine geometry, in terms of model performance compared to full-scale HAHkT using a combination of BEM and systematically varied chord and twist angle distribution. Using an optimization procedure, the Göttingen 804 foil geometry and tip speed ratio (TSR) of 5 were found to have the best performance. The power coefficient and thrust coefficient in the designed turbine model were calculated as $C_P = 0.28$ and $C_T = 0.83$, respectively. The available materials used in the 3-D printing were found to be too weak to manufacture a turbine with blades rigid enough to withstand hydrodynamic forces. Therefore, a commercial turbine was employed in initial tests. A sensitivity analysis was conducted which found the commercial turbine perform similarly to that of the design. It has been shown that thrust coefficient of the turbine is more sensitive to the change of twist angle rather than the chord length. The power coefficient is less sensitive to the change of twist angle distribution compared to the thrust coefficient.

The velocity measurements in flow direction at the hub height of the turbine show that the wake associated with the shallower flow exhibits a faster recovery rate. At 13D downstream of the turbine within the flow depth of 17 cm, the velocity is 80% of the incoming flow at the hub height. For the study of performance of turbine array, the downstream turbines should be located in a way that they avoid the wakes produced by upstream devices as much as possible. The results show that the downstream turbines should be located far from the first turbine if the downstream turbines are aligned with the upstream ones. Using low water depth benefits us to place more turbine in a specific length of river channel.

REFERENCES

Chamorro, L. P., Hill, C., Morton, S., Ellis, C., Arndt, R. E. A., and Sotiropoulos, F. 2013a. On the interaction between a turbulent open channel flow and an axial-flow turbine. J. Fluid Mech., 716: 658–670.

Chamorro, L. P., Troolin, D.R., Lee S.J., Arndt R.E.A., Sotiropoulos, F. 2013b. Three-dimensional flow visualization in the wake of a miniature axial-flow hydrokinetic turbine. Experiments in Fluids, 54: 1459.

EPRI. 2012. Assessment and Mapping of the Riverine Hydrokinetic Resource in the Continental United States. Technical Report.

Glauert, H. 1935. Airplane Propellers. Aerodynamic Theory (W. F. Durand, ed.), Div. L, Chapter XI. Berlin: Springer Verlag.

Hadjerioua, B., Wei, Y. and Kao, S. G. 2012. An Assessment of Energy Potential at Non-Powered Dams in the United States. Report prepared by Oak Ridge National Laboratory Oak Ridge, Tennessee 37831.

Hoseyni Chime, A. 2013. “Analysis of Hydrokinetic Turbines in Open Channel Flows.” MSc Thesis, University of Washington.

Javaherchi, T., Stelzenmuller, N., Seydel, j., and Aliseda, A. 2014. Experimental and numerical analysis of a scale model horizontal axis hydrokinetic turbine. Proceedings of the 2nd Marine Energy Technology Symposium METS 2014 April 15-18, 2014, Seattle, WA.

Kang, S., Borazjani, I., Colby, J.A., and Sotiropoulos, F. 2012. Numerical simulation of 3D flow past a real-life marine hydrokinetic turbine. Advances in Water Resources. 39:33–43.

Kang, S., Chamorro, L., Hill, C., Arndt, R. and Sotiropoulos, F. 2014. Flow structure interaction around an axial-flow hydrokinetic turbine: Experiments and CFD simulations

Kao, S. C., McManamay, R. A., Stewart, K. M., and Samu, N. M., Hadjerioua, B., DeNeale, S. T., Yeasmin, D., Fayzul, M., Pasha, K., Oubeidillah, A. A., and Smith, B. T. 2014. New Stream-reach Development: A Comprehensive Assessment of Hydropower Energy Potential in the United States, Report prepared by Oak Ridge National Laboratory Oak Ridge, Tennessee 37831-6283.

Kolekar, N., Hu, Z., Banerjee, A., and Du, X. 2013. Hydrodynamic design and optimization of hydrokinetic turbines using a robust design method. Proceedings of the 1st Marine Energy Technology Symposium, METS13, April 10-11, 2013, Washington, D.C.,

Mukherji, S. S., Kolekar, N., Banerjee, A., and Mishra, R. 2011. Numerical investigation and evaluation of optimum hydrodynamic performance of a horizontal axis hydrokinetic turbine. Journal of Renewable and Sustainable Energy, 3, 063105.

Perfilev, D. 2013. Methodology for wind turbine blade geometry optimization. PhD thesis, Lappeenranta University of Technology, Finland.

Sforza, P. M, Sheerin, P., Smorto, M. 1981. Three dimensional wake simulated wind turbines. AIAA Journal, 19(9):1101-1107.

Stallard, T; Feng, T; Stansby, P. K. 2015. Experimental study of the mean wake of a tidal stream rotor in a shallow turbulent flow, J. Fluid & Structures. 54:.235–246.

VanZwieten, J., McAnally, W., Ahmad, J., Davis, T., Martin, J., Bevelhimer, M., Cribbs, A., Lippert, R., Hudon, T., and Trudeau, M. 2015. In-Stream Hydrokinetic Power: Review and Appraisal. J. Energy Eng., 141(3), 04014024.

Zhang, W. Markfort, C. D., Porte-Agel, F. 2012. Near-wake flow structure downwind of a wind turbine in a turbulent boundary layer, Journal of Experimental Fluids, 52:1219-1235.