Renewable Energy 97 (2016) 274–283

Contents lists available at ScienceDirect

Renewable Energy

journal homepage: www.elsevier.com/locate/renene

# Hydrokinetic turbine array characteristics for river applications and spatially restricted flows

Jacob Riglin [a], Cosan Daskiran [a, b], Joseph Jonas [a], W. Chris Schleicher [a], Alparslan Oztekin [a, *]

[a] P.C. Rossin College of Engineering and Applied Science, Lehigh University, Bethlehem, PA 18015, USA
[b] Turkish Military Academy, Department of Mechanical Engineering, Ankara, Turkey

ARTICLE INFO

*Article history:*
Received 7 December 2015
Received in revised form
21 April 2016
Accepted 24 May 2016

*Keywords:*
Micro-hydrokinetic turbines
River applications
Turbine performance
Computational fluid dynamics
Turbine array
Hydro-farms

ABSTRACT

Multiple hydrokinetic turbines in three array configurations were characterized computationally by employing Reynolds Averaged Navier-Stokes equations. The simulations were conducted for pre-existing turbines operating at their optimum power coefficient of 0.43 which was obtained by design and optimization process. Mechanical power for two adjacent units was predicted for various lateral separation distances. An additional two-by-two turbine array was studied, mimicking a hydro-farm. Numerical simulations were performed using actual physical turbines in the field rather than using low fidelity models such as actuator disk theory. Steady state simulations were conducted using both Coupled and SIMPLE pressure-velocity solvers. Steady three dimensional flow structures were calculated using the $k$-$\omega$ Shear Stress Transport (SST) turbulence model. At a lateral separation distance of $0.5D_t$, the turbines produced an average 86% of the peak power a single turbine producing. Interaction effects at lateral separation distances greater than $2.5D_t$ were negligible. The wake interaction behind the upstream turbines causes a significant performance reduction for downstream turbines within $6D_t$ longitudinal spacing. Downstream turbines employed for the present study performed around 20% or less of a single unit turbine performance for the same operating conditions. Downstream turbines yielded comparable reductions in power to that of experimental results.



## 1. Introduction

Hydropower will provide renewable energy indefinitely as a result of the natural hydrologic cycle. It provides about 78 GW of energy production per year, which amounts to 8% of total U.S. electricity generating capacity [1]. Hydrokinetic turbines use kinetic energy from streams to drive the rotor, thus reducing the need for large-scale civil construction that conventional hydropower systems require. The technically recoverable hydrokinetic energy estimated in the continental United States is almost 120 TWh/yr [2]. The operating principles behind the hydrokinetic turbines are similar to wind turbines which extract kinetic energy from wind to generate electricity. Although water's density is approximately 815 times greater than air, micro-hydrokinetic turbines are less capable than wind turbines to produce large amount of power in rivers due to limitations in channel cross sectional area. Micro-hydrokinetic arrays will be essential to meet commercial power generation goals for river applications [3].

Rivers generally have lower free stream velocities compared to air speeds. They are also limited by depth, which inhibits using larger hydrokinetic turbines. River data provided by the United States Geological Survey (USGS) regarding free stream velocities and average depths, many rivers in the United States have approximately 3 m or less of average depth. Additionally, only 51.0% of rivers have velocities suitable for hydrokinetic turbine operation assuming a free stream velocity ranging from 0.75 m/s to 2.5 m/s [4].

Works conducted by Mukherji et al. [5] and Kolekar et al. [6,7] optimized a horizontal axis hydrokinetic turbine using the blade element momentum (BEM) theory combined with computational fluid dynamics (CFD) analysis of a 12 kW design. Riglin et al. [8–13] and Schleicher et al. [14,15] designed and characterized a portable hydrokinetic turbine unit. The design introduced by Schleicher et al. [16,17] incorporated both a propeller structure and a higher value of solidity comparable to that observed in the efforts made by

* Corresponding author.
*E-mail address:* alo2@lehigh.edu (A. Oztekin).

http://dx.doi.org/10.1016/j.renene.2016.05.081
0960-1481/

Mukherji et al. [5] and Kolekar et al. [6,7] along with other studies conducted involving traditional hydrokinetic and tidal turbine designs [18]. The unit designed by Shleicher et al. [16] produces a maximum power coefficient ($C_P$) of 0.43. Free surface effects were investigated by Riglin et al. [12] for a hydrokinetic turbine unit and Liu et al. [19] for off-shore structures in marine current applications.

In a hydro farm, an array configuration influences the individual performance of turbines and their stability. Array configurations are an important design consideration for hydrokinetic systems. Downstream turbines in the array produce less energy due to the upstream turbine wakes [20]. Therefore, an optimal downstream turbine position is crucial to maximize total power output. Several experimental, analytical and numerical studies have been conducted to determine the most efficient turbine array configuration. In these studies, the hydrokinetic turbine is typically modelled using actuator line, actuator disk or blade element momentum (BEM) theory. Significant advancements in modelling both wind turbine arrays at the meso-scale [21–24] as well as hydro turbine arrays at both micro and macro scales [3,20,25–29] have been accomplished. In the case of Barthelmie et al. [21–24], due to the size and scope of modelling large wind farms encompassing vast amounts of area, the use of lower fidelity numerical models, consisting predominantly of actuator disk theory, was necessary. Many wind farm analyses included determining each array's normalized power output, the resulting wake dynamics and the wind speed gradients resulting from land-sea surface temperatures.

The first tidal turbine hydro farm in the world is the La Rance tidal power plant. This power plant includes 24 tidal turbine units with a combined peak power output of 240 MW. The Rance River's (Brittany, France) average tide depth is 8.2 m and peaks at 13.5 m [30]. Verdant Power installed a six turbine array to benefit the tidal currents in East River in New York City [20,25]. Analysis using BEM software was used to assess the electrical power availability in the Alderney Race by Myers et al. [31], predicting an annual energy output of 1340 GW.

Gebreslassie et al. [3] used analytical models utilizing linear momentum actuator disk theory and CFD tools to tidal turbine wake interactions. It was concluded that more than 91% of the power generated by a single unit turbine can be extracted when the stream wise spacing between turbines is 20*D*. Harrison et al. [26] used blade element and uniform actuator disk models to simulate five tidal turbines in a single column with $7D_t$ spacing among disks. These investigators documented that the first turbine in the stream-wise direction generates the highest power while the second turbine generates the least power in the array. The decrease in performance of the second unit is due to the substantial decrease in wake velocity following the first unit. Units following the second unit do not experience the wake effects of the same magnitude.

Malki et al. [27] and Turnock et al. [32] used a coupled BEM-CFD model to simulate tidal turbine arrays influenced by lateral and longitudinal separation. Jo et al. [29] studied the performance of multi-arrayed tidal turbines numerically and experimentally using meshed disks. They showed that the upstream turbine affects downstream turbine power generation adversely while turbines placed side by side generate almost the same power compared to a single turbine. Myers et al. [33,34] used mesh disk rotors to model tidal turbines in arrays and under various loading conditions to determine array configurations that would provide potential power increase in turbine arrays. Lee et al. [28] optimized turbine arrangement to extract power from ocean currents more efficiently. They modelled ocean current turbines with detailed geometry and rotation affect is included by frozen rotor approach. Mycek et al. [35,36] provides a comprehensive experimental analysis for a single marine current turbine and a two-turbine setup at various downstream separations distances. Results indicate that at a flow turbulence of three percent and six diameter turbine separation, an approximate 70% drop in power and a 46.3% drop in thrust occur.

This manuscript highlights performance predictions determined numerically for multiple hydrokinetic turbine arrays for river applications. The micro-hydrokinetic turbine simulations are conducted with actual turbine geometries incorporated in the fluid domain, providing more accurate power and thrust results corresponding to a specific design. Using the hydrokinetic design investigated by Schleicher et al. [14,15] and Riglin et al. [8–13], higher fidelity results are obtained for hydro farms for river applications The hydrokinetic turbines are placed in four configurations: single turbine, two side by side turbines, two as upstream-downstream turbines and four turbines in a two-by-two, non-staggered matrix. Comparisons are drawn between the current hydrokinetic design suited for river application to the numerical work of Churchfield et al. [20,25] and Turnock et al. [32] and the experimental work provided by Jo et al. [29] and Mycek et al. [35,36].

## 2. Modelling and numerical method

### 2.1. Turbine design and river domain

The propeller turbine was designed by the present authors using a bottom-up approach. Geometric turbine design parameters are determined based on power and free stream velocity [16]. Schleicher et al. [16] proposes a high solidity ($\sigma$) compact turbine, as shown in Fig. 1. The tip diameter, $D_t$, hub diameter, $D_h$, mean diameter, $D_m$, and wrap angle, $\Delta\theta$, are illustrated in Fig. 1(a). The meridional blade length, $\Delta m$, and the relative blade angle, $\beta'$, are displayed in Fig. 1(b). Schleicher et al. [16] provides the step by step design utilizing an empirical relation of relative blade angles formulated by Cebrián et al. [37]. Solidity, $\sigma$, and blade number, $Z_B$, are chosen *a priori*. All input and output design parameters for the propeller based turbine design are listed in Table 1.

The top and front view of the computational river domain is depicted in Fig. 2. The top view of the turbine array provided in Fig. 2(a) illustrates corresponding lateral, $S_1$, and downstream, $S_2$, separation distances between successive turbines and domain dimensions, width, *W*, length, *L*, and height, H. Lateral separation distance is defined as the distance from blade tip to blade tip between two adjacent turbine units. Fig. 2(b) contains the front view of the domain setup, including the boundary conditions imposed on the velocity field at the walls, *w*. Domain settings variables depicted in Fig. 2 are specified in Table 2 for each case investigated in the present study.

The Cases 1 through 6 investigated had a $5.71D_t$ channel height and a $2.19D_t$ turbine submersion depth from the channel's zero-shear surface. The channel height and the submersion depth are large enough to minimize the wall effects on the turbine performances. Upstream turbines were situated $5.71D_t$ away from the domain inlet. The domain outlet is positioned $11.42D_t$ away from the turbine, minimizing the boundary condition's effect on the computed flow field. For Cases 1 through 5 the blockage ratio (ratio of total turbine swept area to domain cross sectional area) was approximately 1.6%, ensuring that the modeled flow represents a nearly infinite medium. In Case 7, the downstream rotor had a 33.3% reduction in the angular velocity from the value listed in Table 1 to account for the expected drop in the axial velocity entering to rotor. Jo et al. [29] and Churchfield et al. [20,25] considered the same arrangements of actuator disks for Case 1 and Case 6, respectively. The present study is differentiated from these studies by considering actual turbine geometries in the simulations.