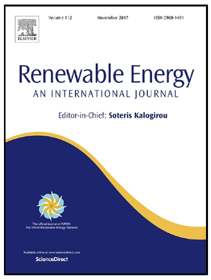
Accepted Manuscript



Numerical Simulation and Dynamical Response of a Moored Hydrokinetic Turbine Operating in the Wake of an Upstream Turbine for Control Design

Parakram Pyakurel, James H. VanZwieten, Cornel Sultan, Manhar Dhanak, Nikolaos I. Xiros

PII: S0960-1481(17)30750-4

DOI: 10.1016/j.renene.2017.07.121

Reference: RENE 9096

To appear in: *Renewable Energy*

Received Date: 23 September 2016

Revised Date: 23 May 2017

Accepted Date: 30 July 2017

Please cite this article as: Parakram Pyakurel, James H. VanZwieten, Cornel Sultan, Manhar Dhanak, Nikolaos I. Xiros, Numerical Simulation and Dynamical Response of a Moored Hydrokinetic Turbine Operating in the Wake of an Upstream Turbine for Control Design, *Renewable Energy* (2017), doi: 10.1016/j.renene.2017.07.121

Numerical Simulation and Dynamical Response of a Moored Hydrokinetic Turbine Operating in the Wake of an Upstream Turbine for Control Design

Parakram Pyakurel[1,a], James H. VanZwieten[a], Cornel Sultan[b], Manhar Dhanak[a], and Nikolaos I. Xiros[c]

[1] Corresponding Author
[a] Florida Atlantic University
[b] Virginia Tech
[c] University of New Orleans

***Abstract*—Numerical simulation of a downstream hydrokinetic turbine operating in the wake of an upstream turbine for feedback control design is presented. Wake effects from an upstream turbine are quantified in terms of wake velocity and amplified turbulence levels. These effects are integrated in an in-stream hydrokinetic turbine numerical simulation that utilizes a Blade Element Momentum approach with a dynamic wake inflow model. Simulations are carried out on a fixed turbine model to simulate operation in river or tidal channels with conventional foundations, as well as on a compliantly moored turbine model such as those designed to operate in open ocean currents.**

***Index Terms*— Hydrokinetic Power, Marine Renewable Energy, Ocean Current Turbines, Numerical Simulation, Wake, Turbulence.**

I. Introduction

IN-STREAM hydrokinetic turbines utilize water currents in oceans, tidal flows and rivers to produce electricity. These turbines have the potential to play a vital role in the future energy supply in many countries around the world [1], and several turbines have been designed [2, 3, 4] to harness this energy. For example, a 300 kW prototype tidal turbine was installed in the United Kingdom in 2003 [2]. Similarly, a 25 kW turbine was deployed to produce power from river currents in a remote village in Alaska [5]. A total of 14 in-stream hydrokinetic projects have been deployed so far and more than 350 other projects are planned for installation according to [6].

In-stream hydrokinetic technologies are beginning to transition from single device testing to the installation of small grid connected turbine farms. Two turbines have been deployed as the start of a small farm in Scotland [7], with eight more planned [8]. Similarly, Verdant Power has conducted prototype and pre-commercial testing of its turbines and is authorized to install up to thirty turbines in the East Channel of the East River, New York [9]. Likewise, Marine Current Turbines Ltd. plans to deploy a 10 MW array in Wales [10]. Therefore, it is important to understand the hydrodynamic interactions among turbines in an array setting to support this transition.

Experimental studies [11, 12, 13] as well as Computational Fluid Dynamics (CFD) simulations [14] have been carried out to examine wake profile behind in-stream hydrokinetic turbines. Experiments were conducted in a flume tank using three porous disks [12] and multiple turbine models [11] at different locations to simulate wake profiles of bottom mounted turbines in an array setting. The experimental setup in [13] was used to study wake profile of a single turbine in shallow water and presents analytical model developed using a wake-similarity approach. The analytical model developed in [13] is different than the model presented here as we quantify wake velocity deficit as a function of ambient turbulence intensity (Equations 3 and 4). Moving beyond the study of the wake field alone, a CFD analysis [14] has been carried out to evaluate power produced by a downstream turbine relative to a turbine operating in the unperturbed environment.

This paper presents an approach for simulating the performance of a downstream turbine suitable for control system development and compares the open loop performance of this downstream turbine with that of a turbine operating in the unperturbed environment. The effects of upstream turbine wake are taken into account in terms of amplified turbulence and wake velocity for the evaluation of the downstream turbine performance. Algorithms utilized to quantify amplified turbulence levels and wake velocity associated with the presence of the upstream turbine [15, 16] are presented, and the integration of these algorithms into an existing numerical simulation of an Ocean Current Turbine (OCT) that uses a Blade Element Momentum (BEM) method with a dynamic wake inflow model [16, 17, 18] are discussed. Standard notations are used with boldfaced mathematical symbols representing vectors or arrays.

The downstream turbine's performance is evaluated for a bottom mounted configuration commonly used for river and tidal turbines, isolating loadings caused by the altered flow field.

Additionally, a compliantly moored turbine design, such as those under consideration for ocean current turbines, is evaluated to better understand the full system response of these devices. The bottom mounted turbine analysis only allows the rotation of