Applied Energy 148 (2015) 121–133

Contents lists available at ScienceDirect

Applied Energy

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

Performance characterization and placement of a marine hydrokinetic turbine in a tidal channel under boundary proximity and blockage effects

Nitin Kolekar, Arindam Banerjee *

Department of Mechanical Engineering & Mechanics, Lehigh University, Bethlehem, PA 18015, United States

HIGHLIGHTS

- Solid and wake blockage on turbine explored using laboratory experiments and CFD.
- Close proximity of hydrokinetic turbine to free surface affects performance.
- Enhanced performance with decreasing tip clearance, maximum at half turbine radius.
- At low values of tip clearance, free surface drop modified upper bypass and wake.
- Blockage from wake modification dependent on tip clearance, rotor rpm and *TSR*.

ARTICLE INFO

Article history:
Received 20 November 2014
Received in revised form 7 March 2015
Accepted 9 March 2015
Available online 28 March 2015

Keywords:
Hydrokinetic turbine
Blockage
Free surface
Wake

ABSTRACT

Marine hydrokinetic turbine, when operating in a shallow channel is subjected to the boundary proximity effects from a deformable free surface on top and the channel bottom. A close proximity of turbine to these boundaries modifies the flow-field around the turbine and affects device performance. Significant flow acceleration occurs in and around the turbine rotation plane; the magnitude of which depends on size of the turbine relative to the channel cross-section and is commonly referred to as solid blockage. In addition, the wake behind the turbine creates a restriction to the flow called wake blockage. We focus on unraveling the influence of boundary proximity and blockage on the turbine performance through coupled experimental and computational studies. The experiments were carried out in an open surface water channel with a three bladed, constant chord, untwisted marine hydrokinetic turbine submerged at different depths and performance was evaluated under various operating conditions. The findings were complimented by a steady state computational fluid dynamics study that was carried out to understand the effect of flow Reynolds number and solid blockage on the turbine performance. A reduction in tip-depth of immersion was observed to improve the turbine performance until it reached an optimum depth beyond which a reduction in performance was observed due to free surface interaction with wake and bypass region. A transient CFD analysis with volume of fluid approach was performed to incorporate free-surface and buoyancy effects and augment flow-field characterization behind the turbine in the wake, upper bypass, and lower bypass regions. For low tip clearance ratios, a significant drop (up to 5–10% of channel depth) in free surface was observed behind turbine with complex three dimensional flow structures that lead to a skewed wake affecting its expansion and restoration process.

1. Introduction

Unlike wind turbines, marine hydrokinetic turbines (MHkT) operate in a bounded flow environment where the flow is constrained between the free surface and the channel (river/sea) bed. In many cases, the channel depth for commercial scale MHkT installation is between 1.5*D* and 3*D* (where *D* is the turbine diameter) which leads to a blockage ratio (*B* = ratio of turbine area to channel area) greater than 0.1 [1]. Under such circumstances, the turbine is subjected to effects of solid blockage that modifies the flow-field around the turbine and hence affects its performance [2–4]. In addition, the rotational motion of the blades creates a low pressure, low velocity *wake* region behind the turbine; the rotating wake presents an additional restriction to the flow called wake blockage.

* Corresponding author at: Packard Laboratory, Lehigh University, Bethlehem, PA 18015-3085, United States. Tel.: +1 (610) 758 4099; fax: +1 (610) 758 6224.
E-mail address: arb612@lehigh.edu (A. Banerjee).

http://dx.doi.org/10.1016/j.apenergy.2015.03.052
0306-2619/

Collectively, these two blockage effects result in accelerated flow near the rotor plane which in turn yields a higher performance compared to a turbine operating in an unblocked environment [5–7]. Quantifying the wake recovery distance behind a MHkT is thus important for designing optimum locations for multiple devices in a farm environment. Although the problem appears to be similar to wind-farm design; important differences exist primarily due to the bounded flow environment which alters the mechanism of wake recovery. This invalidates usage of wind-farm models in which wake restoration takes place by absorbing energy from the atmospheric boundary layer which can be treated as an infinite ambient reservoir [8]. For MHkT, where a limited water depth is available, the wake is tightly restricted between channel bottom wall and free surface that limits free expansion in directions perpendicular to the bounding surfaces. Installation also plays a pivotal role; MHkT can either be bottom mounted on a pier anchored to the channel bed or can be supported from a floating platform moored to the channel bed [9]. In both cases, they are subjected to effects of boundary proximity of either the deformable free surface or the channel bed. Proximity of a turbine to the free surface presents additional complexity to flow structures and hence affects the turbine performance. The deformable free surface allows the water level to drop behind the turbine rotational plane [10]. Though the solid blockage is constant, the drop in free surface height and its deformation behind the turbine with the associated wake modifications are expected to influence the flow-field and hence turbine performance; the effects may vary significantly due to changes in effective flow conditions, operating *TSR* (ratio of blade flow speed to flow speed), Reynolds number (*Re* = ratio of inertial forces to viscous forces) and Froude number (*Fr* = ratio of characteristic flow velocity to gravitational wave velocity). The severity of this effect is expected to be function of various parameters including but not limited to blade shape (airfoil shape, chord and twist distribution), blade pitch angle, *TSR*, free surface proximity, channel wall proximity, and solid blockage itself.

An extensive literature review on wind-turbines was undertaken due to similarity in the working principle with MHkTs. Several experimental studies have been carried out for wind turbines in both unblocked and blocked flow environments to investigate the effects of tip speed ratio, Reynolds number, blade profile, velocity gradient that may exist across the rotor plane and turbulent wind characteristics [8,11–14]. However, the results are not directly applicable as MHkTs operate in a flow medium which is fundamentally different from wind turbines; denser working fluid leads to higher structural stresses on turbine blades [15,16]. In addition, MHkTs operating at higher rotational speeds and high angle of attack in a near free surface environment may get subjected to cavitation effects [15]. Several experimental studies have also been performed to quantify blockage effects on turbine performance. Majority of the early experimental work was done either in wind-tunnels or water channels with the aim of validation and verification of simple physics based models of such flows [2,11,12]. Chen and Liou [12] experimentally investigated effects of tunnel blockage on turbine performance. Blockage effect was quantified in terms of blockage factor by measuring tunnel flow velocity with and without turbine. The blockage factor was found to be strongly related to solid blockage, *TSR*, and blade pitch angle. Higher blockage effects were observed at higher solid blockage and higher *TSR* values. McTavish et al. [3] studied effect of blockage on initial wake expansion for different sized rotors in water channel using dye visualization technique. Higher blockage was found to narrow down the wake expansion and modify the vortex pairing behind the turbine. Several studies have also been performed to analyze free surface effect on marine current turbines; however, majority of these studies used a porous disc to replicate the turbine rotor. Myers and Bahaj [17] carried out experiments with mesh disks to study effect of disk proximity to sea bed/water surface on wake structures behind porous disks. Varying the disk proximity to sea/bed and water surface was found to affect wake structure and its recovery duration. Bahaj et al. [18] performed analytical and experimental study to investigate the effect of surface proximity on turbine performance. Their experiments in a cavitation tunnel and tow tank showed reduction in turbine power with decreasing blade tip-free surface clearance. Experimental investigations by Birjandi et al. [19] with a vertical axis hydrokinetic turbine reported improved performance with increasing free surface proximity. The presence of turbine in a tidal channel not only affects the downstream flow but also the flow upstream of the turbine. Experimental and computational investigations of Medici et al. [11] show influence of blockage on flow up to three turbine diameters upstream of the rotor plane. Near upstream flow showed three dimensional flow structures indicating effect of turbine geometry on incoming flow, similar to near wake flow.

Analytical models for characterizing the turbine performance are based on application of linear momentum theory. Garrett and Cummins [20] applied linear momentum theory for flow constrained between two rigid surfaces and found increase in turbine power with increasing blockage ratio. Houlsby et al. [21] used linear momentum theory to analyze a pressure constrained, parallel-sided tube scenario with an extension to open channel flow. A quartic equation was presented relating flow Froude number, blockage ratio, and flow speeds in wake and bypass region behind turbine. A similar analysis was presented by Whelan et al. [10] yielding a quartic equation relating above quantities. Analytical predictions were comparable with experimental data for mesh disc simulator and two bladed rotor in wind tunnel and water channel for different blockage conditions. Lartiga and Crawford [22] used actuator disc modeling with blockage corrections to predict the performance of tidal turbine in blocked environment. Flow field data from PIV measurements and CFD simulations were used to account for blockage effect. Analytical predictions were in good agreement at lower blockage ratio but showed significant deviations at higher blockage ratios. Computational study by Sun [23] with porous discs reported localized flow acceleration in region between wake and channel bottom. Free surface drop behind mesh disc was observed to affect wake characteristics and turbine performance as well. Consul et al. [4] investigated effect of blockage and free surface deformation on performance of a marine cross-flow turbine for different blockages and free surface boundary conditions using two-dimensional CFD modeling. The deformable free surface boundary condition lead to 6.7% performance improvement compared to closed lid condition due to higher effective blockage caused by free surface deformation. Froude number (over the range studied: 0.08–0.13) was reported to have very small effect on power coefficient (henceforth referred to as C_p = ratio of turbine power to water power) but significantly affected the free surface drop. Recently Bai et al. [24] performed numerical simulations using immersed boundary method to predict marine current turbine performance under free surface flow conditions and validated it with experimental data. No significant free surface deformation was reported with turbine operating with blade tip immersed $\sim 1 \times R$ below the free surface, where R denotes the turbine radius. Computational study by Zhou and Wang [25] investigated effect of Froude number, turbine diameter and depth of immersion on free surface wave induced by tidal turbine and its effect on turbine performance. But the computational study was not able to conclude on effect of depth of immersion on turbine performance. Lee et al. [26] used BEM and CFD based models for performance prediction of a horizontal axis tidal turbine. A new design was suggested with raked tip blades for better cavitation and acoustic performance. Though there is an increasing body of work focusing on experiments, computations and analytical