Renewable Energy 91 (2016) 524–530

Contents lists available at ScienceDirect

Renewable Energy

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

On the near-wall effects induced by an axial-flow rotor

N.J. Möller [a], H. Kim [b], V.S. Neary [c], M.H. García [a], L.P. Chamorro [a, b, *]

[a] Department of Civil and Environmental Engineering, University of Illinois, Urbana, IL 61801, USA
[b] Department of Mechanical Science and Engineering, University of Illinois, Urbana, IL 61801, USA
[c] Sandia National Laboratory, Albuquerque, NM 87123, USA

ARTICLE INFO

Article history:
Received 28 May 2015
Received in revised form
24 December 2015
Accepted 16 January 2016
Available online 11 February 2016

Keywords:
Bed shear stress
Channel flow
Hydrokinetic energy
Axial-flow turbines

ABSTRACT

Laboratory experiments were performed to quantify the near-wall flow characteristics and the bed shear stress in the vicinity of an axial-flow rotor model. It was placed in a smooth-wall open channel flow under subcritical conditions and operated at two heights. A laser Doppler velocimeter was used to collect profiles of the streamwise and vertical velocity at few locations within the rotor symmetry plane as close as one wall unit from the bed. Local estimates of the mean bed shear stress were obtained from the mean velocity profiles in the viscous and logarithmic regions. Results show that the bed shear stress is sensitive to the distance from the rotor and to the relative height of the rotor. Maximum bed shear stress occurred downstream of the rotor with the rotor closer to the bed. Flow statistics shows a reduction of turbulence intensity and turbulent shear stress near the wall and close to the rotor due to flow acceleration and changes in the pressure distribution. Spectral analysis of the streamwise velocity evidences a reduction of the energy content across scales spanning two decades right below the turbine. The increase of bed shear is due to mainly flow accelerating but not to turbulence.

1. Introduction

Marine and hydrokinetic (MHK) energy conversion has the potential to complement the renewable energy portfolio. MHK concepts are targeted to be deployed in various waterways such as streams, rivers, and tidal currents [8,13,16,25,38]. Recent estimates place the technically recoverable resource at 250 TWh/year for tidal, 163 TWh/year for ocean currents [just from Florida to North Carolina], and 120 TWh/year for undammed rivers [18,20]. Compared with standard hydropower sources, these concepts are weakly intrusive, leading to reduced effects on the nearby environment and ecosystem [21].

Quantitative characterization of the induced turbulence, its structure and diffusion features, rotor-induced bed shear stress, and its complex interaction with the ambient environment are still subjects of intensive research, not only to achieve optimal MHK turbine operation, but to determine potential environmental impacts and mitigation strategies associated with the relatively emergent technologies [3,24,39]. Axial-flow turbines can heavily modify the local structure of the flow by generating small-scale turbulence and dampening large-scale motions [4]. Velocity deficit due to energy conversion from the mean flow can persist far downstream with the wake expansion strongly constrained by the flow bed [29]. Rotor wakes are modulated by the dynamics of blade-tip and hub vortices [11] and exhibit complex meandering features in the far-wake region [5,23] that appear to be independent of the turbine operation [36].

Onset and dynamics of sediment motions due to increased shear from MHK generators are areas of particular interest due to their effects on water quality, and ecosystem health, among others [21]. Using a 1D morphological model and tidal data of the Bristol Channel in England [32], predicted a bed shear stress increase and scour in the region surrounding MHK turbines. Coherent turbulence structures can exert instantaneous drag and lift on the flow bed [6,7,15,34,35] and numerical simulations by Ref. [22] have shown that the generated turbulence and wake flow can modify the bed shear, which is key for triggering the onset of sediment motion in natural bedforms [14]. However, Reynolds-averaged boundary shear stresses cannot competently explain the sediment particle movements in the turbulent boundary layer, such as entrainment and transport [6,7], while high-fidelity turbulence measurement in wakes still lacks [5,30,31]. Further, numerical simulations have shown that turbine arrays have potential impact on the environment, including erosion streams alongside the turbine arrays with

* Corresponding author. Department of Mechanical Science and Engineering, University of Illinois, Urbana, IL 61801, USA.
E-mail address: lpchamo@illinois.edu (L.P. Chamorro).

http://dx.doi.org/10.1016/j.renene.2016.01.051

deposition in the upstream [1], while array configuration with a spacing of five rotor diameters have been proposed to limit excessive hydrodynamic impacts from the MHK turbines [10].

Due to the recognized potential impact of the hydrokinetic turbines on the stability of natural beds, our fundamental study aims to offer insights on the flow structure in the vicinity of bed around a rotor model. Uncovering the flow physics in these situations can provide guidelines toward the general problem of placing hydrokinetic turbines without compromising the natural dynamics of the beds, which is key for the ecosystem and the targeted performance of turbine arrays. For this reason, unique and challenging experiments are designed and performed to characterize the flow features near a fix, smooth bed as close as the viscous sublayer around two rotor setups. To accomplish this, a high-resolution 2-component laser Doppler velocimetry (LDV) was used to obtain flow measurements. The LDV system allows for non-intrusive measurements [28] well within the viscous sublayer [33]. The experimental design and measurement procedure are described in Section 2. A quantitative characterization of the flow in the near-wall around the turbine is presented in Section 3 and the main conclusions are discussed in Section 4.

2. Experimental set-up

A three-blade axial-flow type rotor model was placed and operated in a smooth-wall open channel flow under subcritical conditions. The rotor has a diameter $d_T = 0.12$ m and its geometry mimics the reference model from Sandia National Laboratory. The cross section of the blades are part of the MHKF1 family of hydrofoils (MHKF1-180s, MHKF1-240s, and MHKF1-400), which is specifically designed for MHK power applications [40]. Fig. 1 illustrates a general schematic of the flume and the rotor and the relative location of the rotor within the flume. The turbine blades and nacelle were fabricated with PolyJet Vero photopolymers and 3D printed with an Objet Eden 350 machine with resolution of 50 μm. Table 1 summarizes the basic geometry of the turbine blades, which includes the chord length c, and pitch angle α as a function of the radial distance r. The turbine uses Precision Microdrives 112-001 Micro Core 12 mm DC Motor that serves a loading system. Instantaneous voltage from the DC generator was captured at 5 kHz with a Measurement Computing USB-1608HS DAQ, whose internal resistance provided the turbine power output.

The experiments were carried out in the Woods Hole Oceanographic Institute (WHOI) flume at the Ven Te Chow Hydrosystems Laboratory, University of Illinois. The flume is 17 m long, 0.6 m wide, 0.3 m deep, and able to adjust the longitudinal slope. The turbine was placed 13.7 m downstream from the inlet at the center of the flume cross section and under a $H = 0.25$ m water depth, providing a blockage ratio ~0.08, and a bulk flow velocity $U \approx 0.28$ ms^{-1}, which practically coincided with the velocity at the turbine hub U_{hub}. The turbine is installed upside down mimicking some of the existing structure designs [27] and rotor hub heights z_{hub} were set at $z_{hub}/d_T = 1.04$ (case a) and 0.65 (case b), giving submergences $S = (H - z_{hub})/d_T = 1.29$ and 1.68. The resulting Froude number is $Fr = U/(gH)^{1/2} = 0.18$, where g is the gravity acceleration, and the Reynolds numbers, based on the rotor diameter and the middle-span blade chord length (c_m), are $Re = U_{hub}d_T/\nu$~3.1×10^4 and $Re = U_{hub}c_m/\nu$~4×10^3, where ν is the kinematic viscosity. A sharp-crested weir located at the downstream end of the flume controlled the subcritical flow condition. A turbulent boundary layer of $\delta \approx 0.2$ m thickness and friction velocity $u_{*0} \approx 0.0118$ m s^{-1} characterized the incoming flow at ~ 4 rotor diameters upstream of the rotor. The turbine operated at constant tip-speed ratio $\lambda = \pi f_T d_T / U_{hub} \approx 4.0$, where f_T is the turbine frequency in Hz obtained from video recording and from the power output spectrum shown in Fig. 3, which resulted $f_T = 3$ Hz. The spectral distribution of the power output reveals that the rotor is insensitive to turbulence motions of reduced frequency $fd_T/U_{hub} \geq 1.2$ or, invoking Taylor's hypothesis, to length scales $l \leq 1.2d_T$~$O(d_T)$. This behavior is consistent with experiments of [5] with a larger rotor placed in a open channel flow.

Laser Doppler velocimetry (LDV) was used to obtain high-resolution and synchronous measurements of the streamwise (u) and vertical (w) velocity components of the flow. The LDV system includes a Spectra-Physics Stabilite 2017 Ar-Ion laser, TSI TR 60 Series (3-D, non-submersible) probe, TSI Model TLN06-363 (363 mm focal distance) lens, TSI Fiberlight Multi-Color Beam Generator 3-Component optical splitter, TSI PDM1000 photo detector module, TSI FSA3500/4000 signal processor, Tektronix oscilloscope, and TSI FlowSizer software. Flow velocity was

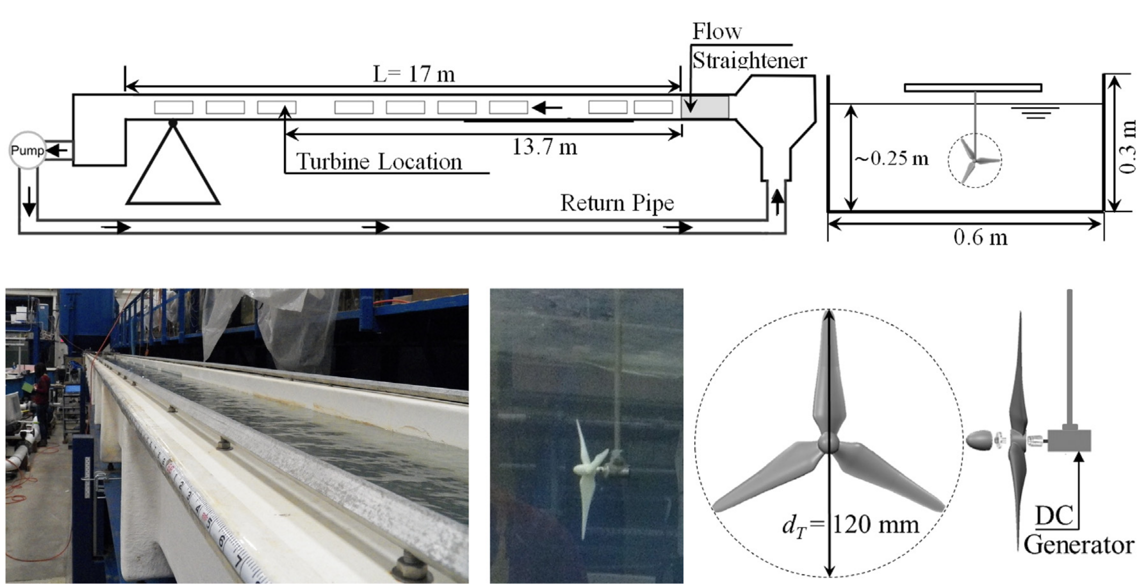

Fig. 1. Schematic of the flume and rotor location (top); and photographs of the flume and schematic the rotor (bottom).