Energy 111 (2016) 104–116

Contents lists available at ScienceDirect

Energy

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

Influences of yaw angle and turbulence intensity on the performance of a 20 kW in-stream hydrokinetic turbine

Wenlong Tian [a, b, *], James H. VanZwieten [b, 1], Parakram Pyakurel [b, 2], Yanjun Li [c, 3]

[a] School of Marine Science and Technology, Northwestern Polytechnical University, Xi'an, Shaanxi, China
[b] Southeast National Marine Renewable Energy Center, Florida Atlantic University, Boca Raton, FL, USA
[c] College of Engineering, Florida Atlantic University, Boca Raton, FL, USA

ARTICLE INFO

Article history:
Received 18 October 2015
Received in revised form
18 April 2016
Accepted 3 May 2016

Keywords:
Yaw
Turbulence intensity
Hydrokinetic turbine
Computational fluid dynamics
CFD (Computational Fluid Dynamics)
Turbine

ABSTRACT

Three-dimensional transient CFD (Computational Fluid Dynamics) simulations are performed to study the hydrodynamic performance of an ocean current turbine with a 3.0 m diameter 3-bladed rotor. Simulations are based on the RANS (Reynolds Averaged Navier–Stokes) equations and the shear stress transport k-ω turbulent model is utilized. The influence of yaw angle and upstream TI (turbulence intensity) on the turbine performance is studied. The CFD method is first validated using existing experimental data and good agreement is obtained. The performance of the turbine, including power, thrust and wake characteristics are then studied at different TSR (tip speed ratios). The turbine obtains a maximum coefficient of power (C_p) of 0.4642 at TSR = 6 and the coefficient of thrust (C_t) increases over the entire evaluated TSR range to a value of 0.8788 at a TSR = 10. Simulations are also performed at four different yaw angles, 0°, 5°, 10° and 15° which show that both C_p and C_t decrease as yaw angle increases. Finally simulations of three different TIs, 3%, 6% and 9%, are performed and analyzed. Results show that TI minimally affects C_p and C_t for the considered TI range, but greatly influences the downstream wake structure.

1. Introduction

The past few years have seen an increase in worldwide interest of using river, tidal, and ocean current hydrokinetic energy to generate electricity without the use of dams, in-stream hydrokinetic energy production. Each of these resource types are distributed globally with riverine energy primarily located in the worlds larger river systems, tidal energy along the coast and primarily at higher latitudes, and ocean current energy primarily along the western boundaries of the world's oceans. Ocean currents with average energy densities greater than 0.5 kW/m^2 are located at numerous locations along the western boundaries of the world's ocean basins [1], with average energy densities measured in excess of 3.0 kW/m^2 [2]. The technically extractable average ocean current energy from US waters has been estimated at 19 GW [3], with several other countries such as Japan, South Africa, Taiwan, the Philippines, and Brazil also having large scale ocean currents accessible off their coasts [1]. Extracting power from these renewable resources could impact the energy portfolios of these countries, providing a potential solution to the energy crisis and environment pollution.

Ocean current turbines (a subset of in-stream hydrokinetic turbines that are designed specifically for ocean currents) are being designed and tested for converting the kinetic energy contained in ocean current to electricity. These turbines can be generally classified into two mail categories according to the angles between the upstream water and the rotating axis of the turbine, the horizontal axis ocean current turbines and the vertical axis ocean current turbines, a detailed description of these turbines can be find in Refs. [4,5]. Currently ocean current turbine prototypes are being designed and tested to extract power from the ocean current resources in the above paragraph (See Refs. [6,7] for example). If deployed in US waters, these systems will most likely be moored in water depths from 300 to 400 m (potentially up to 800 m) and will likely be operated within the top 100 m of the water column where the current is strongest. To help address many of the challenges

---

* Corresponding author. School of Marine Science and Technology, Northwestern Polytechnical University, 127 Youyi Xilu, Xi'an, Shaanxi Province 710072, China.
E-mail addresses: tianwenlong@mail.nwpu.edu.cn (W. Tian), jvanzwi@fau.edu (J.H. VanZwieten), ppyakurel2014@fau.edu (P. Pyakurel), yli2013@fau.edu (Y. Li).
[1] 777 Glades Road, EE319, Boca Raton, FL 33431, USA.
[2] 777 Glades Road, EE318, Boca Raton, FL 33431, USA.
[3] 777 Glades Road, EW210, Boca Raton, FL 33431, USA.

http://dx.doi.org/10.1016/j.energy.2016.05.012
0360-5442/

Nomenclature

c	Blade chord length (m)
C_p	Coefficient of power
C_{paxial}	Axial coefficient of power (W)
C_{ppeak}	Peak coefficient of power (W)
C_{ptotal}	Coefficient of power at zero yaw position (W)
D	Diameter of the rotor (m)
P	Power (W)
Q	The second invariant of the rate of strain tensor
r	Spanwise location of the blade (m)
R	Radius of the rotor (m)
t	Blade Thickness (m)
T	Thrust (N)
TI	Turbulence intensity
TSR	Tip speed ratio
U	Upstream velocity (m/s)
v	Axial velocity (m/s)
x	Downstream distance (m)
y	Radial distance (m)
y^+	y-plus value
θ	Yaw angle (°)
ω	Angular velocity (rad/s)
φ	Blade twist angle (°)
ρ	Water density (kg/m^3)

associated with operating in-stream hydrokinetic turbines in ocean currents the SNMREC (Southeastern National Marine Renewable Energy Center) at FAU (Florida Atlantic University) designed, built, and tested at-sea a 20 kW horizontal axis experimental OCT (ocean current turbine) [8,9]. This OCT (Fig. 1) consists of a single rotor with a radius of $R = 1.5$ m, a torpedo shaped nacelle and two elliptically shaped cylindrical weights designed to make the turbine negatively buoyant and counteract the hydrodynamic rotor torque. The three bladed rotor utilized on this system was geometrically designed using the National Renewable Energy Laboratory's HarpOpt program using the approach described in Ref. [10], with the final blade design presented in Ref. [8]. The nacelle is designed to contain a generator, gear box, and mechanical brake to stop the rotor blades from turning. The nacelle diameter is 0.6 m and the length is 3.7 m.

Experimental trials have been carried out to evaluate the performance of several hydrokinetic rotor designs. Bahaj et al. carried out a power and thrust coefficient study on a 0.8 m-diameter turbine in a towing tank and in a cavitation tunnel, providing comprehensive high-quality data for the validation of numerical computations [11]. Coiro et al. conducted towing tank experiments of a scaled model of a hydrokinetic turbine and provided the power and thrust curves at different water velocities [12]. Galloway et al. studied the power and thrust performance of a 1/20th scale hydrokinetic turbine operating at yaw and in wave by performing towing tank experiments [13]. Later they studied the effect of yaw angle and wave on the loads of the turbine with both experimental and numerical method [14]. Tedds et al. provided many turbine performance curves for hydrokinetic rotor design variants with different numbers of blades, pitch angles, etc [15]. Recently, Mycek et al. studied the upstream turbulence intensity effect and the interaction between two turbines, with emphasis paid on the wake of the turbine [16,17].

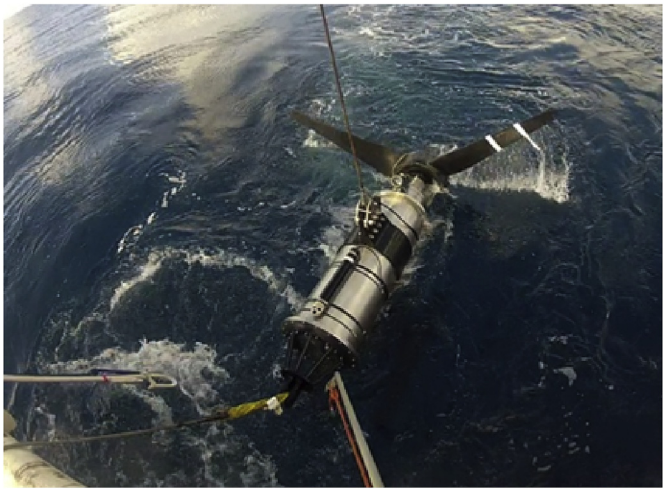

Figure 1. Photograph of the 20 kW experimental ocean current turbine.

Numerical methods utilized to evaluate rotor designs include BEM (blade element momentum) methods, which have been used widely for engineering design because of the low computational overhead associated with this approach. Over the past several years several correction factors have been added to the originally proposed BEM method that account for three-dimensional (3D) effects such as tip loss, rotational flow and dynamic stall [18,19]. 3D inviscid models provide more physics of the turbine hydrodynamics than the BEM method. Current 3D inviscid models include lifting line, panel, and vortex-lattice [20-22]. However, these methods neglect the viscous effects, which need to be considered to achieve the most accurate turbine performance predictions possible.

CFD (Computational fluid dynamics) simulations of the Navier–Stokes equations model fluid flows starting from first principles, and therefore inherently capture viscous effects. Comprehensive CFD simulations of horizontal axis water/wind turbines have been created. Michael et al. numerically modeled a 20 m-tidal turbine at different flow velocities using the commercial CFD code STAR CCM+, including an analysis of the effect of grid density and time step on the calculated torque [23]. Monier et al. designed and optimized a winglet for the NREL Phase VI turbine using the Fine/Turbo of the commercial CFD code NUMECA [24]. They provided a detailed CFD validation study of the NREL Phase VI turbine [25] and the CFD results was in good agreement with the experiment results. Yuwei et al. carried out CFD simulations of the NREL Phase VI turbine utilizing both unsteady RANS (Reynolds-Averaged Navier–Stokes) and DES (Detached Eddy Simulation) methods [26]. Tongchipakdee et al. studied the aerodynamic performance of the NREL (national renewable energy laboratory) Phase VI horizontal axis wind turbine under yawed flow conditions [27]. Schleicher et al. designed a novel portable micro-hydrokinetic turbine and characterized the performance of the turbine with a RANS method [28]. Nak et al. studied the effect of the distance between dual rotors on the performance and efficiency of a counter-rotating tidal turbine using both CFD and experimental methods [29]. More recently, Liu et al. studied the wakes of a horizontal axis ocean current turbine with three different RANS methods, the results show that the Sliding Mesh Method is more