Journal of Fluids and Structures 71 (2017) 78–95

Contents lists available at ScienceDirect

Journal of Fluids and Structures

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

Hydrodynamic loadings on a horizontal axis tidal turbine prototype

Pablo Ouro [a], Magnus Harrold [b], Thorsten Stoesser [a,*], Peter Bromley [b]

[a] Hydro-environmental Research Centre, Cardiff School of Engineering, Cardiff University, The Parade CF24 3AA, Cardiff, UK
[b] Tidal Energy Ltd, CF23 8RS, Cardiff, UK

ARTICLE INFO

Article history:
Received 30 September 2016
Received in revised form 3 February 2017
Accepted 28 March 2017
Available online 8 June 2017

Keywords:
Horizontal axis tidal turbine
Tidal turbine
Immersed boundary method
Large-eddy simulation
Direct forcing
Environmental turbulence

ABSTRACT

Until recently tidal stream turbine design has been carried out mainly by experimental prototype testing aiming at maximum turbine efficiency. The harsh and highly turbulent environments in which tidal stream turbines operate in poses a design challenge mainly with regards to survivability of the turbine owing to the fact that tidal turbines are exposed to significant intermittent hydrodynamic loads. Credible numerical models can be used as a complement to experiments during the design process of tidal stream turbines. They can provide insights into the hydrodynamics, predict tidal turbine performance and clarify their fluid–structure interaction as well as quantify the hydrodynamic loadings on the rotor. The latter can lead to design enhancements aiming at increased robustness and survivability of the turbine. Physical experiments and complementary large-eddy simulations of flow around a horizontal axis tidal turbine rotor are presented. The goal is to provide details of the hydrodynamics around the rotor, the performance of the turbine and acting hydrodynamic forces on the rotor blades. The simulation results are first compared with the experimental and good agreement between measured and simulated coefficients of power are obtained. Acting bending and torsional moment coefficients on the blade-hub junction are computed for idealised flow conditions. Finally, realistic environmental turbulence is added to the inflow and its impact on the turbine's performance, hydrodynamics and rotor loadings is quantified.

1. Introduction

The predictability of tides is a virtue and it allows predicting far in advance and with high accuracy the energy that can be extracted from this renewable energy resource. Great Britain is endowed with vast tidal energy resources, a large tidal range (of up to 14 m in the Severn Estuary), thousands of kilometres of coastline, featuring narrow straits or headlands which often result in powerful tidal streams. Tidal streams are harnessed through the deployment of tidal turbines, basically under water windmills and presently, Horizontal Axis Tidal Turbines (HATT) constitute the majority of commercialised turbines. It appears that some technology is imported directly from Horizontal Axis Wind Turbines (HAWT) (Khan et al., 2009), however, water is approximately 800 times denser than air and therefore water causes significantly greater loads on HATT rotors compared to their HAWT counterpart. Consequently, essential modifications in the tidal stream turbine's design are needed to avoid structural failures, e.g. shorter and thicker blades and hence a smaller rotor diameter, and this has been subject of research from the beginning.

* Corresponding author.
E-mail addresses: OuroBarbaP@cardiff.ac.uk (P. Ouro), Magnus.Harrold@tidalenergyltd.com (M. Harrold), Stoesser@cardiff.ac.uk (T. Stoesser), Peter.Bromley@tidalenergyltd.com (P. Bromley).

http://dx.doi.org/10.1016/j.jfluidstructs.2017.03.009
0889-9746/

Early experimental works on HATTs have focused mainly on their hydrodynamics and performance. Bahaj et al. (2007) studied the effect of free surface proximity, yaw angle and different flow speeds on the performance of a tidal turbine. Mycek et al. (2014b) studied the wake produced downstream of the turbine and its interaction with other turbine(s) towards the design of tidal stream turbine farms, and complementary to this Vennell et al. (2015) presented their vision of the future of large tidal turbine farms. With a more hydrodynamic loadings focus, Milne et al. (2015) investigated the structural moments of a 3-bladed HATT subjected to a sinusoidal motion under uniform flow conditions, and Blackmore et al. (2016) studied the influence of different turbulence intensities and eddy length scales in the approach flow on the turbine loadings and performance.

There has been increasing interest in developing numerical methods that can reproduce the complex flow in the near field of tidal turbines. These can be used in parallel to laboratory experiments or field tests because they can provide more detailed information on the near-field hydrodynamics or can predict turbine performance data for alternative rotor designs. Early works employed the blade element momentum (BEM) method and the actuator disk theory. However, the latter does not consider the flow unsteadiness and the former discretises the turbine's geometry by a set of single points representing blade sections with constant hydrodynamic coefficients. Such simplifications resulted in methods which are computationally cheap and these approaches gave good results for the turbine's far field (Sorensen and Shen, 2002; Sarlak et al., 2015) however they are unable to reproduce the physics around the turbine rotor.

The complex and highly turbulent flow around tidal turbines is governed by fluid–structure interaction including dynamic stall, trailing vortex wake generation or hydrodynamic load unsteadiness. Advanced and accurate models, such as blade resolved methods, are needed to simulate these features (Leishman, 2002). In general, blade resolved methods use three-dimensional meshes that represent explicitly the turbine's geometry and motion and the flow field is computed using Reynolds Averaged Navier–Stokes (RANS) or Large-Eddy Simulation (LES). RANS models are widely used as the computational requirements are affordable (Frost et al., 2015; Mason-Jones et al., 2013) though the time-averaging of the velocity field does not accomplish a realistic representation of the instantaneous fluid–blade interaction. On the other hand, LES resolves the large-scale flow structures present in the velocity field (Stoesser, 2014) as the so-called dynamic stall (Visbal et al., 2013). The main drawback of LES is the large amount of computational resources required to run the demanded fine meshes although exponentially increasing computational resources is making LES more accessible to the research community (Sotiropoulos, 2015).

There have been only few studies applying LES to study the hydrodynamics and performance of HATTs. Kang et al. (2012) were the first to reproduce numerically the operation of a commercial HATT prototype using the immersed boundary method. McNaughton et al. (2014) and Bai et al. (2014) performed LES of the experimental setup from Bahaj et al. (2007), and whilst (McNaughton et al., 2014) used a sliding mesh method, Bai et al. (2014) employed the immersed boundary method. In all three studies an excellent match with the experimental data was achieved showing the potential and accuracy of the method of LES.

The Immersed Boundary (IB) method has demonstrated great potential for the simulation of fluid–structure interaction (FSI). The IB method was first introduced by Peskin (1972) to simulate the FSI of heart valves. Some of the main advantages of the IB method are: the reduction of computational effort in the simulations compared to body-fitted and/or sliding mesh methods by avoiding remeshing and variable re-allocation at each time step or the ability to employ fast (multi-grid) Poisson equation solvers. Different IB methods have been developed since Peskin (1972) and are mainly divided depending on whether the body geometry is considered as a continuous surface or discretised into a finite set of points or markers. While Kang et al. (2012) and Bai et al. (2014) used a continuum IB method, in the following the discrete method developed by Fadlun et al. (2000) and improved by Uhlmann (2005), the so-called direct forcing IB method, is adopted for the simulations. This method has been validated previously for vertical axis tidal turbines (Ouro and Stoesser, submitted for publication; Ouro et al., 2015), particle laden flows (Uhlmann, 2005), bluff body representation (Ouro et al., 2016), and several other applications as summarised in Sotiropoulos and Yang (2014).

During the lifetime of tidal turbines, they are subjected to harsh and highly turbulent environmental conditions. This compromises their structural design in order to avoid any major failure during the project lifespan or at least reduce as much as possible the costly in site maintenance by reducing risks. Therefore, it is essential to identify and quantify the main stresses on the turbine. To date only few research has been dedicated to tidal turbine loadings due to the inherent difficulty of experimentally determining acting forces (Milne et al., 2013). Nicholls-Lee et al. (2013) determined structural loads on the blades made of composite materials while Blackmore et al. (2016) focused on the effect of turbulence on hydrodynamic loads on a HATT. Numerical studies that quantified hydrodynamic loadings of tidal turbines have been presented by Mason-Jones et al. (2013), Frost et al. (2015) and Tatum et al. (2016). These studies were carried out using RANS models which, due to its time-averaging nature, cannot resolve the details of the instantaneous flow.

In the research reported in this paper, a LES-based numerical approach together with the IB method is employed for the simulation of the fluid–structure interaction of a HATT. LES is able to resolve explicitly the large, energetic scales in the flow and hence is expected to provide unprecedented details of the FSI of a HATT rotor subjected to turbulent flow. The method is first validated with data from experimental tests of a HATT prototype. This is followed by revelation of hydrodynamic details of the near-wake of the HATT rotor. Then hydrodynamic loads are quantified in terms of bending and torsional moments at the blade root. Finally, the effect of oncoming environmental turbulence on the turbine's performance and its structural moments is quantified.