Ocean Engineering 78 (2014) 95–111

Contents lists available at ScienceDirect

Ocean Engineering

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

Tidal turbine representation in an ocean circulation model: Towards realistic applications

Thomas Roc [a,*], Deborah Greaves [b,1], Kristen M. Thyng [c,2], Daniel C. Conley [b,3]

[a] IT Power ltd, Department of Marine Renewable Energy, St. Brandon's House, 29 Great George Street, Bristol BS1 5QT, UK
[b] Plymouth University, School of Marine Science and Engineering and Marine Institute, Portland Square A410, Drake Circus, Plymouth, PL48AA Devon, UK
[c] Texas A&M University, Department of Oceanography, 3146 TAMU, College Station TX 77843-3146, USA

ARTICLE INFO

Article history:
Received 20 September 2012
Accepted 6 November 2013
Available online 3 January 2014

Keywords:
Tidal current turbines
Array layout effects
Wake interactions
Power assessment
Scale applicability
Ocean circulation models

ABSTRACT

The present method proposes the use and adaptation of ocean circulation models as an assessment tool framework for tidal current turbine (TCT) array-layout optimization. By adapting both momentum and turbulence transport equations of an existing model, the present TCT representation method is proposed to extend the actuator disc concept to 3-D large scale ocean circulation models. Through the reproduction of experimental flume tests, this method has shown its ability to simulate accurately both momentum and turbulent wake interactions. In addition, through an up-scaling test, this method has shown to be applicable at any scale. Thanks to its short computational time, the present TCT representation method is a very promising basis for the development of a TCT array layout optimization tool. Furthermore, on the basis of the simulations performed for the present publication, a reflection on the quantification of the array layout effects on power assessment and device deployment strategy has been initiated.

1. Introduction

Tidal energy has seen considerable development over the last decade and the first commercial deployments are likely to take place within the next 5 years. After the design and optimization of standalone Tidal Current Turbine (TCT) prototypes, the next step for tidal energy development is towards deployment of multiple device arrays. TCT farms, in which arrays of multiple turbines are deployed, help to minimize the overall cost, by allowing for shared maintenance and grid connection expenses and so maximize the power harvest of a particular site. Such "green energy" generators could be intrusive and potentially harmful for the marine habitat or sediment dynamics if used without understanding the potential impacts and without precautions (Defne et al., 2011; Ingram et al., 2011; Kadiri et al., 2012; Shields et al., 2009; Neill et al., 2009). Therefore it is necessary to develop assessment tools for investigating the flow perturbations on a large scale as well as the power yield for any type of TCT device layout. This is necessary both to find the optimum deployment strategy for power capture and to minimize the impacts on the physical environment and thus preserve the sustainable character of such renewable energy resources (Defne et al., 2011; Ingram et al., 2011; Kadiri et al., 2012; Shields et al., 2009; Neill et al., 2009).

Turbine wake interactions can also have positive impacts on the power harvest (Salter, 2009; Couch and Bryden, 2006; Bryden and Melville, 2004). For instance, Myers and Bahaj (2012) showed that, in TCT arrays, careful device spacing of the upstream row may increase the power capture of the downstream flow by up to 22% (Myers and Bahaj, 2012). Moreover, an optimum vertical location of the device in the water column may decrease structure loading and wake persistence downstream of the device (Myers and Bahaj, 2010). Nonetheless in practise, locating turbines, particularly vertically, have other constraints which need to be considered as well (Willis et al., 2010). Therefore, finding the optimum deployment strategy also requires a detailed understanding of the turbine interactions within farms. Mastering turbine interactions will permit the use of beneficial effects on power yield while limiting their harmful impacts on structure loading and large-scale hydrodynamics (Myers et al., 2011; Myers and Bahaj, 2012; Bai et al., 2009; Churchfield et al., 2011; Lee et al., 2009; Ingram and Smith, 2011).

In the current literature, the large majority of 3-D numerical investigations of TCT wake interactions have been performed with Computational Fluid Dynamics (CFD) models (Bai et al., 2009; Churchfield et al., 2011; Lee et al., 2009; Harrison et al., 2010; MacLeod et al., 2002) using either actuator disc or actuator line approaches (Burton et al., 2008). These high resolution models can give a detailed or better accuracy picture of wake interactions

* Corresponding author. Tel.: +44 1172140514; fax: +44 1752586101, +44 117 214 0525.
E-mail addresses: www.thomas.roc@itpower.co.uk , thomas.roc@gmail.com (T. Roc), deborah.greaves@plymouth.ac.uk (D. Greaves), kthyng@tamu.edu (K.M. Thyng), daniel.conley@plymouth.ac.uk (D.C. Conley).
[1] Tel.: +44 1752586122; fax: +44 1752232638.
[2] Tel.: +1 9798451231; fax: +1 9798456331.
[3] Tel.: +44 1752584561; fax: +44 1752586101.

0029-8018/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.oceaneng.2013.11.010

occurring within TCT farms but can only be applied to simplified cases. Indeed, the computational power needed to run such models limits their use to small spatial and temporal domains of simple bathymetries and geometries in which only basic forcing can be imposed. In realistic TCT farm deployment projects, it is imperative that the optimization layout tool accounts for the complex tidal flows and environment configurations where the devices will operate (Churchfield et al., 2011; Réthoré et al., 2009). Accordingly, the present method consists of integrating an innovative and promising TCT representation (Roc et al., 2013) within an existing 3-D ocean circulation model (Regional Ocean Modeling System, ROMS). It is worth noticing that other academic research converges toward similar approaches (Yang et al., 2013; Hasegawa et al., 2011; Work et al., 2013). The so-adapted numerical platform can potentially tackle any TCT layout in any tidal hydrodynamic system. However, prior to practical application, the relevancy of this TCT array optimization tool has to be validated. Ultimately a numerical tool for regional scale TCT array optimization has to prove its ability to simulate device interactions in a realistic environment in order to be considered as suitable for real-life applications. As this new field is still at an early stage, there is a lack so far of such power plants and real-life observational data and so experimental and analytical benchmark tests have to be used to compensate for this.

In this paper, the accuracy of the present numerical tool for TCT arrays optimization (Roc et al., 2013) in reproducing TCT interactions and real-scale wake decay is investigated. After the description of the theoretical fundamentals of the numerical tool, the experimental results of two physical scale models illustrating device interaction features are reviewed. From the comparison between experimental data and simulated results, the precision of the proposed numerical method to reproduce TCT interactions is estimated. Based on these simulations and their power capture assessment, conclusions are drawn on the best suited TCT layout optimization for the proposed method. Then an analytical exercise of up-scaling is demonstrated to examine the application of the numerical tool in a real-scale flow. The results from these tests indicate that the proposed numerical tool for TCT arrays optimization can be considered suitable for realistic applications.

2. Experimental set-up and simulation features

2.1. Experiment and benchmark description

In order to validate the capability of this proposed turbine modelling method to accurately simulate wake interactions within TCT farm layouts, a model data comparison between physical scale model and numerical model results has been conducted. The physical experiments have been performed and published by Myers and Bahaj (2012) and Myers et al. (2011) in the Chilworth flume at the University of Southampton. The flume experiment set-up is a rectangular channel with a 21,000 mm long (L), 1350 mm wide (l) and 300 mm deep (h) working section in which turbines, represented by a 100 mm diameter (d) perforated disc with a porosity corresponding to $C_t = 0.9$, are deployed. A constant, depth-averaged inflow of 0.27 m/s whose vertical profile closely matches a (1/8) power law is imposed at the upstream boundary of the channel. The ambient turbulence intensity (i.e. Eqs. (2) and (3)) of the flume however fluctuates in the range of 6–8%. More precisely, in Myers and Bahaj (2012) TI equals approximately 8%, whereas in Myers et al. (2011) TI equals approximately 6%. The rest of the experimental set up and characteristics stay the same in both references however. In these experiments, two scenarios have been considered. The first scenario consists of a first row of two discs whose centres are separated by 2.5 diameters and edges are 5 diameters away from the flume sides (i.e. Fig. 1). Such TCT layouts composed of a single row of devices are commonly called first generation TCT arrays (Myers and Bahaj, 2012). For the second scenario, a third disc identical to the other two is positioned at a second row in the centre of the flume, 3 diameters downstream of the upstream pair (i.e. Fig. 1). All of them are placed at mid-depth in the water column with a hub depth of 0.15 m.

In order to assess the ability of the numerical method to reproduce the physical model results, focus is placed on the turbine wake recovery. Here, the momentum behaviour in the flow is examined, in particular perturbations induced by the devices on the flow velocity field and the wake recovery. In addition to the direct flow velocity, a conventional method to characterize wake recovery is the velocity deficit, $U_{deficit}$. This non-dimensional number is a function of the ratio between unperturbed flow velocity, U_∞, and the wake velocity, U_w, both measured at equivalent vertical and lateral locations in the channel (Fig. 2).

$$U_{deficit} = 1 - \frac{U_w}{U_\infty} \tag{1}$$

The turbulence behaviour in the flow is characterised using three different quantities, the turbulence intensity (TI) (i.e. Eqs. (2) and (3)), the turbulent kinetic energy, TKE (or k), and the turbulent length scale, TLS (or l) (i.e. Eq. (15)). It is worth noting that the TI data from the experiment have been measured with a conventional Doppler velocimeter and computed as:

$$TI = \left(\frac{1}{3}(u'^2 + v'^2 + w'^2)\right)^{1/2} / (U^2 + V^2 + W^2)^{1/2}. \tag{2}$$

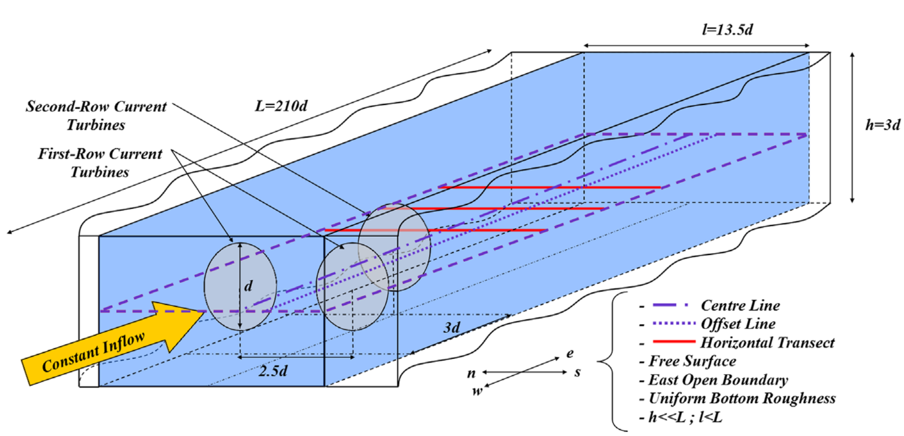

Fig. 1. Idealized channel, multi-disc case – this conceptual diagram represents the multi-disc test geometry and dimensions as well as the locations where horizontal transects (red solid lines), centre and offset lines (purple solid and dotted lines) have been taken have been taken. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)