Renewable Energy 63 (2014) 46–54

Contents lists available at ScienceDirect

Renewable Energy

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

Planning tidal stream turbine array layouts using a coupled blade element momentum – computational fluid dynamics model

Rami Malki, Ian Masters*, Alison J. Williams, T. Nick Croft

Marine Energy Research Group, College of Engineering, Swansea University, Swansea, UK

ARTICLE INFO

Article history:
Received 2 October 2012
Accepted 24 August 2013
Available online

Keywords:
Tidal stream
Marine current
Turbine array
CFD
Blade element momentum theory

ABSTRACT

A coupled blade element momentum – computational fluid dynamics (BEM–CFD) model is used to conduct simulations of groups of tidal stream turbines. Simulations of single, double and triple turbine arrangements are conducted first to evaluate the effects of turbine spacing and arrangement on flow dynamics and rotor performance. Wake recovery to free-stream conditions was independent of flow velocity. Trends identified include significant improvement of performance for the downstream rotor where longitudinal spacing between a longitudinally aligned pair is maximised, whereas maintaining a lateral spacing between two devices of two diameters or greater increases the potential of benefitting from flow acceleration between them. This could significantly improve the performance of a downstream device, particularly where the longitudinal spacing between the two rows is two diameters or less. Due to the computational efficiency of this modelling approach, particularly when compared to transient computational fluid dynamics simulations of rotating blades, the BEM–CFD model can simulate larger numbers of devices. An example of how an understanding of the hydrodynamics around devices is affected by rotor spacing can be used to optimise the performance of a 14 turbine array is presented. Compared to a regular staggered configuration, the total power output of the array was increased by over 10%.

1. Introduction

If the UK is to meet its carbon reduction targets, for which a minimum reduction of 80% by 2050 is required compared to 1990 levels as set out in the Climate Change Act 2008 [1], there needs to be a much more significant emphasis on reducing reliance on fossil fuels and placing a greater emphasis on renewable energy sources for power generation. There are numerous viable renewable energy sources that could be feasibly exploited to meet this goal [2], and considering the highly ambitious targets at hand, all such options will have to be considered. The focus of this work is on the tidal stream option, particularly beyond the initial prototyping and development stage which a number of developers are currently performing.

The UK is fortunate to have one of the best tidal resources in the world, capable of producing an estimated 16.3 TWh/year, which is equivalent to 4.2% of the UK electricity production in 2008 [3]. However, to generate power at this scale, devices will have to be deployed in large numbers. Considering that the few trials on record to date have involved single or few devices, this raises many uncertainties regarding inter-device interaction within the context of a multiple turbine array. The lack of understanding of tidal stream devices and their performance in the natural environment has proven on numerous occasions to be costly, both in terms of time and resources as demonstrated by the failures of the Atlantis device in Orkney, the Marine Current Technology device (SeaGen) in Strangford Lough and the OpenHydro device in the Bay of Fundy shortly after deployment, all of which involved blade failures. This lack of understanding is in part attributed to the complex and unfamiliar operating environment, but also to the novelty of the emerging technologies. This is where science and experience can be applied to inform the industry to minimise such risks in the future and support the marine energy industry.

Despite obstacles in developing the necessary technology and in gaining governmental consent for deployments, tidal stream power generation has emerged in recent years as a potentially reliable form of renewable energy due to the predictability of tide times and magnitudes as well as high concentration of the resource around the UK. Depending on the outcomes of trial deployments currently underway, it is likely that tidal stream deployments will occur at an accelerating rate over the coming decades. To date, a limited

* Corresponding author. Tel.: +44 (0) 1792 295688; fax: +44 (0) 1792 295676.
E-mail addresses: ramimalki@hotmail.com (R. Malki), i.masters@swansea.ac.uk (I. Masters).

0960-1481/$ – see front matter
http://dx.doi.org/10.1016/j.renene.2013.08.039

number of studies, both experimental and physical, have been performed to aid our understanding of how such deployments are likely to perform.

In terms of experimental studies, Myers and Bahaj [4,5] used porous disks to simulate tidal stream turbines in a practical study conducted in a laboratory flume. A range of lateral spacing between devices were assessed before a third disc was introduced further downstream effectively simulating a two-row turbine array. The hydrodynamics downstream of the discs were monitored. There are limitations to the use of porous disks to simulate rotating turbine blades, particular in the near region close to a turbine. However, the authors argue that the porous disks model provides a better representation of turbine wake hydrodynamics at small scales implemented in laboratory flumes where the rotor is less than 0.8 m in diameter due to problems with scaling. On the other hand, Mason-Jones et al. [6] show that non-dimensional rotor performance parameters (power, torque and thrust coefficients) for a tidal stream turbine are independent of scale and validate their model against a single 0.8 m diameter turbine. Nevertheless, the Myers and Bahaj [4,5] studies are some of the few with published data for simulations of multiple turbine arrays with downstream wake characteristics presented and are used to validate the model as presented in Masters et al. [7].

One of the simplest models used for evaluating turbine blade performance implements blade element momentum theory where tabulated hydrofoil data is used to determine lift and drag forces exerted by the blades onto the flow [8]. However, for evaluating multi-turbine arrays, any approach for rotor representation must be combined with a CFD solver to account for the influence of the turbines on the far-field flow structure as this will affect the performance of neighbouring turbines.

Numerical studies on turbine arrays have varied in complexity, ranging from the introduction of additional source terms to models based on solving the shallow-water equations and evaluating the large-scale effects of a turbine array on the environment, to more detailed CFD modelling of rotating turbine blades. For instance, Neill et al. [9] implement an additional bed friction term to a three-dimensional hydrodynamic model, POLCOMS, to simulate a 300 MW array positioned within the Alderney race. The influence of the array on morphodynamics is identified raising a need for careful planning to minimise environmental change. Similarly, Ahmadian et al. [10] modelled an array of turbines using a two-dimensional depth-integrated model whereby the turbines are represented through the addition of thrust and drag forces due to the turbine and supporting structure respectively. Effects of the array on water levels, tidal currents and water quality are predicted by the model. Ahmadian & Falconer [11] used the same model to evaluate the effects of array shapes concluding that denser arrays had more significant, but localised effects on water level and suspended sediment concentration. The significance of interaction between devices within an array setting was highlighted.

Turnock et al. [12] used a coupled model combining blade element momentum theory to represent blade forces on the flow and CFD to simulate flow through the domain to predict the hydrodynamics and performance of a single device. The authors recommend a 6×10^6 element mesh with 40% of the elements in the wake region for modelling a single rotor to achieve convergence. The requirement for minimising lateral spacing whilst maximising longitudinal spacing to achieve optimum power outputs by the array was identified. The authors predict power outputs of multiple turbine arrays by assuming that the power generated by consecutive rows reduces by a constant factor.

More detailed blade modelling was conducted by O'Doherty et al. [13], who simulated a five-turbine array using CFD modelling and identified fluid acceleration around devices which may be used to improve the performance of other turbines further downstream. The authors identified the usefulness of CFD models in improving the design, and Afgan et al. [14] demonstrated that further detail can be obtained by using more computationally expensive models such as large eddy simulation. Wang and Müller [15] conducted CFD simulations using FLUENT of ducted composite material marine current turbines. Each device consisted of a composite wheel, a nozzle and a diffuser. Groups of up to seven devices were considered arranged in three rows. The authors have identified the importance of careful selection of the array arrangement to optimise blockage of the flow and hence, maximise the power output of the array.

The studies published have either focused on over-simplified methods for rotor representation where the main purpose was to reproduce the general effect of the turbines on the flow, or very detailed three-dimensional models that are much more computationally expensive but are able to predict the performance of a small number of turbines. Studies published to date lack detailed modelling of larger turbine arrays that are typical of future deployments. Whilst it is clear that the detailed three-dimensional models can result in a more accurate representation of transient flow features, such simulations are very computationally expensive and are therefore impractical for modelling a large number of turbines [16]. This is addressed here by using the, computationally more efficient, coupled blade element momentum (BEM)–CFD approach [17] to physically simulate multi-turbine arrangements. Here the BEM method is used to simulate the turbine rotor and the blade forces acting on the flow, and flow through the rest of the domain is simulated using a CFD model.

In this study a fixed upstream flow boundary condition is imposed. This is applicable here as the turbine arrays are placed in a wide channel and therefore only extract a small fraction of the energy available to them. However, future large-scale generation will undoubtedly require the deployment of hundreds of devices at high-energy locations. Such locations are fairly limited and hence, the devices are likely to be packed relatively closely to one another along the seabed. A high density of tidal turbines will cause excessive resistance to the flow, or in effect an increase in the drag coefficient of the channel, causing a reduction of flow velocities at the devices [18–20]. Under such circumstances, a different turbine optimisation will be required than for a fixed upstream flow [19,20].

This paper will begin by considering the hydrodynamics downstream of a single device, followed by evaluating inter-device interactions for double and triple device arrays. These interactions will then serve as a guide for choosing suitable arrangements for optimising overall performance for an array containing fourteen devices.

2. Model description

2.1. The BEM–CFD model

2.1.1. Governing equations

The steady-state CFD model solves the Navier–Stokes equations, which consist of the equations for mass continuity (1) and the conservation of momentum (2) whereby the fluid is treated as incompressible and turbulent.

$$\nabla\cdot\left(\rho\underline{u}\right) = 0 \tag{1}$$

$$\nabla\cdot\left(\rho\underline{u}u_i\right) = -\partial p/\partial x_i + \nabla\cdot(\mu_{\mathrm{lam}} + \mu_{\mathrm{t}})\nabla\rho u_i + S_i \tag{2}$$

In these equations, ρ is the fluid density, u_i is the i'th component