Renewable Energy 63 (2014) 222–235

Contents lists available at ScienceDirect

Renewable Energy

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

Near-wake characteristics of a model horizontal axis tidal stream turbine

S.C. Tedds [a], I. Owen [b], R.J. Poole [a,*]

[a] School of Engineering, University of Liverpool, Brownlow Street, Liverpool L69 3GU, UK
[b] School of Engineering, University of Lincoln, Brayford Pool, Lincoln LN6 7TS, UK

ARTICLE INFO

Article history:
Received 1 February 2013
Accepted 13 September 2013
Available online 10 October 2013

Keywords:
Near-wake
Tidal stream turbine
Acoustic Doppler velocimeter measurements
Turbulence statistics
Reynolds stress anisotropy

ABSTRACT

The results of a detailed experimental investigation of the near-wake (up to seven turbine diameters downstream) of a model horizontal axis tidal turbine (HATT) device in a large-scale recirculating water channel facility are reported. An Acoustic Doppler Velocimeter is used to provide detailed three-dimensional mean and turbulent flow field information at five different depths across the full width of the channel downstream of the turbine, giving the most complete three-dimensional velocities and Reynolds normal and shear stress data set yet available. In addition the Reynolds-stress anisotropy tensor is used to illustrate the degree of anisotropy of the Reynolds stress within the turbine's wake. These results reveal the strongly anisotropic nature of the near-wake turbulence suggesting isotropic turbulence models should not be used to model near-wake dynamics. Finally the power-law decay rates of the maximum normalised turbulent kinetic energy differ significantly from those found downstream of grids, meshes or perforated disks, suggesting that previous modelling approaches, which neglected swirl effects and modelled the turbine by absorption discs, may significantly over predict the turbulent kinetic energy decay rate of HATT wakes.

1. Introduction

Throughout the world there is a growing demand for energy produced from sustainable resources, with many governments setting targets for renewable production of electricity. The UK aims to provide 15% of total energy from renewable sources by 2020, this is an increase from 3.3% produced in 2010 [1]. Harnessing the ocean's energy is one way to meet these targets, and the energy can be split into two categories: wave and tidal. One of the main advantages of tidal power is the predictability of the tides. There are two principal ways to harness tidal energy: tidal barrages or lagoons which use the tidal range, and tidal stream turbines that use the tidal current. An advantage of tidal stream turbines is that they minimise the impact on the marine environment as they allow water to pass straight through, and are usually fully submerged with no visual impact.

As well as characterising the power output for a tidal stream turbine it is also important to characterise the wake of a turbine. For example, as turbines are likely to be placed in farms or arrays to make them commercially viable, wake recovery length is crucial for the appropriate spacing between turbines. Also, knowledge of the wake is important so that potential effects on the seabed can be investigated. Only a limited amount of research has hitherto been undertaken into the wakes behind tidal stream turbines. The earliest of this research was conducted using an absorption disc to represent the turbine, both experimentally [2] and using Computational Fluid Dynamics (CFD) [3–5]. Experimental studies into the characterisation of the wake were conducted by Myers and Bahaj [6], Stallard et al. [7,8], Rose et al. [9] and Maganga et al. [10]. Further studies to compare experimental wake data to CFD were completed by Mycek et al. [11] and Rose et al. [12]. Table 1 includes relevant details of studies undertaken to date.

Myers and Bahaj [2] conducted experiments in a 21 m tilting flume, which had a width of 1.35 m and depth of 0.4 m. The vertical velocity profile produced in the flume resembled a modified 1/7th power law which was more uniform closer to the surface and approximated a profile measured at a full scale site and reported by the Carbon Trust [13]. Artificial bed roughness was added to the flume to reduce the velocity and increase the shear stress in the bottom third of the water column, again to create conditions that were as realistic as possible. The absorption discs (mesh discs) used by Myers and Bahaj [2] were 100 mm in diameter and had porosity (ratio of opened-to-closed area) varying between 0.48 and 0.35.

---

[*] Corresponding author.
E-mail addresses: sian.tedds@liv.ac.uk (S.C. Tedds), iowen@lincoln.ac.uk (I. Owen), robpoole@liv.ac.uk (R.J. Poole).

0960-1481/$ – see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.renene.2013.09.011

Nomenclature

A	swept out area of turbine (m^2)
C_P	power coefficient ($P/0.5\rho AU^3$)
C_T	thrust coefficient ($T/0.5\rho AU^2$)
D	disc diameter
P	power (W)
R	radius of turbine (m)
Re	Reynolds number based on upstream flow velocity and turbine radius (UR/ν)
T	thrust (N)
u	streamwise velocity (m/s)
U	mean upstream velocity (m/s)
v	transverse velocity (m/s)
w	spanwise velocity (m/s)
x	streamwise distance across the width behind the HATT (m)
y	transverse distance through the depth behind the HATT (m)
z	spanwise distance behind the HATT (m)
λ	tip speed ratio ($\omega R/U$)
ν	kinematic viscosity of water (m^2/s)
ρ	density of water (kg/m^3)
ω	angular velocity (rad/s)

A load cell was attached to the disc so that thrust acting on it could be measured. An Acoustic Doppler Velocimeter (ADV) device was used, which sampled at a rate of 50 Hz with a sample volume of 0.15 cm^3, to map the wake up to 20 disc diameters (D) downstream. It was found that the porosity alone did not determine the thrust coefficient. Four discs with equal porosity were tested, those with a larger number of small holes had a greater thrust than discs with fewer large holes. For all discs tested, at 10D downstream the mean velocity profiles were virtually identical and any effects of the disc had dissipated. Using a disc with thrust coefficient, $C_T \approx 0.9$, Myers and Bahaj [2] investigated four different discs depths, with the disc centred at 0.33D, 0.5D, 0.66D and 0.75D above the floor of the flume. It was found that as the disc was placed closer to the floor, the mean velocity deficit is longer. The turbulent kinetic energy (TKE) was also measured, where Myers and Bahaj [2] defined TKE as

$$\mathrm{TKE} = \frac{1}{2}\left(\overline{u'^2} + \overline{v'^2} + \overline{w'^2}\right), \tag{1}$$

where u', v' and w' are the varying components of the flow, and the bar above denotes this as a time average. These tests found that there was some evidence that as the distance between the floor and the disc increased, the TKE levels in the wake increased. To simulate a rocky seabed in their flume, Myers and Bahaj [2] placed artificial rocks with lengths between 6.7 and 10 mm over a length of 4 m, as they expected this type of bed to have the greatest effect on turbulence. Again the depth at which the disc was placed was altered and as the distance between the disc and the floor decreased, the wake velocity deficit downstream increased; in turn this increase of the velocity deficit was larger than that with the smoother bed. As a consequence, Myers and Bahaj [2] suggested that a rocky seabed is less suitable for placing tidal turbines than a smoother bed. The experiments conducted by Myers and Bahaj [2] provide useful information on the far wake provided swirl effects are not still important. The changes in wake length with the height of the turbine provides a good indication of where a turbine should be placed in relation to the bed to minimise the effect of the bed.

Sun et al. [3] also used an absorption disc approximation to simulate a tidal turbine, in both two- and three-dimensional CFD calculations. The computational domain used replicated a laboratory water channel, which was 1.5 m wide, 10 m long and had a water depth of 1 m. A volume-of-fluid approach was used to simulate the free surface and a no-slip wall boundary condition was used on the bed. The turbulence model used was the $k-\varepsilon$ model which assumes isotropic turbulence. In the two-dimensional simulations, Sun et al. [3] used an absorption zone centred at a height of 0.5 m. When the water passed through the absorption disc approximately 38% of energy was dissipated. Results from this solution showed a substantial drop in the free-surface behind the absorption disc, to a lowest point of roughly half its initial height. The two-dimensional domain was extended further to a three-dimensional domain, where a square disc was used which had an area ratio to the flume cross-section of 17%, this resulted in a 10% overall loss of kinetic energy in the flume. The results show that as the water flow reaches the disc it accelerates as it flows through, then velocity drops directly behind the disc. Similar to the two-dimensional solution, there was a free-surface drop behind the disc, however the drop was not as large for the three-dimensional model. These simulations gave a first approach for the characterisation of the wake and were concentrated on the near-wake and effects on the free-surface close to the disc.

Harrison et al. [4] compared CFD simulations to the experimental absorption disc results conducted by Myers and Bahaj [2] discussed above. Harrison et al. [4] simulated the free-surface using a homogeneous coupled volume-of-fluid approach. The flow was calculated using the $k-\omega$ shear stress transport (SST) model, which is an extension of the $k-\varepsilon$ model, this was used as preliminary studies indicated that the $k-\varepsilon$ did not accurately model the flow conditions. The model domain represented the flume in which the experiments of Myers and Bahaj [2] were conducted, with a water depth of 0.3 m and with air to a height of 0.15 m above the water. The disc was matched to that of Myers and Bahaj [2] with a diameter of 0.1 m and thickness of 0.001 m, and was located in the centre of the water column. To enable direct comparison with the experimental results, the thrust coefficient was matched. The measured inlet velocity distribution of the flume was used to set the inlet velocity in the CFD model. Both the far-wake and the near-wake were compared, the near-wake had some limitations in its accuracy, in that the effects of swirl were not taken into account, but this, the authors argued, was a reasonable method to estimate the effects of the far wake. The trend and wake recovery of the far wake was found to be similar in the two approaches and the turbulence levels in the wake were comparable.

A computational study was conducted by Daly et al. [14] of an ‘actuator fence’ to simulate an array of tidal turbines, which was placed at various heights in a channel. These results were compared to experimental data of the wake of a actuator fence in a channel, which gave a reasonable approximation to the streamwise velocity downstream of the fence.

Myers and Bahaj [6] measured the wake of a turbine and support structure with a diameter of 0.8 m, using an Acoustic Doppler Velocimeter (ADV), in a recirculating water channel which was 18 m long, 4 m wide and 2 m deep, giving a blockage ratio of 6.3% and had an upstream velocity of 0.8 m/s. The wake of the turbine support structure without the rotor was measured and was found to be significant, especially near the free-surface. The support structure increased the turbulence intensity, again near to the free-surface, by 5D downstream the TI was 10% which was still much higher than the upstream level of 6%. The wake of the support structure and rotor was measured at 5D downstream though the width of the channel at five different heights (−0.2D, −0.1D, centre, 0.1D, 0.2D). The largest deficit was found to be at a height of −0.1D