Renewable Energy 114 (2017) 308–322

Contents lists available at ScienceDirect

Renewable Energy

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

3D modelling of impacts from waves on tidal turbine wake characteristics and energy output

Sufian. F. Sufian, Ming Li*, Brian A. O'Connor

School of Engineering, University of Liverpool, Brownlow Hill, L69 3GH, UK

ARTICLE INFO

Article history:
Received 8 October 2016
Received in revised form
11 April 2017
Accepted 13 April 2017
Available online 21 April 2017

Keywords:
Virtual blade model
Horizontal axis tidal turbine
CFD
Wake characteristics
Wave

ABSTRACT

A Virtual Blade Model is coupled with a CFD model to simulate impacts from a Horizontal Axis Tidal Turbine under combined surface waves and a steady current. A two-equation model is used to represent the turbulence generation and dissipation due to turbine rotation and background wave-current flows. The model is validated against experimental measurements, showing good agreement in both surface elevation and fluid hydrodynamics. It is then scaled up to investigate a steady current with large stream-wise surface waves in the presence of a turbine. A strong interaction is found between surface wave-induced flows and that around the turbine, which clearly impacts on both hydrodynamics within the wake and wave propagation, and produces large fluctuations in power production. Model results show that the wave-period-averaged velocities are similar to those in the steady-current-only condition. However, the wave enhances the turbulence immediately behind the turbine and reduces the length of the flow transition. The wave height reduces by about 10% and the wavelength extends by 12% when propagating over the turbine region in comparison with the no-turbine condition. The wave shape also becomes asymmetric. Compared with the current-alone situation, the model results suggest that the power production is similar. However, wave oscillation produces noticeably larger fluctuations.

1. Introduction

In recent years, the Horizontal Axis Tidal Turbine (HATT) has been regarded as one of the more promising devices for tapping tidal stream energy, which is both reliable and predictable with good potential in many sites around the world. In general, tidal turbines are placed underwater to convert the kinetic energy of tidal flow into electricity through blades rotation. Although the principle is very similar to that for wind turbines, the HATTs are designed differently due to the much larger density of seawater than that of the air [23]. More importantly, at the identified potential sites, the wind-generated surface waves are also often strong and can penetrate to considerable depth and introduce additional oscillatory effects on local flows, see Tatum et al. [21]; Bahaji et al. [2] and Veron et al. [24]. Recent research has shown that when tidal turbines operate under combined current and waves, the changes in free surface has a significant influence on wake characteristics, e.g. Bahaj et al. [2]; Consul et al. [5]; de Jesus Henriques et al. [7]; Lust et al. [12]. Unfortunately, so far, only a handful of studies on offshore HATTs involve surfaces waves. The majority of them also concentrate on turbine performance under much simplified conditions at laboratory scale [21]. The effects of surface waves on the mean flow structure, turbulence, flow-structure interactions and hence the turbine power generation, and vice versa the turbine presence effects on the surface wave dynamics are not been fully understood as yet.

Alongside laboratory experiments, Computational Fluid Dynamics (CFD) modelling has been used in several studies to investigate HATTs under combined waves and current conditions. However, the challenge lies on the modelling of both free surface waves and the flow-turbine interactions. Without resolving details of free surface effects, the wave motion in previous studies has been represented in models via an added periodic oscillatory pressure at the top boundary (rigid lid) of the modelled area, e.g. Holst et al. [10]. Inevitably, the rigid lid limits the motion of fluid near the top boundary and hence the wave induced fluid flow in the vertical direction is missing in the results. This may be adequate for small waves in deep water but not for large storm waves which can affect the seabed. A more realistic approach involves the Volume of Fluid (VoF) method to track the interface between water and air, such as

* Corresponding author.
E-mail addresses: sfuads@liverpool.ac.uk (Sufian.F. Sufian), mingli@liverpool.ac.uk (M. Li), b.a.oconnor@liverpool.ac.uk (B.A. O'Connor).

http://dx.doi.org/10.1016/j.renene.2017.04.030

earlier work of Sun et al. [20]. Similarly, two different approaches are commonly used to represent the stream turbine in a CFD model: a parameterised approach or a blade-resolving approach. The blade-resolving approach requires meshing out each blade in details and rotating multiple frames of reference to compute the flow around the blades, e.g. Mason-Jones et al. [14] and Holst et al. [10]. This type of approach requires over several millions computational nodes to cover the computational domain and each turbine blade for realistic applications, see O'Doherty et al. [18]. The parameterised approach, on the other hand, is a much simpler approach in which the effects of turbine blade rotation is represented by a static porous disk or via added sink terms in the momentum equations, such as the Virtual Blade Model (VBM) based on the Blade Element Method. The porous disk approach is much easier to implement in CFD and the computational cost is the lowest in comparison with other methods [9,20,25]. However, it is unable to resolve the details of flow structure around the turbine and is mainly used for large scale, far-field and multiple turbine simulations. In comparison, the VBM is able to replicate the rotation movement with reasonable computational cost without presenting the actual blades, but instead, simulates the motion of the fluid surrounding the blades. It can be used to simulate near-wake regions from one turbine diameter downstream and provides a useful compromise solution where reasonable accurate results can be achieved when assessing turbine performance and capturing near-wake processes [4].

It is therefore considered that the best optimal approach is obtained by combining the VoF method to resolving the surface wave dynamics alongside with the VBM method to represent the turbine: moderate computational costs than results. However, it should be noted that the VBM was originally designed for a turbine within a single phase fluid, which is strictly speaking not applicable in multi-phase calculations based on the VoF scheme. The present study will test the VBM method by ensuring that the turbine is submerged in the water without any exposure to the air so as to avoid the above complication. The combined approach will be able to provide more evidence on the wave impacts on turbine wake characteristics and power outputs as well as the impact of the turbine on the wave processes. In addition, this study will differ from earlier works [20,21], where more vigorous flow conditions (storm conditions) can be simulated and the impacts from a typical field-scale turbine are considered, thereby benefiting from the lower computational efforts.

The outline of the present paper is as follows. Section 2 presents the modelling system, while the model implementation and validation against de Jesus Henriques et al. [6] experiment are discussed in Section 3. Section 4 presents the model application to a field-scale turbine under combined waves with a steady current. Finally, a summary and conclusions are given in Section 5.

2. Numerical model

2.1. Governing equations

ANSYS FLUENT 14.5 [1] was used to resolve the flow hydrodynamics by solving the Reynolds Averaging Navier-Stokes (RANS) equations via the finite-volume method. The coordinate system is defined as x in the stream-wise, y in the vertical and z in the span-wise directions, respectively, as shown in Fig. 1. The turbine is placed at typically 1/3 of the depth from the surface. Air is assumed to occupy the space above the water.

The pressure and velocity fields are obtained from the Navier-Stokes equations averaged over a time period longer than the turbulent time scale (RANS):

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho \overline{v}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \overline{v}_i) + \frac{\partial}{\partial x_i}(\rho \overline{v}_i \overline{v}_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial \overline{v}_i}{\partial x_j} + \frac{\partial \overline{v}_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial \overline{v}_l}{\partial x_l}\right)\right] + \frac{\partial}{\partial x_j}\left(-\rho\,\overline{v'_i v'_j}\right) + F_i \tag{2}$$

where ρ is the density of the fluid; v_i are the instantaneous flow velocities along the $x\,(u)$, $y\,(v)$ and $z\,(w)$ directions, respectively; p is the total pressure; F_i is the external body force in the i-th direction; and μ is dynamic viscosity. The over-bar denotes time-averaged values and the v'_i refers to the fluctuation in velocity v_i, e.g. $v_i = \overline{v}_i + v'_i$. The RANS equations can be closed using different turbulence models based on the Boussinesq hypothesis:

$$-\rho\overline{v'_i v'_j} = \mu_t\left(\frac{\partial \overline{v}_i}{\partial x_j} + \frac{\partial \overline{v}_j}{\partial x_i}\right) - \frac{2}{3}\rho k \delta_{ij} \tag{3}$$

where μ_t is the turbulence eddy viscosity, $k = \frac{1}{2}\,\overline{v'_i v'_j}$ is the Turbulent Kinetic Energy (T.K.E.) and δ_{ij} is the Kronecker delta. For simplicity, the over-bar is omitted in the following sections.

Following El-Beery [8]; a two-equation turbulence model, Shear Stress Transport (SST) $k-\omega$, is adopted in the present study to simulate turbulence generation and dissipation. In particular, the $k-\omega$ formulation is employed in the main free-stream fluid body and the calculation switches to a viscous sub-layer model near the wall boundary, which combines the advantages of both methods as shown in Menter [15]. The SST modifies turbulent viscosity formulation to account for the transport effects of the principal turbulent shear stress. In addition, the SST model incorporates a damped cross-diffusion derivative term in the ω equation, which makes it better for adverse pressure gradient flows. El-Beery [8] demonstrates that the SST $k-\omega$ is best by considering different turbulence generation and dissipation sources in comparison with other models. The turbulent kinetic energy, k, and special dissipation rate, ω, are computed as follows from the equations,

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k v_i) = \frac{\partial}{\partial x_i}\left(\Gamma_k \frac{\partial k}{\partial x_i}\right) + G_k - Y_k + S_k \tag{4}$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_i}(\rho \omega v_i) = \frac{\partial}{\partial x_i}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_i}\right) + G_\omega - Y_\omega + S_\omega + D_\omega \tag{5}$$

where G_k and G_ω are the generation of k and ω due to turbulent mean-velocity gradients respectively; Γ_k and Γ_ω are the effective diffusivity; Y_k and Y_ω are the dissipation due to turbulence; D_ω is the cross-diffusion term; and S_k and S_ω are user-defined source terms. The effective diffusivity Γ_k and Γ_ω are given by the equations:

$$\Gamma_k = \mu + \frac{\mu_t}{\sigma_k};\;\; G_k = -\rho\overline{v'_i v'_j}\frac{\partial v_j}{\partial x_i};\;\; Y_k = \rho\beta^* f_{\beta*} k\omega \tag{6}$$

$$\Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega};\;\; G_\omega = a_\omega \frac{\omega}{k} G_k;\;\; Y_\omega = \rho\beta_\omega f_{\beta_\omega}\omega^2 \tag{7}$$

where σ_k and σ_ω are the turbulent Prandtl numbers for k and ω respectively, and $a\omega$, β^*, $f_{\beta*}$, β_ω and f_{β_ω} are model coefficients. When SST is employed, the turbulent viscosity μ_t is defined by the equations: