Ocean Engineering 117 (2016) 383–397

Contents lists available at ScienceDirect

Ocean Engineering

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

Wake field studies of tidal current turbines with different numerical methods

Jing Liu*, Htet Lin, Srinivasa Rao Purimitla

Energy Research Institute @ NTU, Singapore

ARTICLE INFO

Article history:
Received 5 July 2015
Received in revised form
6 February 2016
Accepted 28 March 2016
Available online 6 April 2016

Keywords:
Tidal turbine
Wake
Actuator disc model
Computational fluid dynamics
ANSYS Fluent

ABSTRACT

Flow characteristics of the wake of a horizontal axis tidal current turbine are studied by numerically solving Reynolds-averaged Navier-Stokes (RANS) equations. The rotational effects of the three-bladed turbine is modelled with both the Moving Reference Frame (MRF) and the sliding mesh techniques. By comparing the numerical results with theoretical and experimental data, it is found that the sliding mesh technique can more accurately predict the hydrodynamic loads from current and the wake than the MRF technique. Moreover, an alternative turbine modelling technique, Actuator Disk Model (ADM), is also utilised and its results agree well with the sliding mesh results in the far wake velocity if the turbulence length scale (TL) used in the simulation is about half the hub radius and the input porous properties are derived from power coefficient of the turbine. Finally, turbine to turbine interactions have been numerically investigated using the sliding mesh technique.

1. Introduction

In the recent years, a rapid growth has been made in the tidal current turbine development and multi-megawatt scale tidal turbine farms have been planned to reduce cost by sharing transmission cable and devices (Pham and Martin, 2009; Major Boost for Wave and Tidal Industry, 2013; £20 Million Boost for UK Marine Power, 2013; Cobscook Bay Tidal Energy Project, 2012; Tidal Array to be Built off Alderney 'by 2020', 2014). Therefore, the detailed investigations on tidal turbine array optimization become necessary and crucial to maximise the power output and reduce the cost of energy (COE) of tidal stream energy. In addition, unlike offshore wind turbines which can grow up to hundreds of metres in height (Offshore Wind Turbines Keep Growing in Size, 2011), tidal current turbines often have water depth and bathymetric restrictions. Due to these physical limitations of its operational environment, tidal turbine array optimization is necessary to deliver multi-megawatt power for grid distribution cost-effectively and energy-efficiently. The lack of in-depth investigations on tidal turbine array design will result in poor turbine performance of multi-million dollar-worth tidal current generators.

It is well known that the extraction of the kinetic energy by any turbine causes a reduction in the momentum of the downstream wake flow. Hence, for multiple rows and columns of a tidal turbine array, some turbines might be in the wake of the upstream turbines. As a consequence, detrimental influences on the tidal turbines can be lower power output and increased fatigue loads. Therefore, it is crucial to investigate flow characteristics of the wake because it is directly related to the overall turbine array performance, lifetime of turbines and COE.

It is generally expensive and less efficient to use experimental facilities to do hydrodynamic analysis (Martínez et al., 2012; Li and Çalişal, 2010a;) because it is usually not only time consuming in preparation but also complicated or incapable in obtaining a clear physical insight of the flow. Moreover, similarity theory (White, 1999) may be not applicable at different model scales. Recently, an experiment with three tidal turbines was carried out in a flume tank with a blockage ratio of 20, and the different longitudinal distances between turbines with and without lateral offset were tested (Javaherchi et al., 2013). Another experiment involving an array of 10 turbines was carried out by GL Garrad Hassan (Parkinson et al., 2012; Stallard et al., 2013). The scale turbines were employed and arranged in two rows with different lateral spacing. The objective of their study was to provide benchmarking data to validate the numerical results on the wake representation. However, attention should be paid to both scale effect and blockage effect due to the limit of the flume tank size. On the contrary, CFD methods have fewer constraints, and they are able to solve the full-scale model and thus, tunnel blockage ratio can be optimised to eliminate blockage effect. They also can provide a more reliable numerical solution on the fluid–turbine interaction and more reasonable wake characteristics.

Researchers have agreed that numerical modelling of fluid–

* Corresponding author.
E-mail address: ljing2005@gmail.com (J. Liu).
URL: http://erian.ntu.edu.sg (J. Liu).

http://dx.doi.org/10.1016/j.oceaneng.2016.03.061
0029-8018/

turbine interaction becomes tremendously popular over the last decades alongside with the development of powerful computer resources. The computational methods involved in solving flow characteristics of the turbine wake include but not limited to Reynolds-averaged Navier-Stokes (RANS) models coupling with MRF or sliding mesh techniques to fully resolve the actual three-bladed turbine, two-dimensional (2D) porous media models such as Actuator Disk Model (ADM) (Blanco, 2009), and hybrid models like RANS equations coupling Blade Element Methodology (RANS&BEM), and the Actuator Line Model (ALM) (Jourieh et al., 2006). Among these methods, the MRF and sliding mesh techniques require very fine grid around the turbine rotor to capture flow details, hence computationally expensive (Li and Çalişal, 2010a). However, their primary advantage lies in the ability to represent the three-dimensional (3D) wake structure more realistically than other methods.

The ADM technique is a common method to simulate the turbine wake field in the existing literature (Blanco, 2009; Harrison et al., 2010a; Javaherchi, 2010; Sun et al., 2008). A research group from the University of Southampton compared ADM numerical results with experimental results by utilizing porous media discs (Harrison et al., 2010a; Bahaj et al., 2007; Myers and Bahaj, 2010). However, the swirl flow exerted by the turbine blades was neglected in their study; therefore the obtained downstream results of the near-field wake within $7D_r$ (D_r is horizontal turbine diameter) distance were not realistic. A new method called blade-induced turbulence model was proposed by Nishino and Willden (2012). In the ADM technique, the ratio of energy of the turbulence due to turbine rotating over the extracted energy by the turbine was coupled. An additional turbulence kinetic energy κ and a dissipation rate ε were added to the ambient turbulence in the numerical modelling. However, there still have been limited studies discussing the direct comparison on flow characteristics of the wake between the ADM result and the numerical solution of the actual three-bladed turbine or the laboratory test data. In addition, the same input parameters of the ADM are commonly used for all turbines in an array with the assumption that all turbines have the same performance as an isolated turbine in the free-stream (Bai et al., 2009; Myers and Bahaj, 2012). To our best knowledge, there are very few studies in the literature discussing the consequences of using the different ADM parameters for different turbines in the downstream wake region (Harrison et al., 2010a; Javaherchi 2010). To fill this gap, the present study will focus on the sensitivity of input parameters for ADM and the effect of the ambient turbulence on the wake field.

The RANS&BEM method has been proved to be powerful in predicting output power of the turbine and flow characteristics of the wake (Harrison et al., 2010b; Turnock et al., 2011). However, it was found that this method could not represent the blade tip vortices of a rotating rotor, and thereby a fast recovery of the velocity in the downstream was obtained (Batten et al., 2013). Thus, a tip loss correction should be taken into account (Shen et al., 2005). Moreover, the RANS&BEM model cannot resolve the helical vortex of the wake induced by the blade tips and the transient flow situation as presented by Batten et al. (2013). Additionally, this model has been questioned on the possibility to model the downstream wake meandering phenomenon (Li and Çalişal, 2010b; Sanderse, 2009). Although this method has been applied to estimate the performance of turbine array with different configurations, there are very limited numerical and experimental data with respect to full scale turbine array to validate the accuracy of this method (Cobscook Bay Tidal Energy Project, 2012).

The objective of the present study is to investigate flow characteristics of the wake behind tidal current turbines by solving the RANS equations coupled with different turbine modelling techniques such as MRF, sliding mesh, and ADM. Their brief information is explained in Section 2. The first two techniques are commonly employed to deal with the rotational effect of tidal turbines. However, both generally demand computationally expensive resources. So an alternative ADM technique is considered as well in the present paper. The ADM method employs an actuator disk as a porous media to replace the actual turbine rotor. The kinetic energy extracted from the flow by the turbine is simulated by applying a equivalent momentum loss in terms of pressure difference across the disk. In Section 3, the numerical results are compared with the published theoretical results and the flume tank testing data for benchmarking. It is found that the sliding mesh technique could more accurately predict hydrodynamic loads from current and flow characteristics of the wake. For the ADM technique, reliable results are obtained only in the far wake after parametric studies. Consequently, the sliding mesh technique is used to study turbine to turbine interactions. An in-depth analysis of influence of the wake on the downstream turbines is provided including power and thrust output and flow characteristics of the wake. In the end, conclusions are made in the final section.

2. Numerical methodology

2.1. Problem description

In the present study, the tidal current turbine is assumed to be completely submerged in the sea water, so that its performance is not affected by constraints of the free surface and the seabed. In practice, particular clearance has to be given between the deployed turbine and the free surface in order to minimize the wave effect on the performance of the turbine (Walker, 2013). However, Fraenkel (2010) recommended that a turbine rotor should be placed in the top half of water depth which contains approximately 75% of total tidal stream energy. Therefore, a certain distance is required between the turbine rotor and the seabed to minimise the effect of boundary constraints.

The theoretical results of the turbine performance published by Batten et al. (2008) were used to evaluate the present numerical methods. The rotor diameter is 20 m, and the blade profile is NACA 638xx five digit series. The details on turbine specifications and its operational performance are the same to that in the literature (Batten et al., 2008) and are provided in Table 1. It is a three-bladed turbine with a rated rotating speed of 11.5 rpm. The Reynolds number (Re) is approximately 1×10^7, which is calculated using the blade chord length c at approximately 75% of the blade length. The Reynolds number is defined as

Table 1
Turbine specifications and its operational performance data.

Blade profile	NACA 638xx
N: Number of blades	3
U: Free stream velocity [m/s]	2
ρ: Sea water density [kg/m^3]	1025
μ: Sea water viscosity [kg/(m s)]	1.08×10^{-3}
D_r: Rotor diameter [m]	20
D_h: Hub diameter [m]	2.25
TSR: Tip speed ratio	6
R_e: Reynolds number	1×10^7
ω: Rated rotor speed [rpm]	11.5
P_t: Rated power [kW]	580
F_T: Thrust force [kN]	490
C_p: Power coefficient	0.45
C_t: Thrust coefficient	0.76