International Journal of Marine Energy 16 (2016) 41–50

Contents lists available at ScienceDirect

International Journal of Marine Energy

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

# Numerical prediction of the turbulent wakes generated by a row of marine turbines

Arturo Fernandez

*Department of Mechanical Engineering, North Carolina A&T State University, 1601 E. Market St., Greensboro, NC 24711, United States*

ARTICLE INFO

*Article history:*
Received 10 July 2015
Revised 10 February 2016
Accepted 3 May 2016
Available online 7 May 2016

*Keywords:*
Marine turbine
Turbulent wake
Merging
Lateral separation
Modeling

ABSTRACT

Numerical simulations of the turbulent wakes generated by rows of up to five marine turbines are presented. The numerical scheme combines an actuator disk model with a non-uniform velocity distribution, and the solution to the Reynolds averaged Navier–Stokes (RANS) equations. The first computations examine the dynamics of the turbulent wakes when the stream flows perpendicular to the row. The experiments of Stallard et al. (2013) serve as the benchmark for the numerical predictions. The computations also explore the significance of turbulence intensity and yawed conditions. An increase in the ambient turbulent intensity results in a quicker wake recovery. When the fluid stream is yawed to the turbine row, the problem loses its symmetry and some turbines realize lower velocities as they become affected by the wakes generated by other turbines.



## 1. Introduction

Marine energy is still a mostly untapped resource that could potentially supply significant amounts of power, especially to coastal populations, from a renewable resource. One technology currently under development is stream hydrokinetic energy, where turbines located along water streams such as rivers, tidal sites or ocean currents, convert the kinetic energy from these streams into electricity. However, a single turbine is only able to generate a limited amount of energy and installations with multiple turbines or rotors are the only practical way to generate sufficient energy to justify the project cost. An example of this approach is the SeaGen project [1] involving a turbine with a dual rotor. However, multiple rotors generate multiple turbulent wakes leading to a hydrodynamic interaction. One effect from this hydrodynamic interaction is that turbines located downstream realize a lower incident velocity, which results in loss of power production. Another harmful effect is the increase in turbulence intensity levels, which accentuate fatigue, especially on rotating blades, when turbines are interacting. As a nascent technology, the hydrodynamic interaction between marine turbines is still poorly understood and some recent experiments have begun to examine its impact on turbine performance. Stallard et al. [2] measured the flow in a flume tank for turbines aligned in a row perpendicular to the flow direction. These experiments provide valuable information when rotors are close such as in multiple-rotor turbines. Mycek et al. [3,4] conducted experiments with two turbines lining up in the direction of the flow. They observed a decrease in the power generated by the downstream turbine, which depended on separation distance and ambient turbulence intensity.

The cost and complexity of experiments have prompted the development of numerical tools able to study the response of turbines under different ambient conditions, and to isolate the significance of mechanical or environmental parameters. The first computational models capturing the dynamics of turbulent wakes were developed for wind energy and have been

*E-mail address:* afernand@ncat.edu

http://dx.doi.org/10.1016/j.ijome.2016.05.004
2214-1669/

comprehensively reviewed by Sanderse et al. [5]. The two most suitable approaches for turbulent wake modeling are Reynolds averaged Navier–Stokes (RANS) equations and Large-Eddy Simulations (LES). In the former method, the Navier–Stokes equations are averaged and closed with an eddy-viscosity turbulence model. In the latter methodology, the conservation equations are filtered and a subgrid stress model is used to quantify the significance of the unresolved scales. LES provide greater detail of the flow turbulence structure at the cost of higher computational resources. However, and because of their low cost and ability to predict large-scale behavior, RANS is commonly used for engineering purposes. Modeling of marine turbine wakes has built on the experience acquired from wind energy by adapting the capabilities to the specific problems presented by marine energy. Batten et al. [6] reviewed early works for a single turbine using RANS with actuator disk models and how these predictions compare versus the implementation of a blade-element model. Their study indicated a great variability in the numerical predictions, especially in the near wake region. Other computations focused on single turbines include the works of Afgan et al. [7], Frost et al. [8] and Fernandez [9]. In order to examine the interaction between the structure bearing the turbine and the wake, Afgan et al. [7] implemented a sliding mesh technique. These computations captured blades, nacelle and a bearing mast, and included both RANS and LES simulations. The results of Afgan et al. [7] showed wake meandering caused by the interaction between the wake vortical structures and the vortex shedding phenomena due to the mast. Frost at al. [8] used ANSYS to examine the performance of a turbine with a yawing mechanism and a bi-directional turbine in a tidal flow; the simulations predicted a better performance for the former turbine. Fernandez [9] coupled an actuator disk model with a non-uniform velocity distribution to a parabolized RANS solver and the numerical predictions compare more favorably to the experiments of Neary et al. [10] and Mycek et al. [11] than when the standard actuator disk model was used. In addition to single turbine predictions, several groups have recently performed array computations. Bai et al. [12] used a blade element model coupled to RANS simulations to compute arrays of up to twelve turbines. Malti et al. [13] coupled a blade element model with RANS to study one, two, three, and fourteen turbines; for the latter simulations, two staggered seven-turbine rows resulted in a better performance than four rows with a higher later al spacing. Churchfield et al. [14] used OpenFOAM to implement a line actuator model to compute an array of 48 turbines. Vennel et al. [15] have reviewed the progress in the design of tidal turbine arrays where they stress the contributions by numerical methods. Vennel et al. [15] make the distinction between macro-design, which involves the quantification of the total number of turbines necessary to achieve a target power, and micro-design, which focuses on the study of the performance of individual turbines or rows as in the present study. Unfortunately, one of the conclusions from these works is that there is no universal relationship between the power production from an array and the nominal capacity of a turbine based on stream velocity and turbine characteristics. Most of abovementioned works considered the stream to flow perpendicular to the turbines, which will not always be the situation in practical situations. The present work examines the turbulent wakes generated by a row of turbines and how yawing conditions affect wake dynamics. To accomplish this task, the numerical method introduced in Fernandez [9] is expanded to accommodate the computation of several turbines. The paper is structured as follows: Section 2 presents the implementation of the numerical model; the computational results and conclusions are discussed in Sections 3 and 4, respectively.

## 2. Numerical model

The numerical method combines two elements: (i) the continuum solver, which computes the dynamics of turbulent wakes using a RANS formulation, and the $k$–$\varepsilon$ model to close the equations; (ii) the actuator disk model with non-uniform velocity distribution that captures the perturbation in the flow caused by the turbines. Fernandez [9] discusses the implementation of the method, which is briefly outlined in the following subsections.

### *2.1. Continuum solver*

The modeling of the wake is based on the UPMWAKE model [16] originally developed for wind energy. The model uses a perturbation description where the flow velocity, $\mathbf{v}$, is composed by the ambient flow, $\mathbf{u}_0$, which is hereafter referred by the subscript '0', and the perturbation or wake created by the turbine, $\mathbf{u}$. Hence, the simulations compute the total flow as the sum of the ambient and perturbed fields (i.e. $\mathbf{v} = \mathbf{u}_0 + \mathbf{u}$). Using Einstein notation, the equations describing the wake evolution read:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\rho(\bar{u}_j + u_{0j})\frac{\partial(\bar{u}_i + u_{0i})}{\partial x_j} = -\frac{\overline{\partial p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu\frac{\partial(\bar{u}_i + u_{0i})}{\partial x_j} + \tau_{i,j}^{turb}\right] \tag{2}$$

$$(\bar{u}_j + u_{0j})\frac{\partial(k + k_0)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\frac{\nu_t + \nu_{0t}}{\sigma_k}\frac{\partial(k + k_0)}{\partial x_j}\right) + 2(\nu_t + \nu_{0t})\left[\frac{1}{2}\left(\frac{\partial(\bar{u}_i + u_{0i})}{\partial x_j} + \frac{\partial(\bar{u}_j + u_{0j})}{\partial x_i}\right)\right]\left[\frac{1}{2}\left(\frac{\partial(\bar{u}_i + u_{0i})}{\partial x_j} + \frac{\partial(\bar{u}_j + u_{0j})}{\partial x_i}\right)\right] - (\varepsilon + \varepsilon_0) \tag{3}$$