Journal of Fluids and Structures 66 (2016) 69–90

Contents lists available at ScienceDirect

Journal of Fluids and Structures

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

Simulating tidal turbines with multi-scale mesh optimisation techniques

Mohammad Amin Abolghasemi [a,*], Matthew D. Piggott [a], Johannes Spinneken [b], Axelle Viré [c], Colin J. Cotter [d], Sarah Crammond [e]

[a] Applied Modelling and Computation Group, Department of Earth Science and Engineering, Imperial College London, UK
[b] Department of Civil and Environmental Engineering, Imperial College London, UK
[c] Faculty of Aerospace Engineering, Delft University of Technology, The Netherlands
[d] Department of Mathematics, Imperial College London, UK
[e] MeyGen Ltd., 90A George Street, Edinburgh EH2 3DF, UK

ARTICLE INFO

Article history:
Received 19 November 2015
Received in revised form
26 June 2016
Accepted 1 July 2016
Available online 6 August 2016

Keywords:
Tidal turbines
Actuator disc momentum
CFD
RANS turbulence models
Mesh optimisation

ABSTRACT

Embedding tidal turbines within simulations of realistic large-scale tidal flows is a highly multi-scale problem that poses significant computational challenges. Here this problem is tackled using actuator disc momentum (ADM) theory and Reynolds-averaged Navier–Stokes (RANS) with, for the first time, dynamically adaptive mesh optimisation techniques. Both $k-\omega$ and $k-\omega$ SST RANS models have been developed within the Fluidity framework, an adaptive mesh CFD solver, and the model is validated against two sets of experimental flume test results. A brief comparison against a similar OpenFOAM model is presented to portray the benefits of the finite element discretisation scheme employed in the Fluidity ADM model. This model has been developed with the aim that it will be seamlessly combined with larger numerical models simulating tidal flows in realistic domains. This is where the mesh optimisation capability is a major advantage as it enables the mesh to be refined dynamically in time and only in the locations required, thus making optimal use of limited computational resources.

1. Introduction

This study focuses on the extraction of tidal stream energy from coastal waters via horizontal axis tidal turbines which are currently the favoured approach to efficiently harness the vast and reliably predictable tidal resource. The deployment of tidal turbines is a complex and expensive operation and this makes the task of locating the optimal position for such turbines even more important. Maximising the power output of arrays of turbines is essential, but the environmental impacts must also be studied and modelled in depth as it is vital to ensure that the efforts to reduce carbon emissions do not result in new environmental concerns. Previous studies have shown that in order to correctly assess the power extraction from tidal turbine arrays, an *undisturbed flow approach*, also termed *the flux method* (BLACK & VEATCH, 2012), does not suffice and the hydrodynamic influences of the turbines and their wake interactions must be accounted for (Garrett and Cummins, 2007; Whelan et al., 2009; Vennell, 2010; Nishino and Willden, 2012). Therefore, a numerical model that aims to examine the power output and environmental impacts of tidal turbine arrays must be able to capture these features.

* Corresponding author.
E-mail address: m.abolghasemi12@imperial.ac.uk (M.A. Abolghasemi).

http://dx.doi.org/10.1016/j.jfluidstructs.2016.07.007
0889-9746/

Currently many large-scale marine hydrodynamic models employed to study marine energy use the depth-averaged shallow water equations. In order to numerically simulate arrays of turbines, these models usually adopt a discrete approach where the turbines are represented as a region of increased bottom drag (Divett et al., 2013) and the low complexity of such an approach allows for adjoint-based optimisation techniques to be used to improve turbine positions (Funke et al., 2014). More recently, Funke et al. extended this to find the optimal turbine density, i.e. the number of turbines per unit area represented as a continuous field. These models benefit from relatively low computational costs that allow for large scale realistic simulations, such as modelling arrays of tidal turbines in the Pentland Firth (Martin-Short et al., 2015). However, the main shortcoming of such an approach is that it can fail to account for important turbulent physics and 3D effects, e.g. since the flow passing below and above the turbine is not modelled. Alternatively a higher fidelity 3D model coupled with an appropriate turbulence model can be used. A number of these models have been developed and validated against experimental flume tests with promising results (Roc et al., 2013; Afgan et al., 2013), but their high computational expense has generally prevented their application in larger scale regional simulations.

In a recent study, Masters et al. (2015) investigated the performance of a range of computational models of horizontal axis tidal turbines at different spatial scales. As part of that study, both a high resolution blade element momentum (BEM) model and a large scale coastal model were used to simulate the flow past a small tidal turbine array at an idealised headland. It was demonstrated that while the flow velocities in the far upstream were very similar, substantial differences exist in the wake profiles of the two models. This is not surprising given the differences in spatial and temporal scales used, as well as the different treatment of turbulence in the two approaches. Ultimately, it was concluded that the choice of model will depend on the physical scale of interest and the computational resources available (Masters et al., 2015).

Mesh optimisation techniques can help bridge the gap and improve the accuracy of large scale simulations without the need for excessive computational power. Creech et al. (2012) utilised dynamic mesh optimisation to develop a high fidelity ADM-LES model to accurately model the flow past wind turbines. Furthermore, Divett et al. (2013) developed a 2D depth-averaged model coupled with an adaptive mesh flow solver to demonstrate the greater energy extraction that can be achieved from turbines arranged in staggered layouts.

The model presented here is based on the actuator disc momentum (ADM) theory, where the turbines are represented as momentum sink terms. This 3D approach is coupled with a mesh optimisation algorithm that employs a fine spatial resolution only in regions of interest. This allows for an accurate turbine wake characterisation while maintaining a relatively low computational cost. An ultimate aim will be to use this model to assess the effects of deploying tidal turbine arrays in realistic domains, such as the Inner Sound of the Pentland Firth where MeyGen Ltd plan to deliver a fully operational 398MW renewable energy plant powered purely by the tide (MeyGen).

An alternative approach to ADM theory would be to apply BEM theory, whereby radially varying forces in both axial and azimuthal directions are applied. This method better represents the performance of a real turbine as it introduces a swirl component in the wake profile. Stallard et al. (2015) investigated the mean wake properties behind a single three-bladed rotor and demonstrated that between 0.5 and 2 diameters downstream of the turbine, BEM theory can account for the near wake properties reasonably well. However, Batten et al. (2013) demonstrated that the swirl component of the wake dissipates quickly in the streamwise direction and the far wake profiles produced by ADM and BEM methods are very similar. Given that the aim of the current study is to develop a tool for array design, as long as the turbines are not positioned in very close proximity to each other, neglecting swirl remains a reasonable assumption.

The paper is organised as follows. First in Section 2 the Fluidity ADM model is presented along with a description of the turbulence models used and the mesh optimisation techniques employed. Then in Section 3 verification against the Conway solution for flow past an actuator disc (Conway, 1995) as well as a comparison against a similar OpenFOAM model is presented. This is then followed by two more realistic test cases, Sections 4 and 5, where the Fluidity ADM model is validated against experimental flume test results. The paper concludes with a general overview of the results and a discussion on the benefits of the model presented.

2. Methodology

The numerical model presented has been developed within the Fluidity framework, an open source finite element CFD code with 3D mesh optimisation capabilities (Piggott et al., 2008). The main feature of Fluidity that motivates this study is its ability to adapt the mesh dynamically in time and only in the locations of interest. The code is also highly parallelisable which makes it attractive for larger scale computationally challenging applications.

2.1. Governing equations

One of the main challenges when attempting to simulate the flow past a turbine is the ability to correctly account for the turbulence within the flow. This is vital, given that the ambient turbulence intensity has a significant effect on the structure of the turbine wake and its recovery as demonstrated by Blackmore et al. (2014) and Mycek et al. (2014a, 2014b). For the purpose of this study it has been decided to incorporate turbulence models based upon the Reynolds-averaged Navier–Stokes (RANS) approach, given that their computational cost is significantly lower than that required with large eddy simulations (LES). These models are based on the RANS equations, in which the velocity is decomposed into mean and

Please cite this article as: Abolghasemi, M.A., et al., Simulating tidal turbines with multi-scale mesh optimisation techniques. Journal of Fluids and Structures (2016), http://dx.doi.org/10.1016/j.jfluidstructs.2016.07.007