A Numerical Investigation into Tidal Stream Turbine Wake Dynamics and Device Performance in Non-Uniform Flows

Rami Malki, Ian Masters, Alison J. Williams and T. N. Croft
Marine Energy Research Group, College of Engineering, Swansea University
Swansea, UK

ABSTRACT

A numerical modelling study is presented in which tidal stream turbine performance and wake development in offshore conditions are assessed. The model implements the Blade Element Momentum method for characterising turbine rotor source terms which are used within a Computational Fluid Dynamics model for predicting the interaction between the turbines and the surrounding flow. The model is applied to a rectangular domain and a range of slopes are implemented for the water surface to instigate flow acceleration across the domain. Within an accelerating flow, wake recovery occurred more rapidly although rotor performance was not affected.

KEY WORDS: Ocean energy; marine currents; wake recovery; computational fluid dynamics

INTRODUCTION

There are considerable efforts to reduce Carbon emissions at a European scale to comply with ambitious emission reduction targets. In the UK for instance, the Climate Change Act was passed into law recently whereby an 80 % reduction in greenhouse gases compared to 1990 levels is legally required by 2050 (Climate Change Act, 2008). One of the strategies for achieving this is a move from the burning of fossil fuels towards renewable energy alternatives for power generation. In Wales, one attractive option is the exploitation of tidal stream energy. Some of the most suitable tidal streams for power generation in Europe, let alone the UK, in terms of both energy levels and accessibility, lie along the Welsh coastline, particularly around Anglesey to the North and Ramsey to the West.

Many such suitable sites tend to lie in very environmentally sensitive and protected locations (Willis et al., 2010). Environmental impact assessments are therefore required prior to deployment of tidal stream turbines to assess their likely influence on existing hydrodynamics and their effects on the environment. Considering the large financial investment into the development and production of these devices, their performance in such complex environments must also be assessed and detailed feasibility studies would be required prior to deployment. There is clearly a need to improve our understanding of how tidal stream turbines are likely to affect the environment, and how they are likely to perform in natural offshore environments.

Full scale trial deployments can produce realistic performance data and, indeed a number of such schemes are currently underway (Fraenkel, 2007; Gilson, 2010; Paish et al., 2010). This is a relatively expensive option, but is very useful in assessing the performance of these devices and inform decisions on future larger-scale deployments.

Flume experiments are a convenient alternative for assessing turbine performance and wake hydrodynamics under controlled conditions (e.g. O’Doherty et al., 2006; Batten et al., 2007; Myers et al., 2010). This can be very useful for collecting repeatable measurements to assist in the understanding of how these devices can perform and how they are affected by minor variations in flow conditions. However, such experiments are often conducted using simplified or hypothetical flow conditions. In practice, tidal stream turbines are likely to be deployed in offshore environments with much more complex and highly turbulent conditions, hence there is still a need for actual offshore deployments to complement laboratory studies.

A cheaper option is the use of numerical models to simulate turbines in a wide range of flow conditions. Such models require calibration, hence the need for reliable measured data, however, once this is achieved, numerical modelling results can be very reliable. The majority of numerical studies conducted on tidal stream turbines to date have focused on the efficiency of the devices for different flow and operating conditions (e.g. O’Doherty et al., 2009; Masters et al., 2010; Masters et al., 2011). However, CFD models are also capable of capturing the influence of tidal stream turbines on hydrodynamics in the far-stream.

Velocity and turbulence structures in natural environments can be fairly complex depending on the local bathymetry as well as other factors affecting the flow such as wind-generated waves. Yet many studies on turbine performance and wake characteristics are conducted for rectangular flumes or domains where the flow structure is likely to be less complex. A more complex flow structure is likely to affect turbine performance and wake development. For this reason, there is a requirement for site-specific evaluations of the feasibility of a tidal stream turbine array prior to deployment.

Detailed CFD modelling of turbines can produce accurate predictions (O’Doherty et al., 2009), although this may be quite demanding on computational resources, particularly considering the complexity of such problems. An alternative modelling approach is the Blade Element Momentum (BEM) method (Glauert, 1935). This has been used widely, particularly within the helicopter (Rajagopalan and Mathur, 1993) and wind power (Sørensen and Kock, 1995) industries, and has been applied successfully to tidal stream turbines (Masters et al., 2011). The modelling approach implemented here combines the two whereby the