CFD Techniques for Modelling Marine Current Turbine Arrays under Oceanic Flows

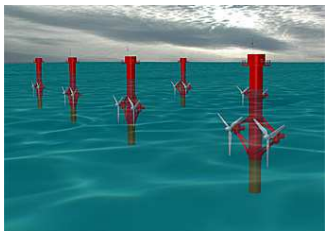
M E Harrison, A S Bahaj, W M J Batten

Work Stream 4: Arrays, wakes and near field effects

Introduction

The work will consider the effect of wake induced turbulence and oceanic flows on the performance of an array of horizontal axis tidal turbines.

Commercial installations will probably see horizontal axis tidal turbines installed in arrays. This work will consider two key hydrodynamic factors that affect array performance.



Site induced perturbations:

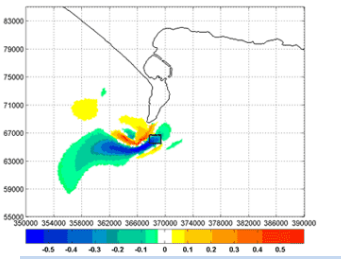
- Ebb and flood tides tend to be asymmetric.
- Stream-wise shear, & turbulent kinetic energy vary with depth.
- Wave induced velocities also affect conditions.

Turbine Wake Effects:
Operation of the turbine will result in significant perturbations to the flow up to at least 20 diameters downstream of the device [1].

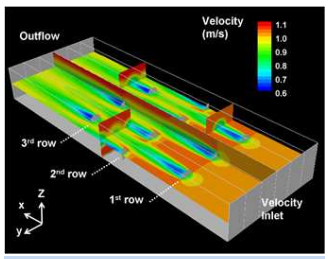
Wake model measurements [1]

Background

Previous array models have taken a 2D boundary layer approach under simulated tidal conditions [2], or have relied on a an actuator disc approximation in CFD, without considering ocean conditions [3].



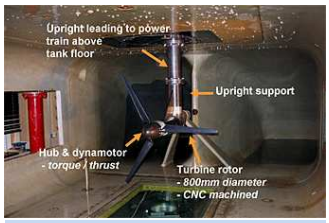
An array model at Portland Bill [2]



A model of 3 rows of turbines [3]

This work will combine aspects of both models: A 3D unsteady CFD model with realistic inflow conditions derived from tidal measurements.

The CFD Models will be validated with data from ongoing experiments studying turbine wakes within the Sustainable Energy Research Group [4].



Scale model testing [4]

Methodology & Current Work

- RANS CFD simulations will be used to give hi-resolution data about the flow around each individual turbine.
- Each device will be modelled as a distribution of momentum sources, derived to simulate a turbine. This approach will be compared with a numerical model of a turbine blade, and experimental results.
- Large arrays of turbines will be modeled using this validated technique to examine turbine wake effects
- Measured unsteady flow data from real tidal sites will be used to derive appropriate flow boundaries in the models.
- In Initial studies an array of turbines has been modeled in 2D with each row approximated as a porous fence (based on actuator disc theory). Experimental work has been done to validate the model. [5]

References

[1] Myers L. E., Bahaj A. S., (2006). Flow effects in marine current turbine arrays. Proceedings World Renewable Energy Congress (WREC-IX), Florence, 19-25 Aug 2006

[2] Blunden L.S. and Bahaj A.S. (2007) Effects of tidal energy extraction at Portland Bill, southern UK predicted from a numerical model. Proceedings Seventh European Wave and Tidal Energy Conference, Porto, Portugal, 11-14 September 2007.

[3] Batten W.M.J. and Bahaj A.S. (2006) CFD simulation of a small farm of horizontal axis marine current turbines. Proceedings World Renewable Energy Congress (WREC-IX), Florence, 19-25 August 2006.

[4] Bahaj A. S., Molland A.F., Chaplin J. R., Batten W.M.J., (2007). "Power and thrust coefficients of marine current turbines operating under various hydrodynamic conditions of flow in cavitation tunnels and towing tanks." Renewable Energy, 32,3, 407-426

[5] Harrison et al., Comparisons of a large tidal turbine array using the Boundary layer and Field wake interaction models, to be presented at the Second International Conference on Ocean Energy (ICOE 2008), 15th to 17th October, Brest.

University of Southampton
School of Civil Engineering and the Environment

Sustainable Energy Research Group
www.energy.soton.ac.uk

EPSRC
Engineering and Physical Sciences Research Council