CFD Techniques for Modelling Wake Effects in Tidal Turbine Arrays

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

Work stream 4: Arrays, wakes and near-field effects

Introduction

At a commercial scale marine current turbines are likely to be installed in groups, or arrays. Turbine performance will be affected by the wake of those upstream. This work focuses on modeling arrays using CFD, to understand the effect of wakes and other flow characteristics on power output and optimal spatial arrangement.

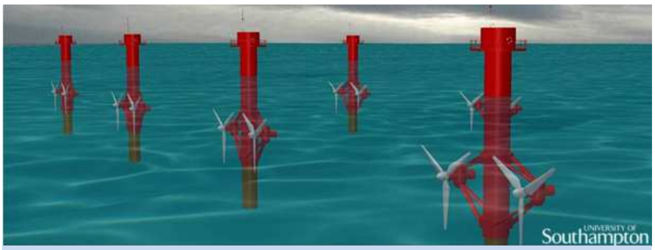


Illustration of a turbine array

Background

CFD (Computational Fluid Dynamics) provides a useful tool for understanding the flow within arrays. However a full rotor CFD model (e.g. [1]) would be very computationally expensive when modelling a number of devices. This work concentrates on the use of three, more efficient, actuator disc rotor approximations which are implemented in a commercial CFD-RANS solver:

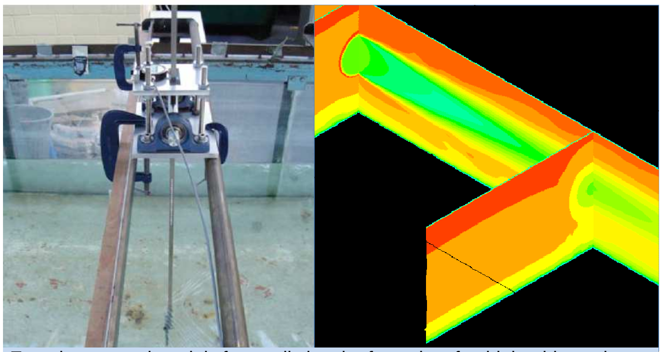
- The constantly loaded actuator disc [2] applied to tidal turbine arrays by Batten in [3].
- The blade element actuator disc, discussed in [4]
- The actuator line, also discussed in [4].

Firstly a verification and validation study is being undertaken for each of these techniques. This includes studies of grid, time step and iterative convergence, and quantitative comparison with theoretical and experimental results.

Secondly, once satisfactory parameters have been defined, the most suitable rotor model (in terms of a trade off between accuracy and computational time) will be selected and used to model array configurations.

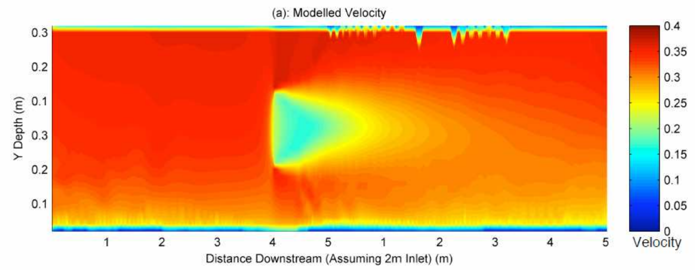
Preliminary Results

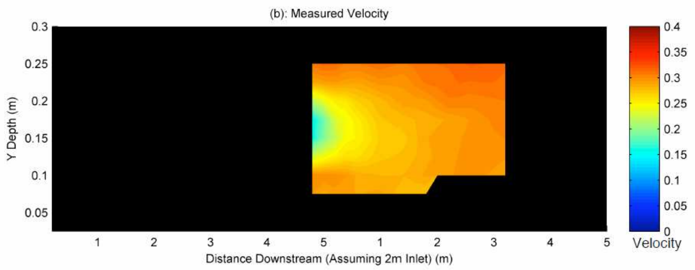
Preliminary modeling has compared the constantly loaded disc to experimental results [5,6,7].



Experiments and models for predicting the far wake of a tidal turbine using a constantly loaded actuator disc.

The results showed good agreement in terms of the wake velocities, however the methods for modelling turbulence levels require further work.





References

1.Mason-Jones, A., O'Doherty, T., O'Doherty, D. M., Evans, P. S, Wooldridge, P. S. (2008) Characterisation of a HATT using CFD and ADCP site data. in Proceedings World Renewable Energy Congress (WREC-X). 2008.Glasgow.

2.Froude, R E (1889) On the part played in propulsion by differences in fluid pressure. Transactions of the institute of naval architects. 30 p390

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

4. Mikkelsen, R E. (2003) Actuator Disc Methods Applied to Wind Turbines. PhD Thesis. Mechanical Engineering,Technical University of Denmark

5. Harrison, M.E., Batten, W. M., Bluden, L. S., Myers, L. E., Bahaj, A. S., (2008) Comparisons of a Large Array of Tidal Turbines Using the Boundary and Field Wake Interaction Models, in Second International Conference on Ocean Energy. 2008: Brest, France.

6. Harrison, M. E., Batten W. M., Myers L. E., Bahaj, A. S. (2009). A comparison between CFD simulations and experiments for predicting the far wake of horizontal axis tidal turbines. 8th European Wave and Tidal Energy Conference. Uppsala, Sweden, 2009

7. Myers, L. E., Bahaj, A.S., Thomson, M. D., Jorge, N. (2007) Characterisng the Wake of Horizontal Axis Marine Current Turbines. in Proceedings Seventh European Wave and Tidal Energy Conference. 2007. Porto.

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