Proceedings of the 2nd International Conference on Environmental Interactions of Marine Renewable Energy Technologies (EIMR2014), 28 April – 02 May 2014, Stornoway, Isle of Lewis, Outer Hebrides, Scotland.
www.eimr.org

EIMR2014-7118

The modelling of tidal turbine farms using multi-scale, unstructured mesh models

Stephan C. Kramer[1] **Matthew D. Piggott**[1,2] **Jon Hill**[1]
Louise Kregting[3] **Daniel Pritchard**[3] **Bjoern Elsaesser**[3]

[1] Applied Modelling and Computation Group,
Department of Earth Science and Engineering,
Imperial College London

[2] Grantham Institute
for Climate Change,
Imperial College London

[3] School of Planning, Architecture
and Civil Engineering,
Queen's University Belfast

ABSTRACT

A model intercomparison study is presented between MIKE 21 and Fluidity for the modelling of tidal turbine farms. Close agreement is observed in the outcomes of both models. An important aspect is the parameterisation of turbines in tidal models that typically do not resolve the individual turbine scale. Here we present a correction to the applied drag force to ensure results that are less mesh resolution dependent.

INTRODUCTION

Tidal energy generation through free stream turbines placed in high tidal currents is now progressing to commercialisation with a number of sites planning to become operational in the near future. Numerical modelling of the tidal flow and the influence of turbines on this play a key role in the development of new projects, in particular where large numbers of turbines are placed together in a farm. Typical questions that can be answered through hydrodynamic models are: How much energy is available, what is the optimal configuration of a farm and what is its impact on the environment? Because of the often complicated geometry of high tidal-current environments, and the large range of length scales, from large-scale tidal flow down to the turbine scale, unstructured mesh models are a natural candidate to efficiently answer these questions.

The flow through and around turbines is a complex, fully three-dimensional phenomenon that is typically modelled using 3D CFD models. However, for a tidal model that may have to extend out to 100s of kms to avoid boundary effects and requires to be run for a significant period of time to capture the full tidal cycle, a two-dimensional, depth-averaged approximation is often the only feasible approach. The parameterisation of turbines in such a model however is not straightforward. The challenge is to incorporate them in a consistent and mesh-independent way. In addition, a correct representation of the turbine wake and parameterisation of the turbulence is essential, especially for farm configuration optimisation studies.

Here we present the results of an inter-model comparison of Fluidity and MIKE 21 and discuss issues surrounding the correct parameterisation of turbines. Fluidity [1] is an open source, mesh adaptive, finite element modelling framework that has a wide range of applications from CFD problems to large scale oceanographic simulations. It allows for the simulation of both turbine scale flow and large scale tidal flow within the same modelling framework and thus to study the effects of the interaction between the two. MIKE 21 [7] is a well-established, depth-averaged unstructured mesh model for coastal simulations, that is already used by many in the marine renewable industry.

TURBINE PARAMETERISATION

The drag force that a single turbine exerts on the flow is usually given as:

$$F = \tfrac{1}{2}\rho_w C_T A_e \|u\|^2, \qquad (1)$$

where ρ_w is the water density, C_T the thrust coefficient, A_e the effective cross-sectional area of the turbine exposed to the current and $\|u\|$ is the upstream current speed.

Various approaches exist to implement this force in a tidal model. Draper et al. [2] make use of actuator disc theory to describe the effect of a single turbine or a row (fence) of turbines on the depth-averaged flow, and show how its effect can be implemented as a line momentum source in a Discontinuous Galerkin model via a jump condition. See also Serhadlıoğlu et al. [5] for implementation of this approach in the ADCIRC model. In those works, the fence of turbines needs to be represented in the computational mesh as a line. Alternatively, in [6] and [3] individual turbines are represented in the mesh as rectangles over which the drag force is

---

[1] Corresponding author: s.kramer@imperial.ac.uk

spread. This requires a very high mesh resolution with grid distance that are smaller than the turbine diameter.

The turbine parameterisation approach followed in MIKE 21 is targeted at lower resolution models where the turbine scale is not necessarily well represented. The drag force, equation (1), is applied as a momentum point source, which is spread over the single grid cell that contains the turbine (the cell typically being much larger than the turbine). For the purposes of model intercomparison we have used the same parameterisation in Fluidity.

A complication of applying (1) is that it depends on the upstream current speed, whereas the local speed is expected to be reduced due to the presence of the drag force representing the turbine. For low resolutions, where the drag is applied over a large cell, the local velocity may not be very different from the upstream. With increasing resolution however, the drag is more focussed and the reduction in local velocity grows, and thus the exerted drag force keeps decreasing. As an example, in figure 1 it is shown how the local velocity at the turbine decreases in MIKE as the mesh resolution is increased while the upstream velocity is kept constant at $u = 3.05$ m/s (a similar reduction was observed in Fluidity using the same turbine parameterisation). In the same figure, it is also shown how using actuator disc theory the drop in velocity can be predicted. Reversely, this theory can therefore be used to compute an upstream velocity from the local velocity, and this can be easily incorporated in MIKE 21 and Fluidity as a correction on the drag force. The corrected force is given by:

$$F = \tfrac{1}{2}\rho_w C_T A_e \frac{4}{\left(1+\sqrt{1-\gamma}\right)^2} \|u_{\text{local}}\|^2,$$

where $\gamma = C_T A_e / (\Delta x H)$ with water depth H and grid distance Δx (in the triangular grids that were studied here this should be the width of the triangle perpendicular to the flow direction). This factor incorporates a correction for the fact that instead of applying the force over the actual turbine cross section A_e, we apply the force over the entire water column and over a width Δx that may be much larger than the turbine width. It is easily verified that for $\gamma = C_T$, we obtain the standard result from actuator disc theory. This is used in for instance [4] where indeed in a higher resolution 3D model, the numerical drag force is applied over the same cross-sectional area as A_e. Our method ensures that the amount of momentum that has to be extracted from the flow is more accurately calculated in the case that individual turbines are not resolved. This also leads to a more accurate estimate of the energy extraction.

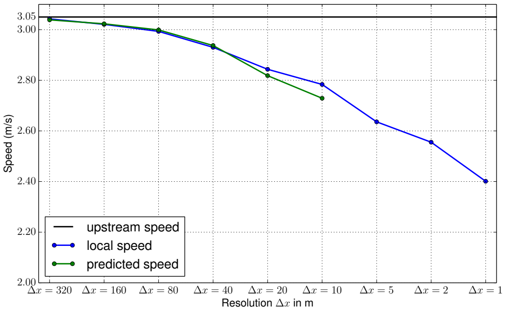
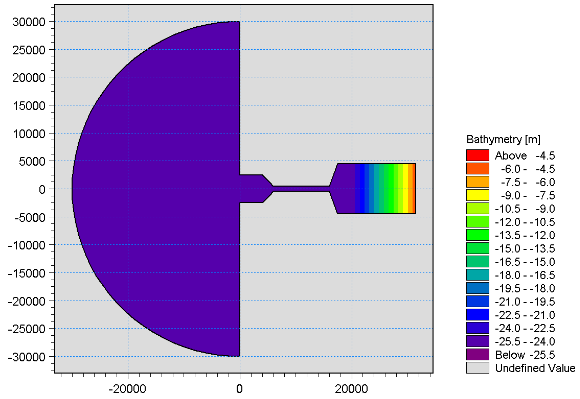


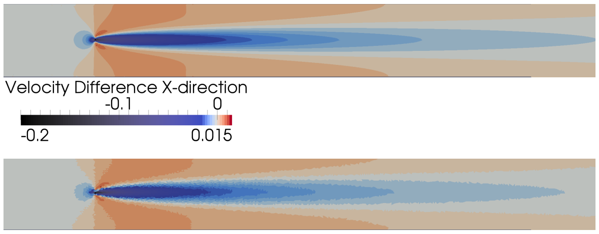
Figure 1. Local turbine velocity in MIKE 21 (blue) decreases with increasing mesh resolution. Actuator disc theory can be used to predict (green) this drop in velocity as a function of Δx. This prediction is valid as long as Δx is bigger than the turbine scale. Note that a 10% drop in the local velocity leads to a 20% drop in the quadratic drag force that represents the turbine.

TIDAL BASIN BENCHMARK

To gain more confidence in the hydrodynamic model solutions, a benchmark test case was set up to study the performance of both models. It consists of an idealised tidal basin, inspired by the Strangford Lough geometry, with a tidal free surface forcing on the open boundary.



For the comparison, various mesh resolutions, ranging from $\Delta x = 1000$ m to $\Delta x = 31.25$ m, and turbine configurations have been used. As an example, the following two figures show the velocity deficit (difference in solution with and without a turbine) in the wake of a turbine for Fluidity (top) and MIKE (bottom on a fine mesh, $\Delta x = 31.25$ m).



The agreement of both models has been studied in different ways. One way is to study the convergence with mesh resolution, by projecting the coarser solutions onto the finest mesh and taking the integrated L^2-norm of the difference with the finest solution. In figure 2, this is shown for the velocity solution at a flood tide. It can be seen that both models converge between first and second order for $\Delta x \geq 250$ m (the channel width is 1000 m) and their solutions stay relatively close.

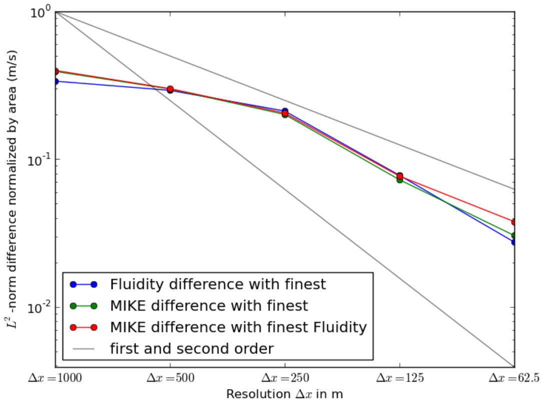


Figure 2. Integrated L^2-norm difference between model results at various mesh resolutions and the result at finest resolution ($\Delta x = 31.25$ m). Blue shows the convergence of Fluidity's results to its finest resolution result, and green the convergence of MIKE's results to its finest resolution. Finally, red shows the convergence of MIKE's results to the finest resolution Fluidity results.

CONCLUSIONS

Both MIKE 21 and Fluidity have been shown to provide reliable results for the modelling of tides in marine renewable projects. However, in the parameterisation of tidal turbines, the drag force, which is implemented as a quadratic function of the local velocity, may become dependent on the mesh resolution in an unrealistic way. This is a result of the fact that the turbines in large scale tidal models are typically underresolved and the local velocity will therefore be somewhere between the free stream velocity and the local turbine velocity predicted by actuator disc theory. Here, we suggest a correction that allows estimation of the free stream velocity from the local velocity taking into account the mesh resolution. It is shown that this leads to a drag force that is much less mesh dependent, as opposed to the unmodified drag force which may show a drop of 20% when the resolution is refined down to the turbine scale.

ACKNOWLEDGEMENTS

This work is part of the SuperGen UK Centre for Marine Energy Research (UKCMER) project ‘‘Large Scale Interactive Coupled Modelling of Environmental Impacts of Marine Renewable Energy Farms’’ (LINC) funded by EPSRC (reference EP/J010065/1).

REFERENCES

[1] Applied Modelling and Computation Group, Imperial College London. Fluidity manual v4.1. `http://www3.imperial.ac.uk/earthscienceandengineering/research/amcg/fluidity`.

[2] S. Draper, G. Houlsby, M. Oldfield, and A. Borthwick. Modelling tidal energy extraction in a depth-averaged coastal domain. *IET renewable power generation*, 4(6):545–554, 2010.

[3] S. W. Funke, P. E. Farrell, and M. Piggott. Tidal turbine array optimisation using the adjoint approach. *Renewable Energy*, 63:658–673, 2014.

[4] T. Roc, D. C. Conley, and D. Greaves. Methodology for tidal turbine representation in ocean circulation model. *Renewable Energy*, 51:448–464, 2013.

[5] S. Serhadlıoğlu, G. Houlsby, T. Adcock, S. Draper, and A. Borthwick. Assessment of tidal stream energy resources in the UK using a discontinuous Galerkin finite element scheme. In *17th Int. Conf. on Finite Elements in Flow Problems, San Diego, CA, 24–27 February 2013*, 2013.

[6] R. Vennell. Tuning turbines in a tidal channel. *Journal of fluid mechanics*, 663:253–267, 2010.

[7] I. Warren and H. Bach. Mike 21: a modelling system for estuaries, coastal waters and seas. *Environmental Software*, 7(4):229–240, 1992.