VI International Conference on Computational Methods in Marine Engineering
MARINE 2015
F. Salvatore, R. Broglia and R. Muscari (Eds)

A COMPARISON OF NUMERICAL MODELLING TECHNIQUES FOR TIDAL STREAM TURBINE ANALYSIS

I. MASTERS, A. J. WILLIAMS, T. N. CROFT, M. TOGNERI, M. EDMUNDS, E. ZANGIABADI AND I. FAIRLEY

Marine Energy Research Group
Swansea University
Singleton Park, Swansea, UK
E-mail: I.Masters@swansea.ac.uk, http://www.swansea.ac.uk/engineering/marine-energy/

Key words: CFD, BEMT, coastal/shelf scale, comparison, validation

Abstract. To fully understand the performance of tidal stream turbines for the creation of ocean renewable energy, a range of computational models is required. We review and compare results from several models at different length scales. Models under review include blade element momentum theory (BEMT), blade element actuation disk RANS-CFD, blade-resolved RANS-CFD and coastal models based on the shallow water equations. Three sets of experimental results are used for model validation.

1 INTRODUCTION

Attempts to fully understand the mechanics of extraction of energy from tidal currents is a challenging computational engineering problem. The main form of a tidal renewable energy device is an axial flow turbine using hydrofoils that generate lift and drag. The rotation of the turbine and the movement of the tidal current result in an angle of attack at the hydrofoil that provides suitable torque to extract energy. Therefore, the primary requirement of a numerical scheme is to be able to describe the relative rotational movement of the aerofoil and consequently to estimate the lift and drag forces generated. One approach is to utilise a moving reference frame containing the rotating turbine blades, which we describe in this paper. Difficulties of this scheme occur at the boundary of the moving part of the mesh, which manifests itself as a pressure discontinuity. More computationally efficient schemes treat the rotor as an actuator disk, blade element disk or actuator line. These schemes are also discussed, with focus on the blade element disk approach. Finally, the most efficient numerical scheme is a blade element momentum theory (BEMT) approach, widely used for analysis of propellers and other rotors, where the flow field is assumed and the computation reduces to a 1D treatment of the blade elements, comparing tabulated lift and drag co-efficients with axial and rotational induction factors. This scheme is computationally very fast and can be used to study in detail transient effects, such as the effects of waves and rotor control strategies.

Secondly, the difficulty of validation of these schemes is the lack of field data. Several sets of results are in the literature based on experiments with relatively small diameter rotors.

Calculation of Reynolds number based on the blade chord length shows that Reynolds numbers are significantly lower than those found in typical datasets for aerofoils in air [1]. Computationally efficient schemes such as the BEM-CFD approach rely on pre-determined lift and drag data and are therefore sensitive to uncertainty in these estimates. We present a sensitivity study of these effects and propose some solutions to improve such schemes.

2 BEMT

The basic theory of BEMT is widely described in the literature [2,3], so we do not go into detail here. Briefly, one may describe the principle of the approach as reconciling two different models of a turbine: a blade element (BE) model that treats the turbine as a collection of foil sections generating lift and drag forces in response to the oncoming flow, and a momentum theory (MT) model that treats the turbine as a series of annular elements that absorb linear momentum (representing the slowing-down of the current due to the turbine) and impart angular momentum (representing the swirl induced in the turbine wake). Two parameters are defined that capture the salient details in both models, and we obtain our solution by determining the values of these parameters that bring the two models into closest agreement.

Our BEMT code [4] incorporates several extensions to the classical theory, including tip/hub losses, high induction effects [5] and the ability to model an arbitrary inflow. Here, we present its ability to capture the sensitivity of turbine performance to small changes in the hydrodynamic properties of the blade profiles.

We begin by considering three rotors for which experimental results have already been published. These turbines will be referred to by the institute at which the experimental work has been carried out. Thus we have the Liverpool rotors, reported in [6,7]; the IFREMER turbine, reported in [8]; and the Manchester turbine, reported in [9,10]. All rotors have a three-bladed configuration, and the blade geometries for each are shown in figure 1.

The Liverpool rotor uses a Wortmann FX63-137 section for the entire blade length, the IFREMER rotor uses a NACA 63418 section, and the Manchester rotor uses a Göttingen 804 foil. BEMT relies on a table of lift and drag coefficients to calculate the forces generated by the blade elements, and the data for each of these foils was taken from different sources. The coefficients for the Wortmann foil were taken from a flume study carried out at Swansea [11]; for the NACA 63418 section, data was taken from standard tables [1] for moderate angles of attack, with flat-plate theory used for extreme angles of attack; for the Göttingen foil data was taken from a wind tunnel study [12]. Data for the Göttingen foil is much less complete than that for the other sections; as a result, the BEMT results are noticeably discontinuous, since in many cases a relatively small change in inflow angle results in a large jump in the lift and drag properties of the foil. This is also noticeable in the BEMT work carried out at Manchester, as can be seen in the studies that report their experimental results [9].

As we mentioned above, the purpose of the work presented here is to show the sensitivity of turbine performance to small changes in the lift and drag properties of the hydrofoils used in the rotor blades. For each rotor, we calculated five sets of performance data: one with the original geometry, two with decreased lift (i.e., C_L decreased by 5% and 10% at all inflow angles) and two with increased drag (i.e., C_D increased by 5% and 10% at all inflow angles). These results are presented graphically in figure 2, along with experimental data for the IFREMER and Manchester turbines.

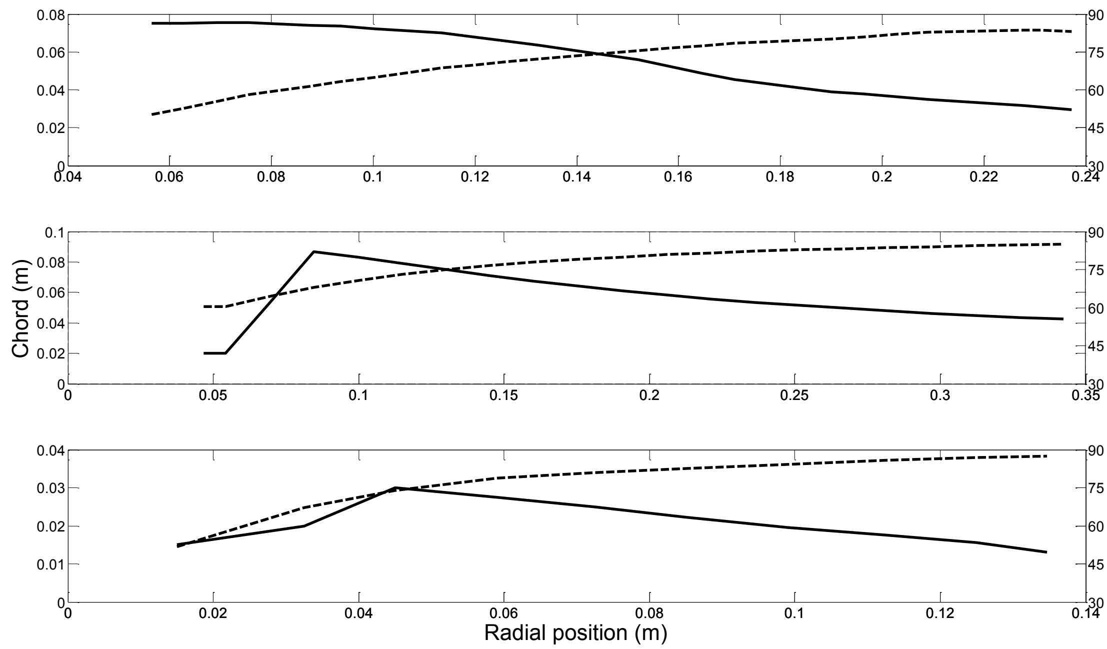

Figure 1: Blade geometries for the Liverpool (top), IFREMER (middle) and Manchester (bottom) turbines

It is immediately apparent that there are no significant qualitative changes in turbine performance as a result of lift/drag changes of this magnitude. This is reassuring from an operational point of view, as it implies that relatively small changes to the blade characteristics that may occur as a results of biofouling or blade erosion/pitting are unlikely to cause a catastrophic drop in performance.

The details of the performance changes vary between rotors: we tabulate the most important parameters in table 1. It is easily seen that the Liverpool rotor is more sensitive to the lift/drag changes than the IFREMER rotor. Maximum axial force on the IFREMER rotor, for instance, is effectively unchanged by increasing the drag, while for the Liverpool rotor, both the 5% and 10% drag increases result in the peak axial force decreasing by 2.13%. For both rotors, we can see that reduction in section lift produces a more significant drop in axial force.

Power output is usually the more salient criterion for a device developer. Again restricting our consideration to the changes at peak power, we see that the performance of the Liverpool rotor is more significantly affected by the profile changes. The peak power of both rotors is less sensitive to changes in drag than changes in lift - this is to be expected, as it is the lifting action of the rotor blades that actually creates the torque necessary for a turbine to generate power.

		Liverpool	IFREMER	Manchester
Max. C_P	Original	0.4622	0.4122	0.3313
	C_D +5%	0.4432 (*-4.11%*)	0.4094 (*-0.69%*)	0.3181 (*-3.99%*)
	C_D +10%	0.4427 (*-4.21%*)	0.4066 (*-1.37%*)	0.3063 (*-7.53%*)
	C_L - 5%	0.4425 (*-4.26%*)	0.4077 (*-1.09%*)	0.3198 (*-3.46%*)
	C_L - 10%	0.4392 (*-4.98%*)	0.4035 (*-2.11%*)	0.3016 (*-8.97%*)
Max. C_{Fa}	Original	0.8189	0.7531	0.9631
	C_D +5%	0.8014 (*-2.13%*)	0.7532 (*+0.01%*)	0.9591 (*-0.42%*)
	C_D +10%	0.8014 (*-2.13%*)	0.7533 (*+0.02%*)	0.9705 (*+0.76%*)
	C_L - 5%	0.7857 (*-4.06%*)	0.7374 (*-2.08%*)	0.9432 (*-2.07%*)
	C_L - 10%	0.7702 (*-5.95%*)	0.7224 (*-4.08%*)	0.9196 (*-4.52%*)
Optimum TSR	Original	3.78	4.52	4.64
	C_D +5%	3.54 (*-6.35%*)	4.50 (*-0.44%*)	4.58 (*-1.29%*)
	C_D +10%	3.78 (*-5.29%*)	4.46 (*-1.33%*)	4.68 (*+0.86%*)
	C_L - 5%	3.70 (*-2.12%*)	4.58 (*+1.33%*)	4.54 (*-2.16%*)
	C_L - 10%	3.78 (*+0.00%*)	4.62 (*+2.21%*)	4.62 (*-0.43%*)

Table 1: Changes to maximum power and thrust coefficients and optimum TSR for Liverpool, IFREMER and Manchester turbines in response to small changes in lift and drag properties of blade sections

It is not only the value of the optimum C_P that is altered by these changes in blade profile properties, but also the TSR at which this optimum is attained. An increase in sectional drag coefficient always results in a downwards shift of the optimum TSR, while a decrease in lift has different effects: for the Liverpool rotor, the optimum TSR shifts slightly downwards, but for the IFREMER rotor the shift is upward. A sophisticated control scheme, then, may be able to use sensitivity analyses such as those presented here to partially compensate for blade degradation by altering the TSR at which the turbine is operated.

We can also see that the agreement between the BEMT code's predictions of the unmodified rotor performance and the experimental data for the IFREMER and Manchester turbines is good throughout the range of experimental TSR values; the apparent exception is the axial force coefficient for the IFREMER turbine, but this discrepancy is attributable to the fact that the measured axial force included the force on the supporting structure and not simply the rotor disc itself. Despite this difference, it can nevertheless be seen that the trend of axial force's dependence on TSR matches the experimental observations.

3 CFD AND BEM-CFD

The most complete computational model of a full-scale tidal turbine that can feasibly be

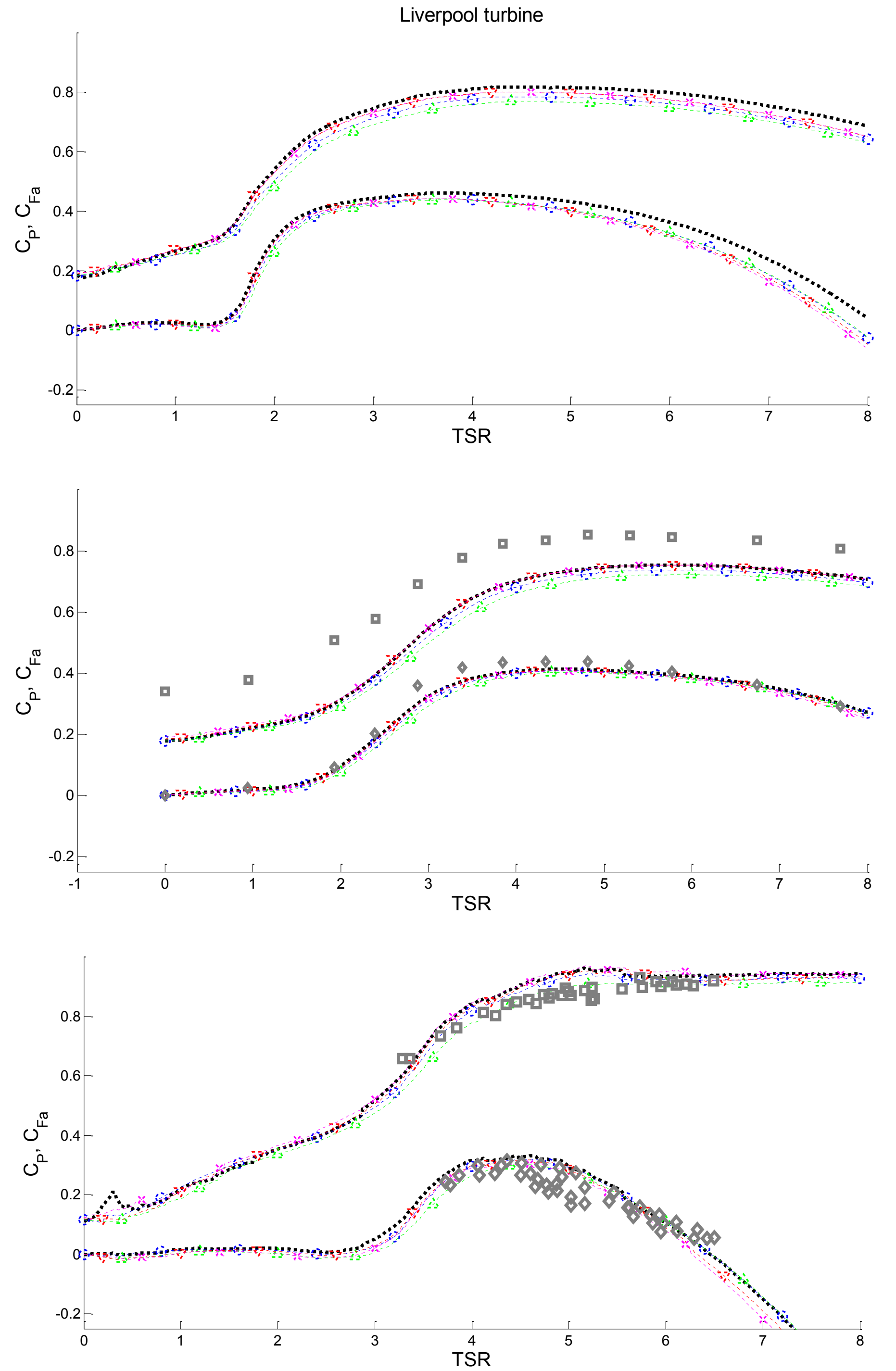

Figure 2: Power (C_P) and axial force (C_{Fa}) coefficients for Liverpool, IFREMER and Manchester turbines (from top to bottom). In all cases, the upper cluster of lines shows C_{Fa} and the lower shows C_P. Black dashed lines show original turbine performance; blue lines (with circles) show the 95% C_L case; green lines (with triangles) show the 90% C_L case; red lines (with triangles) show the 105% C_D case; magenta lines (with crosses) show the 110% C_D case. Experimental results are also shown for the IFREMER and Manchester turbines, with data taken from [8] and [10] respectively.

run is LES, although unsteady RANS remains more common [13,14]. Such a model will necessarily rely on a moving mesh to account for blade rotation rotation. The moving boundary in the mesh presents difficulties, manifesting as a pressure discontinuity and longer computational costs. An alternative method is the BEM-CFD model in which the flow properties are resolved by interaction of the BEM and CFD methods [15-18]. As BEM by itself does not provide any useful information about the effect of a turbine in the far field, we use CFD to resolve the full domain [19]. In other words, the BEM method is used to model the turbine, and CFD is employed to model the flow properties elsewhere in the domain, thus giving us a time-averaged estimate of the turbine wake while significantly reducing the computational cost compared to a geometry-resolved CFD model. Here, we will present two sets of results from a BEM-CFD model validated against CFD and against experimental measurements.

We start by modelling a single tidal turbine configuration in both the BEM-CFD method and a blade-resolved geometry (BRG) CFD method. In the comparison between these two models, we are primarily interested in the velocity deficit in the turbine wake and the pressure behaviour, particularly immediately downstream of the turbine. An understanding of turbine wakes is vital if tidal turbines are to be deployed in arrays (as turbine wakes will inevitably impinge on other turbines located downstream), and such arrays are the only way tidal turbines can be economically viable.

The CFD model, including the finite volume model construction and set up, was provided by the Marine Energy Research Group of Cardiff University in Fluent. It has a 10m diameter turbine in a rectangular domain with the dimension of 506m × 50m × 50m, in which the turbine is located 104 m from the inlet to allow the flow to settle before reaching the turbine. The inlet boundary condition is set as a uniform flow of speed 3.086 ms^{-1}, and a no-slip condition is imposed at the bed and blade surfaces. For all the side walls and the top of the domain, symmetry boundary conditions have been enforced. At the outlet boundary, a zero diffusion flux condition for all flow variables is imposed. The rotational speed of the blade was set to 2.25 $rad \cdot s^{-1}$, the optimum rotational speed for the rotor in normal operation (TSR 3.64). The zone that represents the rotation of the rotor is set to have 17 m diameter and 6 m width.

An equivalent model has also been implemented using the BEM-CFD method. The domain geometry is the same, and is meshed with 6.03 million tetrahedral elements. The boundary conditions are the same as those used in the Fluent model. Rather than representing the geometry of the turbine directly as is done in the CFD model, we treat the rotor as momentum source/sink in the domain. This BEM-CFD method has been validated in the previous research of the group [20]. Both the BRG-CFD and BEM-CFD models employ a standard finite volume approach with a k-ε turbulence model, in order to make the comparison more meaningful. Note also that in both cases we are modelling only the rotor itself: no support structure (nacelle, tower etc.) is included.

In figure 3, we compare the two models' predictions of velocity deficit at five diameter (5D) intervals between the distances of 5D and 40D behind the turbine. The values are extracted along the horizontal line passing behind the turbine at the hub level. We see that the models agree well throughout most of the wake. Five diameters downstream of the rotor, the velocity has dropped by more than 50% in comparison to the inlet velocity, although on the centreline the BRG predicts a higher wake velocity than the BEM-CFD. Further downstream, the wake profiles are very similar across the span of the domain, both in terms of the magnitude of velocity deficit and wake spreading. The results also show that the wake shape for the BEM-CFD method is symmetric, while the BRG method's wake is slightly asymmetric in the direction of rotation. The lower minimum velocity observed in the BEM-CFD method could be because of the over-prediction of turbine power in the BEM-CFD method compared to the BRG model. It should be borne in mind that for simplicity this comparison is done in a uniform flow above a flat bed; in the real environment, non-uniform flows and sloped surfaces will influence the results [21].

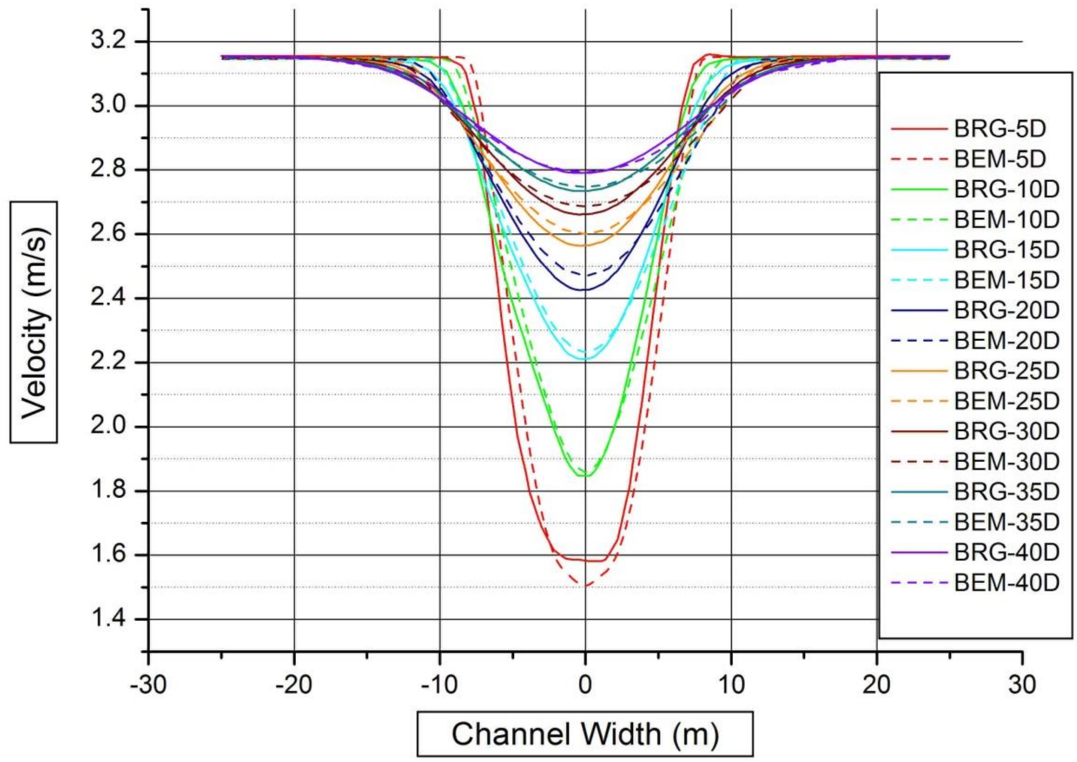

Figure 3: Comparison of the velocity deficit in the rotor wake from BEM-CFD and blade-resolved geometry (BRG) models at a range of downstream locations

The pressure at the wake for both of the models is shown in figure 4. We see that the pressure is very low at 5D, but beyond this the rate of pressure recovery is very swift. With the BEM-CFD method it recovers much faster in comparison to the BRG model. The reason that the pressure comparisons are not as close as the velocity comparison could be because of the amount of turbulence that exists in the flow. The BRG model imparts a greater amount of turbulence to the downstream flow due to the presence of the physical blades while in the BEM-CFD method the effects of the blade are specified as a source term. Thus, in validating our BEM-CFD model's predictions of turbine wakes against BRG-CFD, we can say that the velocity deficit is satisfactory, but the pressure profile is not so well-matched, and that this is probably attributable to the implementation of turbulence production at the blades.

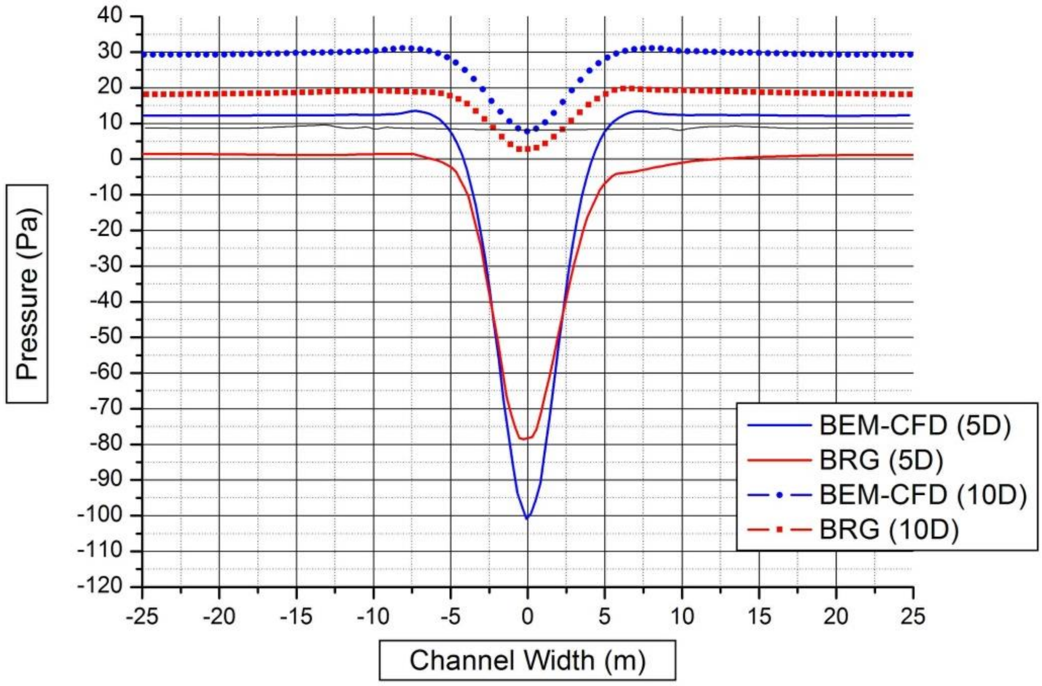

Figure 4: Comparison of wake pressure between BRG and BEM-CFD models at locations 5D and 10D downstream of the rotor.

In addition to validating the BEM-CFD model against the BRG-CFD results, we have also carried out a comparison with experimental results carried out in the IFREMER flume, the same results used as one of the test cases for the BEMT simulations reported in section 2. Figure 5 demonstrates that results from the BEM-CFD method match experimental measurements well, in comparison with figure 9a of [8].

4 COASTAL AND SHELF SEA SCALE MODELLING

While CFD and BEMT approaches are focussed very much on the turbine itself, whether in terms of structural loading, blade performance or detailed wake structures and intra-array interactions, computational limitations mean that different techniques are reqquired for the modelling of larger-scale impacts. Coastal area models are therefore used, which typically solve the RANS equations in two or three dimensions. Horizontal meshes are either rectangular or, increasingly, unstructured, and models are either 2D depth averaged or, in the 3D case, cater for vertical resolution with a series of layers. Length scales of coastal area models range from the order of 10km for site-specific modelling to the order to 1000km for shelf-scale studies of multiple arrays or far-field impacts. Simulation lengths typically range from a single spring-neap cycle to more than a year, with time-steps of minutes to hours. Coastal scale modelling typically investigates available resource [23] or the impacts of energy extraction on the wider environment for array or inter-array scenarios. Resource modelling is

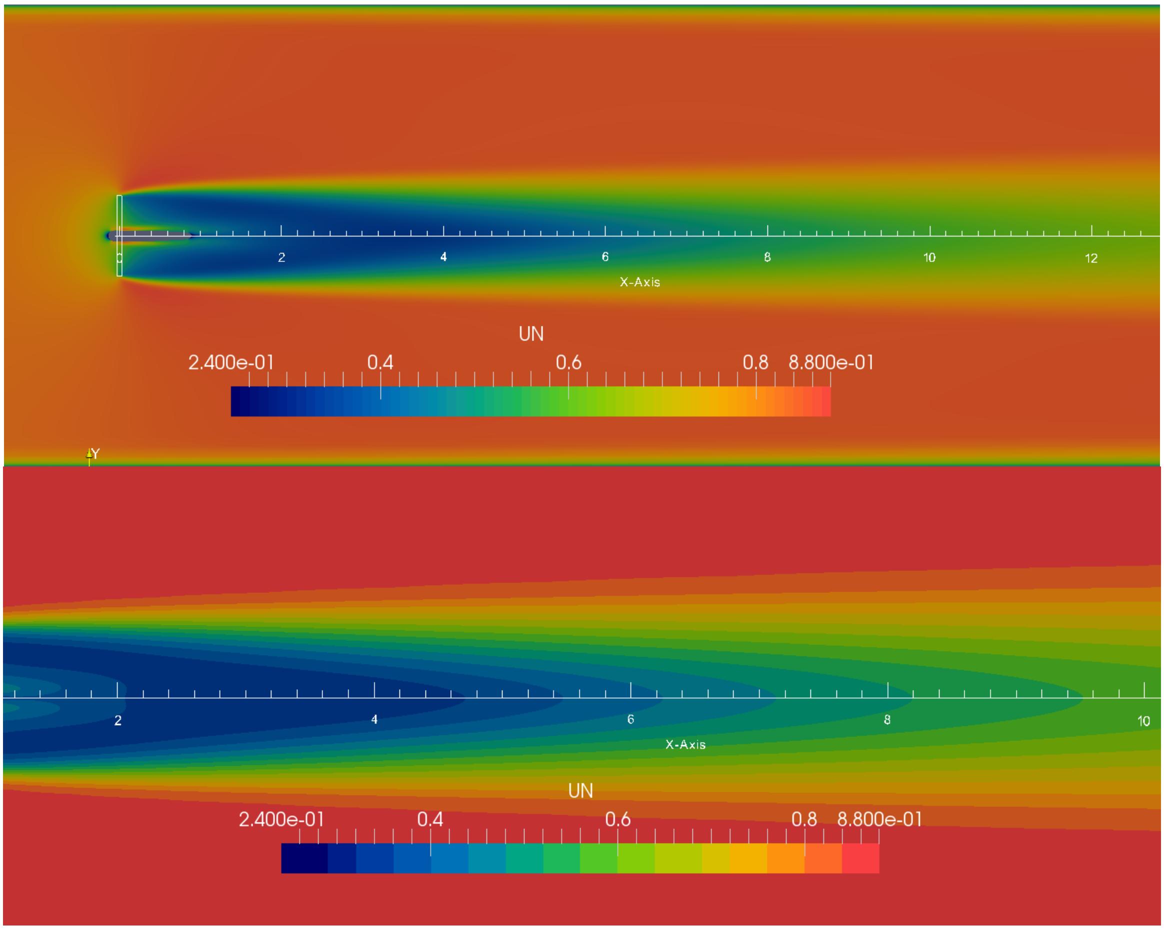

Figure 5: Wake characteristics: axial velocity slice taken at hub centre. Free stream velocity is 0.8 ms-1, TSR is 3.67 and background turbulence intensity is set at 3%. The top visualisation shows the full wake to the end of the domain, while the lower visualisation is staged to compare with figure 9a from [8]

conducted to provide greater spatio-temporal coverage than can be achieved with field measurements [24].

Impacts on the hydrodynamic regime can be observed over a much larger area than covered by CFD. Hydrodynamic responses can be noticed on an ocean scale for tidal barrages and on a regional to shelf-sea scale for tidal stream turbines [25,26]. Changes to hydrodynamic regime can lead to second-order effects. Researchers have also investigated impacts on sediment transport and associated changes to morphology [27-29]. Wave-current interactions [30] can also be an important process both in affecting the tidal resource and the changes to currents impacting the wave climate [31-33], the wáter quality [34] or aquatic organisms. More recently optimal siting for power production over a large area has been considered [35].

A variety of methods have been used for the implementation of turbines in coastal array models: in two-dimensional models the impact of turbines is often included as an additional component of bottom friction within the array footprint, either averaged over the whole array or as individual turbines [28, 29, 36]; in three-dimensional models such an approach would give unrealistic vertical velocity profiles and thus an additional sink term must be introduced to the model [37]. A review of the commonly used methods is presented in [38]. Importantly, the complexity and detail available in coastal area modelling will always be lower than for CFD or similar techniques, meaning that the two methodologies are complementary. It is conceivable that a CFD model could be nested within a coastal area model to provide accurate inflow conditions for CFD and energy extraction patterns for coastal area models.

CONCLUSIONS

Computer modelling has always been a compromise between three issues: the numerical description, computational resources and experimental validation. We have presented here several numerical schemes to describe the extraction of useful energy from a tidal turbine and the interaction that it has with the flow. It is clear that the correct choice of scheme is not always obvious, and depends on the physical scale where answers are required. Each scheme described used very different assumptions. At the largest scale, turbines are smaller than the grid cells and are described as sources. Finite volume approaches use a large number of cells to describe the rotor and its immediate vicinity, with a contrast between blade resolved and embedded blade element formulations. At the very smallest scale, BEMT assumes the wake properties and is purely interested in the rotor loadings. The choice of model will depend strongly on the availability of computational resources. The existence of efficient models is due to the limits on computational power available on a day to day basis to the turbine modelling community. While very large models are possible, they are not practical, and may not add value when the uncertainties in boundary conditions are taken into account.

It is clear that reasonable characterisation of lab-scale flows can be achieved with good instrumentation, and the experiments used for validation can be replicated with reasonable accuracy. However, attempts to model real flows have a very high uncertainty in the physical geometry of the problem and characterisation of the boundary conditions and care should be taken when making comparisons to real turbines in real channels.

REFERENCES

[1] I. H. Abbott and A. E. von Doenhoff, *Theory of Wing Sections, Including a Summary of Airfoil Data*, Dover Books (1959)
[2] H. Glauert, "Airplane Propellers" in *Aerodynamic Theory* ed. W. F. Durand (1934)
[3] T. Burton, D. Sharpe, N. Jenkins and E. Bossanyi, *Wind Energy Handbook*, John Wiley & Sons, Ltd. (2001)
[4] I. Masters, J.C. Chapman, J.A.C. Orme and M.R. Willis, "A robust blade element momentum theory model for tidal stream turbines including tip and hub loss corrections", *Proc. Inst. Marine Engin. Sci. Tech. Pt A – J. Marine Engin. Tech.*, **10**:25-25 (2011)
[5] J.C. Chapman, I. Masters, M. Togneri and J.A.C. Orme, "The Buhl correction factor applied to high induction conditions for tidal stream turbines", *Renew. Energy*, **60**:472-480 (2013)
[6] S.C. Tedds, I. Owen and R.J. Poole, "Near-wake characteristics of a model horizontal axis

tidal stream turbine", *Renew. Energy*, **63**:222-235 (2014)
[7] T.A. de Jesus Henriques, S.C. Tedds, A. Botsari, G. Najafian, T.S. Hedges, C.J. Sutcliffe, I. Owen and R.J. Poole, "The effects of wave-current interaction on the performance of a model horizontal axis tidal turbine", *Int. J. Marine Energy*, **8**:17-35 (2014)
[8] P. Mycek, B. Gaurier, G. Germain, G. Pinon and E. Rivoalen, "Experimental study of the turbulence intensity effects on marine current turbines behaviour. Part I: One single turbine", *Renew. Energy*, **66**:729-746 (2014)
[9] E. Fernandez-Rodriguez, T.J. Stallard, P.K. Stansby, "Experimental study of extreme thrust on a tidal stream rotor due to turbulent flow and with opposing waves", *J. Flu. Struc.*, **51**:354-361 (2014)
[10] T. Stallard, T. Feng, P.K. Stansby, "Experimental study of the mean wake of a tidal stream rotor in a shallow turbulent flow", *J. Flu. Struc.* (in press) (2014)
[11] M. Togneri, I. Masters, R. Malki, A. Rio, "Experimental study of the mean wake of a tidal stream rotor in a shallow turbulent flow" (in review).
[12] U. Hassan, "A wind tunnel investigation of the wake structure within small wind turbine farms", Technical Report ETSU WN 5113
[13] D. O'Doherty, A. Mason-Jones, C. Morris, T. O. O'Doherty, C. Byrne, P. Prickett and R. Grosvenor, "Interaction of marine turbines in close proximity", *9th European Wave and Tidal Energy Conference* (2011), Southampton, UK
[14] A. Mason-Jones, D. O'Doherty, C. E. Morris, T. O'Doherty, C. Byrne, P. W. Prickett, R. I. Grosvenor, I. Owen, S. Tedds *et al.*, "Non-dimensional scaling of tidal stream turbines", *Energy* **44**(1):820-829 (2012)
[15] M. Willis, I. Masters, S. Thomas, R. Glalie, J. Loman, A. Cook, R. Ahmadian, R. Falconer, B. Lin, G. Gao *et al.*, "Tidal turbine deployment in the Bristol Channel: a case study", *Proc. ICE-Energy* **163**(3):93-105 (2010)
[16] M. E. Harrison, W. M. J. Batten and A. S. Bahaj, "A blade element actuator disc approach applied to tidal stream turbines", *OCEANS 2010* (2010), Seattle, USA
[17] R. Malki, A. J. Williams, T. N. Croft, M. Togneri and I. Masters, "A coupled blade element momentum-computational fluid dynamics model for evaluating tidal stream turbine performance", *Appl. Math. Model.* **37**(5):3006-3020 (2013)
[18] A. J. Williams, T. N. Croft, I. Masters, C. R. Bennet, S. G. Patterson and M. R. Willis, "A combined BEM-CFD model for tidal stream turbine", *3rd International Conference on Ocean Energy* (2010), Bilbao, Spain
[19] T. R. McCombes, "An Unsteady Hydrodynamic Model for Tidal Current Turbines", Ph.D Thesis (2013)
[20] R. Malki, I. Masters, A. J. Williams and T. N. Croft, "A Numerical Investigation into Tidal Stream Turbine Wake Dynamics and Device Performance in Non-Uniform Flow", *22nd International Offshore and Polar Engineering Conference*, (2012), Rhodes, Greece
[21] I. Masters, R. Malki, A. J. Williams and T. N. Croft, "The influence of flow acceleration on tidal stream turbine wake dynamics: A numerical study using a coupled BEM-CFD model", *Appl. Math. Model.* **37**(16-17):7905-7918 (2013)
[22] P. Mycek, B. Gaurier, G. Germain, G. Pinon and E. Rivoalen, "Experimental study of the turbulence intensity effects on marine current turbines behavious. Part I: One single turbine", *Renew. Energy* **66**:729-746 (2014)
[23] S. P. Neill, M. R. Hashemi and M. J. Lewis, "The role of tidal asymmetry in

characterizing the tidal energy resource of Orkney", *Renew. Energy* **68**:337-350 (2014)

[24] I. Fairley, P. Evans, C. Wooldridge, M. Willis and I. Masters, "Evaluation of tidal stream resource in a potential array area via direct measurements", *Renew. Energy* **57**:70-78 (2013)

[25] R. Ahmadian, R. Falconer and B. Bockelmann-Evans, "Far-field modeling of the hydro-environmental impact of tidal stream turbines", *Renew. Energy* **28**:107-116 (2012)

[26] R. Ahmadian and R. A. Falconer, "Assessment of array shape of tidal stream turbines on hydro-environmental impacts and power output", *Renew. Energy* **44**:318-327 (2012)

[27] A C. Chatzirodou and H. Karunarathna, "Impacts of tidal energy extraction on sea bed morphology", *34th International Conference on Coastal Engineering*, (2014), Seoul, South Korea

[28] S. P. Neill, J. R. Jordan and S. J. Couch, "Impact of tidal energy converter (TEC) arrays on the dynamics of headland sand banks", *Renew. Energy* **37**:387-397 (2012)

[29] P. E. Robins, S. P. Neill and M. J. Lewis, "Impact of tidal-stream arrays in relation to the natural variability of sedimentary processes", *Renew. Energy* **72**:311-321 (2014)

[30] J. Wolf and D. Prandle, "Some observations of wave-current interaction", *Coast. Eng.***37**:471-485 (1999)

[31] M. R. Hashemi, S. P. Neill, P. E. Robins, A. G. Davies and M. J. Lewis, "Effect of waveson the tidal energy resource at a planned tidal stream array", *Renew. Energy* **75**:626-639 (2015)

[32] I. Fairley, R. Ahmadian, R. A. Falconer, M. R. Willis, I. Masters, "The effects of a Severn Barrage on wave conditions in the Bristol Channel", *Renew. Energy* **68**:428-442 (2014)

[33] M. J. Lewis, S. P. Neill, M. R. Hashemi and M. Reza, "Realistic wave conditions and their influence on quantifying the tidal stream energy resource", *Appl. Energy* **136**:495-508 (2014)

[34] M. Kadiri, M. R. Hashemi, B. Bockelmann-Evans, W. Rauen and R. Falconer, "A review of the potential water quality impacts of tidal renewable energy systems", *Renew. Sust. Energy Rev.* **16**:329-341

[35] S. P. Neill, M. R. Hashemi and M. J. Lewis, "Optimal phasing of the European tidal stream resource using the greedy algorithm with penalty function", *Energy* **73**:997-1006 (2014)

[36] R. Martin-Short, J. Hill, S. C. Kramer, A. Avdis, P. A. Allison and M. D. Piggott, "Tidal resource extraction in the Pentland Firth, UK: Potential impacts on flow regime and sediment transport in the Inner Sound of Stroma", *Renew. Energy* **76**:596-607 (2015)

[37] I. Fairley, I. Masters, H. Karunarathna, "The cumulative impact of tidal stream turbine arrays on sediment transport in the Pentland Firth", *Renew. Energy*, under review (2015)

[38] S. Baston, S. Waldman and J. Side, "Modelling energy extraction in tidal flows", MASTS Position Paper (2014)