European Association for the Development of Renewable Energies, Environment and Power Quality (EA4EPQ)

International Conference on Renewable Energies and Power Quality
(ICREPQ'10)
Granada (Spain), 23rd to 25th March, 2010

Combined BEM-CFD Modelling of Tidal Stream Turbines Using Site Data

A. J. Williams, T. N. Croft, I. Masters, M. R. Willis and M. Cross

Marine Energy Research Group
School of Engineering, Swansea University, Singleton Park, Swansea, SA2 8PP, UK
alison.j.williams@swansea.ac.uk, t.n.croft@swansea.ac.uk, i.masters@swansea.ac.uk, m.r.willis@swan.ac.uk

Abstract. Marine currents have the potential to provide a large proportion of Britain's energy generation requirements. Whilst a number of devices capable of utilizing this resource are under development, there is at present little exploitation of marine energy. One potential generator of energy in this area is tidal stream turbines (TST). However, since they are expensive to install, engineers need to ensure that the TST will deliver optimum performance once they are in place. This performance is dependent on a number of features that are specific to the surrounding environment, including the underlying bathymetry and variation of the current both temporally and within the water column. Computational fluid dynamics (CFD) is a useful tool for predicting what impact the surrounding environment and supporting structure will have on the performance of a TST. In this paper the importance of using measured site data to develop CFD models for TSTs is demonstrated. A CFD model of a TST is presented and two designs for the supporting structure are investigated. A parametric study is carried out using a flat bed model. Finally, a series of results is presented for a site in the Severn Estuary using a bathymetry defined bed.

Keywords

CFD, Blade Element Momentum Theory, Tidal Stream Turbines, Environment

1. Introduction

As one of the signatories of the Kyoto Protocol the United Kingdom is committed to reducing its emissions of greenhouse gases by 12.5 % relative to the 1990 level by 2012. Exploitation of renewable energy sources offers a potential solution to achieve these aims. Certainly to achieve the government's stated desire to reduce greenhouse gas emissions by 80 % by 2050 there is necessity to invest in research and technology associated with energy generation with low or zero carbon footprints.

An area that has not been exploited to the same level as wind is the generation of power from tides. One approach to harness the available energy in the tides is to use a device that resembles the standard wind turbine. The advantage of tidal power generation is that it is a predictable resource. Whilst the turbine will generate no power at the changing of the tides, probably for around 30 minutes every 6 hours, the device may be operational for the remainder of the time. Although the tidal flows are much slower than wind the extra density of water over air means the available energy in the tide is much greater. The disadvantages are mainly associated with installation and maintenance.

This work is part of a collaborative project between 7 groups at Swansea and Cardiff Universities and sponsored by the Low Carbon Research Institute [1]. The project examined all aspects related to the installation of tidal stream turbines, including determining the process necessary to reach consensus regarding installation of the device, surveying techniques necessary to obtain site related data and numerical modelling. In the strand of the project reported here a computational model of a single tidal stream turbine device has been developed.

When modelling turbines with CFD a decision has to be made about how the turbines should be represented in the model. Moving mesh methods and time averaged momentum sources are two approaches that may be implemented. In a moving mesh method the rotor is physically part of the solution domain, whereas in the time averaged momentum source approach, based on blade element momentum theory [2, 3], the rotor characteristics are averaged over a disk of mesh elements. The prediction of the power extracted from the tide is consistently predicted by the two methods as is the nature of the large scale fluid flow. The techniques differ in the vicinity of the rotor where the moving mesh method allows the transient nature of the flow over the moving turbine blades to be fully predicted. Thus the moving mesh method is commonly used where effects local to the rotor are important, such as the calculation of temporal loads on the structure. However, the time averaged method is computationally more efficient than the moving mesh method. As a result it can be used in simulations associated with large scale phenomena such as scouring and sedimentation.

The tidal stream turbine model presented in this paper uses a time averaged momentum source approach. The details of this approach will be presented. The case study will show how the performance of the rotor varies for two supporting structure designs across a range of tip speed ratios. The model will then be used to predict performance at a site in the Severn Estuary using bathymetry data measured by other teams in the collaborative project. The results will investigate the effect that using measured on-site data in a CFD model, instead of using idealised parameters, has on predictions of TST performance.

2. Computational Approach

A. Governing Equations

In this paper the water is modelled as an incompressible turbulent fluid. The Navier-Stokes equations

$$\nabla.\boldsymbol{u} = 0 \tag{1}$$

$$\frac{\partial \rho u_i}{\partial t} + \nabla.(\rho \boldsymbol{u} u_i) = -\frac{\partial p}{\partial x_i} + \nabla.(\mu_{lam} + \mu_t)\nabla u_i + S_i \tag{2}$$

are used to represent the conservation of mass (1) and momentum (2) in the domain, where ρ is the density, u_i is the i'th component of the velocity vector, p is the pressure, μ_{lam} and μ_t are the dynamic laminar and turbulent viscosities respectively, and S_i represents any sources into the equations (for example the source due to the moving rotor).

The effect of turbulence is resolved through the k-ε model [6]. In this model the following two equations are solved:

$$\frac{\partial(\rho k)}{\partial t} + \nabla.(\rho \boldsymbol{u} k) = \nabla.\left(\mu_{lam} + \frac{\mu_t}{\sigma_k}\right)\nabla k + \mu_t G - \rho\varepsilon \tag{3}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \nabla.(\rho \boldsymbol{u}\varepsilon) = \nabla.\left(\mu_{lam} + \frac{\mu_t}{\sigma_\varepsilon}\right)\nabla\varepsilon + \frac{\varepsilon}{k}(C_{1\varepsilon}\mu_t G - C_{2\varepsilon}\rho\varepsilon) \tag{4}$$

Equation (3) represents the energy k contained within the turbulence, and equation (4) represents the dissipation ε of this energy.

Equations (3) and (4) are used to calculate the turbulent viscosity μ_t as follows

$$\mu_t = \frac{\rho C_\mu k^2}{\varepsilon} \tag{5}$$

In equations (3), (4) and (5) σ_k, σ_ε, $C_{1\varepsilon}$, $C_{2\varepsilon}$ and C_μ are taken to be constants, and *G* represents the turbulent generation rate.

B. Blade Element Momentum Method

The concept behind the CFD turbine model presented in this paper is that at a large enough time scale the turbine rotor would photographically appear as a blur. At this time scale the rotor will apply the same force to all locations at the same radial distance from the rotor axis on a given axial plane. This time averaged approach allows sources that represent the force on the fluid due to the blades of the turbines to be applied to each of the momentum equations. The advantage of this approach is that the physical characteristics of the blade are built into the source rather than the mesh consequently allowing the use of better quality meshes. The disadvantage is that because of the time average principle of the approach it fails to resolve any transient flow features due to blade position.

To define the characteristics of the rotor according to axial and radial position the blade element momentum (BEM) [1] method is employed. Figure 1a shows diagrammatically how a three bladed rotor is discretised using the BEM approach. The blade properties for blade element at a certain radius r are determined and are then averaged throughout the whole of the shaded region in this diagram. This is done for each blade element throughout the radius of the rotor.

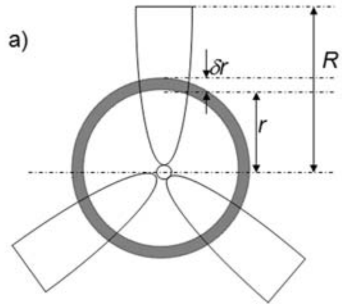

Figure 1 Schematic of discretisation of the rotor

Each blade element has a chord length c_{Fb}, element thickness t_{Fb} and radial width δr. Each element experiences forces acting on it due to the fluid. These are broken down into axial and tangential components as shown in Figure 2. Here *dT* and dF_A represent the torque and axial forces respectively. The lift *dL* and drag *dD* forces are dependent on the angle of attack α between the blade element and the resultant velocity v_R. The angular velocity is represented by Ω and U is the upstream horizontal velocity. The chord inclination angle and flow inclination angle are denoted by β and ϕ respectively.

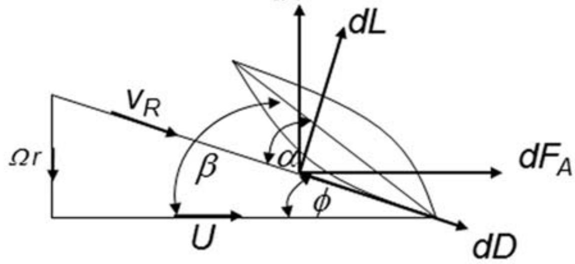

Figure 2 Resolution of lift and drag forces

Following the approach described by Orme [3] an axial force on a blade element is defined by

$$dF_A = dL\sin\phi + dD\cos\phi \qquad (6)$$

and the tangential force on a blade element, which is equal to the torque/radius, i.e. *dT/r*, is defined by

$$dF_T = dL\cos\phi - dD\sin\phi \qquad (7)$$

The flow inclination angle ϕ is defined by

$$\phi = \tan^{-1}\left(\frac{\Omega r - u_\theta}{u_z}\right) \qquad (8)$$

where u_θ, u_z are the local fluid tangential and axial velocities respectively, and Ω is the angular velocity of the blades in rad/s. The lift force *dL* and drag force *dD* are given by

$$dL = \frac{1}{2}\rho\left|v_R\right|^2 c_L c_{Fb}\delta r \qquad (9)$$

$$dD = \frac{1}{2}\rho\left|v_R\right|^2 c_D c_{Fb}\delta r \qquad (10)$$

Here c_L and c_D are the lift and drag coefficients respectively, and

$$\left|v_R\right|^2 = u_z^2 + \left(\Omega r - u_\theta\right)^2 \qquad (11)$$

Substituting (9) and (10) into (6) and (7) gives the following equations

$$S_z = dF_A = \frac{1}{2}\rho\left|v_R\right|^2 c_{Fb}\delta r\left(c_L\sin\phi + c_D\cos\phi\right) \qquad (12)$$

$$S_\theta = dF_T = \frac{1}{2}\rho\left|v_R\right|^2 c_{Fb}\delta r\left(c_L\cos\phi - c_D\sin\phi\right) \qquad (13)$$

which, when resolved into Cartesian components and converted into a force per volume, are substituted into the momentum equations (2) through S_i.

C. Computational Solution of the Equation Set

The solution to the governing equations (1)-(4) is achieved using a collocated cell-centred finite volume (FV) scheme [4] within the software PHYSICA [5]. CFD solution software tools are often based on FV schemes as they are locally as well as globally conservative and computationally efficient when compared to alternative techniques.

In the FV scheme the physical domain is represented by a mesh consisting of non-overlapping polyhedral elements. In the cell-centred scheme a solution node is placed at the centre of each of these elements. The governing equations are integrated over time and each mesh element, and these discretised equations lead to a series of relationships between the unknown values in an element and the values in neighbouring elements. The solution procedure is based on a variant of the SIMPLE [7] algorithm, and, since a collocated scheme is used, the Rhie-Chow approximation is used to prevent checker-boarding of the pressure field [8]. The CFD equations are typically solved for using a diagonally preconditioned conjugate gradient algorithm. Further details of the solution procedure can be found in [9] and [10].

D. Definitions

The tip speed ratio (TSR), which is the ratio between the rotational speed of the tip of the blades and the free-stream, horizontal velocity is defined by

$$TSR = \frac{R\Omega}{U} \qquad (14)$$

where R is the blade radius.

The performance of a rotor can be characterised using the power coefficient, Cp, the torque coefficient, Ct, and the force coefficient, Cf which are expressed as follows

$$Cp = \frac{P}{\frac{1}{2}\rho U^3 A} = \frac{\int_0^R \Omega dT}{\frac{1}{2}\rho U^3 A} \qquad (15)$$

$$Ct = \frac{T}{\frac{1}{2}\rho U^2 AR} = \frac{\int_0^R dT}{\frac{1}{2}\rho U^2 AR} = \frac{\int_0^R r dF_T}{\frac{1}{2}\rho U^2 AR} \qquad (16)$$

$$Cf = \frac{F_A}{\frac{1}{2}\rho U^2 A} = \frac{\int_0^R dF_A}{\frac{1}{2}\rho U^2 A} \qquad (17)$$

In the computational model the integrals in (15)-(17) are calculated from a summation over each element of the blade of the blade force applied to that element as calculated from (12) and (13).

3. Case study

The aim of this case study has been to investigate the impact that the surrounding environment has on the performance of a generic 10 m rotor tidal stream turbine at a particular location in the Bristol Channel. Of particular interest is the shape of the supporting structure and a bathymetry defined realistic sea-bed.

A. The effect of supporting structure on performance

For wind turbines changes in flow behaviour in the vicinity of the rotor can have an effect on both the power output and the structural loads on the rotor. Since the nacelle length is usually kept as small as possible to reduce structural imbalance the rotating plane of the rotor is usually close to the supporting structure. This means that the rotor performance is influenced by the flow around the structure, and for an upstream rotor it results in slower flow in front of the tower, which leads to lower power output. However, for wind turbines, it is believed that the effects on rotor loading are minimized provided that a clearance of approximately one tower diameter between the rotor blade and tower is maintained [11].

In order to investigate the effect that the supporting structure has on the performance of a tidal stream turbine a CFD model with a flat base was initially used. The dimensions of the solution domain for a 10 m rotor are shown in Figure 3. The width of the domain has been chosen to be six times the diameter of the rotor as this ensures that boundary conditions are having little effect on performance, i.e., blockage is not occurring. The downstream channel length is 400 diameters. The height of the domain is taken from navigational chart depths of the sea-bed at the site to be 32 m below LAT (lowest astronomical tide).

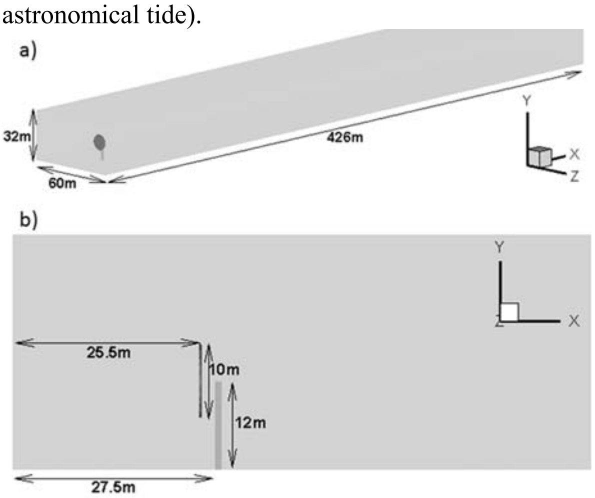

Figure 3 Schematic of the flat-bed geometry a) 3D dimensions b) cross-sectional view showing dimensions of rotor and supporting structure relative to inlet.

In all the following simulations the TST model consists of a vertical tower (either a cuboid or a cylinder) of height 12 m and a 4-blade 10 m rotor. The centre of the model is positioned on the z mid-plane, the centre of the rotor is 25.5 m downstream of the inlet and the centre of the tower a further 2.5 m downstream - see Figure 3b. This gives a clearance of approximately 1.5 m between the rotor and the tower. The cross-sectional area of the cuboid tower is 1 m x 1 m, and the diameter of the cylindrical tower is 1 m. The resulting meshes consist of approximately 350k hexahedral elements.

The following boundary conditions were imposed:

- Upstream flow speed of 1.6 m/s corresponding with velocity measurements taken at the water surface at the proposed site.
- A zero pressure BC on the outlet and symmetry BC on the top and vertical sides of the domain.
- No-slip BC on the base of the domain and the surface of the tower.

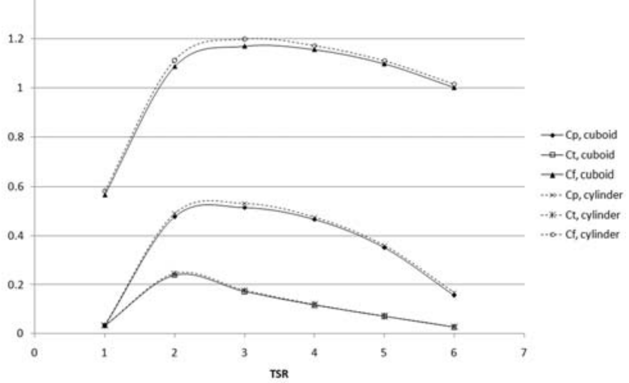

Figure 4 A comparison of *Cp*, *Ct* and *Cf* for the cuboid and cylindrical towers on the flat bed domain.

Figure 4 shows how the power coefficient, *Cp*, torque coefficient, *Ct*, and axial force coefficient, *Cf*, vary with increasing TSR for both cuboid and cylindrical towers.
It can be seen that for both towers the optimum performance occurs between TSR=2 and TSR=3, with the performance tailing off quickly after TSR=4. The model with the cylindrical tower consistently gives higher performance than the cuboid tower. This is unsurprising since the cylinder is more hydrodynamic than the cuboid, which leads to more power being available in the flow stream at the front of the rotor. This is confirmed in Figure 5 which shows the velocity contours along the z mid-plane for both the cuboid structure and cylindrical structure. The figure clearly shows that upstream flow slows down in front of the cuboid structure more quickly than for the cylindrical structure.

Figure 5 also shows the impact that the structure has on the downstream flow. Immediately behind the cuboid structure there is a region with very slow flow both at the sea-bed level and at the top of the structure which is not present when the cylindrical structure is used. However this slow flow region is quickly dissipated and by x=180 m the size of the wake is of a similar magnitude for both structures. These results demonstrate that the design of the supporting structure can affect both the rotor performance and the surrounding environment. It is perhaps not surprising that the more hydrodynamic structure delivers the better performance and shows better wake recovery immediately behind the TST.

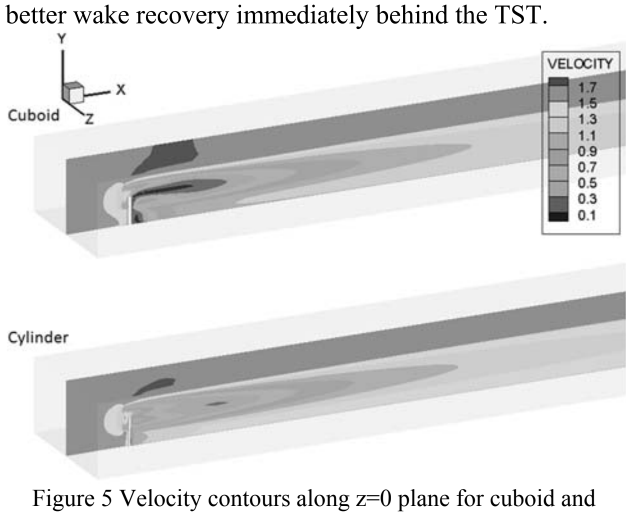

Figure 5 Velocity contours along z=0 plane for cuboid and cylindrical towers in flat sea-bed domain at TSR=3

B. Effect of bathymetry on performance

Bathymetry data of the proposed test site in the Bristol Channel was gathered by one of the project partners [12]. The data was measured for a region of approximately 1 km^2 and was used to pinpoint a suitable location for the turbine. When the turbine location was specified the CFD model was modified using the bathymetry data to define the base of the solution domain. The bathymetry, magnified by a factor of 10 in the vertical plane is shown in Figure 6. Note that the supporting structure is shown in this figure but the rotor is not.

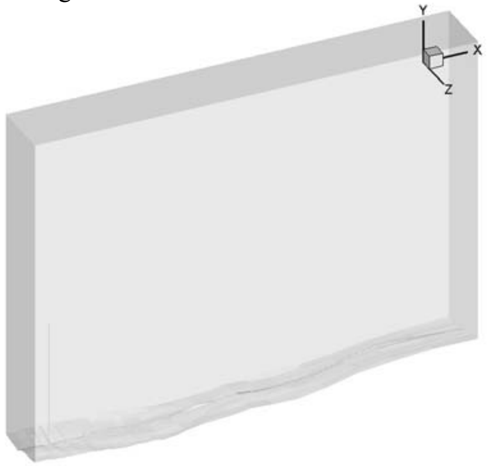

Figure 6 Bathymetry defined solution domain magnified by a factor of 10 in vertical plane.

A series of simulations were carried out on the bathymetry domain using the cuboid and cylindrical tower for $1 \leq \text{TSR} \leq 6$. As in the flat seabed model an upstream velocity of 1.6 m/s is imposed at the inlet. The graph in Figure 7 shows how Cp, Ct, and Cf, vary with increasing tip speed ratio (TSR) for the flat seabed model and the bathymetry defined model using the cuboid structure. It can be seen that both models follow the same trends and there is very little difference in performance between the two domains. This is because the cross-sectional areas of the two domains upstream of the rotor are very similar, resulting in a similar upstream velocity distribution for each model - see the velocity profiles at x=20 m, y=12 m at TSR=3 in Figure 8 for each domain.

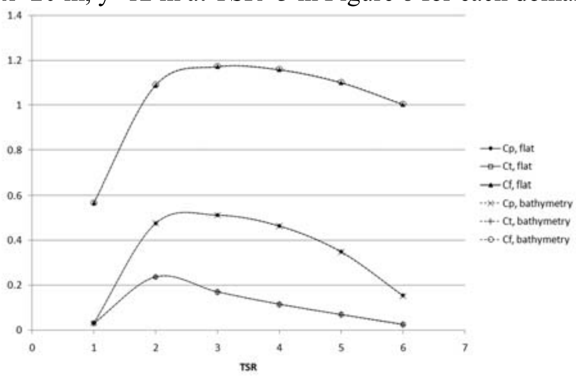

Figure 7 A comparison of Cp, Ct and Cf for the flat and bathymetry defined seabed domains.

Although the performance of the TST is very similar for both domains, the shape of the external geometry has a more significant effect on the strength of the wake downstream of the rotor. Figure 8 also shows a comparison of the behaviour of the wake along y=12 m at x=226 m and x=426 m for the flat and bathymetric domain. It can clearly be seen that at x=226 m the resultant velocity profile for both models is very similar with a maximum difference of the bathymetry results from the flat results of 0.8 %. However, for x=426 m the corresponding value is 6.2 %. This is due to the cross-sectional area of the domain at x=426 m being approximately 5 % smaller than the cross-sectional area of the flat domain. Since the flow is incompressible this obviously results in faster flow speeds in this section of the bathymetry domain in order to ensure continuity. It can also be observed that for the flat domain the minimum velocity at x=426 m, y=12 m (i.e. 40 rotor diameters downstream) has only recovered to 87.5 % of its inlet value.

These results demonstrate that if it is possible to choose the dimensions of a flat bed domain to give a reasonable approximation to the bathymetry data upstream of the proposed location of the TST the performance of the rotor can be predicted as well by the flat bed domain as it can by the bathymetry domain. This of course relies on little change in the shape of the sea-bed bathymetry upstream of the proposed site. However, the results for wake recovery also demonstrate that the bathymetry can significantly change the flow profile in the solution domain. This means that it is important to use bathymetry data if it is available, particularly if the model is to be extended to predict the behaviour of arrays of turbines.

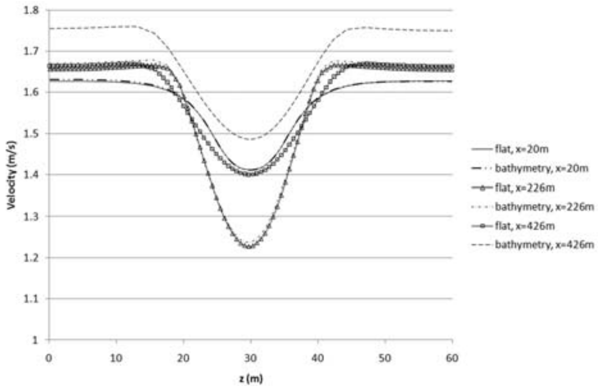

Figure 8 Resultant velocity along y=12 m for flat and bathymetric domain with cuboid structure and TSR=3 at x=226 m and x=426 m

4. Conclusions

The aim of this paper has been to demonstrate a blade element momentum method for predicting the performance of a tidal stream turbine at a particular site in the Bristol Channel. The method has been demonstrated on two computational domains, a domain with a flat base and a domain with the base defined by bathymetry data measured at the site.

Two supporting structures for the rotor have been implemented in the flat domain model. The results over a range of TSRs demonstrate that the performance of the rotor increases with the more hydrodynamic structure design. The results also show that the flow patterns in the immediate vicinity of the structure vary considerably and this could have a significant impact on environmental issues such as scouring and sedimentation.

Comparing the performance of a TST in the flat domain and bathymetry domain showed that for this particular site there was very little difference in performance characteristics on each domain. This was partially due to the height of the flat domain being chosen to be a good match to the bathymetry data at the proposed site, and also because the cross-sectional area of the bathymetry domain upstream of the TST did not show much variation from the cross-sectional area of the flat domain. However, downstream of the TST the flow did show more variation which was caused by the change in cross-sectional area of the domain.

These results demonstrate that if a single TST is being considered and the bathymetry is fairly uniform upstream of the TST then a flat domain model will give a good prediction of the performance of the TST. This can be useful as using bathymetry data can sometimes lead to complex mesh shapes, which can in turn lead to increased computation time. However if the bathymetry upstream of the TST is not fairly uniform then it is not appropriate to use a flat domain model as the bathymetry will have a more significant effect on the upstream flow profile, which in turn will affect the amount of energy available to be extracted. In addition, if the geometry varies a lot downstream of the TST the flat domain model will not be able to predict the wake behaviour as well as the bathymetry model. This is particularly important when considering fields of turbines as the wake recovery of a TST will have an impact on the performance of any TST placed behind it.

Although the BEMT CFD model does not allow the transient nature of the flow in the vicinity of the blades to be predicted it does enable the user to investigate the impact of the TST on more large scale phenomena, and some research which uses the model to investigate scouring, sedimentation and the impact on sea-life has already been carried out by the authors [13].

Acknowledgement

This work was undertaken as part of the Low Carbon Research Institute (www.lcri.org.uk). The Authors wish to acknowledge the financial support of the Welsh Assembly Government, the Higher Education Funding Council for Wales, the Welsh European Funding Office, and the European Regional Development Fund Convergence Programme

References

1 M. R. Willis et.al., “Tidal Turbine Deployment in the Bristol Channel – A Case Study”, Proceedings of the Institute of Civil Engineers Energy, accepted, in press, (2010)
2 O. L. Hansen, Aerodynamics of wind turbines, Earthscan, London, (2008).
3 J.A.C. Orme and I. Masters, “Design and Testing of a Direct Drive Tidal Stream Generator”, in Proc. Institute Marine Engineering Science and Technology Part B: J. Marine Design and Operations, Vol. 9, (2005/6) pp. 31-36.
4 H. K. Versteeg and W. Malalasekera, An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson Prentice Hall, London, (2007).
5 PHYSICA, http://www.physica.co.uk.
6 B. E. Launder and D. B. Spalding, Mathematical models of turbulence, Academic Press, London, (1972).
7 S. V. Patankar, Numerical Heat Transfer and Fluid Flow, McGraw-Hill, New York, (1980).
8 C. M. Rhie and W. L. Chow. “Numerical study of the turbulent flow past an aerofoil with trailing edge separation”. *AIAA J*, 1983, Vol. 21, pp. 1525-1532.
9 P. Chow, M. Cross and K. Pericleous, “A natural extension of standard control volume CFD procedures to polygonal unstructured meshes”, Applied Mathematical Modelling, 1995, Vol. 20, pp. 170-183.
10 T. N. Croft, K. Pericleous and M. Cross, “PHYSICA: A multiphysics environment for complex flow processes”, in Numerical Methods in Laminar and Turbulent Flows Vol. IX, (TAYLOR C. et al (eds.)). Pineridge Press: Swansea, 1995.
11 E. Hau Wind Turbines: Fundamentals, Technologies, Application, Economics. Springer-Verlag, Berlin, 2006.
12 P. S. Evans, A. Mason-Jones, C. F. Woolridge, D. M. O'Doherty, T. O'Doherty and I. Fryett. “The utilisation of ADCP and CFD to investigate the feasibility of siting a tidal stream turbine in the inner Bristol Channel”, Estuarine and Coastal Sciences Association (ECSA) Symposium of "The Severn Estuary, Cardiff, 2008, http://www.severnestuary.net/sep/pdfs/ecsa/23amason-jones.pdf (accessed 09/06/2009).
13 A. J. Williams, T. N. Croft, N. Bouwmann, M. Cross and M. Muhasilovic, “CFD Modelling of Environmental Issues Related to the Installation of Tidal Stream Turbines”, Estuarine and Coastal Sciences Association (ECSA) Symposium of "The Severn Estuary, Cardiff, 2008, http://www.severnestuary.net/sep/pdfs/ecsa/10ncroft.pdf (accessed 09/06/2009).