Characterisation of the Coastal Hydrology of Oceans Using 3D Computational Fluid Dynamics

E. Zangiabadi[1], I. Masters[2], A. J. Williams[3] and T. N. Croft[4]

[1,2,3,4] Marine Energy Research Group, College of Engineering
Swansea University, Singleton Park, Swansea, SA2 8PP, UK
[1] Email: 502830@swansea.ac.uk
[2] Email: I.Masters@swansea.ac.uk
[3] Email: Alison.j.Williams@swansea.ac.uk
[4] Email: T.N.Croft@swansea.ac.uk

Abstract

Knowing the characteristics of the current is necessary in order estimate the energy a tidal stream turbine (TST) could extract from a site. These characteristics are strongly influenced by the topography of the seabed. Often in locations of fast currents, which are ideal sites for TSTs, there is significant detail in the bathymetry. Modelling of the flow in an estuary or a channel therefore offers a challenge as a large domain is required to reduce the influence of the boundary assumptions but this leads to problems resolving the effects of the bathymetry on the flow and turbulence.

This paper describes a computational fluid dynamics model of a potential TST deployment site. The bathymetry of the deployment site was surveyed using an echosounder and the resulting data was used in the development of the geometric detail. Two different turbulence models (*k-e* and *LES*) were investigated and the simulation results were then compared with the flow data gathered by Acoustic Doppler Current Profiler (ADCP) survey. Despite some simplifications in the model, the result showed good agreement with the real site data.

Keywords: CFD, Finite Volume, Real Environment, Tidal Stream Turbine, Turbulence

1. Introduction

Devices to harness zero head hydrokinetic energy of tides, waves and currents in oceans, rivers and streams for electricity generation at the commercial level have been in development for a number of years. The significant contribution of these devices to the future supply of clean energy, both in the UK and around the world, is evident [1]. Amid the different methods of marine renewable energy, the development of tidal energy has been ahead of the other schemes, mainly because this kind of marine renewable energy is totally predictable. In their recent report, the Low Carbon Innovation Co-ordination Group (LCICG) published that by current knowledge the feasibly exploitable resource by 2050 could deliver around 20-30 TWh/year of electricity from tidal. Comparing this number with the current UK electricity consumption which is around 360 TWh/year the figures shows it could meet about 5% of expected 2050 total UK electricity needs [2]. Despite their bright future, marine energy technologies are not yet commercial. Only 4 tidal technologies have been deployed at full scale demonstration, with an additional 2 devices expected to be installed over the year 2013 [2].The progress rate is slow, mainly because of environmental specification of the suitable locations. Roughness and irregularity in topography and three-dimensional (3D) turbulent flows characterise marine and estuarine environments. Both of these characteristics can significantly affect the performance of Tidal Stream Turbines (TST) and should be investigated carefully [1].

In general, when a fluid flows over a solid surface, the velocity of fluid at the surface is zero. Close to the surface the velocity increases rapidly and forms the boundary layer. Near the surface the shear stress is more significant and in the turbulent flows considered here, beyond the boundary layer the effect of viscosity is negligible [3]. The boundary layer, along with the main stream flow, determines the velocity profile of the flow above the surface. The roughness of the surface is one of the important parameters which affect the shape of the boundary layer and the velocity gradient is recreated by variation of the bed friction coefficient. Rocks, pinnacles and severe changes in bathymetry have the influence to change the shape of the velocity profile and consequently affect the overall performance of a TST. Because the installation site for the TST farms usually covers a vast area, different channels with length of more than 3 km were considered and the equations were solved for various parameters to produce the shape of the velocity gradient above the seabed with different boundary conditions in the early stages of the research in order to use the results as input conditions for modelling the tidal stream turbine [4].

Next, the bathymetry of a possible TST installation site was surveyed and added to the model together with the simulated effect of material and vegetation of the seabed in order to generate a more realistic environment. The survey to collect the real site data was done during Operation Celtic Odyssey by a group of Welsh universities for environmental assessment of St David's peninsula and the adjacent coastline around Ramsey Sound. The area has already been identified by the Welsh Government as a potential site for tidal energy generation**.**

The results were validated by comparing the results with flow data gathered by transect Acoustic Doppler Current Profiler (ADCP) survey in Operation Celtic Odyssey I in the spring of 2011 [5].

2. Theory and Method

2.1 k - ε Turbulence Model

The k-ε model is the most commonly used two-equation turbulence model. It is said that a greater number of equations lead to more realistic models but it is very hard to prove that in practice [6]. The governing equations for the transport of any fluid are the continuity equation:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho \underline{u}) = S_m$$

And the momentum equations:

$$\frac{\partial(\rho \underline{u})}{\partial t} + \nabla.(\rho \underline{u}\, \underline{u}) = -\nabla p + \nabla.[\mu \nabla(\underline{u})] + \underline{S_u}$$

In the above-mentioned equations $\underline{u}$ is the velocity vector, p is the pressure, ρ is the fluid density, μ is the dynamic effective, laminar and turbulent, viscosity and the terms S_m and $\underline{S_u}$ are additional source terms into the respective equations [7]. The effects of turbulence are resolved through the k-ε model by using the following equations to calculate the turbulent kinetic energy, k, and the turbulent dissipation rate, ε, respectively:

$$\frac{\partial(\rho k)}{\partial t} + \nabla.(\rho \underline{u} k) = \nabla.\left[\left(\mu_l + \frac{\mu_t}{\sigma_k}\right) \nabla(k)\right] + \mu_t G - \rho\varepsilon$$

$$\frac{\partial(\rho \varepsilon)}{\partial t} + \nabla.(\rho \underline{u} \varepsilon) = \nabla.\left[\left(\mu_l + \frac{\mu_t}{\sigma_\epsilon}\right) \nabla(\varepsilon)\right] + C_{1\varepsilon}\mu_t G \frac{\varepsilon}{k} \ldots$$

$$\ldots - C_{2\varepsilon}\, \rho \frac{\varepsilon^2}{k}$$

The values $C_{1\varepsilon}$,$C_{2\varepsilon}$, σ_k and σ_ε are constants and the values are respectively 1.44, 1.92, 1.0 and 1.3[8]. G is the turbulent generation rate, μ_l is the laminar dynamic viscosity and μ_t is the dynamic turbulent viscosity which is calculated using [8]

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

The coupling between the turbulence and the momentum equations is through the dynamic viscosity which is a sum of the laminar and turbulent values. In all of the equations, as we deal with incompressible flow, the derivative of density over time is zero and there are no external source terms [7].

2.2 Large Eddy Simulation

Nowadays Large Eddy simulation (*LES*) is becoming more popular for solving complicated turbulent flow and in near future will be a real alternative to RANS [9]. Using *LES* allows one to solve explicitly the large eddies by using a filtered approach of Navier-Stokes equation in which the filter is the grid size and implicitly account for the small eddies by using a subgrid-scale model (SGS model) [10].

The filtered continuity and momentum equations use filtered variables

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho \tilde{u}_j}{\partial x_j} = 0$$

and

$$\frac{\partial \rho \tilde{u}_i}{\partial t} + \frac{\partial \tilde{u}_i \tilde{u}_j}{\partial x_j} = -\frac{\partial \tilde{p}}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \frac{\partial \sigma_{ij}}{\partial x_j}$$

In which

τ_{ij} is the filtered stress tensor.

σ_{ij} is the subgrid-scale Reynolds stresses [11].

Physically, the dynamic coupling between large and small scales in turbulence is described by the SGS stress. Dimensionally, it scales quadratically with turbulent velocity differences at scales of order Δ [12]. To solve the SGS model the most known and simple method is the Smagorinsky model.

$$\mu_t = (C_s \Delta)^2 \sqrt{2 S_{ij} S_{ij}}$$

Where

μ_t is the turbulent viscosity

Δ is the length scale of the filter

S_{ij} is the filtered strain-rate tensor.

C_s is the Smagorinsky constant

The length scale of the filter usually considered to be cube root of the volume of the cell [10].

One of the most important issues of the *LES* method is setting the boundary condition for the inlet. Among a few methods for generating inlet velocity fluctuation, a most recent one is called the Vortex Method. This method, unlike the random noises has some spatial correlations to make the fluctuations. This approach is based on creating random vortices on the inlet flow plane (normal to the streamwise velocity) for the wall-normal components which gives a spatial correlation, and on the generation of the streamwise component using a Lagevin equation, which provides a temporal correlation, as well as with the two cross-stream components [10]. The Vortex Method is based on a Lagrangian approach of Navier-Stokes equation. The centres of the vortices are transported and the velocity is given a certain distribution [10].

Vorticity is calculated by the following equation:

$$\omega_i(\underline{x}) = \sum_{i=1}^{n} \Gamma_i \xi_{\sigma_i}(\underline{x} - \underline{x_i})$$

in which

$\underline{x_i}$ is the centre of the vortex

σ_i is the vortex diameter

Γ_i is the circulation

$\xi_{\sigma_i}$is the shape function

n is the number of vortices

v and *w* which are the wall normal and the spanwise velocity components can be obtained by using a Biot and Savart Kernel

$$\underline{V_x}(v, w) = -\frac{1}{2\pi}\sum_{i-1}^{n} \Gamma_i \underline{K}_{\sigma_i}(\underline{x})$$

with

$$\underline{K}_{\sigma_i}(\underline{x}) = \frac{-1}{2\pi}\iint \frac{\underline{x}-\underline{y}}{\left\|\underline{x}-\underline{y}\right\|^2}\xi_{\sigma_i}\left(\underline{y}\right) dy$$

$$= \left(1 - e^{\frac{\|x\|^2}{2\sigma_i^2}}\right) e^{\frac{\|x\|^2}{2\sigma_i^2}} \frac{\underline{x}^{\perp}}{\left\|\underline{x}\right\|^2}$$

With the shape function of

$$\xi_{\sigma_i}(\underline{x}) = \frac{1}{2\pi}\frac{1}{\sigma_i^2} e^{\frac{\|x\|^2}{2\sigma_i^2}}\left(2e^{\frac{\|x\|^2}{2\sigma_i^2}} - 1\right)$$

The most important parameter is the circulation Γ_i, because the intensity of the fluctuation is related to this parameter. It is possible to define this parameter as a function of the turbulent kinetic energy which can be calculated from a RANS calculation.

$$\Gamma_i \approx 4\sqrt{\frac{\pi S k(\underline{x_i})}{3n[2\ln(3) - 3\ln(2)]}}$$

Where

S is the surface of the inlet

k is the turbulent kinetic energy

At the first time step, the position of the vortex is set randomly in the 2D domain. Then the vortices are given in a random rotation sense. For each vortex a characteristic time (which is also known as the turbulent time scale$= k/\varepsilon$) is defined. This characteristic time is used for the life time of a vortex and if the time exceeds this characteristic time, then it is destroyed and another random vortex is created on the 2D plane in the inlet. The diameter of the vortices σ_i can be set to a fixed number or can be calculated by using a turbulent length scale approach ($l = {k^{3/2}}/{\varepsilon}$). Then the vortices carry a random walk in the inlet plane to develop fluctuation in time [10].

3. Real Environment and Bathymetry

The ultimate purpose of this research is to create a model which covers all aspects of a natural turbulent flow and applying this to test cases around the coast of Wales [13] One of the potential locations to install TST devices is Pembrokeshire coast which will be the site of Wales' first tidal stream turbine, named DeltaStream. The company has recently been granted consent to install a 1.2MW pre-commercial prototype device for 12 months. Scientists spent time collecting data on tidal flows, seabed bathymetry, underwater noise, marine mammal movements, fish behavior and public perception of the marine environment during a two week survey called "Operation Celtic Odyssey" which brought together engineers, marine biologists, computer modelers, oceanographers, marine mammal experts and water quality scientists on board research boats from Cardiff and Swansea Universities [13].

The team used an echosounder to measure the seabed profile. The device was mounted on one of the research vessels and hundreds of transects were done around the area. The resolution of data points varied between 0.5 to 5 meters.

Because modeling of the whole area was very time consuming, at this stage it was decided to select an area in vicinity of the installation site which is located between the Ramsey Island and the mainland.

The data set was refined and the first version of the bathymetry was created by using more than fifteen thousands of data points provided by Olex Charts System.

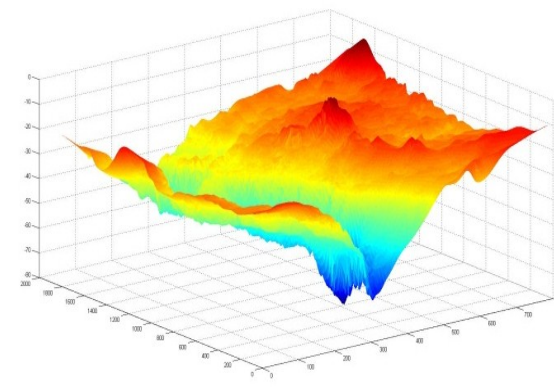

Figure 1: Bathymetry created with the original data points (The elevation was exaggerated to show the features clearly).

This bathymetry was too complicated to be meshed as the seabed was too spiky and the quality of data was not assured in the deep valley. Eventually, it was decided to use two other datasets and recreate the bathymetry. One of the datasets was provided by SeaZone and had the resolution of 30 meters and the other one was a historic dataset made available by the Royal Navy. The final bathymetry have the resolution of 12 meters and an attempt was made to have as much detail as possible, particularly the main features of that area. One of the visible changes was the height and shape of Horse Rock. This feature was missing from the Olex data because of the risk of collision damage to vessels from surveying too close to these surface piercing features.

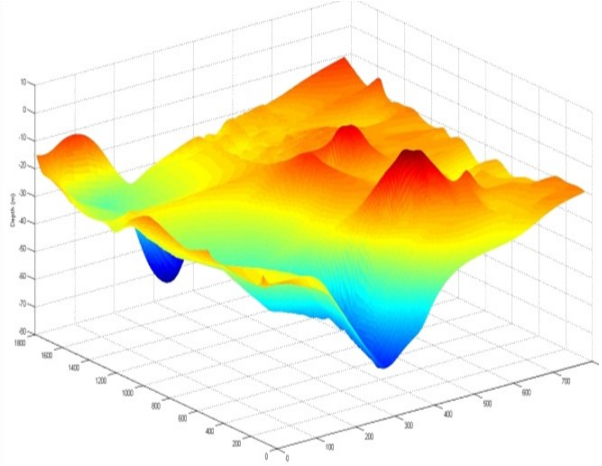

Figure 2: Bathymetry created with the combination of the three data sets (The elevation was exaggerated to show the features clearly).

4. Result

As mentioned above, a potential deployment site in Ramsey Sound has been chosen to run the simulation. The selected bathymetry covers an area of about 1800m length and 800m width. The water elevation for the selected area was varying between +5 and -74 meters considering that the zero altitude has been set to be the highest peak on the seabed. A tetrahedral mesh with around 3.7 million elements was generated and size of the smallest element in the streamwise direction was nearly 1 meter.

The same geometry was used to run the simulation for the both k-ε and *LES* problems. For running the simulation the commercial code ANSYS FLUENT®13.00 was used.

The inlet velocity for the k-ε case was set to be 1.5 m/s and the turbulent intensity of 20 % along with the turbulent viscosity ratio of 100 were chosen. For the bed boundary condition a no slip wall with roughness height of 1.29 meters with roughness constant of 0.75 was selected.

The SIMPLE Algorithm was selected to couple pressure-velocity equations. Spatial discretisation of momentum, turbulent kinetic energy and turbulent dissipation rate was calculated using a Second Order Upwind method. For calculation of the gradients the Least Squares Cell Based method was used. The case was run for 1000 iterations in steady state condition. The running time was nearly 6 hours.

The velocity contour results are shown in figure 3, along with a graph for velocity across the channel in various locations at the depth of -10 is shown in figure 4. Note the influence of the surface piercing rock on the flow patterns.

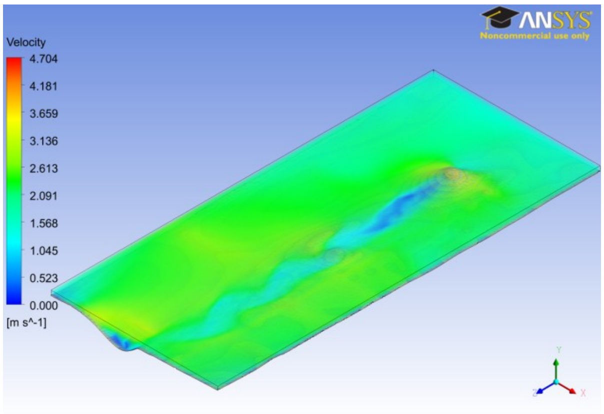

Figure 3: Volume rendering of the velocity for the area (k-ε Method)

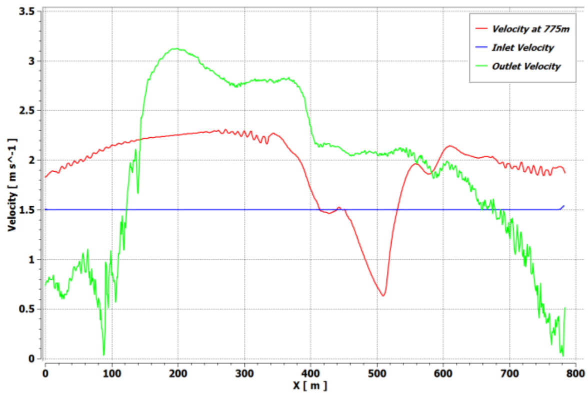

Figure 4: Velocity across the channel at various locations from the inlet (k-ε Method)

To run the transient condition, *LES* with Smagorinsky-Lilly method was selected to model the subgrid scale. Again the same inlet velocity has been chosen for the flow. By using the Vortex Method to create random fluctuation at the inlet, in two different cases, 800 and 1000 vortices were applied at the inlet. The maximum number of vortices is restricted to 1000 in ANSYS FLUENT®13.00. Turbulent intensity of 20 % along with the turbulent length scale of 1 meter was chosen. The bed boundary condition set to be a stationary wall.

Like the k-ε case, the SIMPLE Algorithm was selected to couple the pressure-velocity equations. The momentum was calculated by using bounded central differencing method for spatial discretisation. For calculation of the gradients the least squares cell based method was used. 550 time steps, with constant step size of 6 second which was selected to achieve a multiple of 10 of the time taken for the flow to pass the smallest element in streamwise direction and the maximum number of iteration for each time step was set to be 35. The total running time was about 162 hours.

The computational result is shown in the following figures:

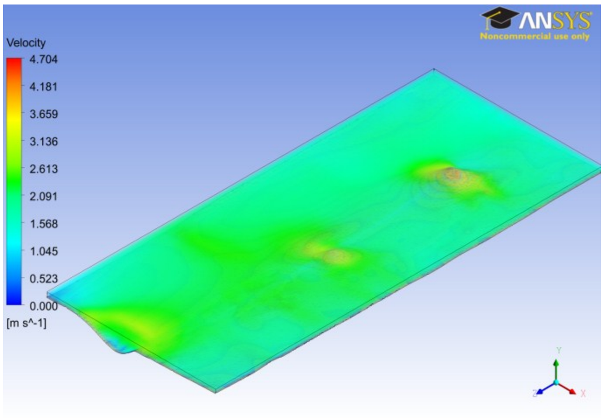

Figure 5: Volume rendering of the velocity for the area (*LES* Method)

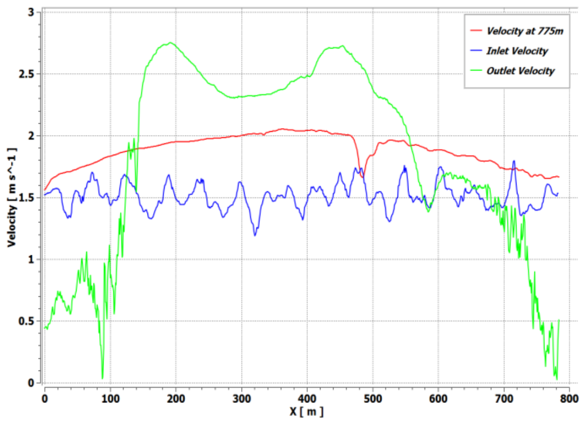

Figure 6: Velocity across the channel at various locations from the inlet (*LES* Method)

The final result was almost the same for the both conditions with the different number of vortices at the inlet. It has been proven before that the generated vortices at inlet decay rapidly and do not have significant effect on the downstream flow if the outlet is far enough from the inlet [10] and in this case as the outlet is located about 2000 meters from the inlet, it has led to the same conclusion.

The measured flow data was acquired from the site, by using an ADCP device, by Swansea and Cardiff universities research boats and processed using WinADCP software.

The results of the both simulations were compared with the real data:

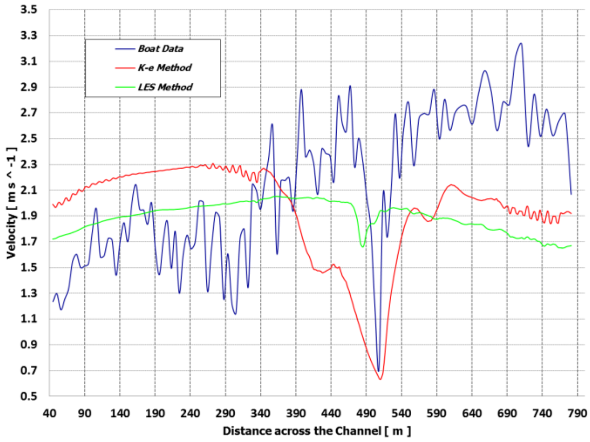

Figure 7: Comparison of the velocity profile across the channel at 775m from the inlet (k-ε, *LES, Boat data*)

The comparison shows that both of the models picked up some features of the real environment and although there is a difference between the amount and fluctuations of the velocity, there is a relatively good adaptation in the pattern of the flow. In all of the cases the velocity generally increase as we travel toward the centre and the sudden drop which is caused by the two big rocks (Horse Rock and Pony Rock) is also picked by the models.

The reason that the velocities calculated by the simulations have less fluctuation could be the difference between the generated bathymetry and the real one. As the real bathymetry was very spiky at some regions and it was nearly impossible to create fine mesh over the seabed, those locations converted to a relatively smooth surfaces. The effect of these kind of changes can be clearly seen in the locations between 100m to 300m across the channel. On the other hand there might be a number of rocks and boulders with dimension of less than 12 meters that did not considered in the model which can also cause lots of disruption and turbulence generation in the real case.

These model have been simulated a flood situation in the channel and in the real world it means that the flow direction is from south to the north in which there are lots of pinnacles upstream of the inlet and the effect of them is not considered because of lack of data.

The simulation showed that although using traditional k-ε and *LES* models can give reasonable results but to have a more precise model that can cover all the aspects of coastal flows it is needed to customise the bed roughness parameters in a way that can compensate the effects of simplification of the bathymetry.

5. Conclusion

In this paper two different turbulence models have been investigated for a possible site of deployment of TST and the results of them were compared with the real flow data acquired from the location.

It has been understood that the *k-ε* method can predict the flow pattern with good accuracy and although the *LES* model has the ability to demonstrate some features of the bathymetry, it still needs more investigation before considering it for a possible robust method for solving these kinds of problems. *LES* can provide more precise results in this current problem if there had been more knowledge about the inlet boundary conditions. In proposing to model an ocean using CFD approaches the target area is very extensive, and consequently the vortices created by the vortex method at the inlet are not propagating throughout the length of the domain, as occurs in real life. Therefore, future work will attempt to define a suitable bed boundary condition that can generate and maintain complex turbulent structures in the flow.

Acknowledgements

This work was undertaken as part of the Low Carbon Research Institute Marine Consortium (www.lcrimarine.org). 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. The authors also acknowledge with grateful thanks the collaboration and assistant of LCRI Marine partners, RNLI St David's, St Justinian's Boat Owners Association and Swansea and Cardiff universities research boats for their enormous effort to collect the data.

References

[1] S. Kang, I. Borazjani, J. A. Colby, F. Sotiropoulos. (2012): Numerical simulation of 3D flow past a real-life marine hydrokinetic turbine. Advances in Water Resources (2012) Vol 39: P 33-43

[2] Technology Innovation Needs Assessment (TINA), Marine Energy Summary Report (2012): Available: http://www.decc.gov.uk/assets/decc/11/funding-support/fund-opportunities/6170-technology-innovation-needs-assessment-tina-mar.pdf (Accessed at August 2012)

[3] B. Massey (1998). Boundary layer and wakes. In: John Ward-Smith Mechanics of Fluid. 7th ed. UK: Nelson Thornes Ltd. 312.

[4] E. Zangiabadi, I. Masters and A.J. Williams (2012): Characterisation of the Coastal Hydrology of Oceans Using 3D Computational Fluid Dynamics. Proceedings of the 20th UK National Conference of the Association for Computational Mechanics in Engineering (ACME2012), Manchester, UK

[5] P. Evans, C. Wooldridge, A. Mason-Jones, T. O'Doherty D. O'Doherty, S. Neill, R. Hashemi, R. Malki, I. Masters and E. Zangiabadi. (2012): "From Vessel to Model": Bridging the Gap between Real and Modelled Flow Data. Oxford Tidal Energy Workshop, March 2012, Oxford, UK

[6] C. D. Scott-Pomerantz (2004): The K-Epsilon Model in the Theory of Turbulence. Ph.D. Thesis, University of Pittsburgh, USA

[7] A. J. Williams, T. N. Croft, I. Masters, M. R. Willis and M. Cross (2010): Combined BEM-CFD Modelling of Tidal Stream Turbines Using Site Data. Proceedings of International Conference on Renewable Energies and Power Quality (ICREPQ'10), Granada (Spain)

[8] W. P. Jones and B. E. Launder (1972): The Prediction of Laminarization with a Two-Equation Model of Turbulence, International Journal of Heat and Mass Transfer, Vol. 15, p. 301-314.

[9] P. G. Tucker and S Lardeau (2009): Applied Large Eddy Simulation. Phil. Trans. R. Soc. A 2009 367, 2809-2812

[10] S. Benhamadouche, N. Jarrin, Y. Addad and D Laurence (2006): Synthetic turbulent inflow conditions based on a vortex method for large-eddy simulation. Progress in Computational Fluid Dynamics Vol.6, No. 1-3, p 50-57

[11] A. Bakker: Introduction to large eddy simulation (LES). Available: www.bakker.org/dartmouth06/engs150/12-les-l.ppt (Accessed Aug 2012)

[12] C. Meneveau (2010): Turbulence: Subgrid-Scale Modelling. Scholarpedia, 5(1):9489. Available: http://www.scholarpedia.org/article/Turbulence:_Subgrid-Scale_Modeling (Accessed Aug 2012)

[13] M. Willis, I. Masters, S. Thomas, R. Gallie, J. Loman, A. Cook, R. Ahmadian , R. Falconer, B. Lin, G. Gao, M. Cross, N. Croft, A. Williams, M. Muhasilovic, I. Horsfall, R. Fidler, C. Wooldridge, I. Fryett, P. Evans, T. O'Doherty, D. O'Doherty, and A. Mason-Jones. (2010): Tidal Turbine Deployment in the Bristol Channel – A Case Study. Proceedings of the Institute of Civil Engineers - Energy, Volume 163, Issue 3, pages 93 –105