Near wake properties of horizontal axis marine current turbines

L. Myers and A.S. Bahaj

Sustainable Energy Research Group,
School of Civil Engineering and the Environment,
University of Southampton,
United Kingdom.
www.energy.soton.ac.uk
E-mail: L.E.Myers@soton.ac.uk

Abstract

The flow field in the near wake region of a tidal current turbine is strongly driven by the combined wake of the device support structure and the rotor. Accurate characterisation of this region of flow is important but it is characterised by highly turbulent, slow-moving fluid.

Wake flow field characteristics of a 1:20th-scale horizontal axis turbine has been have been measured in a large water channel facility. A downstream map of both the channel base flow and downstream wake was made using both laser and acoustic doppler velocimeters. Wake mapping was conducted with the rotor in both operational an inactive states to quantify the effect of the turbine support structure upon the near wake flow properties.

Results indicate that the wake created by the turbine support structure has a significant effect upon the near wake flow field with strong synergistic effects of both rotor and supporting structure close to the downstream centre plane of the rotor which diminish with increasing lateral (cross channel) distance. Determination of higher order flow properties was difficult close behind the turbine (<4 rotor diameters) due to extremely high levels of turbulence. Furthermore, it is clear that the form of the rotor support structure has a strong influence upon near wake properties and this will change for the various forms of tidal energy devices either deployed or at the design stage.

Keywords: ADV, Flow, Turbine, Wake

Nomenclature

a	=	Axial induction factor
A	=	Area
C_t	=	Rotor thrust coefficient
I	=	Turbulence intensity
N	=	Newton force
U	=	Longitudinal velocity
W	=	Watts (power)
ρ	=	Fluid density

Subscripts

d	=	disk (rotor)
w	=	downstream (wake) region
0	=	upstream condition

Superscripts

–	=	mean value

1 Introduction

Free stream tidal energy conversion technology is presently at the commercial demonstrator stage where single devices of appreciable scale are operating at various tidal stream sites around the world. Evidence of large amounts of available power at certain sites has long been known [1] and thus the next stage for the technology will be the installation of farms or arrays of devices. The layout of such arrays may well be site dependant as a number of variables will determine inter-device spacing. Some sites with strong flow characteristics do not have the classic bi-directional flow characteristics usually associated with tidal streams. In such cases available surface area for energy extraction may be limited and wake meandering from individual devices as well as that from the entire array may be expected [2]. At present there are a wide range of device designs that may have differing access requirements and this could be a driver in array layout. A previous publication by the authors has attempted to ascertain energy yields and layout of a large-scale array in UK waters [3] where rows of devices were spaced at 15 rotor diameters, equating to 240m for 16m-diameter turbines. Such a distance could be regarded as a minimum for the manoeuvring of an installation/maintenance vessel hence could be the driver towards larger row spacing. Indeed, previous publications by the authors as part of the current research area would suggest that 15-20 rotor diameters would be a minimum for reasonable

© Proceedings of the 8th European Wave and Tidal Energy Conference, Uppsala, Sweden, 2009

wake velocity recovery [4, 5] of between 85% and 92%. It was found that the close proximity of the sea bed and water surface reduces the mixing of free stream and wake fluid coupled with different vertical shear and turbulence distributions.

Offshore wind farms are perhaps the most closely related field of research in terms of inter device spacing although wind turbines are often spaced at much closer intervals; typically 7 diameters perpendicular to the prevailing wind direction and approximately 10-diameters downwind. There has been evidence to suggest that farm losses due to turbine operating in wake flow are significant at such close spacing [6, 7].

This paper addresses a subject more specific to tidal energy generation; that of the near wake structure and the influence of the rotor support structure upon this region of downstream flow. The present state of tidal energy technology entertains a variety of methods and forms for fixing the hydrodynamic subsystem in the water. Foundation types include mono and multi-leg piles, gravity base and even semi-submersible/neutrally buoyant concepts. The shape of the support structure immediately downstream of the rotor is also of importance. Present concepts and demonstrators show little regard for minimising flow disturbance generated by the support structure and hence large bluff objects will create a wake that will be entrained within that created by the rotor when in operation. This will lead to stronger velocity deficits and turbulence intensities immediately downstream of the turbine. The near wake region is important for numerical models as many solve the flow conditions in a downstream stepping process. Additional turbulence and a wake shape that cannot be related to the circular horizontal axis rotor will lead to difficulties for simulation packages and could potentially lead to errors in predicting far wake flow, the region in which downstream devices will be positioned within an array.

2 Wake properties

The wake of a horizontal axis turbine is commonly separated into two distinct regions; the near and far wake. The near wake lies immediately downstream of the rotor and is characterised by slow moving fluid (having just passed through the rotor) with strong pressure/shear gradients and intense turbulence structure. Horizontal shear stresses are greatest in the annular region close to the blade tip region. This is principally caused by the motion of the rotating blades and the near wake region is often characterised by a large amount of 'swirl' which is dissipated within 4-5 rotor diameters downstream of the rotor [8, 9]. High vertical shear stresses exist towards the top of the rotor region as ambient and rotor generated shear are added. At the bottom of rotation these are subtracted giving a slightly lower value. In general turbulence and shear diminish with distance away from the rotor centreline. The near wake velocity deficit is strongly influenced by the momentum drop across the rotor disk as defined by the following expression derived from axial momentum of actuator disk theory:

$$U_w = (1-2a)U_0 \qquad (1)$$

A coefficient of rotor thrust can be defined and related to the axial induction factor a:

$$C_t = \frac{thrust}{0.5\rho U_0^2 A_d} = 4a(1-a) \qquad (2)$$

Thus for increasing rotor thrust the initial wake velocity deficit should increase until a = 0.3 and C_t = 0.9. C_t values in excess of 0.9 are possible at which point actuator disk theory breaks down and is substituted by an empirical relationship.

The far wake is a region where the rotor wake is principally dissipated by turbulent mixing between the wake and ambient flow that passed around the rotor disk. Rotor swirl has generally dissipated by this distance downstream and the rotor wake gradually expands and the outer edges are entrained within the free stream flow. Previous experimental work by the authors [5] has shown that the wake of a tidal turbine will not necessarily be axis-symmetric in the lateral plane due to the bounding effects of the sea bed and water surface. The transition from the near to far wake region has not been definitively defined in literature but it is suggested that the point where the shear layer reaches the wake centre height defines the transition between the near and far wake [10].

Whilst the far wake region is undoubtedly of great importance to inter-device spacing within an array there is still a requirement to understand the flow development within the near wake region. Numerical models employing Blade Element Momentum (BEM) theory and Navier-Stokes equations solve in a down stream direction from the near wake region. Strong rotor/support structure flow interactions will complicate the near wake flow field and could lead to reduced accuracy when modelling the important far wake region. Merging of the rotor and support structure wakes also raises the issue of separating the flow effects from each to better understand the flow downstream of a tidal turbine. To this end the wake shed by a model horizontal axis turbine has been measured with the rotor inoperative to assess the nature of the flow field. With the rotor in operation further flow mapping has been conducted to map the near wake region and to investigate the effects further downstream into the far wake.

3 Experimental setup

Experimental work involved the design, construction and testing of a scale horizontal axis tidal turbine. Details of the device, test facilities and method of flow measurement are described herein.

3.1 Design and test requirements of turbine

Previous experimental studies have demonstrated that there is an acceptable minimum scale for horizontal axis turbines in order to maintain appropriate levels of thrust, power and to ensure representative water surface profiles [11, 12]. Thus the diameter of the rotor was set to be approximately 800mm. In addition to this the turbine was designed to meet the following requirements:

i. to allow installation and operation at a centreline depth of 1m or greater
ii. an instrumented test rig to allow accurate measurements of rotor torque and thrust upstream of any bearing or seal losses
iii. a split rotor hub to allow easy changes of rotor blades and a mechanism for accurate blade pitching
iv. the upright support to be at least one rotor radius downstream of the rotor in order to minimise the effect upon performance
v. the horizontal main body (nacelle) to have a diameter as small as possible

There were a number of available forms that the turbine could have taken in terms of power take off and the relative position of the main components. Almost immediately it was evident that a right-angle drive train where power was transmitted above the water line was not practical due to the large distance from the rotor to the water surface. Long transmission runs would at best be awkward to adjust and both belt and solid shaft transmission was deemed to be unacceptably problematic.

An axial transmission similar to those used for modern horizontal axis wind turbines would allow a cleaner and more reliable design fulfilling requirement (i). There were several options for power dissipation. A simple mechanical (friction) brake had to be discounted due to the long operational duty of the turbine required for flow mapping to be conducted. Brake wear would be significant and longer testing periods would require brake replacement that would impact negatively upon testing time. Heat build-up within the nacelle was also a concern. Eddy-current brakes were a promising solution but at lower rotational speeds rotor torque was excessive for the small diameter brakes required for a small bore nacelle design (requirement (v)). Series installation of disks would require several meters of additional nacelle length and so this option also had to be discounted. The final solution was perhaps the most conventional using a mechanical gearbox and generator.

The experimental rig consisted of a three bladed rotor attached to a main shaft driving an axial transmission fully contained within a tubular nacelle. A gearbox increased the rotational speed (reducing torque) for the generator which was located at the rear of the nacelle. The turbine could accommodate a rotor power of up to 400W depending upon rotational speed and an axial thrust of up to 1000N. A fully contained axial transmission meant that deep immersion was possible via increasing the length of the dummy upright tower support. Stainless steel construction minimised nacelle diameter whilst increasing life of threaded components. An image of the turbine installed in the water channel is shown in Fig 1.

A rotor diameter of 800mm was chosen as a compromise between maximising Reynolds number and not incurring excessive tunnel blockage correction. Hub diameter of the rotor was 100mm with the main nacelle body measuring 114mm in diameter. The blades were composed of NACA 48XX aerofoil sections for good lift/drag performance at lower Reynolds numbers and reduced stall characteristics.

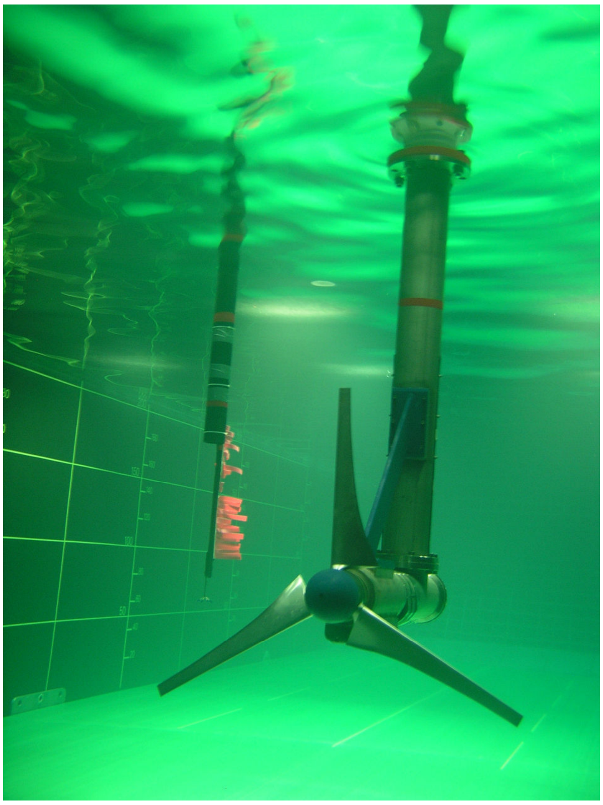

Figure 1: 0.8m diameter turbine installed in the circulating channel.

An in-line strain gauge dynamometer mounted immediately behind the rotor hub, and running wet, was used to measure the rotor thrust. There was sufficient torsion in the aluminium shaft to measure rotor torque by directly gauging the hollow shaft. The dynamometer had the capability of measuring 1000N of thrust although this figure would not be approached due to hydraulic effects of the wake flow downstream of the rotor being disturbed at high flow speeds. The dynamometer was CNC machined from solid Aluminium bronze. The rotor hub was in contact with the outer rim of the dynamometer meaning that thrust force was transmitted to the drive shaft via 4 flexures. Strain gauges were laminated to these flexures in a bridge circuit arrangement to increase output response and reduce temperature effects. A wireless data telemetry system was employed to convey signals from the turbine above the waterline. The system included means to smooth, amplify and digitize outputs directly from the bridges to ensure maximum data quality and accuracy.

The power take-off subsystem consisted of a planetary gearbox and permanent magnet generator. Electrical power generated by the turbine was dissipated above the waterline via an umbilical cord that also conveyed rotor data and power for the telemetry system.

3.2 Circulating water channel

Full experimental tests were conducted in the IFREMER circulating water channel at Boulogne-sur Mer in France. The channel has the following characteristics:

- Length 18m
- Breadth 4m
- Depth 2m
- Maximum flow speed $2ms^{-1}$

The turbine was fixed at a position towards the upstream end of the working section. The location within the water channel is shown in Fig 2.

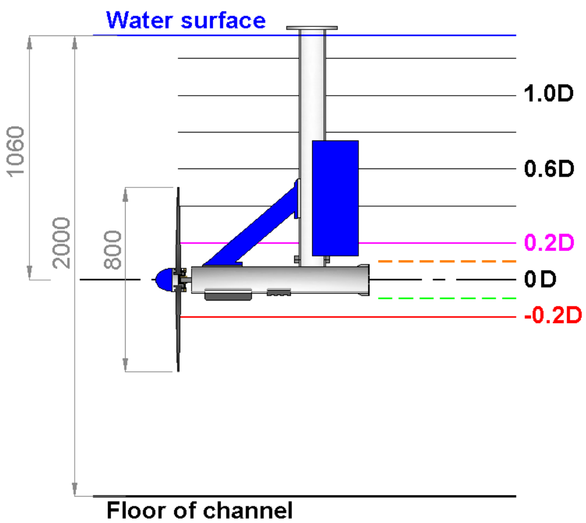

Figure 2: Diagram of side elevation of turbine showing relevant dimensions.

Depths are shown in millimetres on the left whilst to the right distances are expressed in terms of rotor diameters (D). A streamlined tapered section with an aspect ratio of approximately 3 was placed on the vertical support tower (shown in blue). The aim was to reduce the turbulence from the cylindrical tower section into the near wake region generated by the rotor.

With a channel depth equating to 2.5D and width of 5D the blockage ratio of the rotor in the channel was an acceptable 6.3%.

3.3 Flow measurement

In order to visualise the flow field around the mesh disk rotor simulators, a large number of point measurements were taken within the channel. A Nortek Acoustic Doppler Velocimeter (ADV) device was used for high frequency velocity sampling in 3 planes. The uptake of such devices has been swift due to affordability and ease of use. The functionality and general accuracy of ADV devices has been addressed elsewhere [13-16]. The Vectrino ADV used for this work incorporated advanced firmware and was set to sample at 50Hz with a sample volume of $0.15cm^3$. Sample periods were a minimum of 3 minutes giving 9000 discrete values of velocity in 3 planes. A long sample time was chosen to ensure mean conditions were accurately quantified at each location and to ensure a good amount of data remained following filtering of spikes, aliasing errors and other erroneous data. Measurements were taken downstream in the wake region of the turbine in a dense 3-dimensional grid to ensure good resolution at locations of specific interest.

Upstream inflow or base flow measurements were acquired using both the ADV and a Laser Doppler Velocimeter (LDV). This was for the function of providing a non-dimensionalised downstream flow map to allow cross comparison for differing rotor operational parameters.

4 Data reduction and presentation

Horizontal axis rotor wake recovery can be defined in terms of velocity deficit, which is a non-dimensional number relative to the free-stream flow speed at hub height (U_0) and the wake velocity (U_W):

$$U_{deficit} = 1 - {U_w}/{U_0} \tag{3}$$

Turbulence intensity is commonly defined:

$$I \cong \frac{\sigma}{U} \tag{4}$$

Where σ is the root-mean-square of the turbulent velocity fluctuations and U is the mean velocity (3-deminsional average). Fig 3 shows the principal data sets presented in this paper. 2-dimensional contour plots are presented in downstream longitudinal (left) and cross channel lateral planes (centre). The rotor centreline refers to a downstream line perpendicular to the rotor plane (right of Fig 3).

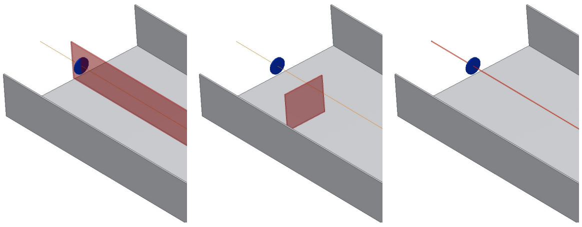

Figure 3: Regions of flow mapped in results section of this report. Centre plane vertical slice (left), lateral slice (centre) and centre line (right).

5 Results and discussion

The flow speed in the channel was set to approximately $0.8ms^{-1}$ for wake flow experiments. The chosen speed was a compromise between providing sufficient energy flux for good rotor operation and minimising changes in water surface elevation usually associated with high Froude numbers.

5.1 Support structure wake

The tower /support structure wake was mapped in both lateral (cross channel) and longitudinal (downstream) directions with the rotor in a non-operational state to ascertain the extent of the wake created by the supporting structure. A total of 156 single point measurements were acquired in order to characterise the extent and intensity of the structure wake. Fig 4 shows a contour plot of the downstream centre plane velocity which shows a number of interesting features. The lower of the two triangular flow structures, centred at 0D vertical offset, is shed from the rear of the nacelle.

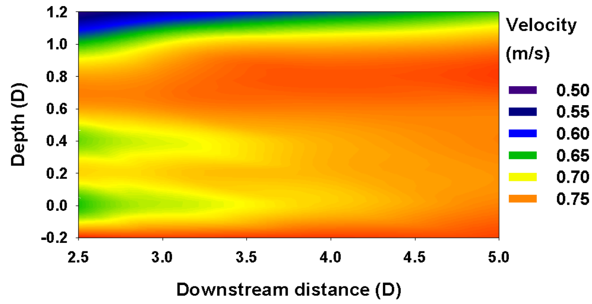

Figure 4: Centre plane velocity plot of the turbine support structure wake.

The upper structure is at the same depth as the cross reinforcement strut mounting on the vertical tower section which is centred at +0.4D vertical offset (see Figs 1 & 2). Despite the presence of the short streamlined section immediate downstream a wake is still formed that persists to almost 4D downstream of the rotor. Closer to the water surface the circular tower section creates a wake that persists further downstream with increased intensity. Fig 5 shows a similar trend occurring for the downstream u-turbulence intensity.

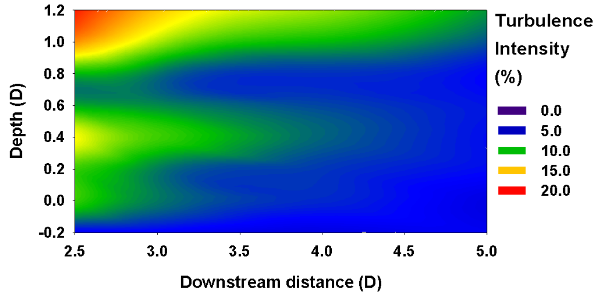

Figure 5: Centreplane u-turbulence intensity downstream of the turbine support structure.

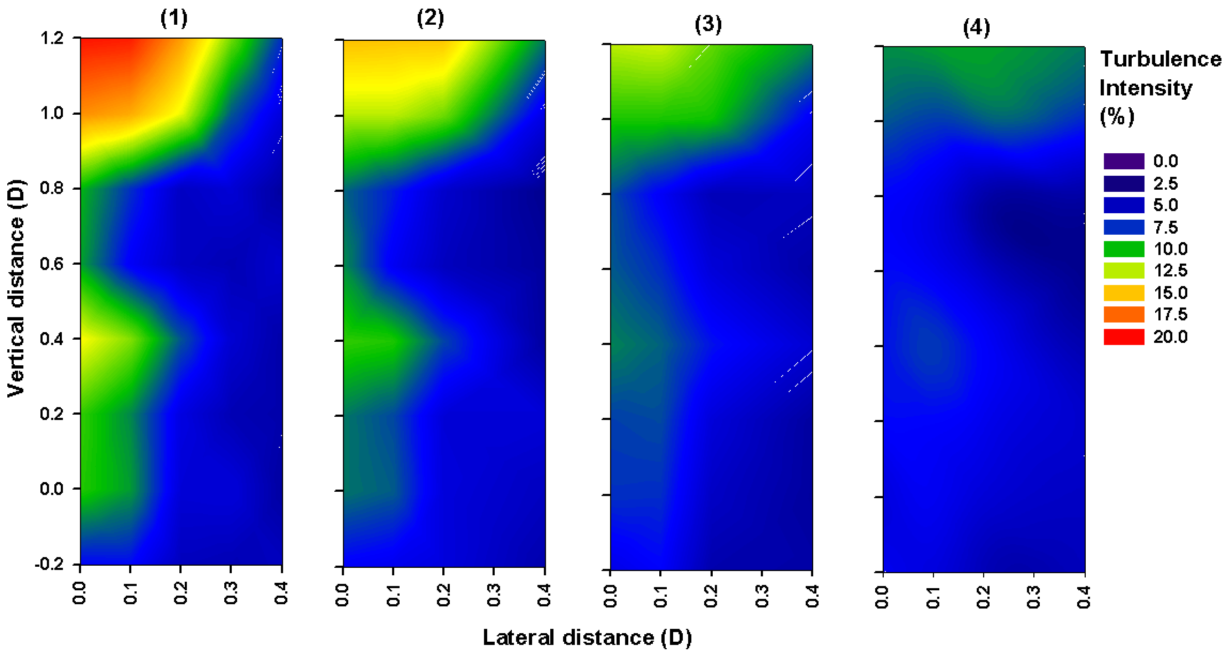

Figure 6: Lateral plane u-turbulence intensity downstream of the turbine support structure. 2.5D (1), 3D (2), 4D (3) and 5D (4).

Baseflow turbulence intensity in the water channel in the u-direction (downstream) was approximately 6% thus the added turbulence generated from the support structure is evident. The pattern of the turbulence is similar to the velocity shown in Fig 4 although there is a merging of turbulence between the lower support tower flow features. The turbulence generated from the upper section of circular tower migrates downwards to lie between +0.8 to1.0D above the rotor centre height. In this near wake region this turbulence will likely sit above any wake generated from the rotor. Fig 6 shows a series of lateral planes of turbulence intensity at increasing distance downstream. It can be observed that lateral migration of the wake is small, little more than the width of the nacelle at around 0D vertical offset. The wake from the tower section closer to the water surface expands laterally with increasing downstream distance gradually dissipating becoming confined to the surface.

The strength of the wake generated from the vertical tower and other rotor support structure is significant. Despite the low amount of lateral migration many numerical models calculate velocity deficits along the rotor centreline as this is the most turbulent and slow moving part of the wake.

Despite the installation of a streamlined section extending up the vertical tower to a point above the top of the rotor there were still significant flow effects generated by the support tower from flanges, the trailing edge of the horizontal nacelle and the upper part of the tower.

5.2 Combined rotor/support structure wake

With the support structure wake persisting into the near wake region created by the rotor it was deemed important to quantify any synergistic effects between both flow structures.

Initial inspection of the downstream vertical centre plane wake profile with the rotor in operation seemed to suggest that the wake lifted slightly with the strongest deficits at 0.1D height in the near wake region. This would have been in contradiction to previous work by the authors involving experiments with actuator disks [4,5] that suggested that increased mass flow rates over the top of the rotor (due to the non-linear vertical velocity profile) would serve to push the wake downwards. A full- width wake map of the turbine with rotor in operation seemed to suggest that the flow over the support structure was contributing to the lifting of the wake along the centreline. Fig 7 shows a number of lateral wake profiles at varying depth 5D downstream of the turbine with the rotor in operation. At the centreline it can be seen that the strongest deficit lies just above the centre height. Flow speeds at depths less than the rotor centre height are generally greater as the rotor wake is able to flow more or less uninterrupted past the support structure (Fig 2). Inspection of the full-width combined wake velocity profiles appear to show that the strongest deficit does indeed occur at the 0D vertical offset having the lowest average value of velocity for the data range shown in Fig 7. The horizontal plane at +0.1D depth exhibits more strength along a narrow band at the centre plane consistent with the presence and merging of the support structure wake whilst the average velocity across the wake width is greater than at 0D vertical offset.

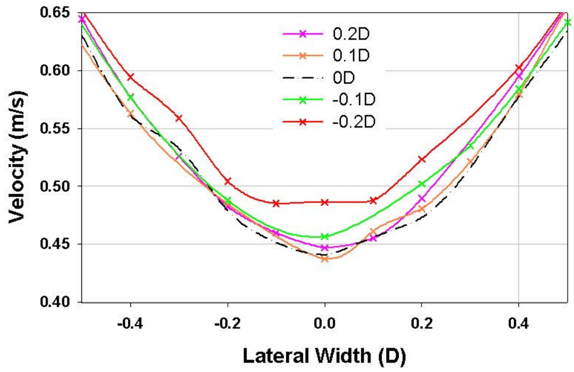

Figure 7: Lateral wake velocity at 5D downstream of the turbine with the rotor in operation.

Fig 8 further highlights the increasing of wake interaction effects between the support structure and rotor.

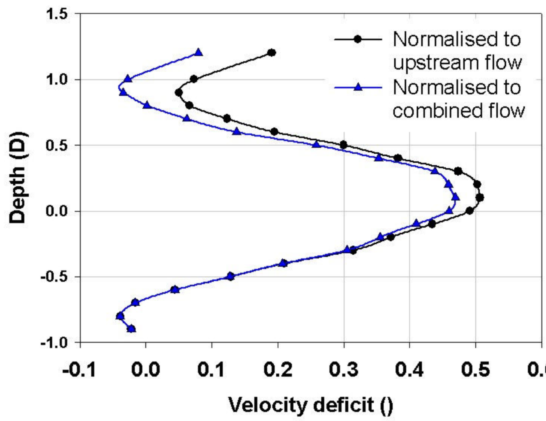

Figure 8: Centre plane vertical velocity deficit profiles at 5-Diameters downstream

Vertical velocity has been normalised as defined in equation (1) using two different values of U_0. Generally wake flow is normalised to the flow conditions immediately upstream of the device. The combined flow case also takes into account the wake created by the support structure. With a rotor thrust coefficient of 0.77 the vertical velocity deficit profiles are generated and have been measured throughout a large proportion of the water column. The effects of the tower wake are clear close to the centre height and at the surface with the source of these flow features having been discussed earlier. The implication of using the combined flow field to non-dimensionalise the wake flow is that a lower velocity deficit is seen most significantly at the centreline where many simulations focus their computational effort.

Inspection of the centreline velocity deficit profiles reveals the extent and influence of the structure wake (Fig 9).

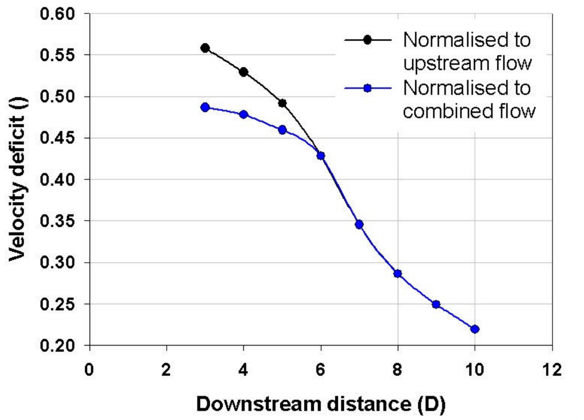

Figure 9: Centreline velocity deficit profiles for the turbine in normal operation

The wake shed from the support structure has effectively dissipated by 6-diameters downstream. Initiation of a simulation code at 3D downstream would yield a discrepancy of approximately 10% if not accounting for the effects of the support structure wake.

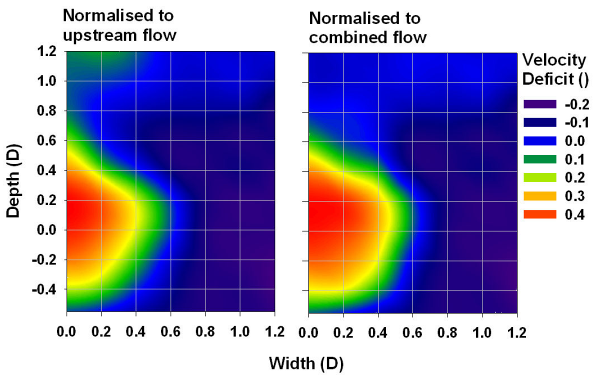

Figure 10: Lateral plane velocity deficits at 5D downstream with rotor in operation

Fig 10 visualizes the effects of normalizing to both the upstream and combined flow in the near wake region. Each plot contains 80 discrete point measurements. Removing the effects of the structure wake (right plot) provides an improved velocity distribution of the rotor wake with a more axis-symmetric shape. It can be seen that at 5D downstream the wake has expanded to a degree, beyond a radius of 0.5D. Some of the tower wake cannot be separated from the overall flow field. It can be observed as a slight lift at the top-centre of the wake on the right hand plot. However, it is much cleaner than the left plot that is normalized to the upstream flow field only. This plot is useful in visualizing the true form of the wake as the upstream flow is relatively constant with depth and width hence normalizing has a smaller effect. Features of the velocity field generated by the support structure wake as shown in Fig 6 can be seen in the left plot of Fig 10.

The wake is stronger along the vertical centerline is pinched to a narrower shape at the top centre. It follows that the effect of the structure wake and its effect upon the combined flow field at closer distances to the turbine are more pronounced.

6. Conclusions

The wake created by the supporting structure of a tidal turbine can have a significant effect upon the flow field in the near wake. Flow mapping downstream of a scale horizontal axis turbine has demonstrated that increased turbulence and velocity deficits exist along the rotor centreline and downstream of non-streamlined bluff elements of the support structure. There was a small amount of lateral flow migration from the support structure which generally dissipated between 4 and 6 rotor diameters downstream.

Combination of wakes shed from the rotor and support structure can lead to improper conclusions being drawn as to wake behaviour. In the case presented it was initially believed that the wake was being lifted vertically above the rotor centreline until it was discovered that it was due to a disturbance created by the support structure.

The influence of the wake shed from the support structure may influence how wake data is presented in a non-dimensional format. One approach is to take the undisturbed upstream flow speed and assume this persists over the longitudinal distance of the turbine wake. When non-dimensionalised this will provide a more true picture of the absolute near wake flow field but the wake from only the rotor is distorted and cannot be investigated with much effect which may be important for development and validation of numerical models. A second approach is to remove the effect of the support structure wake from the flow field (laterally and longitudinally) over its extents. This give a more complete picture of the rotor wake but some synergistic effects are created especially in areas of heavy wake mixing between rotor and support structure that cannot be removed.

This work has highlighted the requirement for highly streamlined and narrow supporting structures if the rotor wake (in isolation) requires characterisation. Investigations utilising actuator disks [4, 5] avoid this problem as narrow streamlined support stems are easy to construct and adequately strong allowing the effects of the actuator disk to be measured in isolation. Larger horizontal axis rotors require more substantial support and thus minimising downstream disturbance is more difficult.

Ongoing work in this project has investigated the effect of varying rotor thrust coefficient upon the wake structure at distances up to 10-diameters downstream for the scale turbine presented here. Rotor interaction effects utilising two of the turbines and at smaller scale with multiple actuator disks have also been completed and will be disseminated in due course.

Acknowledgements

This work is funded by the UK Technology Strategy Board. "Performance characteristics and optimisation of marine current energy converter arrays", project number T/06/00241/00/00. Additional appreciation is given to the staff at the IFREMER water channel facility, Boulogne sur Mer, France for their assistance during the testing.

References

[1] European Commission. The exploitation of tidal marine currents. Report EUR16683EN, 1996.

[2] L.S. Blunden and A.S. Bahaj. Comparison of different approaches to site selection for tidal stream energy resource assessment. *Proceedings 2nd International Conference of Renewable Energy in Maritime Island Climates (REMIC 2)*, Dublin, 26th - 28th April 2006, pp 139-144, 2006.

[3] L.E. Myers and A.S. Bahaj. Simulated electrical power potential harnessed by marine current turbine arrays in the Alderney Race. *Renewable Energy*, Volume 30, Issue 11, September 2005, pp 1713-1731, 2005.

[4] A.S. Bahaj, L.E. Myers, M.D. Thomson and N. Jorge Characterising the wake of horizontal axis marine current turbines. *Proceedings Seventh European Wave and Tidal Energy Conference*, Porto, Portugal, 11-14 September 2007.

[5] L.E. Myers, A.S.Bahaj, R. Rawlinson-Smith and M. Thomson, The effect of boundary proximity upon the wake structure of horizontal axis marine current turbines. *Proceedings 27th International Conference on Offshore Mechanics and Arctic Engineering (OMAE 2008)*, Estoril, Portugal, 15-20 June 2008.

[6] R.J. Barthelmie, Power losses due to wakes in large wind farms, Proceedings of the tenth World Renewable Energy congress, Glasgow, UK, 19th-25th July 2008.

[7] R.J. Barthelmie, S.T. Frandsen, M.N. Nielsen, S.C. Pryor, P.E Rethore and H.E. Jrgensen, Modelling and measurements of power losses and turbulence intensity in wind turbine wakes at middelgrunden offshore wind farm, *Wind Energy*, Volume: 10, Issue: 6, Pages: 517-528, 2007.

[8] P.M.Sforza, P.Sheerin and M. Smorto, Three-dimensional wakes of simulated wind turbines, *AIAA Journal*, Vol. 19, No. 9, pp 1101-1107, 1981.

[9] J.R.Connel and R.L. George, The wake of the MOD-0A1 wind turbine at two rotor diameters downwind on 3 December 1981, Report no. PNL-4210, Pacific Northwest Laboratory, Battelle, U.S, 1982.

[11] A.S.Bahaj, A.F.Molland, J.R. Chaplin and W.M.J. Batten, Power and thrust measurements of marine current turbines under various hydrodynamic flow conditions in a cavitation tunnel and a towing tank. *Renewable Energy*, Volume 32, Issue 3, March 2007, pp 407-426, 2007.

[12] L.E. Myers, Operational parameters of horizontal axis marine current turbines, Thesis, School of Civil Engineering and the Environment, University of Southmapton, 2005.

[13] A.Lohrmann, R.Cabrera and N.C. Kraus, Acoustic Doppler velocimeter (ADV) for laboratory use. In: *Proceedings of symposium on fundamentals and advancements on hydraulic measurements and experimentation*. ASCE, pp 351–365, 1997.

[14] G.Voulgaris and J.H. Trowbridge, Evaluation of the acoustic Doppler velocimeter (ADV) for turbulence measurements. *Journal of Atmospheric and Oceanic Technology*, Vol. 15, pp 272–289, 1998.

[15] P.J.Rusello, A. Lohrmann, E. Siegel and T. Maddux, Improvements in acoustic doppler velocimetry, *The 7th Int. Conf. on Hydroscience and Engineering*, ICHE-2006, Sept. 10-13, 2006.

[16] K. Blanckaert and U. Lemmin, Means of Noise Reduction in Acoustic Turbulence Measurements, *Journal of hydraulic Research*, Vol. 44, pp 3-17, 2006.