Near-wake characteristics of a model Horizontal Axis Tidal Stream Turbine

S. C. Tedds[a], I. Owen[b], R. J. Poole[a,*]

[a]*School of Engineering*
University of Liverpool,
Brownlow Street,
L69 3GU.
E-mail: sian.tedds@liv.ac.uk; robpoole@liv.ac.uk
[b]*School of Engineering,*
University of Lincoln,
Brayford Pool,
Lincoln,
LN6 7TS.
E-mail: iowen@lincoln.ac.uk

Abstract

The results of a detailed experimental investigation of the near-wake (up to seven turbine diameters downstream) of a model horizontal axis tidal turbine (HATT) device in a large-scale recirculating water channel facility are reported.. An Acoustic Doppler Velocimeter is used to provide detailed three-dimensional mean and turbulent flow field information at five different depths across the full width of the channel downstream of the turbine, giving the most complete three-dimensional velocities and Reynolds normal and shear stress data set yet available. In addition the Reynolds-stress anisotropy tensor is used to illustrate the degree of anisotropy of the Reynolds stress within the turbine's wake. These results reveal the strongly anisotropic nature of the near-wake turbulence suggesting isotropic turbulence models should not be used to model near-wake dynamics. Finally the power-law decay rates of the maximum normalised turbulent kinetic energy differ significantly from those found downstream of grids, meshes or perforated disks, suggesting that previous modelling approaches, which neglected swirl effects and modelled the turbine by absorption discs, may significantly over predict the turbulent kinetic energy decay rate of HATT wakes.

Keywords:
Near-wake, Tidal Stream Turbine, Acoustic Doppler Velocimeter Measurements, Turbulence Statistics, Reynolds Stress Anisotropy

*Corresponding author

Nomenclature

A	=	swept out area of turbine (m^2)
C_P	=	power coefficient ($\frac{P}{1/2\rho AU^3}$)
C_T	=	thrust coefficient ($\frac{T}{1/2\rho AU^2}$)
D	=	disc diameter
P	=	power (W)
R	=	radius of turbine (m)
Re	=	Reynolds number based on upstream flow velocity and turbine radius ($\frac{UR}{\nu}$)
T	=	thrust (N)
u	=	streamwise velocity (m/s)
U	=	mean upstream velocity (m/s)
v	=	transverse velocity (m/s)
w	=	spanwise velocity (m/s)
x	=	streamwise distance across the width behind the HATT (m)
y	=	transverse distance through the depth behind the HATT (m)
z	=	spanwise distance behind the HATT (m)
λ	=	tip speed ratio ($\frac{\omega R}{U}$)
ν	=	kinematic viscosity of water (m^2/s)
ρ	=	density of water(kg/m^3)
ω	=	angular velocity (rad/s)

1. Introduction

Throughout the world there is a growing demand for energy produced from sustainable resources, with many governments setting targets for renewable production of electricity. The UK aims to provide 15% of total energy from renewable sources by 2020, this is an increase from 3.3% produced in 2010 [1]. Harnessing the ocean's energy is one way to meet these targets, and the energy can be split into two categories: wave and tidal. One of the main advantages of tidal power is the predictability of the tides. There are two principal ways to harness tidal energy: tidal barrages or lagoons which use the tidal range, and tidal stream turbines that use the tidal current. An advantage of tidal stream turbines is that they minimise the impact on the marine environment as they allow water to pass straight through, and are usually fully submerged with no visual impact.

As well as characterising the power output for a tidal stream turbine it is also important to characterise the wake of a turbine. For example, as turbines are likely to be placed in farms or arrays to make them commercially viable, wake recovery length is crucial for the appropriate spacing between turbines. Also, knowledge of the wake is important so that potential effects on the sea bed can be investigated. Only a limited amount of research has hitherto been undertaken into the wakes behind tidal stream turbines. The earliest of this research was conducted using an absorption disc to represent the turbine, both experimentally [2] and using Computational Fluid Dynamics (CFD) [3, 4, 5]. Experimental studies into the characterisation of the wake were conducted by Myers and Bahaj [6], Stallard et al. [7, 8], Rose et al. [9] and Maganga et al. [10]. Further studies to compare experimental wake data to CFD were completed by Mycek et al. [11] and Rose et al. [12]. Table 1 includes relevant details of studies undertaken to date.

Myers and Bahaj [2] conducted experiments in a 21 m tilting flume, which had a width of 1.35 m and depth of 0.4 m. The vertical velocity profile produced in the flume resembled a modified $\frac{1}{7}$ th power law which was more uniform closer to the surface and approximated a profile measured at a full scale site and reported by the Carbon Trust [13]. Artificial bed roughness was added to the flume to reduce the velocity and increase the shear stress in the bottom third of the water column, again to create conditions that were as realistic as possible. The absorption discs (mesh discs) used by Myers and Bahaj [2] were 100 mm in diameter and had porosity (ratio of opened-to-closed area) varying between 0.48 and 0.35. A load cell was attached to the disc so that thrust acting on it could be measured. An Acoustic Doppler Velocimeter (ADV) device was used, which sampled at a rate of 50 Hz with a sample volume of 0.15 cm^3, to map the wake up to 20 disc diameters (D) downstream. It was found that the porosity alone did not determine the thrust coefficient. Four discs with equal porosity were tested, those with a larger number of small holes had a greater thrust than discs with fewer large holes. For all discs tested, at $10D$ downstream the mean velocity profiles were virtually identical and any effects of the disc had dissipated. Using a disc with thrust coefficient, $C_T \approx 0.9$, Myers and Bahaj [2] investigated four different discs depths, with the disc centred at 0.33 D, 0.5 D, 0.66 D and 0.75 D above the floor of the flume. It was found that as the disc was placed closer to the floor, the mean velocity deficit is longer. The turbulent kinetic energy (TKE) was also measured, where Myers and Bahaj [2] defined TKE as

$$TKE = \frac{1}{2}(\bar{u'^2} + \bar{v'^2} + \bar{w'^2}), \tag{1}$$

where u', v' and w' are the varying components of the flow, and the bar above denotes this as a time average. These tests found that there was some evidence that as the distance between the floor and the disc increased, the TKE levels in the wake increased. To simulate a rocky seabed in their flume, Myers and Bahaj [2] placed artificial rocks with lengths between 6.7 and 10 mm over a length of 4 m, as they expected this type of bed to have the greatest effect on turbulence. Again the depth at which the disc was placed was altered and as the distance between the disc and the floor decreased, the wake velocity deficit downstream increased; in turn this increase of the velocity deficit was larger than that with the smoother bed. As a consequence, Myers and Bahaj [2] suggested that a rocky seabed is less suitable for placing tidal turbines than a smoother bed. The experiments conducted by Myers and Bahaj [2] provide useful information on the far wake provided swirl effects are not still important. The changes in wake length with the height of the turbine provides a good indication of where a turbine should be placed in relation to the bed to minimise the effect of the bed.

Sun et al. [3] also used an absorption disc approximation to simulate a tidal turbine, in both two- and three-dimensional CFD calculations. The computational domain used replicated a laboratory water channel, which was 1.5 m wide, 10 m long and had a water depth of 1 m. A volume-of-fluid approach was used to simulate the free surface and a no-slip wall boundary condition was used on the bed. The turbulence model used was the $k-\varepsilon$ model which assumes isotropic turbulence. In the two-dimensional simulations, Sun et al. [3] used an absorption zone centred at a height of 0.5 m. When the water passed through the absorption disc approximately 38% of energy was dissipated. Results from this solution showed a substantial drop in the free-surface behind the absorption disc, to a lowest point of roughly half its initial height. The two-dimensional domain was extended further to a three-dimensional domain, where a square disc was used which had an area ratio to the flume cross-section of 17%, this resulted in a 10% overall loss of kinetic energy in the flume. The results show that as the water flow reaches the disc it accelerates as it flows through, then velocity drops directly behind the disc. Similar to the two-dimensional solution, there was a free-surface drop behind the disc, however the drop was not as large for the three-dimensional model. These simulations gave a first approach for the characterisation of the wake and were concentrated on the near-wake and effects on the free-surface close to the disc.

Harrison et al. [4] compared CFD simulations to the experimental absorption disc results conducted by Myers and Bahaj [2] discussed above. Harrison et al. [4] simulated the free-surface using a homogeneous coupled volume-of-fluid approach. The flow was calculated using the $k-\omega$ shear stress transport (SST) model, which is an extension of the $k-\varepsilon$ model, this was used as preliminary studies indicated that the $k-\varepsilon$ did not accurately model the flow conditions. The model domain represented the flume in which the experiments of Myers and Bahaj [2] were conducted, with a water depth of 0.3 m and with air to a height of 0.15m above the water. The disc was matched to that of Myers and Bahaj [2] with a diameter of 0.1 m and thickness of 0.001 m, and was located in the centre of the water column. To enable direct comparison with the experimental results, the thrust coefficient was matched. The measured inlet velocity distribution of the flume was used to set the inlet velocity in the CFD model. Both the far-wake and the near-wake were compared, the near-wake had some limitations in its accuracy, in that the effects of swirl were not taken into account, but this, the authors argued, was a reasonable method to estimate the effects of the far wake. The trend and wake recovery of the far wake was found to be similar in the two approaches and the turbulence levels in the wake were comparable.

A computational study was conducted by Daly et al. [14] of an 'actuator fence' to simulate an array of tidal turbines, which was placed at various heights in a channel. These results were compared to experimental data of the wake of a actuator fence in a channel, which gave a reasonable approximation to the streamwise velocity downstream of the fence.

Myers and Bahaj [6] measured the wake of a turbine and support structure with a diameter of $0.8m$, using an Acoustic Doppler Velocimeter (ADV), in a recirculating water channel which was $18m$ long, $4m$ wide and $2m$ deep, giving a blockage ratio of 6.3% and had an upstream velocity of $0.8m/s$. The wake of the turbine support structure without the rotor was measured and was found to be significant, especially near the free-surface. The support structure increased the turbulence intensity, again near to the free-surface, by $5D$ downstream the TI was 10% which was still much higher than the upstream level of 6%. The wake of the support structure and rotor was measured at $5D$ downstream though the width of the channel at five different heights ($-0.2D$, $-0.1D$, centre, $0.1D$, $0.2D$). The largest deficit was found to be at a height of $-0.1D$ due to the presence of the structure support and the rotation of the blades. The smallest deficit is found at the deepest point measured ($-0.2D$) as the support structure is not present here. The streamwise velocity was also measured from $3D$ to $10D$ downstream of the centre of the turbine, it was found that by $10D$ downstream the velocity was still less than 80% of the upstream velocity. Overall it is clear that the support structure in this particular set-up had a dominant effect on the overall wake.

Batten et al. [5] investigated the use of the actuator disc- Reynolds-average Navier-Stokes (RANS) approach of predicting the wake of a tidal turbine. The RANS results were compared to the experiments conducted by Myers and Bahaj [6], discussed above, therefore the turbine diameter, size of channel and velocity and TI was matched. Two different computational models were investigated, firstly, a RANS actuator disc approach + blade element (BE) approach and RANS + disk approach where turbulence is injected at the disc. The RANS + BE can be used to predict the power coefficient (C_P) and thrust coefficient, this model under predicted the C_P at a tip speed ration greater than 6.5 compared to the results from blade element momentum theory (BEMT), but gave a reasonable approximation to the experimental results. The thrust coefficient was overpredicted by the RANS + BE approach compared to both the BEMT and experimental results. The time averaged velocity and TI downstream of the turbine at the centreline of both the RANS + BE and the RANS + disk approaches showed a reasonable agreement to experimental data of [6] between $5D$ and $10D$ downstream. At $5D$, $8D$ and $10D$ downstream time average velocities and TI of the two computational approaches and experimental data were analysed through the depth of the channel. At $5D$ downstream the TI from the RANS + BE model was greater than that of the experimental data, but further downstream the TI levels of the model were in reasonable agreement with the experimental data. Batten et al. [5] concluded that the RANS + BE model was a better method of modelling a turbine compared to the RANS + disk model as the wake measurements showed a better agreement to that of the experiments and it removed the pragmatic approach of turbulence source terms added at the turbine location.

Stallard et al. [7, 8] probed the effect of a series of turbines on the wake structure. All turbines were three bladed and had a diameter of $0.27m$ (corresponding to an estimated $\frac{1}{70}$th scale), with a Reynolds number based on the blade chord of $30,000$. These experiments were conducted in a flume with a water depth of $0.45m$, width of $5m$ and length of $12m$, the upstream flow had a turbulence intensity (TI $= \sqrt{\frac{1}{3}(u'^2 + v'^2 + w'^2)} = \sqrt{\frac{2}{3}TKE}$) of approximately 10% and a mean velocity of $0.45m/s$. The number of turbines tested at a time were 1, 2, 3, 5, 6, 7 and 10. The turbines in the arrays were configured into one, two, and three rows. The lateral spacing between the devices was 1.5, 2 or $3D$), the longitudinal spacing of devices considered was between 4 and $10D$. The wake was measured using an ADV, sampling at $200Hz$ for $60s$ at each location (i.e. collecting $12,000$ samples per spatial point). For a single turbine, the wake was measured directly downstream of the centreline of the turbine. Rapid recovery of the wake was observed for the first $5D$ downstream. Further downstream the recovery was much slower and by $20D$ downstream the wake still has a velocity deficit of just under 20%. It was found that with the 2, 3 and 5 turbine configurations the individual deficit was identifiable up to $4D$ downstream but further downstream all the wakes merged into a single structure. The effect of waves was briefly investigated, with a waveheight of $50mm$ and a peak period of $1.25s$ (frequency $0.8Hz$). These waves increased the turbulence intensity to around 14.4% at the surface but the TI decayed through the depth. The wake recovery in these conditions was similar over a range of $5D$ to $10D$ downstream, however the turbulence intensity remains above 12% at $10D$ downstream.

Rose et al. [9] tested three devices and measured the wake using different techniques. The wake of a four-bladed, $0.14m$ diameter, turbine, with a tip chord Reynolds number of $25,000$, was measured using a Particle Imaging Velocimetry (PIV) system which acquired images at $100Hz$ with $1000\mu s$ between pairs of images. A total of 1000 image pairs were taken for each test, over a $10s$ period. This experiment was undertaken in a flume which was $20m$ long, $0.75m$ wide and $0.52m$ deep, at a velocity of $0.57m/s$. The second experiment took place in a flume $35m$ x $0.4m$ x $0.92m$ (length x width x depth) at a mean velocity of $0.42m/s$ using a two-bladed turbine with a diameter of $0.25m$, and tip chord Reynolds number of $27,000$, placed at $0.23m$ below the surface. A Nortek ADV was used to measure the velocities downstream, sampling at $25Hz$ for $33s$ at each location, therefore collecting a total of 825 samples per location. A third, larger scale experiment was undertaken in a lake by towing a turbine. The site, Montgomery Lough, is a lake of approximately $400m$ x $133m$. The turbine used was four-bladed with a diameter of $1.5m$, with a Reynolds number based on the chord length of $246,000$; it was placed $1.285m$ below the water surface. An ADV system was again used which had a sampling rate of $64Hz$, each point was sampled for $170s$ therefore producing $10,000$ samples per point. Measurements were taken between $0.5D$ and $3D$ in the longitudinal direction and $1D$ laterally. It is difficult to compare these three results produced by Rose et al. [9] because of the differing thrust coefficients for each turbine, combined with the sparse data sets. As other work has shown that the wake is strongly effected by the thrust on the turbine and C_T is not matched in these cases [2, 4] it is not entirely unexpected for the wakes to be different. The general pattern produced by the three experiments are consistent as the centreline always has the maximum velocity deficit.

Maganga et al. [10] measured the wake of a three-bladed turbine, with a diameter of $0.7m$ in a flume $18m$ long, $4m$ wide and $2m$ deep, giving a blockage ratio of 5%. Tests were conducted with velocities between $0.5m/s$ and $1.5m/s$, and with different upstream turbulence intensity levels of 4.6% and 14.4%. Different yaw angles of the turbine were investigated; $0^o, -10^o, 10^o, 15^o$ and 20^o. Power and thrust were also measured at these yaw angles, and it was found that as the yaw angle increased from the optimum position (i.e. correctly aligned (0^o)), the thrust decreased. It was also noticed at the higher turbulence intensity level, 14.4%, both C_P and C_T decreased in comparison to the base case (4.6% TI). A two dimensional Laser Doppler Velocimetry (LDV) system was used to measure the velocities behind the turbine, with a sampling rate of $50Hz$ for $100s$. Measurements were taken every diameter in the streamwise direction behind the turbine, up to $10D$ downstream; measurements were also taken in the spanwise direction between $-1.7D$ and $1.7D$ with a spacing of $100mm$ between points. Results from the study showed that with the higher turbulence intensity level (14.4%) there is a much faster wake recovery and that after $5D$ downstream the deficit is almost negligible. At the lower turbulence intensity level (4.6%) the mean velocity deficit is still clearly visible $10D$ downstream and the turbulence intensity levels in the wake remained higher than upstream levels. These results confirm a number of experimental studies conducted on the influence of ambient turbulence intensity on the near wake of different structures: a wind turbine, a circular cylinder and prisms [15, 16, 17, 18].

Mycek et al. [11] furthered the work of Maganga et al. [10] by studying the interaction between two tidal turbines placed in series, by first measuring the wake of a single turbine and then the interaction of wakes of two turbines one placed downstream of the other; this was done both experimentally, using two component LDV and numerically using CFD. The turbines investigated were three-bladed with a diameter of $0.7m$ ($\frac{1}{30}$th scale). The water channel used was the same as that of Maganga et al. [10], with a flow velocity of $0.8m/s$ and upstream turbulence intensity levels of 2.9% and 14.4%. The velocities and turbulence intensities were measured with a two component LDV system and were collected at each point for $100s$ with a frequency of $7 - 17Hz$ (therefore $700 - 1700$ samples per point). The measurements were taken up to $10D$ downstream of the turbine and 25 measurements were taken in the spanwise direction across the width of the flume every $100mm$. For the wake behind a single turbine with TI of 14.4% it was found that by $7D$ downstream the wake had recovered to 90% of its upstream turbulence intensity level and its uniformity had returned. Whereas with a TI of 2.9% even after 10 diameters the profile was still non-uniform and the TI was still much greater than the upstream levels. Thus high upstream turbulence levels significantly reduce the wake, presumably by enhancing the transfer of momentum between the free stream and the wake regions. The numerical study conducted by Mycek et al. [11] investigated an upstream flow with 0% TI, using a Lagrangian vortex particle method with a large eddy simulation turbulence model. A comparison of these results with experimental data was undertaken at $1.2D$ behind the turbine. The results showed a qualitatively similar profile, however the mean velocity was underestimated by the computational model. Experimental work with two turbines, also reported by Mycek et al. [11] compared the power output of the second turbine at $4D$, $6D$, $8D$ and $10D$ downstream from the first turbine. As expected there was a drop in maximum power coefficient, C_P, for the downstream turbine (based on the velocity upstream of the first turbine) and the further downstream the second turbine was placed from the first, the higher the maximum C_P of the downstream turbine.

Other comparisons of CFD and experimental work have been conducted by Rose et al.[12], in which a single rotor with a diameter of $0.3m$ was used. The turbine had interchangeable blades and 2 and 3 blade configurations were tested at different blade pitch angles (0^o, 2^o, 4^o and 6^o). The experiments were conducted in a tank which was $0.92m$ wide, $0.42m$ deep and $35m$ long, therefore giving a blockage ratio of 18%. An ADV was used to measure the three-dimensional velocities and TI, which were sampled at $25Hz$ for $33s$ at each location (i.e $\approx$ 825 points per location). The results showed that there was a rapid and significant recovery between $2D$ and $8D$ downstream, however the wake had not fully returned to the mean upstream velocity by $25D$ downstream. The increase in blade pitch angles showed a smaller wake impact and the three-bladed turbine had a higher deficit than the two-bladed due to the corresponding higher C_T. The CFD comparison conducted by Rose et al. [12], used a three-dimensional RANS simulation using the same width and height of the flume in which the experiments were conducted, but they made the channel longer. A $k - \omega$ turbulence model was used near the walls and $k - \epsilon$ was used in the main flow. The assumptions of isotropic turbulence in this latter model meant that it was under-predicting the wake compared to that measured in the experimental study.

The research studies conducted to date have investigated the mean streamwise flow and turbulence intensity in single planes downstream of the turbine, with measurements confined to a centre plane through the turbine or planes normal to the flow. In the experimental studies the number of locations at which the flow has been characterised has been sparse or been concerned with the far-wake effects. This study investigates the near-wake ($x/D \leq 7$) at five different depths across the full width of the channel downstream of a turbine, giving the most complete three-dimensional velocities, turbulence statistics and Reynolds shear stresses data set available. Previous numerical studies have generally under-predicted the wake due to the assumption of an isotropic turbulent flow field. In this study the Reynolds-stress anisotropy tensor is used to illustrate the degree of anisotropy of the Reynolds stress within the turbine's wake.

Table 1: Overview of previous tidal stream turbine wake studies

Author / Year	CFD / Experimental	Turbine diameter	Blockage ratio	TI	Reynolds Number	C_T	Conclusions
Sun et al. (2008) [3]	CFD	$0.5m$	17%				First approach at estimating the near wake.
Myers and Bahaj (2009) [6]	Exp	$0.8m$	6.3%	$\approx 6\%$	$3.2\text{x}10^{5}$*	0.9	Support structure had a large influence up to $5D$ downstream. By $10D$ the velocity had recovered to 80% of the upstream velocity.
Myers and Bahaj (2010) [2]	Exp	$100mm$ - mesh discs	1.5%	$\approx 5\%$	$2.5\text{x}10^{4}$*	≈ 0.9	When disc was placed closer to the floor the velocity deficit persisted for longer.
Harrison et al. (2010) [4]	CFD	$100mm$ - mesh discs	1.5%	$\approx 5\%$	$2.5\text{x}10^{4}$*	≈ 0.9	The trend of the wake recovery and turbulence levels are qualitatively similar in the CFD and experimental results.
Maganga et al. (2010) [10]	Exp	$0.7m$	5%	4.6% and 14.4%	$3.5\text{x}10^{5}$*		With 14.4% TI after $5D$ downstream the deficit is almost negligible, with 4.6% TI at $10D$ downstream the TI is sill higher and the velocity deficit still visible.

Author / Year	CFD / Experimental	Turbine diameter	Blockage ratio	T.I.	Reynolds Number	C_T	Conclusions
Rose et al. (2011) [9]	Exp	$0.14m$, $0.25m$ and $1.5m$	3.9%, 13.3%		Based on the tip chord $25,000$, $27,000$ and $246,000$	0.959, 0.373 and 0.567	Centreline measurements of velocity are consistent for the three devices, the maximum deficit is always at the centreline.
Rose et al. (2011) [12]	CFD and Exp	$0.3m$	18%		$1.2\text{x}10^5$*		Rapid recovery between $2D$ and $8D$ but by $25D$ still not fully recovered. The CFD results underpredicted the experimental results.
Mycek et al. (2011) [11]	Exp and CFD	$0.7m$	5%	2.9% and 14.4% experimentally and 0% in CFD	$3.5\text{x}10^5$*		With 14.4% TI at $7D$ downstream the TI is 90% of the initial TI. With the TI levels of 2.9% at $10D$ downstream the flow is still non-uniform and TI levels are still much greater. The CFD results were similar for the velocity measurements but underestimate the TI.

Author / Year	CFD / Experimental	Turbine diameter	Blockage ratio	T.I.	Reynolds Number	C_T	Conclusions
Stallard et al. (2011) [7], (2013) [8]	Exp	$0.27m$	2.5%	10% and 25% with waves	based on blade chord $30,000$	0.82	Rapid recovery up to $5D$ downstream with a recovery of 70% of upstream velocity, but by $20D$ downstream the velocity was 80% of the upstream velocity. With two or more turbines the individual wakes of the devices can be seen up to $4D$ downstream but after the wakes merge to form a single deficit. The effect of waves on the wake had a similar velocity trend to the uniform flow between $5D$ to $10D$ but the TI remained a lot higher downstream.
Batten et al. (2013) [5]	CFD	$0.8m$	6.3%	5%	3.2×10^5	0.9	RANS + BE model gives a better agreement to experimental data than the disc model. Results showed that by $22D$ downstream the velocity had not returned to the upstream value.
Daly et al. (2013) [14]	CFD						An actuator fence was used to simulate arrays of turbines, the computational approach showed good agreement to the experimental results of a fence.

*based on the radius and upstream velocity which is estimated by present authors based on limited data provided in the original papers.

2. Test Facilities

Testing was completed at the University of Liverpool using the high-speed recirculating water flume and a turbine with diameter of $0.5m$.

2.1. Description of Turbine

The original Horizontal Axis Tidal Turbine (HATT) blade profile was selected by [19] using methods suggested by the National Advisory Committee for Aeronautics (NACA) for aerofoil sections, and subsequently optimised using Blade Element Modelling theory. Figure 1 shows schematically the assembled $0.5m$ diameter, three-blade configuration HATT with the stanchion attachment point and nose cone that has been used in this study. The turbine has been designed such that the number of blades can be varied from 2 up to a maximum of 6 blades and the blade pitch angle can be varied. Complete details of the turbine can be found in [20].

The turbine was connected to a Baldor brushless AC servomotor in order to measure/calculate the torque, angular velocity and power generated via hydrodynamic loading. A regen resistor or dynamic brake was used to apply an opposing torque to that developed by the hydrodynamic forces on the turbine. This motor was combined with a control system which in turn was programmed via Workbench V5 [21, 22].

The torque applied was proportional to the maximum drive rate current, in this case $10.1A$ (max), which is related to the torque by a proportionality constant of $0.82Nm/A$. The torque throughout the tests was set to 20% of the current limit. The rotational speed of the motor was measured as a percentage of the rated rotational velocity for the motor ($500rpm$). Therefore, the angular velocity of the turbine was calculated from:

$$\omega = \frac{speed(\%)}{100} \text{x} \frac{500}{60} \text{x} 2\pi = speed\,\text{x}\frac{\pi}{6} \tag{2}$$

2.2. Experimental Set-up

Testing was undertaken in the University of Liverpool high-speed re-circulating water flume, a schematic of which is shown in figure 2. The flume uses a $75kW$ motor-driven axial-flow impeller to circulate $80,000$ litres of water. The water flows into the working section which is $3.7m$ long by $1.4m$ wide and can provide a depth range of between $0.15m$ and $0.85m$. This provides the possibility of a uniform velocity profile ranging between $0.03m/s$ and $6m/s$.

The conditions under which the experimental tests were made were $0.8m$ water depth with uniform velocity in the range 0.5 - $1.5m/s$ with a measured turbulence intensity of 2%. The turbine was located at a depth of $0.425m$, midway along the working section, giving a blockage ratio of approximately 16% [23, 22].

To determine the thrust on the turbine a $50kg$ strain gauge dynamometer was used, the design of which is described in detail by [24]. The force block was calibrated by applying a load from $1kg$ to $25kg$ in steps of $1kg$. It is estimated that this produced a calibration which is accurate to about 1%.

An Acoustic Doppler Velocimeter (Nortek Vectrino+) was used to measure velocities and turbulence statistics. ADV is a well known technique [25] for simultaneously measuring velocity components in three dimensions using the acoustic Doppler principle. The ADV used in this study had four $10MHz$ receiving elements positioned around a $10MHz$ transmitter. The probe was submerged in the flow and focused on a location $50mm$ away from the probe head to minimise flow interference. The ADV had a cylindrical sampling volume set to $198mm^2$ (height of $7mm$ and diameter of $6mm$). Data was collected at a sampling rate of $200Hz$ and at least $10,000$ individual velocities were measured to ensure the mean velocities and turbulence statistics were meaningful. The statistical uncertainty in these mean velocities is estimated to be better than 1%. Velocity measurements were taken upstream of the turbine at distances of $0.25m$, $0.5m$ and $1m$ at a number of heights, and taken in the spanwise direction across the width of the flume with a maximum of $50mm$ between measurement locations. The near-wake was measured at five different depths (transverse distances) at a number of streamwise distances behind the turbine; from $1.5D$ behind the turbine to $7D$, and in the spanwise direction at least every $50mm$, where the origin is defined at the point where the blades meet in the centre of the hub. These measurement locations are illustrated in Figure 3.

3. Results and Discussion

For all the measurements downstream of the three bladed turbine the optimum blade pitch angle was set (6^o). As stated previously the torque was set to 20% of the motor's maximum torque, this results in $C_P = \frac{P}{1/2\rho AU^3} = 0.34$, $\lambda = \frac{\omega R}{T} = 6.15$, $C_T = \frac{T}{1/2\rho AU^2} = 1$ (this value was determined by measuring the force on the whole turbine structure, including body and support, and then subtracting the drag of the body and support alone without blades at identical upstream velocities) and $Re = \frac{UR}{\nu} = 2.22 \text{ x } 10^5$.

3.1. Mean flow

Figure 4 shows the mean streamwise velocities behind the three-bladed turbine measured in horizontal planes. The largest velocity deficit is at the tip of the blades, in figure 4(a) this can be seen at $z/D = \pm 0.5$ and in figures 4(d),4(e) at the centre (i.e. $z/D = 0$). By seven diameters downstream there is an overall recovery of the streamwise velocity to about 80% of the upstream velocity. There is a small asymmetry in the results due to free-surface effects. This asymmetry in figures 4(b) and (c) can be observed by noting that the velocities are larger at $y/D = -0.25$ than at $y/D = +0.25$, similarly it can be seen at $y/D = -0.5$ the velocities are greater than at $y/D = +0.5$ in figures 4(d) and (e).

The mean transverse (vertical) velocities in figure 5 show the effect of the swirl induced by the turbine blade rotation, this is clearest in figure 5(a) where there are both negative velocities and positive velocities corresponding to the clockwise rotation of the blades; this same trend can be seen at $\pm 0.25D$ depths, again, asymmetry due to the surface effects can be seen between $y/D = +0.5$ and $y/D = -0.5D$. Significant transverse velocities are still evident at the edge of the measurement window (i.e. $6 - 7$ diameters downstream).

Figure 6 shows the mean spanwise (side-to-side) velocities, which are all less than 10% of the streamwise velocities, the regions of high absolute velocities occur generally at the higher turbulent regions at the edge of the blades. These higher turbulent regions alter between negative and positive velocities, in particular when we look at figure 6(b) between 2 and $3D$ downstream there are negative values on one side and positive on the other, further downstream this is reversed, showing the effect of two rotations of the large-scale swirling motions.

3.2. Turbulent flow field

Figure 7 show the streamwise Reynolds normal stresses (u') at the centre height and up to seven diameter distances downstream. The Reynolds normal stresses of the three velocity components, u', v' and w' are defined by the standard deviation in each direction;

$$u' = \sqrt{(\frac{1}{N}\sum_{i=1}^{N}(u_i - \bar{u}_i)^2)}, \tag{3}$$

$$v' = \sqrt{(\frac{1}{N}\sum_{i=1}^{N}(v_i - \bar{v}_i)^2)}, \tag{4}$$

$$w' = \sqrt{(\frac{1}{N}\sum_{i=1}^{N}(w_i - \bar{w}_i)^2)}, \tag{5}$$

where N is the number of samples at a particular point and $\bar{u}$, $\bar{v}$ and $\bar{w}$ are the mean velocities at a particular location. The key point to take from the data for the Reynolds normal stresses is that there is a maximum effect around the tip of the blades ($z/D = \pm 0.5$) and this effect can be seen up to $7D$ downstream showing the turbine still has an impact on the flow field at this location. The Reynolds normal stresses emanating from the blade tip can be expected to be higher due to the blade speed being maximum at this point, and the generation of tip vortices. The other two velocity components (v' and w') exhibit similar variations and are not shown for conciseness.

Figure 8 show spanwise distributions of the normalised turbulent kinetic energy (TKE) downstream of the three bladed turbine, where TKE is defined in equation (1). In figure 8, it can be seen that the TKE levels peak at the edges of the blades at $z/D = \pm 0.5$ and can still be observed in all measurements up to seven diameters downstream. In figure 8(a), it can be seen that at $1.5D$ downstream there is a third smaller peak at $z/D = 0$, which is probably due to the presence of the stanchion.

3.3. Decay of turbulence anisotropy downstream of the turbine

To investigate the anisotropy of the turbulent flow field behind the turbine the Reynolds-stress anisotropy tensor **b** can be used and is defined by Choi and Lumley [26];

$$b_{i,j} = \begin{pmatrix} \frac{u'^2}{u'^2+v'^2+w'^2} - \frac{1}{3} & \frac{\overline{uv}}{u'^2+v'^2+w'^2} & \frac{\overline{uw}}{u'^2+v'^2+w'^2} \\ \frac{\overline{uv}}{u'^2+v'^2+w'^2} & \frac{v'^2}{u'^2+v'^2+w'^2} - \frac{1}{3} & \frac{\overline{vw}}{u'^2+v'^2+w'^2} \\ \frac{\overline{uw}}{u'^2+v'^2+w'^2} & \frac{\overline{vw}}{u'^2+v'^2+w'^2} & \frac{w'^2}{u'^2+v'^2+w'^2} - \frac{1}{3} \end{pmatrix}. \tag{6}$$

The Reynolds-stress anisotropy can be more simply, and is commonly, characterised by a pair of invariants, η and ξ, defined by;

$$6\xi^2 = b_{ii}^2 = b_{ij}b_{ji}, \tag{7}$$

$$6\eta^2 = b_{ii}^3 = b_{ij}b_{jk}b_{ki}, \tag{8}$$

[27, 26, 28, 29]. As discussed in the introduction, most studies have modelled the turbine wake using standard turbulence models which assume isotropic turbulence. Therefore knowledge of the degree of turbulence anisotropy induced by the turbine and the rate of return to isotropy is important. At any point and time in any turbulent flow these invariants can be defined from the Reynolds-stresses, and are restricted with respect to the realisability of the flow to the anisotropy invariant map, which is often referred to as the 'Lumley triangle' [27] which is shown in figure 9. At the origin when both invariants are zero, i.e. $\xi = \eta = 0$, this situation corresponds to isotropic turbulence: all three diagonal components of the Reynolds stress tensor are equal and therefore $u' = v' = w'$. Emanating from this isotropic origin there are two limiting sides of the triangle which are said to be axisymmetric. The side of the 'Lumley triangle' when $\xi > 0$, is where a single diagonal component of the Reynolds stress tensor is dominating over the other two components and the shape of the tensor can be said to be 'rod-shaped'. As ξ increases further this side leads up to the limit point where two fluctuating components have tended to zero resulting in a one-component state of turbulence. The other limiting line of the 'Lumley triangle' when $\xi < 0$, is where two equal diagonal components of the Reynolds stress tensor are dominating over the third smaller diagonal component, and the shape of the tensor is said to be 'disc-shaped'. As η increases along this side until the smallest diagonal component has tended to zero the limit point at the side's end corresponds to a two-component state of turbulence. The line connecting the one- and two-component axisymmetric turbulence is all other possible states of the Reynolds stress tensor where there are only two non-zero diagonal components of the tensor. To aid in the visualisation of the eight turbulence anisotropy shapes (i.e. rod-shaped, disc-shaped etc) a graphic representation of the different possible ellipsoid shapes formed by the diagonal components of the Reynolds-stress tensor are shown in figure 10.

The Reynolds stress anisotropy data plotted in figure 9, has been undertaken to compare the effect of the turbine and the outer areas to observe the effect around the 'outside' of the turbine. Plotted in this way it can be seen that the upstream (inlet) velocity has two dominant turbulent components (u' and w'), shown in figure 9(a). Figure 9(a) shows the downstream data closest to the turbine, and there are larger values for ξ and η, showing the flow is highly anisotropic, further downstream and the figures 9(b) and (c) show that the flow is mainly in the so-called 'disc-shaped' region, where two of the diagonal components of the tensor are dominant, but further downstream the flow gradually becomes, as expected, more isotropic as the two dominant axisymmetric components decrease (i.e. u' and w' become smaller). However at $7D$ the turbulence is still strongly anisotropic and the effect of the turbine is still being felt. Thus turbulence models which assume turbulent anisotropy, such as the $k - \epsilon$, will always fail to capture the near-wake turbulent flow field correctly.

3.4. Wake decay characteristics

Figure 11(a) shows the maximum velocity deficit in the streamwise direction at the five different heights behind the turbine, the data for different heights exhibit a similar trend; the first three diameters behind shows a rapid velocity recovery, further downstream the recovery is much slower. The centreline component of the streamwise velocity can been seen to be recovering with a power law with a decay exponent ≈ 1.2. The maximum of the transverse and spanwise velocities (figures 11(b-c)) can be seen to essentially linearly decrease at the different heights, but there is no general trend to the heights which have the maximum deficit.

The maximum of the Reynolds normal stresses of the three-dimensional velocity field is shown in figure 12 and shows a decrease in the maximum values further downstream. In the streamwise and spanwise directions (figures 12(a) and (c)), the maximum Reynolds normal stresses decay at a similar rate with a power-law decay exponent of approximately 0.5, where the values closest to the turbine are approximately double in magnitude to that of the furthest measurement ($7D$ downstream).

The maximum of the Reynolds shear stresses and turbulent kinetic energy is shown in figure 13. All the shear stresses decay with a power law, in the $u - v$ plane and $v - w$ plane, and by four disc diameters downstream have decayed to a minimum level of 20% of the maximum level at two disc diameters downstream. The maximum turbulent kinetic energy decays with a power law with an exponent of ≈ 0.9, which is a slower rate than that of isotropic grid-generated (or mesh generated) turbulence which has a decay exponent most commonly found to be around 1.3 [28]. Such differences would indicate that simplified methods to model the wake characteristics of tidal stream turbines, such as the absorption disc approach [3, 2, 4], may significantly overpredict the rate of decay of the maximum turbulent kinetic energy downstream of a model HATT. The experimental data of Harrison et al. [4], when replotted as TKE (TI $=\sqrt{\frac{2}{3}TKE}$) as shown in figure 13(d), exhibit power-law decay rates of 1.3 to 1.6 dependent on exact C_T value for example. Thus at identical C_T they predict a TKE decay rate almost twice as high as that measured here.

4. Conclusions and outlook

The near-wake ($x/D = 7$) of a model horizontal axis tidal stream turbine has been determined experimentally using detailed three-dimensional ADV measurements. These measurements reveal the complex structure of both the mean velocity field and the turbulent flow field induced by the presence of the turbine. They provide a systematic data set which may be used by numerical modellers to benchmark their results against for the simplified case of a single turbine in a confining flume (blockage ratio = 16%). Determination of the Reynolds stress anisotropy tensor confirms the strong degree of turbulence anisotropy induced by the blade rotation and suggests the use of turbulence models based on isotropy (such as the $k - \epsilon$) be avoided. Finally the rate of decay of the maximum turbulent kinetic energy (well fit by a power-law with exponent -0.94) is significantly different to that observed downstream of grids, meshes or perforated disks, suggesting that previous modelling approaches, which neglected swirl effects and modelled the turbine by absorption discs, may significantly over predict the TKE decay rate of HATT wakes. Data at further downstream distances than could be obtained in the current experimental facility are required to fully test this hypothesis.

Acknowlegements

The authors would like to thank A. Mason-Jones, D.M. O'Doherty, C.E. Morris, T. O'Doherty, C.B. Byrne, P.W. Prickett and R.I. Grosvenor for design and production of the turbine. Also, we would also like to thank H. Eyre and T. McCormick for help with ADV measurements.

References

[1] Department of Energy and Climate Change . UK Renewable Energy Roadmap. Tech. Rep.; 2011.

[2] Myers LE, Bahaj AS. Experimental analysis of the flow field around horizontal axis tidal turbines by use of scale mesh disk rotor simulators. Ocean Eng 2010;37(2-3):218–27.

[3] Sun X, Chick JP, Bryden IG. Laboratory-scale simulation of energy extraction from tidal currents. Renewable Energy 2008;33(6):1267–74.

[4] Harrison ME, Batten WMJ, Myers LE, Bahaj AS. Comparison between CFD simulations and experiments for predicting the far wake of horizontal axis tidal turbines. IET Renewable Power Generation 2010;4(6, Sp. Iss. SI):613–27.

[5] Batten WMJ, Harrison ME, Bahaj AS. Accuracy of the actuator disc-RANS approach for predicting the performance and wake of tidal turbines. Philos Trans Royal Soc 2013;371:20120293.

[6] Myers L, Bahaj AS. Near wake properties of horizontal axis marine current turbines. In: Eighth European Wave and Tidal Energy Conference. Uppsala, Sweden; 2009,.

[7] Stallard T, Collings R, Feng T, Whelan JI. Interations Between Tidal Turbine Wakes: Experimental Study of a Group of 3-Bladed Rotors. In: Ninth European Wave and Tidal Energy Conference. Southampton, UK; 2011,.

[8] Stallard T, Collings R, Feng T, Whelan J. Interactions between tidal turbine wakes: experimental study of a group of three-bladed rotors. Philos Trans Royal Soc 2013;371:20120159.

[9] Rose S, Good A, Atcheson M, G. H, Johnstone C, MacKinnon P, et al. Investigating Experimental Techniques for Measurement of the Downstream Near Wake of a Tidal Turbine. In: Ninth European Wave and Tidal Energy Conference. Southampton, UK; 2011,.

[10] Maganga F, Germain G, King J, Pinon G, Rivoalen E. Experimental characterisation of flow effects on marine current turbine behaviour and on its wake properties. IET Renewable Power Generation 2010;4(6, SI):498–509.

[11] Mycek P, Gaurier B, Germain G, Pinon G, Rivoalen E. Numerical and Experimental Study of the Interaction Between Two Marin Current Turbines. In: Ninth European Wave and Tidal Energy Conference. Southampton, UK; 2011,.
[12] Rose S, Ordonez S, Lee KH, Johnstone C, Jo CH, McCombes T, et al. Tidal Turbine Wakes: Small Scale Experimental and Initial Computational Modelling. In: Ninth European Wave and Tidal Energy Conference. Southampton, UK; 2011,.
[13] Carbon Trust . United Kingdom Wave and Tidal Energy Study. Variablity of UK marine resources. Final Report; 2005.
[14] Daly T, Myers LE, Bahaj AS. Modelling of the flow field surrounding tidal turbine arrays for varying positions in a channel. Philos Trans Royal Soc 2013;371:20120246.
[15] Adaramola MS, Akinlade OG, Sumner D, Bergstrom DJ, Schenstead AJ. Turbulent wake of a finite circular cylinder of small aspect ratio. J Fluid Struct 2006;22(67):919 –28.
[16] Vermeer LJ, Sorensen JN, Crespo A. Wind turbine wake aerodynamics. Prog Aerosp Sci 2003;39(6-7):467–510.
[17] Devarakonda R, Humphrey JAC. Experimental study of turbulent flow in the near wakes of single and tandem prisms. Int J Heat Fluid Flow 1996;17(3):219–27.
[18] Wussow S, Sitzki L, Hahm T. 3D-simulation of the turbulent wake behind a wind turbine . In: Science of Making Torque from Wind; vol. 75 of *J. Phys. Conf. Ser.* 2007, p. 12033.
[19] Egarr DA, O'Doherty T, Morris T, Ayre RG. Feasibility study using Computational Fluid Dynamics for the use of a turbine for extracting energy from the tide. In: 15th Australasian Fluid Mechanics Conference. The University of Sydney, Sydney, Australia; 2004,.
[20] Mason-Jones A. Performance assessment of a Horizontal Axis Tidal Turbine in a high velocity shear environment. Ph.D. thesis; Cardiff University; 2010.
[21] O'Doherty T, Mason-Jones A, O'Doherty DM, Byrne CB, Owen I, Wang Y. Experimental and Computational Analysis of a Model Horizontal Axis Tidal Turbine. In: Eighth European Wave and Tidal Energy Conference. Uppsala, Sweden; 2009,.
[22] Mason-Jones A, O'Doherty DM, Morris CE, O'Doherty T, Byrne CB, Prickett PW, et al. Non-dimensional scaling of tidal stream turbines. Energy 2012;44(1):820 –9.
[23] Tedds SC, Poole RJ, Owen I, Najafian G, Bode SP, Mason-Jones A, et al. Experimental Investigation Of Horizontal Axis Tidal Stream Turbines. In: Ninth European Wave and Tidal Energy Conference. Southampton, UK; 2011,.
[24] Millward A, Rossiter J. The Design of a Multi-purpose Multi-component Strain Gauge Dynamometer. Strain 1983;19(7):27–30.
[25] Lohrmann A, Cabrera R, Kraus NC. Acoustic-Doppler velocimeter (ADV) for laboratory use. In: C.A. Pugh , editor. Fundamentals and advancements in hydraulic measurements and experimentation. 1994,.
[26] Choi KS, Lumley JL. The return to isotropy of homogeneous turbulence. J Fluid Mech 2001;436:59–84.
[27] Lumley J. Computational modelling of tubulent flows. Adv Appl Mech 1978;18:123–75.
[28] Pope SB. Turbulent Flows. Cambridge University Press; 2011.
[29] Simonsen AJ, Krogstad PA. Turbulent stress invariant analysis: Clarification of existing terminology. Phys Fluids 2005;17(8):88–103.