

Experimental and Computational Analysis of a Model Horizontal Axis Tidal Turbine

T O'Doherty¹, A Mason-Jones¹, D M O'Doherty¹, C B Byrne¹
I Owen² and Y X Wang²

¹Cardiff School of Engineering,
Cardiff University,
Queen's Buildings, The Parade, Cardiff, CF24 3AA, UK
E-mail: mason-jonesA@cf.ac.uk; odoherty@cf.ac.uk;
odohertydm@cf.ac.uk; byrne@cf.ac.uk

² Department of Engineering,
University of Liverpool,
Liverpool L69 3GH
E-mail: I.Owen@liverpool.ac.uk; YX.Wang@liverpool.ac.uk

Abstract

Tidal stream turbines provide a predictable and sustainable source of energy. They can be sized to suit the requirements of the local environment, and can be placed in either an individual or 'farm' configuration. The work described in this paper provides CFD validation data from a series of laboratory tests undertaken on a scaled model of a horizontal axial tidal turbine (HATT). The laboratory tests used a 0.5 m diameter three bladed turbine in a water flume which had a uniform flow profile with a magnitude of 1 m/s (~2 knots). Experimental data for power and torque were generated using a Baldor servomotor, load and control system. The motor's speed and torque were controlled and logged. The power and torque data are compared to that produced from a series of CFD models of the same turbine, rotating over the full range of angular velocities with the flume boundary conditions, using the software package FLUENT. The comparative study shows that the CFD models provide excellent predictions of power and torque.

Keywords: CFD, HATT, power, torque, water flume

Nomenclature

a	= swept area (m ²)
C _p	= power coefficient
C _t	= axial thrust coefficient
F _a	= axial thrust (N)
I	= current (A)
P	= power (W)
R	= turbine radius (m)

s	= percentage of the rated rotational velocity for the motor (%)
t	= time (s)
T	= torque (Nm)
v	= velocity (m/s)
ρ	= density (kg/m ³)
ω	= angular velocity (rad/s)

Acronyms

BEM	= beam element method
CFD	= computational fluid dynamics
HATT	= horizontal axis tidal turbine
MRF	= moving reference frame
RANS	= Reynolds Averaged Navier-Stokes
RSM	= Reynolds Stress model
TSR	= Tip Speed Ratio ($\omega R/v$)
UDF	= user defined function

1.1 Introduction

The emerging technologies of tidal energy generation provide two very different approaches. Electricity can either be generated through impoundment schemes, such as barrages and lagoons, or directly from the tidal current. Tidal energy can provide a highly predictable and sustainable level of energy that is dependent on the tidal cycle.

In contrast to barrage and tidal lagoons, tidal stream turbines use the kinetic energy of the tide directly. They are fully submerged below the water level and thus do not provide a visual obstruction to the landscape. They can be seabed mounted for example via a pile driven stanchion or floated to a desired depth through the water column. There are a number of devices currently under development that fall into a number of categories, such as the horizontal and vertical axis turbines. Others include venturi devices that can be used to concentrate the flow and oscillating hydrofoils that move up and down through

the water column generating electricity via the pumping of hydraulic fluids. Unlike the impoundment schemes, tidal stream turbines allow the water to pass through, and around them and do not require water to be stored.

Much of the technology associated with HATTs is derived from the wind industry, however the medium in which they operate produces higher structural loading with the addition of biological fouling from marine life, increased material corrosion from salts and the possibility of blade cavitation at shallower water depths. As a result the design criterion for a HATT requires a high degree of robustness with a limited maintenance schedule to reduce both operational cost and embodied CO₂ emissions [1].

Of the 382.5 TWh of electricity demand in the UK [2], tidal stream resource has the potential to generate 15.6 TWh or approximately 4% of the UK electricity demand. However, this figure is slightly reduced by the 'significant impact factor, (SIF) which is defined as 'the percentage of the total resource that can be extracted without significant economic or environmental effect, to give the available resource' [3]. The resulting figure of 12 TWh/y represents the UK tidal stream resource that could be economically exploited if the technology were to be fully developed and deployed.

To date UK tidal stream technology has resulted in a number of installed full scale devices. Marine Current Turbines (MCT) introduced the world's first offshore tidal stream turbine, the 11 m diameter twin bladed Seaflow, which was built into the seabed 1.5 km off shore from Lynmouth, Devon. It is capable of producing 300 kW of electricity at a tidal flow of about 2.8 m/s (5.5 knots) [4]. MCT has also developed the more recent 1.2 MW SeaGen project at Strangford Lough off the coast of Northern Ireland. This should supply up to 1000 homes with electricity [2]. MCT have also commenced studies for a small array of 12 turbines off Foreland Point, North Devon.

Resulting from recommendations made by the House of Commons Science and Technology Select Committee in 2001 the European Marine Centre (EMEC) with its five tidal stream test sites was established. EMEC is situated 2 km offshore at the Fall of Warness, off Eday, Orkney and is fast becoming a major centre for the testing of tidal stream and wave devices [2,5]. The Dublin based OpenHydro Group Ltd have installed a 250 kW prototype Open centre turbine at the site as part of their plans to develop a deep sea application.

The clear advantage of tidal stream turbines is that they can be sized to suit the requirements of the local environment, i.e. coastal restrictions, tidal flow, tidal range, seabed topography, etc., and can be placed on either an individual or 'farm' configuration.

As such, no large civil works are required and this method would therefore be less disruptive to wildlife, marine activity (and possibly the coastline) and would not present a significant barrier to water transport. It has been stated that the ideal site for a tidal stream turbine is

to be within 1 km of the shoreline and at a depth of 20 to 30 m [6]. The ideal tidal speed is 2 to 3 m/s (between approx. 4 and 6 knots) as higher speeds can lead to blade loading problems [7].

Although various test site facilities have been established, there is limited public information on the detailed performance of tidal stream turbines. As with all engineering problems, particularly those on such large scales in hostile environments such as the sea, cost is often an issue. In order to reduce such capital expenditure associated with experimental iteration there is a need to numerically model the system, thus establishing a theoretical solution prior to producing large scale systems. However, validation of numerical models must be completed to ensure confidence in the design.

The aim of this work was therefore to provide validated data for a three bladed free stream tidal turbine. To date, modelling has taken place on diameters ranging from 0.5 m to 30 m. [8] In order to validate these models, a 0.5 m diameter laboratory scale turbine was constructed and tested in the water flume housed at Liverpool University. The blades used the Wortmann FX 63-137 profile as the basis of the design with the key geometric values being: length 0.19 m, tip and root chord lengths 0.029 and 0.075 m respectively and a twist of 34°. The blade design was originated using a mixture of an in-house BEM programme and CFD modelling. The flume data were then compared to a CFD model of the turbine operating within the flume.

2.1 CFD modelling

The CFD modelling described in this paper used the package FLUENT. Initially a reference model was created to investigate grid independency and establish the optimal blade pitch angle for the turbine. The reference model used the Reynolds Stress Model (RSM) to close the 'Reynolds Averaged Navier-Stokes' (RANS) equations that relate the Reynolds Stresses to the mean velocity gradients. This viscous model was chosen since it was expected that the rotation of the turbine, would introduce anisotropic turbulence and the RSM is recommended for such situations, [9].

The reference model, a rectangular channel 2.5 m (width) x 2.5 m (depth) x 20 m (length) was established. The boundary conditions used were a uniform velocity-inlet of 1 m/s with 5% turbulence intensity and a pressure-outlet. The side walls and the top surface were defined as zero stress boundaries whilst the floor was defined as no-slip. The turbine was positioned centrally in the vertical plane, a distance of 5 m from the inlet. The reference domain dimensions were chosen to ensure that the interference from the side walls was insignificant. This was determined iteratively with consideration to the wake characteristics.

The reference domain included two volumes, one to model the volume around the turbine and the second to

model the remaining volume of the rectangular channel. Following tests to ensure that grid dependency was not an issue and that the model was not unnecessarily computationally intensive, the meshed model included ~1 million cells around the turbine and a further 90 000 cells for the remaining flow field. The cell count was controlled by the number of cells initially generated while meshing the faces of each turbine blade and hub. A finer mesh density was placed towards the tip of the blade within the last 1/3 of the blade length. The upstream and downstream faces were meshed with increasing mesh densities. The turbine volume was modelled using a cylindrical Moving Reference Frame (MRF) volume with its axis of rotation through the centre of the hub to allow the angular velocity of the turbine to be varied. Given the complex shape of the blades and geometry between the blades and the hub, the MRF cylindrical volume was meshed with a tetrahedral hybrid scheme (Figure 1). The MRF volume was subtracted from the rectangular channel representing the remainder of the model.

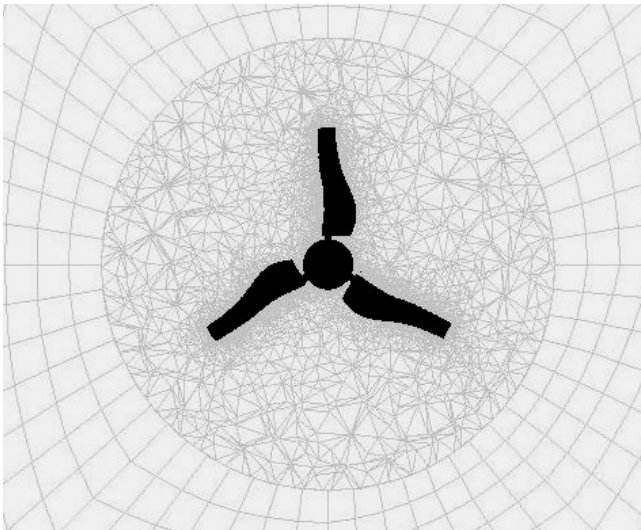


Figure 1 Tetrahedral MRF and quadrilateral channel mesh with non-conformal interface

Since it was known that the flume tests would utilize mains water, the fluid in the reference model was assumed to be the same. As such, a density of 1000 kg/m^3 and a constant viscosity of 0.00103 kg/ms at 20°C were used.

The work investigating the effect of blade pitch angle on the power and torque characteristics focused on addressing two aims; - the first being to determine the optimum blade pitch angle to maximize the power generated, and the second to evaluate the sensitivity of the power generated to the blade pitch angle. To achieve this, seven CFD models were created using the technique identified for the reference domain where the only difference was the blade pitch angle which was varied between 0° and 12° . The blade angle was defined as the

angle between the chord at the tip of the blade and the normal to the rotational axis of the turbine hub assembly. All the models were run for different angular velocities, up to the freewheeling velocity, with a uniform velocity profile of 1 m/s and 5% turbulence intensity. The torque, and consequently the power were calculated for each angular velocity, using a user defined function (UDF). In this way a torque and power curve could be plotted for each case, facilitating the comparison of the data at either peak power or peak torque.

2.2. CFD model of experimental conditions

In order to validate the CFD predictions, it was important to have a CFD model of the experimental conditions. Since all the experimental work was undertaken in the Liverpool University re-circulating water flume, the flume's working section was modelled. This CFD model was created using the same protocol as the reference domain model, and as such included the exact MRF turbine volume, previously described. The walls defining the boundaries were now defined to replicate the working section of the flume. In addition, the stanchion, keeping the turbine centrally within the working section, was added. This meant that the number of cells within the turbine volume was still ~ 1 million, but the number of cells in the remaining flow field was increased to ~250 000 in order to accurately incorporate the stanchion. The boundary conditions used were a velocity-inlet and a pressure-outlet which matched that of the experimental test conditions. These were a uniform velocity profile of 1 m/s with a turbulence intensity of 5% as this was typical for the flume. The walls and base were modelled using wall cells (i.e. no-slip boundaries) and the water surface boundary as a 'no free surface' with zero shear. The appropriateness of this boundary condition will be discussed later in the paper.

To determine if the RSM was indeed the best viscous model to use for this study, in terms of both accuracy and computational expense, five different viscous models were used to close the RANS equations. These were the Spalart-Allmaras, Standard $k-\epsilon$, RNG $k-\epsilon$ and Realizable $k-\epsilon$ models which all assume isotropic turbulence and the RSM which assumes anisotropic turbulence.

3.1 Experimental testing

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

uniform velocity profile ranging between 0.03 m/s and 6.4 m/s.

The conditions under which the experimental tests were made were 0.84 m water depth with a 1 m/s uniform velocity with a measured turbulence intensity, over the profile, of 5%. The maximum uncertainty in the velocity profile, measured by a pitot tube, was estimated to be better than 5% over the water depth. This was limited by the resolution of the instrumentation. The boundary layers were of the order of 16 mm at the walls. The centre of the turbine was located at a depth of 0.425 m, midway along the working section.

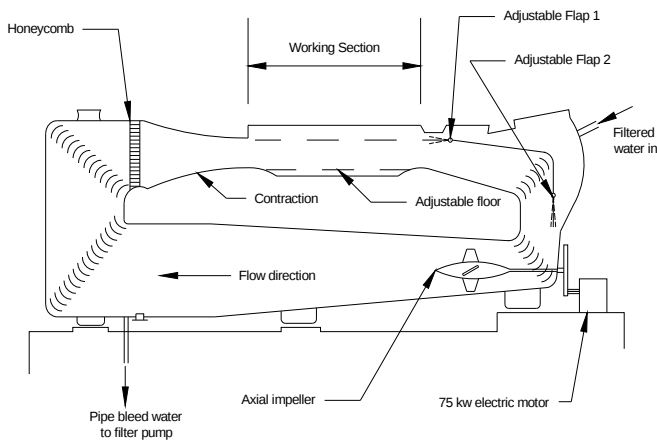


Figure 2 Schematic of the re-circulating water flume

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 load to that developed by the hydrodynamic forces from the turbine. This was combined with a control system which in turn was programmed via a laptop computer.

The drive coupling was taken from the back of the housing and since the servomotor was not waterproof it was necessary to locate it out of the water. The optimum blade pitch angle, for this turbine design, of 6°, as calculated from the CFD modelling was set using precision machined blocks and marking table. The optimum angled block was aligned with the chord at the tip of the blade and the hub assembly. The turbine was positioned mid way with the blades normal to the flow, Figure 3. The turbine blade and hub assembly were then attached to a cylindrical supporting stanchion which in turn was attached to a rectangular cross beam spanning the width of the water flume (1.4 m).

The assembly holding the support bar to the cross beam also housed the strain gauges used to measure the axial load exerted by the turbine throughout each angular velocity sweep. The strain gauges were zeroed while the flume water velocity was set to zero and a test file created on the strain logging computer. The water velocity within

the flume was then slowly increased to 1 m/s and the turbine allowed to freewheel with zero load applied by the servomotor.

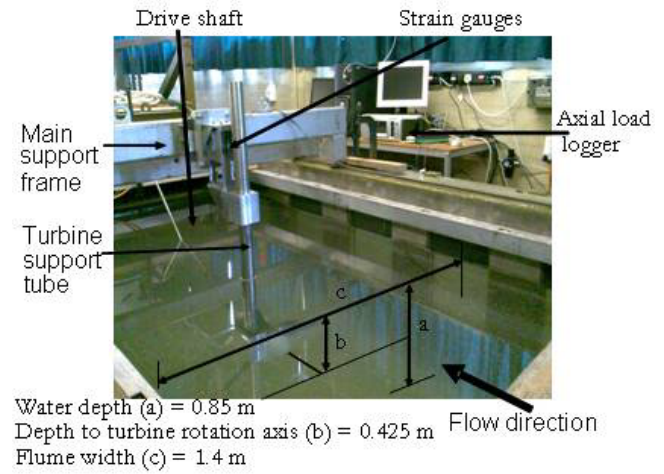


Figure 3 Turbine positioned in circulating water flume

The servomotor torque was controlled using MintMT in WorkBench v5 [10]. By using the motion-specific keywords contained in MintMT control over the speed, torque, interpolation and synchronization of multiple axes is obtained. For the flume tests the torque supplied to the turbine was controlled via the TORQUEREF command embedded in an incremental macro. Values between ± 100 give the torque demand as a percentage of the drive rated current. The rotational direction of the turbine was controlled via the sign of the value. For the torque macro the FOR loop was used with limits, incremental steps and the WAIT command between increments to allow the rotational speed of the turbine to stabilise once the next value of torque was applied.

The torque generated by the servomotor was proportional to the drive rated current, in this case 6 A (max). The current limit throughout the tests was set to 70% of the current limit giving a peak current of ~ 4.2 A. The relationship between current and torque for the servomotor was linear [10] with a proportionality constant of 0.906 (Nm/A) producing a peak torque of ~ 3.8 Nm. The rotational speed of the motor was measured as a percentage of the rated rotational velocity for the motor (1000 rpm).

Therefore, the angular velocity of the turbine was calculated from:

$$\omega = \frac{s}{100} \times \frac{1000}{60} \times 2\pi = s \frac{\pi}{3} \quad (\text{Equation 1})$$

3.2. Variation of optimal blade pitch angle

The optimal blade pitch angle, as defined by the reference CFD domain, was found to be 6° (Figure 4). As previously stated, this value was then used as the datum

blade pitch angle in the experimental tests. These tests were extended to include the angles of 3° and 9° in order to verify the CFD prediction. The turbine blades were set using angled machined spacers and a marking table. As with the optimal blade pitch angle, the edge of the angled spacer was aligned with the chord of the blade tip profile. As in the previous case for the calculated optimum blade pitch angle the servomotor and the turbine were allowed to freewheel. Using the same methodology as previously discussed the load was applied to the turbine via the servomotor.

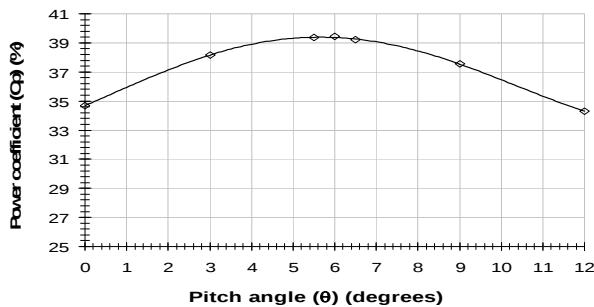


Figure 4 CFD predicted optimal blade pitch angle

4.1 Flume test results

With the blade angle set to 6°, the turbine was tested with the flume velocity set to 1 m/s. With no load applied to the turbine the rotational velocity is at a maximum and therefore freewheeling. The control circuit was activated and the load applied. The load test data starts at the high angular velocity, low torque end of the curve. As the load is applied the torque applied by the servomotor starts to reduce the angular velocity of the turbine.

From F it is clear that there is some scatter in the angular velocity data across each sample period. This could be due to proximity of the stanchion to the turbine blades as discussed later in the paper in section 6.1. The angular velocity (%) reduces as the load current is increased to the servomotor. This is demonstrated by the downward trend from left to right as the turbine slows from freewheeling at the start of the test.

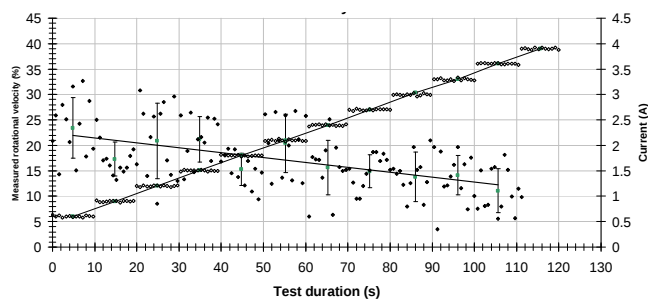


Figure 5 Angular velocity (%) with standard deviations and curve fit and servomotor current (A) showing average point for each sample period

Figure 5 also gives the measured servomotor current for a sample period, here however there is considerably less scatter in the data, indicating a stable load current. Initially a data sampling period was determined to ensure that steady state was achieved for each data set. This time period was reduced from 30 to 5 seconds, all of which provided steady state conditions, with no significant change in the level of scatter. For the data used in this work the sample period used was 10 seconds before the current was increased.

Since the data were collected at 10 second intervals for each angular velocity setting, the raw data defined by Equation 1 were averaged for each individual time period. A curve fit was then applied to smooth the data for each sample period in order to average the averaged angular velocity data. For each of the tests the curve fit for the average angular velocity is given by Equation 2, as shown in Figure 5, where 23.5 represents the freewheeling angular velocity.

$$\omega(\%) = -0.01(t) + 23.5 \quad (\text{Equation 2})$$

To allow comparisons between the flume tests and the CFD flume model Equations 2 and 3 were used to calculate the average angular velocity (%) and torque (Nm) for each test.

$$T = I(0.906) \quad (\text{Equation 3})$$

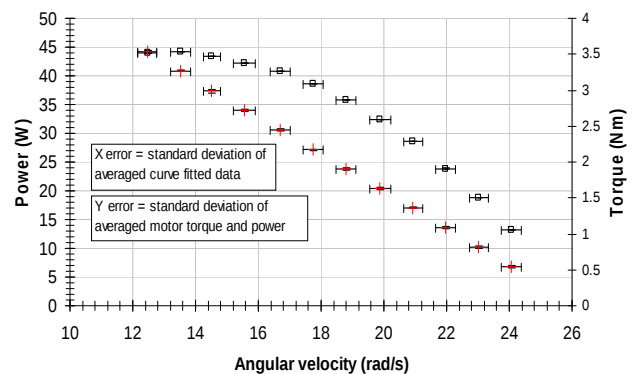


Figure 6 Torque (red) and power (black) versus angular velocity

The data from each individual sample interval were then combined to generate torque and power curves for each test, shown in Figure . As the angular velocity of the turbine decreases the torque generated by the hydrodynamic lift forces start to increase towards a maximum torque of 3.5 Nm and an angular velocity of approximately 10 rad/s. The torque given in Figure represents the servomotor torque and not that directly generated by the turbine, however as the angular velocity stabilises the torque generated by the turbine equals that generated from the servomotor. In this way the torque

curves generated for each test are linear since they directly match the output of the motor and not the torque characteristics that would be generated by the turbine from lift and drag forces via hydrodynamic interaction between the water and blade profiles.

The linearity of the torque curve is shown in Figure 6. As the turbine slows from the freewheeling state, at around 24 rad/s, the hydrodynamic forces generated from the lift characteristics of the turbine profile start to increase. The servomotor torque is gradually increased until it matches the maximum torque generated by the turbine for the 1 m/s water velocity. Below an angular velocity of 13.5 rad/s the rotational direction of the turbine alters as the servomotor torque overcomes the hydrodynamic torque generated by the turbine. At this point the turbine starts to oppose the direction of flow and pump the water in the opposite direction.

The combined torque curves for each of the 6 tests show some variance in the point at which the rotational direction of the turbine switches. For the combined test data the average switch point for the rotational direction occurred at an average torque of 3.36 ± 0.22 Nm and an average angular velocity of 6.5 ± 3 rad/s. For each flume test the average maximum torque generated from the servomotor, before switching the rotational direction, was 2.81 ± 0.41 Nm at an angular velocity of 10.9 ± 3.1 rad/s.

The power generated was calculated from the product of the averaged torque and average of the curve fitted angular velocity data, i.e. $T\omega$ (W), Figure . Since the torque is increased from a freewheeling state the corresponding power curve also starts from a high angular velocity of 24 rad/s. The power curve has a peak occurring at a higher angular velocity than that of the maximum torque.

The power was non-dimensionalised, using Froude's momentum theory [11] to define the power coefficient C_p , where

$$C_p = \frac{T\omega}{0.5\rho av^3} \quad (\text{Equation 4})$$

Table 1 Summarised results for flume tests.

	Test							
	1	2	3	4	5	6	ave	sd
$T_{\max}$ (Nm)	3.27	2.99	3.53	3.54	3.54	3.54	3.36	0.22
T at $P_{\max}$ (Nm)	3.27	2.99	3.26	2.45	2.44	2.45	2.81	0.41
$P_{\max}$ (W)	44.4	45.2	41.3	39.9	35.7	32.7	39.8	4.85
C_p	0.45	0.46	0.42	0.41	0.36	0.33	0.41	0.05

The tests were repeated a number of times in order to build up a series of data sets. Each test was performed with the same boundary conditions and an average for the

torque, power and power coefficient determined. Table 1 shows a summary of the torque, power and power coefficient for all the tests, including the overall average values and associated standard deviation. The combined power curves from the flume tests give a maximum average power of 39.8 ± 4.85 W at an angular velocity of 14.3 ± 1.3 rad/s and an average peak torque of 3.4 ± 0.2 Nm. What is clear is that the scatter found for the angular velocity, shown in Figure 5, is present for all tests.

Data at lower angular velocities were difficult to collect due to the fact that the turbine would reach stall at an angular velocity <12 rad/s.

4.2. Effects of blade angle.

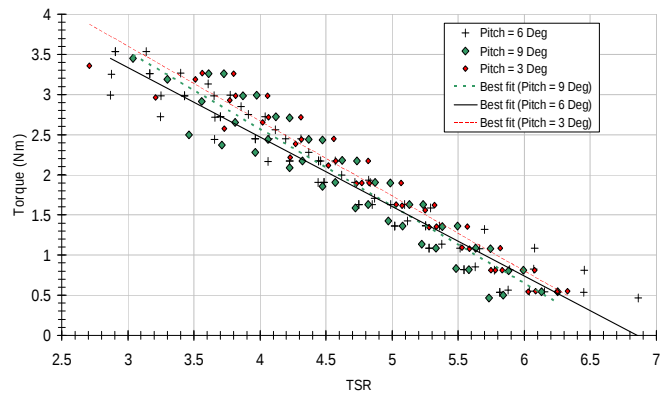


Figure 7 Torque curves for 3°, 6°, 9° blade angles.

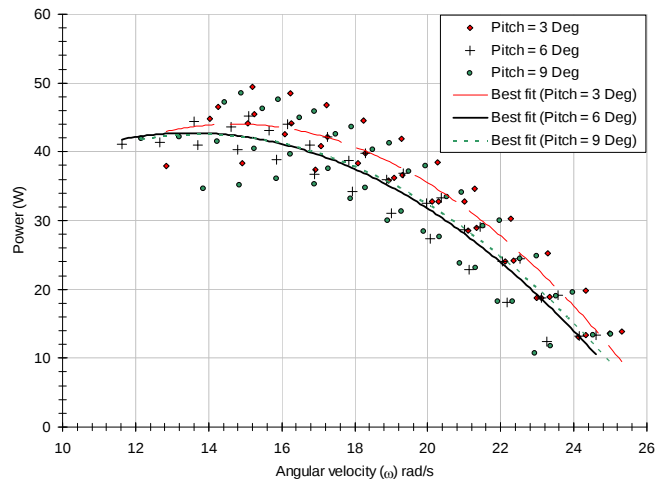


Figure 8 Turbine power at 3°, 6° and 9° blade angles

Using the methodology discussed previously, the blade pitch angle for the turbine was changed to 3° and 9° with the flume water velocity maintained at 1 m/s. The best fit angular velocity was again used to smooth the fluctuations as previously discussed. The power is calculated as before using the measured current and the linear relationship between current and torque (0.906 Nm/A). As in the 6° tests, torque curves were generated

for each blade angle. To give insight to the general trend a best fit curve was added to each combined 3°, 6° and 9° data sets, Figure 7. From Figure 8 it is clear that there is variability between the power curves for each test.

5.1 CFD results

The CFD model of the reference domain was used to predict the optimal blade angle. The results (Figure 4) clearly show that the power coefficient is a maximum when the blade pitch angle is 6° and that the power coefficient is not highly sensitive to the blade pitch angle. By varying the optimal angle by $\pm 0.5^\circ$, the C_p value only fell by 0.5% ($C_p \approx 0.398$) and fell a further 4.5% ($C_p \approx 0.38$) when the optimal angle was changed by $\pm 3^\circ$.

The CFD model of the flume was used to correlate the flume test results. The torque and power curves were extracted using the power UDF. The results of this study are shown in Figures 9 and 10 for torque and power against angular velocity respectively. Both figures contain a series of curves each represented by a different viscous model to close the RANS equations. Using this method it is possible to justify the use of the RSM model when anisotropic turbulence and swirl need to be considered. All of the viscous models utilized use the Reynolds averaging methodology and all except the RSM model assume that the turbulence characteristics of the flow field are isotropic.

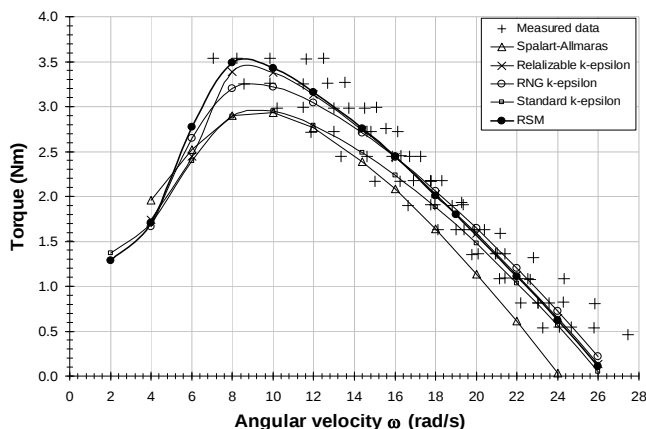


Figure 9 Comparison between measured and predicted torque.

In general, all the models showed good agreement with the measured data, at least at some point through the torque and power curves. The average maximum measured torque during the flume tests equated to 3.36 ± 0.22 Nm. From the CFD model using different viscous models the closest correlations were made with the use of the Realizable k- ϵ , RNG k- ϵ and RSM with a maximum torque of 3.39 Nm, 3.22 Nm and 3.49 Nm, respectively.

The average torque at maximum power for the flume tests was 2.81 ± 0.41 Nm. The Spalart-Allmaras and Standard k- ϵ CFD models gave the weakest correlation at 2.39 Nm and 2.49 Nm respectively, whilst the RSM

results provided the best correlation with a predicted torque of 2.75 Nm.

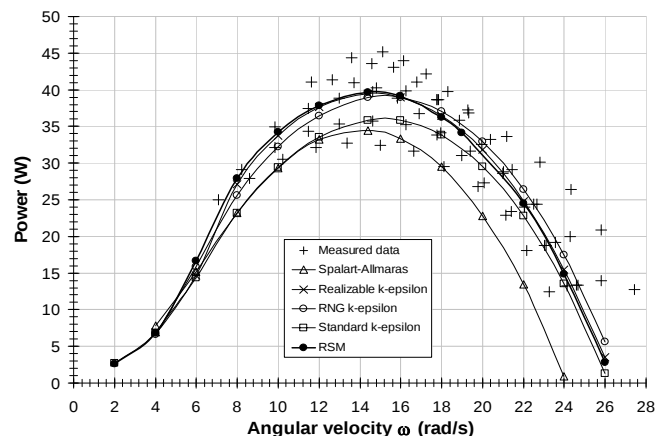


Figure 5 Comparison between measured and predicted power

The respective average power was calculated to be 39.82 ± 4.85 W. The correlation between the CFD predictions and the experimental data for the maximum power is in the same order as seen for the maximum torque, with the RSM predicting a power of 40.4 W. As before the Spalart-Allmaras and Standard k- ϵ viscous models show slightly lower power extraction at 34.39 W and 36.52 W. Due to the interrelation of the parameters the viscous model pattern extends into the power coefficient as calculated from the measured flume data and the CFD model. The average power coefficient for the flume tests equated to 0.41 ± 0.05 and 0.404 from the RSM predictions.

In general, even though there are differences in their formulation, all the viscous models correlated reasonably well with the measured flume data, in terms of the trends. In terms of the power curves all the viscous models have a good correlation with the RSM except for the Standard k- ϵ and Spalart-Allmaras models. Differences in the shape of the torque and power curves are apparent over the complete angular velocity sweep. Starting from the freewheeling state, all of the viscous models, except for the Spalart-Allmaras, correlate well down to an angular velocity of approximately 18 rad/s. Below this value the Standard k- ϵ model starts to diverge with a maximum divergence coinciding with the point of maximum torque where it then converges with the peak torque as given by the Spalart-Allmaras model.

Both the RNG k- ϵ and Realizable k- ϵ models track the RSM curve very closely up to 12 rad/s and 8 rad/s, respectively. The largest difference between these two models and the RSM is demonstrated by the RNG k- ϵ model around the position of maximum torque. As the turbine slows however the RNG k- ϵ model starts to converge with the RSM and Realizable k- ϵ model at approximately 6 rad/s. As the torque curves approach zero angular velocity the torque reduces to the stationary torque of the turbine for the given flow conditions, for the

RSM this is approximately 1.2 Nm. Table 2 summarises the results for each of the viscous models.

It should be noted at this point that the prediction of the axial thrust on the turbine is ~ 94.5 N, which gives a C_T at freewheeling value of ~ 0.97 . An initial value recorded during the experimental work, using strain gauges, gave the axial thrust value of ~ 96.2 N and a C_T at freewheeling of ~ 0.98 . Obviously since the axial thrust is a function of density then the forces are particularly large. However, since the axial force is also a function of water velocity, (v^2) then the force will become significant as the water velocity increases.

Table 2 Summary of CFD flume models

	Spalart-Allmaras	Realizable k - ϵ	RNG k - ϵ	Standard k - ϵ	RSM	Exptl
T_{max} (Nm)	2.93	3.39	3.22	2.95	3.49	3.36
T at P_{max} (Nm)	2.39	2.74	2.70	2.49	2.75	2.81
P_{max} (W)	34.39	39.41	39.67	36.52	40.4	39.8
C_p	0.350	0.401	0.397	0.365	0.404	0.41
Axial thrust at P_{max} (N)	81.71	83.11	81.50	80.46	82.65	-
C_T at P_{max}	0.832	0.847	0.830	0.819	0.842	-
F_a at freewheeling (N)	94.60	94.79	96.79	96.37	94.46	96.2
C_T at freewheeling	0.964	0.966	0.986	0.982	0.962	0.98

6.1 General Discussion

The recirculating flume with its well defined velocity profile, known level of turbulence intensity and size of the boundary layer in the working section provided a test facility that could be easily replicated in the CFD model of the working section. The only parameter not included in the CFD model was the free surface interaction. Although it is recognised that this omission could have an effect on the flow, its effect is depth dependent and its effects will be reduced the greater the distance between the free surface and the turbine blade tip. Therefore by placing the centre of the turbine 0.425 m below the surface, the tip of the blades was 0.175 m below the surface (minimum distance). It was therefore assumed that this distance was large enough for the surface effects to be minimal. The results show this assumption to be reasonable for the conditions described in this paper, however, if the inlet velocity was increased the blockage effects of the turbine may cause substantial surface interaction as indicated by Bahaj et al [12]. This is an issue that will need further investigation.

On inspection of the experimental results it is clear that the measurement of the torque provided an accurate data set with an average of 3.36 Nm and a standard deviation over the tests of 0.22 Nm. The same level of accuracy could not be said of the measurement of the angular velocity where there is a definite scatter. The accuracy of

the torque is due to the fact that this is a measure of the applied torque from the servomotor. The scatter found with the angular velocity could be due to either the resistance applied to the turbine via the shaft connecting to the servomotor or the proximity of the blades to the stanchion. The shedding from the blades would have resulted in an interference with the stanchion as they passed in front of it. What is interesting is the fact that when a best line fit is applied to the angular velocity data and then used to calculate the power, the data can be seen to provide a good comparison, i.e. a maximum power of 39.8 W against a predicted (RSM) value of 40.4 W. Whilst the validation of the CFD models are reasonable there is evidence that in order to reduce the scatter in the experimental data, particularly the angular velocity the blades should be set clear of the stanchion. Hau, [13] suggests that for the wind turbine industry that the distance between the stanchion and the rotor should be around 1 tower diameter upstream of the tower where the relative free stream velocity is around 0.9 of the upstream velocity. Using this methodology, i.e. achieving a 0.9 ratio the separation between the turbine blades and the stanchion should be approximately 3 stanchion diameters. In fact the separation between the turbine blades and the stanchion on the experimental model was one only stanchion diameter. Mason-Jones [8] modelled a 10 m diameter turbine with and without the interference of a stanchion. He found that there were definite fluctuations in the power extracted over a complete rotation of the turbine when a stanchion was in close proximity, which did not exist when there was no interference from the stanchion. Mitigation of the scattering of the data may also be produced by integrating the servomotor with the blade hub, so providing a direct drive from the turbine shaft into the motor.

Examining the CFD data shown in Table 2 it is clear that the Realizable k- ϵ model and the RSM are the most reliable for this work, providing very close global results which are comparable to the experimental data. The RSM has been primarily used for this work due to its capability to reasonably model anisotropic turbulence and flow separation from the turbine blades.

There is a small discrepancy between the experimental and CFD data in the power produced, as a result of the blade angle. Figure 4 shows that whilst there is only a small difference between 3° and 9° , the maximum power occurs at 6° . The experimental data, using the averaged values of torque and angular velocity results in the maximum power being derived at 3° . The differences in the maximum values are, however, also showing only small differences which are in agreement with the CFD data. This provides an interesting point which is that this turbine is reasonably insensitive to the blade angle over this range

7.1 Conclusions

There is good correlation between CFD and experimental data for power and torque curves.

Whilst limited data was collected, the axial thrust at freewheeling also showed a good correlation.

The viscous models all performed well at predicting the power characteristics, but the higher order turbulence models captured the full operational range of the measured power and torque curves better, with the RSM being the preferred model.

The predicted C_p for this blade design of ~ 0.4 was confirmed by the experimental study which also confirmed the relative insensitivity to the blade pitch angle, within the discussed range of $3^\circ - 9^\circ$.

Acknowledgements

This research was funded by a European Regional Development Fund grant through the Welsh Assembly Government and the Knowledge Exploitation Fund.

References

- [1] C.A. Douglas, G.P. Harrison and J.P. Chick. Life cycle assessment of the Seagen marine current turbine. Proc. IMechE Vol. 222 Part M: *J. Engineering for the Maritime Environment* (2007).
- [2] Sustainable Development Commission. *Turning The Tide, Tidal Power in the UK*, 2007. <http://www.sd-commission.org.uk/publications.php?id=6072007>. [March 2009].
- [3] Black and Veatch – Phase II UK Tidal Stream Energy Resource Assessment. Carbon Trust, 2005
- [4] Department for Trade and Industry *SEAFLOW Marine Current Turbines Summary Report*. Report PS244. DTI, London 2006.
- [5] T. O'Doherty, D. A. Egarr, A. Mason-Jones and D. M. O'Doherty. An Assessment of Axial Loading on a 5 Turbine Array. *Proceedings of the Institution of Civil Engineers, Energy*, 2009, 162, No. 2 .
- [6] Fujita Research. *Wave and Tidal Power*, Web version July 2000, www.fujitaresearch.com/reports/tidalpower.html. [March 2009].
- [7] C. Soares. Tidal Power: the next wave of electricity, 2002, *Pollution Engineering*, www.pollutionengineering.com. [March 2009].
- [8] A. Mason-Jones. Assessment of Tidal Stream Turbines in a High Shear Environment. PhD Thesis, Cardiff University, 2009
- [9] Fluent Inc., Fluent Documentation, 2006.
- [10] Baldor motion products. User manual. MintMT Help files. Version 2005-02 ©Baldor 2005
- [11] J. Carlton, (2007), *Marine Propellers and Propulsion*. Butterworth-Heinemann, ISBN 0750681500, 9780750681506
- [12] A.S. Bahaj, A.F. Molland, J.R. Chaplin, 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*. January 2006. 32(2007)407-426
- [13] E. Hau. Wind turbines: fundamentals, technologies, application, economics. Edition 2. ISBN 3540242406, 9783540242406 (2006).