<table>
<thead>
<tr>
<th>Title</th>
<th>A two and three-dimensional CFD investigation into performance prediction and wake characterisation of a vertical axis turbine</th>
</tr>
</thead>
<tbody>
<tr>
<td>Author(s)</td>
<td>Mannion, Brian; Leen, Sean B.; Nash, Stephen</td>
</tr>
<tr>
<td>Publication Date</td>
<td>2018-05-24</td>
</tr>
<tr>
<td>Publisher</td>
<td>AIP Publishing</td>
</tr>
<tr>
<td>Link to publisher's version</td>
<td><a href="https://doi.org/10.1063/1.5017827">https://doi.org/10.1063/1.5017827</a></td>
</tr>
<tr>
<td>Item record</td>
<td><a href="http://hdl.handle.net/10379/15595">http://hdl.handle.net/10379/15595</a></td>
</tr>
<tr>
<td>DOI</td>
<td><a href="http://dx.doi.org/10.1063/1.5017827">http://dx.doi.org/10.1063/1.5017827</a></td>
</tr>
</tbody>
</table>
A two and three-dimensional CFD investigation into performance prediction and wake characterisation of a vertical axis turbine.

Brian Mannion*1,2, Seán B. Leen1,2 & Stephen Nash*1,2

1College of Engineering & Informatics, National University of Ireland Galway, Ireland
2SFI Centre for Marine Renewable Energy Ireland.

Abstract

The emergence of tidal energy as a key renewable energy source requires the development of computational design models for accurate prediction of turbine performance and wake effects whilst also being computationally efficient. In this paper, we develop and validate a three-dimensional CFD model for vertical axis turbines, which achieves high accuracy. We also investigate the limitations of two-dimensional models and present a blockage correction for improved prediction. The two-dimensional blockage correction model is potentially attractive for preliminary design studies due to its computational advantage over three-dimensional models.

1. Introduction

In recent times, there has been a significant increase in research and development of efficient tidal stream energy converters. This involves physical and numerical modelling at various scales and for different prototype designs (Nash and Phoenix, 2017). Laboratory testing of turbines is expensive, and suitable facilities are relatively sparse. Field testing is even more expensive and is further hampered by the difficulties of working in the highly dynamic marine environment. Numerical modelling offers a cost-effective alternative for assessment and optimisation of prototypes at any scale. A critical issue with numerical modelling is computation cost, with higher levels of accuracy incurring significant computational costs. This research investigates the ability of two-dimensional (2D) and three-dimensional (3D) models to accurately simulate energy extraction and resulting impacts of a vertical axis tidal turbine (VATT).

In terms of turbine performance modelling, and specifically in relation to vertical axis turbines (VAT), several candidate modelling techniques can be utilised. Many of these techniques were initially developed for wind turbine modelling. Blade element momentum theory (BEMT) is a technique that was pioneered by Glauert (1926), Strickland (1975) and Templin (1974). It combines blade element and momentum theories via the use of actuator discs and stream-tubes. The advantage of this technique is that it has low computational cost compared to other modelling techniques such as computational fluid dynamics (CFD). BEMT models allow for rapid evaluation of turbine design iterations. Most BEMT codes are based on an iterative approach in determining the local axial induction factor and rely upon the use of experimental aerofoil data or data that has been predicted using a panel method (e.g. XFOIL®); see Sheldahl and Klimas (1981) for example. The BEMT method is most effective with low blade loadings. For high loadings, this iterative scheme can lead to convergence difficulties (Klimas and Sheldahl, 1978; Paraschivoiu, 1983; Gupta and Leishman, 2005). An alternative method to BEMT is the free-vortex model. It is based on the replacement of the
aerofoil blade with a bound vortex filament, called a lifting line, that changes its strength as a function of azimuthal position (Ponta and Jacovkis, 2001). The first of these free-vortex models applied to VATs was carried out by Strickland et al. (1979). Blade forces are calculated within the free-vortex model using the blade element method (BEM) based on experimental aerofoil data (e.g. Sheldahl and Klimas (1981)), and the forces are applied with knowledge of blade local velocity vectors. Li and Çalışal (2010) used a vortex model to accurately reproduce the vertical axis wind turbine performance test data of Templin (1974); however, when the same model was used to simulate power performance data from tow tests of a scale model VATT conducted by University of British Columbia (UBC) it resulted in a 20 % (approximately) over-prediction of the peak power coefficient. Thus, while the computation times of free-vortex models are a fraction of CFD models, their accuracy is lower. One of the causes for inaccuracy is that they use a bound vortex to represent the blade and thus are incapable of modelling near blade effects (Li and Çalışal, 2010).

The most common modelling method for VATs is computational fluid dynamics (CFD), which offers a large variety of techniques that can be used for modelling turbines. The governing equations of flow in CFD models are the Navier-Stokes equations. One primary difference between these CFD approaches is how turbulence is modelled. The Reynolds-averaged Navier-Stokes (RANS) approach is time-averaged and requires a turbulence model to close the RANS equations. For engineering applications, the most commonly used turbulence models include the Spalart-Allmaras, \( k-\varepsilon \), \( k-\omega \) and shear stress transport (SST) models. The Large Eddy Simulation (LES) approach is based on filtering rather than time-averaging. All flow scales larger than the selected filter size will be calculated directly with scales smaller than the filter size being approximated. LES requires a highly-refined mesh and is therefore extremely computationally expensive. For this reason, the RANS approach is more commonly used and is employed in this research. The CFD software used is ANSYS® FLUENT®; which has been extensively used in turbine modelling studies. (Zanette et al. 2010; Ghosh et al. 2015)

Modelling of turbines can be done in steady-state or transient modes depending on the objectives of the research and the computational resources available. If computational resources are scarce, relatively simple, steady flow models can be used to model the turbine blades in different azimuthal positions and the results aggregated, e.g. Gupta and Biswas (2010). A more common approach is transient modelling of the moving blades through the use of the unsteady Reynolds Averaged Navier-Stokes (URANS) approach (Howell et al. 2010; Almohammadi et al. 2012; Biadgo et al. 2013; Mohamed, 2013; Lanzafame et al. 2014; Balduzzi et al. 2016). Although more complex, URANS is more accurate, particularly where blade interaction occurs. With regard to solution procedures, there is general agreement that 2nd order spatial discretisation schemes provide the most accurate results (Balduzzi et al. 2016). When it comes to selecting an appropriate solver, there does not seem to be any explicit agreement for vertical axis turbines and all of the following Fluent® solvers have been used: SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) (Beri and Yao, 2011), SIMPLEC (Semi-Implicit Method for Pressure-Linked Equations-Consistent) (Mcnaughton et al. 2014), PISO (Pressure-Implicit with Splitting of Operators)(Lanzafame et al. 2014) and COUPLED (pressure-velocity coupling method) (Balduzzi et al. 2016). Many different turbulence models have been employed in CFD RANS VAT models. These include the \( k-\varepsilon \) model (e.g. Mohamed, 2012a), the \( k-\omega \) model (e.g. Yang and Lawn, 2011), the \( k-\omega \) Shear
Stress Transport (SST) model (e.g. Lain and Osorio, 2010) and the Transition SST model (e.g. Lanzafame et al. 2014). It is generally accepted that Shear Stress Transport models give the best accuracy.

3D models of VATs tend to be quite similar in nature to 2D models, i.e. they tend to use the same discretisation schemes, turbulence models and mesh resolutions. For example, Orlandi et al. (2015) carried out a study of a VAT in misaligned flow using 3D URANS and a $k-\omega$ SST turbulence model, and Zamani et al. (2016) implemented 3D URANS in OpenFOAM to evaluate the enhanced mechanical performance of “J-blades” as compared to conventional NACA 0015 blades. Marsh et al. (2015) used 3D URANS with the $k-\omega$ SST turbulence model to investigate the performance of a three (straight) bladed vertical axis turbine with included strut supports. Other researchers that have carried out three dimensional CFD studies of VAT’s include: Howell et al. (2010); Li et al. (2013); Nini et al. (2014); Alaimo et al. (2015); Lam and Peng (2016). Levels of accuracy vary widely between model studies but for maximum power coefficient, differences of less than 10 % between measured and modelled values are rare. Marsh et al. (2012), for example, achieved agreement of only approximately 58 % at peak power in their 3D VATT model study.

With regard to 2D models, the representation of the channel blockage ratio (the ratio of the turbine frontal area to the channel cross-sectional area) can also affect model accuracy, particularly in cases where the blockage ratio is significant. For example, Bachant and Wosnik (2016b) over-predicted peak mechanical performance of a scaled VATT by approximately 96 % using a 2D model and attributed the over-prediction to the elevated blockage of the 2D model relative to the actual blockage of the scale model setup. By comparison, in a study where the blockage was not significant Balduzzi et al. (2016) achieved agreement within 15 % of peak performance for their 2D model of a scaled vertical axis wind turbine.

The primary aim of this research is to develop 2D and 3D CFD models of a three-bladed VATT and assess their levels of accuracy in prediction of power performance and wake properties. The turbine models are developed using the sliding mesh technique and model performance is assessed using the results of experimental tow tank testing of a scale model VATT conducted by Bachant and Wosnik (2016a and 2015). A secondary aim of this research is to test the hypothesis that over-prediction of power performance by 2D models is a result of their higher blockage ratio. This was achieved by testing a blockage correction approach for 2D models. A similar study was undertaken recently by Bianchini et al. (2017). In addition to the general convergence studies, 2D sensitivity studies are presented with respect to modelling decisions such as the diameter of the rotating domain, and specification of inlet parameters for turbulence intensity (TI) and turbulence viscosity ratio (TVR).

The key novelty of the paper is the detail of the modelling approach which is shown to achieve very high levels of accuracy. The approach is intended to provide a template for the development of highly accurate 3D models of vertical and horizontal axis tidal turbines. Particular focus is placed on mesh spacing, particularly in the boundary layer region, the size of the rotating mesh and the turbulence model. Turbine models can be validated against both power performance and wake characteristics; however, model developers tend to focus on one or the other. Of the three dimensional CFD studies of VATTs reviewed as part of this research, only Bachant and Wosnik (2016b) validate their model against both and, even then,
power performance validation was conducted for just a single TSR. The current research provides validation of the full power performance curve as well as downstream wake velocities and turbulent kinetic energy. A holistic approach like this is crucial, particularly with a view to subsequent turbine array modelling where both hydrodynamic impacts and power capture are intrinsically linked. An additional novelty of the research is the use of the Transition SST turbulence model which, based on a review of the literature, is a first in 3D modelling of vertical axis tidal turbines.

2. Methodology

2.1. Device Background and Experimental Testing

The turbine modelled in this research is a straight-bladed VATT built and tested by researchers at the University of New Hampshire (UNH). It is henceforth referred to as the UNH Reference Vertical Axis Turbine (UNH-RVAT). A simple schematic of the device is shown in Figure 1. The model was designed as a generic test case, which would produce an extensive set of measured data against which various numerical modelling approaches could be calibrated and validated. The UNH-RVAT turbine is 1 m in diameter and consists of three 1 m long NACA 0020 aluminium hydrofoils with a chord length of 0.14 m. These are connected to a 95 mm diameter aluminium central shaft via mid-span support struts. The struts are also NACA 0020 aluminium hydrofoils used to reduce restrictive drag. Table 1 presents a summary of device dimensions. The turbine has a relatively high solidity, \( Nc/\pi(RD) = 0.134 \), where, \( N \) is the number of blades, \( c \) is the blade chord length and \( RD \), is the rotor diameter. Solidity correlates with the chord to rotor radius ratio \( c/R \). According to Bachant and Wosnik (2016a), rotors with \( c/R > 0.1 \) are considered to have high solidity; for the UNH-RVAT \( c/R = 0.28 \).

![Turbine schematic with outlining dimensions.](image)

Figure 1. Turbine schematic with outlining dimensions. - Reproduced with permission from *Energies*, pp. 1–18, (2016). Copyright 2016 MDMI.

<table>
<thead>
<tr>
<th>Turbine Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rotor diameter (( D ))</td>
<td>1 m</td>
</tr>
<tr>
<td>Blade profile type</td>
<td>NACA 0020</td>
</tr>
<tr>
<td>Blade chord (( c ))</td>
<td>0.14 m</td>
</tr>
<tr>
<td>Blade length (( L ))</td>
<td>1 m</td>
</tr>
<tr>
<td>Number of blades</td>
<td>3</td>
</tr>
<tr>
<td>Mid-span struts profile type</td>
<td>NACA 0020</td>
</tr>
<tr>
<td>Shaft diameter</td>
<td>0.095 m</td>
</tr>
</tbody>
</table>

Table 1. UNH RVAT Device Details.
Details of the experimental testing are given in Bachant and Wosnik (2016a) and are summarised as follows. The turbine was tow-tested in UNH’s tow tank which measures 2.44 m deep, 3.66 m wide and 36 m long. The blockage ratio, defined as the ratio of turbine frontal area to tank cross-sectional area, was 11.2%; this is within the range of <20% recommended by Bahaj et al. (2008) for testing of scale model tidal turbines. The turbine was positioned on a specially designed towing rig constructed of NACA 0020 hydrofoils and suitably instrumented. The turbine’s rotational velocity was controlled through a permanent magnet servomotor which incorporated a 20:1 speed increasing gearbox. Carriage motion was controlled by an additional permanent magnet servomotor and timing belt. Both servomotors were encoded into the central motion controller, allowing for high accuracy control of the turbine tip speed ratio (TSR), which is defined as the ratio between the tangential speed of the blade tip and the ambient velocity (\(\omega R/V\)). The torque produced by the turbine was measured via a 200 Nm rotary torque transducer. Wake velocities were measured using a Nortek Vectrino+ acoustic Doppler velocimeter (ADV) which had an approximately 6 mm diameter sampling volume and was sampled at 200 Hz. The ADV and data acquisition system sampling times were synchronized by triggering the start of data acquisition via a pulse sent from the motion controller. Approximately 1,500 tows were carried out in total for different Reynolds numbers, with 31 tows required for each Reynolds number dependent power curve.

2.2. CFD Modelling Background

One of the most critical elements of transient CFD modelling of rotating turbines is the mesh structure. Spatial resolution, the number of nodes over the aerofoil edges and the extent of the domain upstream and downstream of the turbine, all vary significantly between studies and have been shown to affect performance. Model accuracy has also been shown to be quite sensitive to the ratio of the rotating domain to turbine diameter. Raciti Castelli et al. (2011) and Trivellato and Raciti Castelli, (2014) conclude that a ratio of rotating domain diameter of approximately two is required for accurate solutions.

Mesh resolution near walls is particularly essential in order to capture correctly flow properties in the boundary layer region. The dimensionless wall distance for a wall-bounded flow, \(y^+\), can be used to help determine the appropriate mesh resolution near solid boundaries. \(y^+\) is defined as:

\[
y^+ = \frac{u_* y}{v}
\]

where \(u_*\), the friction velocity is further defined as:

\[
u_* = \sqrt{\frac{1}{2} C_f V^2}
\]

Here, \(V\) is the ambient velocity and \(C_f\) is the skin friction which is defined by Schlichting and Gersten (1979) for a flat plate as:

\[
C_f = [2 \log_{10}(R_e) - 0.65]^{-2.3}
\]
where \( R_e \) is the Reynolds number for the flow. Recommended \( y^+ \) values vary between studies and depend on the selected turbulence model. Mohamed et al. (2011) and Mohamed (2012) implement the \( k - \varepsilon \) model on a VAT with a mesh designed to result in a \( y^+ \) value in excess of 30. Lain and Osorio (2010) and Maître et al. (2013) use the \( k - \omega \) SST model in their VAT model and choose to directly resolve the boundary layer using a \( y^+ \) of approximately 1. Rolland et al. (2015) and Velasco et al. (2017) use a similar \( y^+ \) value with the Transition SST model in their VAT model studies.

Selection of the computational time-step is a perennial modelling problem. Too large a time-step can lead to solutions that are not entirely parameter independent, while too small a time-step may yield accurate results but at excessive computational cost. For transient turbine modelling, the time-step (\( \Delta t \)) is generally expressed as a function of the change in azimuthal position with time. Beri and Yao (2011) use a time-step reflective of 2° azimuth, while Maître et al. (2013) use a time-step value reflective of 1° azimuth. Simão Ferreira et al. (2010) show that a time-step reflective of <0.5° azimuth is required to provide a time-step independent solution.

Due to the sensitivity of model performance to mesh structure and time-step, it is critical to carry out mesh and time-step independence studies to achieve an independent solution. For transient turbine modelling studies, this is generally done by comparing the difference in turbine torque between solutions. Solution convergence can be assessed in a similar manner by comparing torque calculated from one complete rotation to the next. The number of rotations required to achieve convergence varies and depends on the torque variation criteria used. The general consensus (Raciti Castelli et al. 2011; T Maître et al. 2013) is that a solution is converged if the difference in torque between two rotations is less than 1 %. Previous modelling studies show that it can take between eight to fifteen rotations for solution convergence (Lain and Osorio, 2010; T. Maître et al. 2013). The moment coefficient monitor (\( C_m \)) outputted from FLUENT® is a measure of the turbine torque and is commonly used as the parameter of interest for convergence and independence studies. \( C_m \) is defined as:

\[
C_m = \frac{M}{\frac{1}{2} \rho V^2 AL}
\]  

where \( M \) is the turning moment created around a predefined axis, i.e. the centre of the turbine, \( \rho \) is the fluid density, \( A \) is the reference area (taken as the turbine frontal area), and \( L \) is the reference length (taken as the turbine radius).

2.3. 2D Model Development

2D model meshing was implemented using ANSYS® Workbench Meshing. The mesh is primarily an unstructured tri-element mesh with quad element inflation layers around the walls. The model contains two mesh domains, an inner rotating domain representing the turbine which sits within an outer static domain. An interface boundary condition is applied to the edges where the domains meet. The rotating domain had a diameter of 2 m. Presented in Figure 2 are images of the 2D mesh that lead to a mesh independent solution. The static domain of the mesh extended ten rotor diameters (RDs) upstream and thirty-five RDs downstream.
In order to avoid any potential divergence issues that may have arisen from using 2\textsuperscript{nd} order schemes and sliding meshes immediately upon simulation start-up, a progressive solution procedure is employed. Initially, the turbine is modelled as a moving reference frame (MRF) problem with 1\textsuperscript{st} order discretisation schemes included. The model is run for 10,000 iterations in this setup before switching to mesh motion. The model is run using 1\textsuperscript{st} order schemes until the torque has reached a quasi-periodic state, before finally switching to 2\textsuperscript{nd} order schemes for the remainder of the simulation. The pressure-based solver was used throughout this process in conjunction with the coupled pressure-velocity scheme. The most prevalent turbulence models used in CFD performance prediction of VAT’s (wind or tidal) are the $k - \omega$ SST and Transition SST models. Both turbulence models were implemented here for comparative assessment.

2.3.1. Solution Convergence and Independence

The criteria to determine convergence and independence was $\Delta C_m < 0.1 \%$ between rotations and simulations. The convergence and independence studies were carried out for TSR = 1.9 as this was the TSR that gave the peak mechanical power performance coefficient ($C_P$) during testing. Figure 3 shows that 12 rotations were required for convergence of the model solution.

![Figure 3. Average moment coefficient output from the model for each rotation.](image)
Mesh independence was determined by developing a series of models of varying densities (see Table 2). Higher mesh densities were achieved by primarily increasing the number of nodes around the edges of the hydrofoils. The number quoted in Table 2 is the total number of nodes over a single hydrofoil; this was split evenly between the upper and lower surfaces of the foil. The edges on both sides of the domain interface were selected to ensure a corresponding number of elements on either side of the interface. Figure 4(a) shows the effect of the mesh on $C_m$ (calculated over the final rotation) for three different tip speed ratios. Mesh M3 was chosen for subsequent model set-ups. Its geometric characteristics are presented in Table 3. All of the values presented are within the recommended limits for the FLUENT® solvers.

A time dependence study was carried out using a number of different time-steps and comparing the final $C_m$ as seen in Figure 4(b) compares $C_m$ values computed using the different time steps. A time-step value representing $0.2$ azimuthal degrees per time-step ($0.2^\circ / \Delta t$) was identified as the optimum time-step value, with smaller time-steps resulting in a negligible difference in $C_m$.

**Table 2. Mesh Parameters for Mesh Independence Study.**

<table>
<thead>
<tr>
<th>No. of nodes over hydrofoils</th>
<th>No. of Quad rows</th>
<th>Interface No. of divisions</th>
<th>1st layer element height (m)</th>
<th>Total number of elements</th>
</tr>
</thead>
<tbody>
<tr>
<td>M1</td>
<td>500</td>
<td>25</td>
<td>600</td>
<td>$1 \times 10^{-5}$</td>
</tr>
<tr>
<td>M2</td>
<td>750</td>
<td>35</td>
<td>700</td>
<td>$1 \times 10^{-5}$</td>
</tr>
<tr>
<td>M3</td>
<td>1000</td>
<td>45</td>
<td>900</td>
<td>$1 \times 10^{-5}$</td>
</tr>
<tr>
<td>M4</td>
<td>1200</td>
<td>55</td>
<td>1000</td>
<td>$1 \times 10^{-5}$</td>
</tr>
</tbody>
</table>

**Table 3. Details of M3 mesh used in the 2D model.**

<table>
<thead>
<tr>
<th>Description</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Average skewness</td>
<td>0.0059</td>
</tr>
<tr>
<td>Max skewness</td>
<td>0.76</td>
</tr>
<tr>
<td>Average quality</td>
<td>0.82</td>
</tr>
<tr>
<td>Max $y^+$</td>
<td>1</td>
</tr>
<tr>
<td>Number of elements</td>
<td>980 x 10^3</td>
</tr>
</tbody>
</table>
2.3.2. 2D Models Used for Sensitivity Analyses

As mentioned earlier a key finding of this study is the effect of blockage, especially for 2D models. In a 1:1 2D model of the experiment, the blockage ratio is 27.3 %. It is, therefore, logical to hypothesise that this considerably higher blockage might result in turbine performance over-prediction. To test this hypothesis, a Low Blockage 2D model was developed where the side walls of the tank were extended (see Figure 5) to give the same blockage ratio as the experiment set-up. The results of this model were compared with those from the High Blockage model.

Figure 5. Schematic comparing static domain sizing, High Blockage model domain with 3.66 m width and Low Blockage model with 8.93 side width.

2D model sensitivity to the size of the rotating domain was carried out using three different diameters of rotating domain. Measured relative to the turbine rotor diameter, the domain diameters are 2RD, 1.4RD and 1.1RD. The domain extents are presented in Figure 6. The prescription of turbulence properties (e.g. turbulence intensity (TI) and turbulence viscosity ratio (TVR)) at the inlet and outlet boundaries may have an effect on the model results. The effect of their specification was investigated for varying inlet and outlet values of TI and TVR. The FLUENT® user manual (ANSYS FLUENT 17.0 user’s guide, 2016) specifies ranges for these parameters for interior and exterior flows. Models were run for TI values of 1 %, 5 %, 10 % and 15 % and TVR values of 0.2, 1, 2 and 5. Table 4 lists the different models used in the sensitivity analyses. A naming convention used henceforth is also presented.

Table 4. Details of M3 mesh used in the 2D model.

<table>
<thead>
<tr>
<th>Model Description</th>
<th>Acronym</th>
</tr>
</thead>
<tbody>
<tr>
<td>Lateral tank walls at 3.66 m with blockage of 27.3 %</td>
<td>High Blockage</td>
</tr>
<tr>
<td>Lateral tank walls at 8.93 m with blockage of 11.3 %</td>
<td>Low Blockage</td>
</tr>
<tr>
<td>Low blockage model with rotating domain diameter of 2RD</td>
<td>2RD</td>
</tr>
<tr>
<td>Low blockage model with rotating domain diameter of 1.4RD</td>
<td>1.4RD</td>
</tr>
<tr>
<td>Low blockage model with rotating domain diameter of 1.1RD</td>
<td>1.1RD</td>
</tr>
</tbody>
</table>
2.4. 3D Model Development

3D model development followed a similar approach; the static domain was modelled based on the tow tank dimensions. The full turbine was modelled, including the mid-span support struts, the turbine shaft and full-length blades. The rotating domain extended the full depth of the tank and had a diameter of 1.8RD. Figure 7 and Figure 8 show images of the independent solution mesh. Similar to the 2D model, the static domain of the mesh extended 10 RD upstream and 35 RD diameters downstream. *Mapped Faced Meshing* ensures a uniform mesh; this was applied to the blades, struts, shaft, domain interface and the strut-shaft hubs. The rotor was divided into a number of separate sections to enable *Mapped Faced Meshing* be used.

Figure 7. (a) Three-dimensional view of the mesh, with rotating and static domains visible, (b) horizontal section view through the model domains showing the hydrofoils and shaft and (c) section view of the mesh around a blade showing the 35 inflation layers.
Convergence and independence studies were used to determine the optimal number of turbine rotations, mesh density and time-step. The final 3D model details are summarised in Table 5.

Table 5. Details of 3D mesh and model parameters that lead to a converged and independent solution.

<table>
<thead>
<tr>
<th>Definition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>No. of quad inflation layers</td>
<td>35</td>
</tr>
<tr>
<td>1\textsuperscript{st} layer element height</td>
<td>1 x10\textsuperscript{-5} m</td>
</tr>
<tr>
<td>No. of elements spanwise on the blades</td>
<td>600</td>
</tr>
<tr>
<td>No. of elements spanwise on the struts</td>
<td>150</td>
</tr>
<tr>
<td>No. of nodes around hydrofoil edge</td>
<td>500</td>
</tr>
<tr>
<td>Total number of elements</td>
<td>21.84 x 10\textsuperscript{6}</td>
</tr>
<tr>
<td>Max skewness</td>
<td>0.899</td>
</tr>
<tr>
<td>Average skewness</td>
<td>0.49</td>
</tr>
<tr>
<td>Converged (degrees/\Delta t)</td>
<td>0.25</td>
</tr>
<tr>
<td>No of Rotations to convergence</td>
<td>9</td>
</tr>
</tbody>
</table>

3. Results

The 2D and 3D models were employed to characterise the effect of turbine tip speed ratios on mechanical power performance in comparison with the experimental data, for a free stream velocity of 1 m/s, giving a rotor diameter dependant Reynolds number of 1x10\textsuperscript{6}. Predicted wake velocities and turbulent kinetic energy were also compared to the test data to assess the wake dynamics prediction.

3.1. 2D Model Results

Figure 9 shows the comparison of 2D power curves for the High and Low Blockage models with the experimental test data, using two different SST turbulence models. It is clear from Figure 9(a) that the High Blockage model vastly over-predicted power, consistent with Bachant and Wosnik (2016b), using a similar 2D modelling approach. Also, the peak predicted power occurs at a higher TSR. The Low Blockage model (Figure 9(b)) showed much better agreement with the experimental data. RMSEs for all models are listed in Table 7. The root mean squared error (RMSE) for the Low Blockage model was 0.06, as compared to 0.27 for
the *High Blockage* model. The *Low Blockage* model correctly predicted the optimum TSR of 1.9; however, it under-predicted $C_p$ at lower TSRs and over-predicted it at higher TSRs. There was relatively little difference in performance for the turbulence models but the Transition SST gave a little more accuracy, particularly at the lower TSR values as a transitional phase is more likely to occur with lower Reynolds numbers.

![Figure 9. Comparison between 2D modelled and experimentally measured $C_p$ for (a) High Blockage model and (b) Low Blockage model using two different turbulence models: the Transition SST and the $k - \omega$ SST.](image)

<table>
<thead>
<tr>
<th>2D model</th>
<th>RMSE ($C_p$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Trans. SST <em>High Blockage</em></td>
<td>0.265</td>
</tr>
<tr>
<td>$k - \omega$ SST <em>High Blockage</em></td>
<td>0.266</td>
</tr>
<tr>
<td>Trans. SST <em>Low Blockage</em></td>
<td>0.057</td>
</tr>
<tr>
<td>$k - \omega$ SST <em>Low Blockage</em></td>
<td>0.061</td>
</tr>
</tbody>
</table>

Figure 10(a) shows the variation in $C_m$ for a single hydrofoil over the course of a full rotation. Output from the *High Blockage* and *Low Blockage* models are presented for a TSR value of 1.9. As shown in Figure 10(b), an azimuthal position of 0° corresponds to three o’clock with the turbine rotating anti-clockwise. As would be expected, the vast majority of the turning moment is created from the upstream part of the rotational cycle (40° to 140°). The figure shows the effect of tank blockage with the higher blockage resulting in higher torque being computed during the upstream part of the rotation, thus leading to over-prediction of $C_p$. 

![Figure 10](image)
Figure 10. (a) Comparison of moment coefficient calculated by the *high* and *low blockage* models during one complete rotation cycle and (b) schematic showing $0^\circ$ blade azimuth position and flow direction and turbine rotation direction.

To investigate the difference in performance between the turbulence models, Figure 11 presents a comparison of $C_m$ output from the Transition SST and $k$-$\omega$ SST models for the *low blockage* model. It is clear from the figure that there are two azimuth regions where noticeable differences in $C_m$ can be observed, namely at approximately $140-170^\circ$ and $260-270^\circ$. The $k$-$\omega$ SST model computed higher $C_m$ values for both of these blade positions. To investigate further, velocity streamline plots were output for a blade azimuth position of $150^\circ$ (Figure 12). The streamline plot from the Transition SST model (Figure 12(b)) shows the occurrence of detached and reattached flow near the trailing edge of the blade which is not predicted by the $k$-$\omega$ SST model. This flow detachment and reattachment suggests the onset of stall, causing lower lift and therefore lower torque. These images present visual confirmation of one of the often-stated benefits of the Transition SST model. (Lanzafame *et al.* 2014; Rezaeiha *et al.* 2017)

Figure 11. Comparison of $C_m$ output from the $k$-$\omega$ SST and Transition SST for low blockage model setup of a single hydrofoil.
Figure 13 shows normalised velocity contour plots from the 2D High (Figure 13(a)) and Low Blockage models. Figure 13(a) shows that higher blockage results in larger flow acceleration. The velocities at the entrance and at either side of the turbine are higher than in the low blockage case (Figure 13(b)), leading to higher $C_p$ values. The higher blockage also affects the wake characteristics. The smaller domain width in the high blockage model results in a more constrained flow and therefore a more persistent wake.

Figure 12. Velocity streamline plots for a single hydrofoil for: (a) k-ω SST at 150° azimuth, and (b) Transition SST at 150° azimuth.
Figure 13. Two dimensional model velocity contour plots for (a) High Blockage and (b) Low Blockage.
Flow velocities in the wake were sampled along Transect B as shown in Figure 5. Corresponding instantaneous velocity data was output from the High Blockage and Low Blockage models. A comparison of measured and modelled along-stream (x-direction) velocities for a TSR value of 1.9 is presented in Figure 14. The velocities are normalised against the free-stream velocity. Overall, the Low Blockage model more accurately reproduces the experimental data. The High Blockage model RMSE was 0.204 m/s, while the Low Blockage model had an RMSE of 0.128 m/s. There are some peaks and troughs within the modelled data that do not occur in the experimental data. It is believed these discrepancies are caused by differences in the positions of the turbulent eddies. While the model data was sampled at discrete time instances, the experimental data were collected over the course of several tows.

To further understand the reasons for the differences between the high and low blockage models, velocities were also extracted from Transect A (see Figure 5). These data are presented in Figure 15. It is clear that flow velocities are noticeably higher in the High blockage model. The transect average was 1.15 m/s for the High blockage model compared to 1.06 m/s for the Low blockage model; an 8.5 % increase on the Low Blockage value. Since power is proportional to velocity cubed, this will have a significant effect on predicted available and extracted powers.
3.2. Sensitivity Studies
Two sensitivity studies were carried out using the 2D models to study the effects of (1) rotating domain diameter (RDD) and (2) inlet and outlet values for TI and TVR. Figure 16 shows the effect of RDD in CFD-predicted power coefficient for different TSRs. Clearly, there is a significant effect of RDD on predicted $C_P$ at lower TSR values; this is attributed to the fact that angle of attack varies dramatically at low TSR values and therefore so does the flow characteristics. A larger domain could help to curtail these flow features.

The results of the inlet parameter study are presented in Figure 17. The model is not overly sensitive to inlet specifications of TI and TVR but predicted device power is seen to decrease marginally with increasing TI and TVR.

![Graph of $C_P$ computed by models with rotating domains of 1.1RD and 1.4 RD normalised against those from the model with a rotating domain of 2RD model as a function of TSR.](image)

![Graphs of $C_P$ computed by models with different boundary values for (a) TI and (b) TVR normalised against those from the model with boundary values of TI = 1 % and TVR = 0.2.](image)

3.3. 3D Model Results.
Similar to the 2D case, 3D model accuracy was evaluated by comparing the modelled and measured turbine power curves. Figure 18 compares measured $C_P$ with corresponding model values predicted using the Transition SST and $k - \omega$ SST turbulence models. It can be seen that the models reproduce the measured data quite accurately. Both models predicted that the peak performance of the device occurred at a TSR of 1.9 which concurs with the experimental data. In general, both models behave similarly, underestimating the turbine
performance at TSRs less than the optimum and overestimating performance for higher TSRs. The Transition SST model performs better than the $k-\omega$ SST model. The Transition SST model data had a RMSE of 0.028, while the $k-\omega$ SST model data had a value of 0.035.

The Transition SST model performs better than the $k-\omega$ SST model. The Transition SST model data had a RMSE of 0.028, while the $k-\omega$ SST model data had a value of 0.035.

The improved power performance prediction of the 3D model in comparison to the 2D model also manifested itself in improved agreement in downstream velocities. Figure 19 shows a comparison of the modelled and measured velocities for a TSR of 1.9. Both of the 3D models accurately reproduce the bypass velocities outside of the wake. While both models give good correlation relative to the experimental data inside the wake, the Transition SST model outperforms the $k-\omega$ SST model here. The Transition SST model had a RMSE of 0.067 m/s while the $k-\omega$ SST model had a value of 0.079 m/s, relative to the measured data.

Presented in Figure 20 is a comparison of measured turbulent kinetic energy levels against corresponding values outputted from the 3D models along Transect B (see Figure 5). As expected, turbulent kinetic energy levels are highest at the extremities of the turbine, due to the shedding of eddy vortices from the blades. At this extreme near-field location, low levels of turbulent kinetic energy are expected in the middle of the wake with these increasing downstream due to turbulent mixing of the wake and the bypass flow. Once again, the Transition SST model is seen to be more accurate. Excluding the outlier in the experimental
data at $RD = 0.45$, the RMSE values for the Transition SST and $k - \omega$ SST models are 0.012 m$^2$/s$^2$ and 0.015 m$^2$/s$^2$ respectively.

![Figure 20](image1.png)

**Figure 20.** CFD model comparison to experimental data for Turbulent Kinetic Energy.

Figure 21 shows velocity contour plots of a horizontal sectional plane through the centre of the turbine from the Transition SST model (Figure 21). Comparing the contour plot with those from the 2D models, the velocity gradients immediately upstream, and to either side of the device are most similar to those of the Low Blockage model in Figure 13(b).

![Figure 21](image2.png)

**Figure 21.** CFD predicted velocity contour at a tip speed ratio of 1.9 for the Transition SST model.

### 3.4. Aspects of Turbine Design

Figure 22 shows a 3D contour of the vorticity generated by the turbine. It is seen that the large 95 mm diameter shaft shed significant vortices into the path of the oncoming downstream blades, thus potentially reducing the effectiveness of the downstream blades. This demonstrates the importance of shaft sizing in the design process; there is a trade-off between using a larger shaft diameter to give a higher factor of safety, for better reliability and using a smaller diameter shaft, to reduce incoming flow vorticity on the downstream blades.
To evaluate the potential effects of the horizontal support struts on turbine performance, an additional 3D model was developed. This model retained the same mesh details and model setup but omitted the struts. The model was run at five different TSR values, and the results were normalised ($C_p$ with struts / $C_p$ without struts). The normalised $C_p$ curve is shown in Figure 23. It is clear that inclusion of the struts leads to a reduction in $C_p$. Also, this effect increases with increasing TSR.

![3D vorticity contour plot at a tip speed ratio of 1.9.](image)

Figure 22. 3D vorticity contour plot at a tip speed ratio of 1.9.

![Predicted effect of struts on normalised $C_p$ (relative to the case without struts included) on turbine performance.](image)

Figure 23. Predicted effect of struts on normalised $C_p$ (relative to the case without struts included) on turbine performance.

3.5. Comparison Between 2D and 3D Model Results.

A comparison between the performance predictions of the 2D blockage corrected and 3D models are presented in Figure 24. Even with the improvements in power prediction arising from the blockage correction, the 3D results are clearly more accurate with an RMSE of 0.028 compared to 0.057 for the 2D model. However, as seen in Table 7, this higher level of accuracy comes at the cost of a significant increase in CPU time; the 3D model runtime was almost 5 times that of the 2D. There is again a trade-off here in relation to model accuracy versus
available computational resources when deciding which modelling approach to use. In some cases, a blockage corrected 2D model may be of sufficient accuracy for assessment of the effect of design decisions such as shaft sizes, strut design, etc. For example, the inclusion of the struts in the 3D model resulted in 30% additional CPU time over an already very high CPU time compared to that of the 2D model.

![Figure 24. Comparison of 3D and 2D Low Blockage model performance for $C_p$.](image)

Table 7. Comparison between runtimes for 2D and 3D CFD models.

<table>
<thead>
<tr>
<th>Model</th>
<th>CPU Time (hrs)</th>
</tr>
</thead>
<tbody>
<tr>
<td>2D High Blockage model</td>
<td>3600</td>
</tr>
<tr>
<td>2D Low Blockage model</td>
<td>4200</td>
</tr>
<tr>
<td>3D model without struts</td>
<td>17500</td>
</tr>
<tr>
<td>3D model with struts</td>
<td>23000</td>
</tr>
</tbody>
</table>

4. Discussion

The developed 3D CFD model accurately reproduced the measured mechanical performance coefficient of the UNH RVAT turbine with less than 5% difference in peak performance coefficient. By comparison, the 3D model of Bachant and Wosnik, (2016b) of the same turbine over-predicted the peak performance by 30% using the $k - \omega$ SST model. The high level of accuracy is attributed to the rigorous and systematic approach to mesh development, model convergence and sensitivity studies and the validation of the model against both performance and wake characteristics. Comparison to other research is also quite favourable, Zamani et al. (2016) and Marsh et al. (2012) over-predicted peak performance by approximately 8% and 56% respectively, using the $k - \omega$ SST model. Whereas, Lei et al. (2017) under-predicted peak performance by approximately 11%, also using the $k - \omega$ SST model.

This high level of accuracy shows that URANS CFD models can be successfully used in the design and optimisation of tidal turbines, both individually and in arrays. For the latter, it is critical that turbine models are validated against both power performance, and wake
characteristics as both are important in identifying optimum array layouts. Accurate wake modelling is vital in identifying optimum distances for placement of downstream turbines so as to avoid excessive turbulence and/or reduced power availability. Successfully validated CFD models can also be integral in the areas of device reliability. Li et al. (2014) conducted a 2-year experimental study and showed that the natural frequency of a vertical axis tidal turbine can be in the same range as that of the turbulence flow conditions it is subjected to. If this is the case device resonance is likely, and may ultimately lead to device failure. Accurate CFD models, such as that presented here, could be used to predict turbulence loads for use in preliminary design checks, including vibration analyses, to improve device reliability and avoid resonance.

The 2D model studies investigated whether blockage contributes to poor model accuracy, and to over-prediction of the power performance in particular. The blockage in a 2D model of an in situ turbine will be higher than the actual 3D value, and this will be exacerbated if the true blockage is high, to begin with. In this study, the 2D model without blockage correction significantly over-predicted the turbine mechanical performance coefficient, yielding an average RMSE of >0.26. By comparison, when corrected for blockage, the model’s performance was notably more accurate, with an average RMSE value in the region of 0.05. Comparing the two model’s predictions for peak power performance, the uncorrected model over-predicted the maximum power coefficient by approximately 100 % compared to an approximate 20 % over-prediction by the corrected model. In this instance blockage, therefore, accounted for 80 % of the model over-prediction. The study proves that 2D models can be corrected for blockage effects by extending the domain width to give the equivalent 3D blockage. Correcting for blockage also gave better predictions of both bypass velocities and wake velocities. The RMSE value was reduced from 0.2 m/s to 0.12 m/s when blockage was corrected.

The 2D study on the effect of the size of the rotating domain diameter showed that model performance can be adversely affected by the size of the rotating domain, this effect is more noticeable at lower TSRs. Larger diameter domains of at least 1.5 times the rotor diameter are therefore required at lower tip speed ratios to avoid incurring substantial errors. At higher TSRs (>1.9 in this research), one can use a smaller rotating domain without significantly affecting the model accuracy.

The effects of the turbulence models were more apparent in the validation of the turbine wakes. It is thought that this may be the result of some flow reattachment to the hydrofoil that is being modelled by the Transition SST but which the $k-\omega$ SST model is incapable of doing. The close similarity of the 3D wake velocity contour plot with the corresponding 2D plot from the blockage corrected model, particularly upstream and to either side of the device, provides further confirmation that the 2D blockage correction approach is valid. In summary, if model accuracy is paramount, it is recommended that 3D models are used. If not, then 2D models that account for blockage may be attractive as a device evaluation tool due to their significantly lower computation costs. The potential saving can be demonstrated by comparing the CPU runtimes of 22,000 CPU hours for the 3D model with struts to 4,200 CPU hours for the blockage corrected 2D model.

Now that we have developed a highly accurate 3D tidal turbine model, future work involving fluid-structure interaction is planned. This would involve extracting pressure values
from the 3D model as inputs into a finite element program, where parameters such as stress and deflection could be determined. Li et al. (2014) have previously conducted similar work using a discrete vortex model coupled with a finite element analysis approach.

5. Conclusion
A range of 2D and 3D CFD models of a vertical axis tidal turbine have been developed to reproduce experimental test data. A 2D model blockage correction approach was assessed where the correct level of blockage is achieved by extending the width of the 2D model domain. A rigorous approach to model development was employed involving detailed convergence and sensitivity studies and extensive validation of both power performance and wake characteristics. The main conclusions of the research are as follows.

- 2D CFD models can be corrected for blockage by extending the domain width to give a blockage value equivalent to the correct value in three dimensions. By doing so, one can achieve a reasonable level of accuracy, but if accuracy is paramount, it is recommended that 3D models are used.
- Model accuracy can be sensitive to the choice of size of the rotating domain but is dependent on the TSR. To avoid these effects, a domain diameter of twice the turbine rotor diameter is recommended. Model performance is much less sensitive to domain size at higher TSRs (>1.5), therefore smaller rotating domains can be used to reduce computation costs.
- The rigorous model development approach resulted in a highly accurate 3D model. This proves that URANS CFD models can give sufficient levels of accuracy for turbine design without the need for the development of higher order, and much more computationally expensive, CFD methods like large eddy simulation (LES).

Acknowledgements
This material is based upon works supported by Science Foundation Ireland under Grant No. 12/RC/2302 through MaREI, the national centre for Marine and Renewable Energy Ireland.

The authors wish to acknowledge the contribution of Prof. Martin Wosnik of the School of Marine Science and Ocean Engineering from the University of New Hampshire, USA, for providing all experimental data for this research.

The authors wish to also acknowledge the DJEI/DES/SFI/HEA Irish Centre for High-End Computing (ICHEC) for the provision of computational facilities and support.

References


ANSYS Fluent 17.0 user’s guide (2016) Ansys Inc.


26


