Accuracy of the actuator disc-RANS approach for predicting the performance and wake of tidal turbines

W. M. J. Batten¹, M. E. Harrison² and A. S. Bahaj¹

¹Sustainable Energy Research Group, Energy and Climate Change Division, Faculty of Engineering and the Environment, University of Southampton, Southampton SO17 1BJ, UK
²Marine Current Turbines Ltd, Bristol & Bath Science Park, Dirac Crescent, Emersons Green, Bristol, BS16 7FR

The actuator disc-RANS model has widely been used in wind and tidal energy to predict the wake of a horizontal axis turbine. The model is appropriate where large-scale effects of the turbine on a flow are of interest, for example, when considering environmental impacts, or arrays of devices. The accuracy of the model for modelling the wake of tidal stream turbines has not been demonstrated, and flow predictions presented in the literature for similar modelled scenarios vary significantly. This paper compares the results of the actuator disc-RANS model, where the turbine forces have been derived using a blade-element approach, to experimental data measured in the wake of a scaled turbine. It also compares the results with those of a simpler uniform actuator disc model. The comparisons show that the model is accurate and can predict up to 94 per cent of the variation in the experimental velocity data measured on the centreline of the wake, therefore demonstrating that the actuator disc-RANS model is an accurate approach for modelling a turbine wake, and a conservative approach to predict performance and loads. It can therefore be applied to similar scenarios with confidence.

1. Introduction

The actuator disc model has widely been used to represent a propeller or turbine in a solution of the Reynolds-averaged Navier–Stokes (RANS) equations. The approach approximates the forces that the device
applies to the flow, over a disc with the same diameter. It significantly improves computational efficiency over a full rotor computational fluid dynamics (CFD) simulation, since it does not require fine mesh at the rotor to capture the boundary layer on the blade, and can be solved in a steady-state solution.

The actuator disc-RANS model was first applied in the mid-1990s to study propulsion problems such as the influence of helicopter rotors on the fuselage [1] and the influence of ship propellers on rudders [2]. Since then it has been applied to several other applications such as the wake of wind turbines in a farm [3]. Recently, it has also been used in a number of cases to study unducted tidal stream turbine wakes [4–14]. Although this technique has computational advantages for modelling tidal stream turbines, the assumptions can cause inaccuracies in the representation of the near wake of a turbine. The effect of how the incorrect representation of the near wake will influence far-wake predictions is currently unknown. The three main limitations are detailed below.

Firstly, as the disc is not rotating it will not introduce any swirl into the flow, unlike rotating blades. The assumption of the actuator disc approach is that, beyond the near wake, the flow will have a similar structure to that of a turbine since most swirl components will have dissipated, together with the turbulence generated by the turbine blades. The downstream limit of the near wake is defined as the point at which the shear layer (developing between the edge of the wake and the free stream flow) reaches the wake centreline. This is generally between 2D and 5D downstream [15]. Swirl in the near wake can persist further downstream [16] potentially interacting with local flow boundaries and can cause the wake to meander.

Secondly, tip vortices are shed from the blades of a rotating turbine. These vortices will not be replicated by an actuator disc. (This may be represented by a line model [17] or a fully meshed turbine blade.)

Thirdly, the steady actuator disc-RANS methodology does not account for transient flow characteristics. It provides information about mean flow characteristics and assumed isotropic turbulence. Therefore, the model can be usefully applied for understanding the time-averaged characteristics of the flow behind the turbine. This approach would not be suitable if there is a specific interest in vortices or irregularity in the wake; if this is the case then a large eddy simulation would be more appropriate.

In previous studies where the actuator disc-RANS model has been applied to tidal stream turbines, predictions of the velocities in the wake have varied significantly. Figure 1 shows normalized centreline velocities from examples in the literature where these data were available. All of the turbines in this comparison were operating at a thrust coefficient of approximately 0.9, apart from [6]. Therefore, the wake predictions should be similar, and it ought to be possible to explain any small variations between them by considering the physical situation.

However, figure 1 shows that the centreline velocities vary significantly between models, with minimum normalized velocities ranging from −0.2 in the case of Sun et al. [6] to 0.65 in the case of MacLeod et al. [4] (which has a reported thrust coefficient of 0.9). It would be expected that models with a lower thrust coefficient would correspond to lower minimum velocity in the wake [15], but the model results show an opposite trend. Models in this plot which have the same reported thrust coefficient of 0.9 have a range of minimum normalized velocities from 0.25 in the case of Roc et al. [11] to 0.65 in the case of MacLeod et al. [4].

Similarly, rates of velocity recovery also vary significantly between the models, with the fastest rate of recovery between D and 12D observed in Sun et al. [6] and the slowest in the far wake in MacLeod et al. [4]. Rate of recovery is dependent on ambient turbulence intensity and turbine-induced turbulence; however, only Harrison et al. [12] specifically reports ambient turbulence values so it is not possible to correlate ambient turbulence to the rate of wake recovery and understand why significant differences occur. The maximum velocity on the centreline also varies, with Gant & Stallard [7] showing the lowest maximum velocity at 7D downstream, and MacLeod et al. [4] showing the highest. Ambient turbulence data are not available for either of these cases, so it is difficult to specify why the variation between the models occurs.
Validation studies to date have not satisfactorily demonstrated the accuracy of the model for wake and performance estimation for tidal turbines. Some studies vary significantly from experimental data in the wake \cite{6,18,19}. Some present improved agreement, but the validation lacks estimations of discretization error, which can improve agreement in the wake \cite{11,20}. Therefore, the aim of this paper is to contribute to the verification and validation of the actuator disc-RANS model for modelling the wake of a tidal stream turbine. This is achieved by comparing with experimental data measured in the wake of a scaled device \cite{21} and performance data measured for the device itself \cite{22}.

2. Model theory

(a) Reynolds-averaged Navier–Stokes equations

The approximated forces applied by a turbine to the flow are applied as source terms in the RANS equation of momentum conservation (2.1), and solved together with a mass continuity equation (2.2). The source terms are only applied at elements within the turbine region.

\[
\frac{\partial (\rho U_i)}{\partial t} + \frac{\partial (\rho U_i U_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ (\mu + \mu_t) \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] + \rho g_i + S_i \tag{2.1}
\]

and

\[
\frac{\partial U_i}{\partial x_i} = 0. \tag{2.2}
\]

Einstein notation is used in these equations for brevity. \( \rho \) is the density of water, \( U_i \) is the velocity of the water averaged over time, \( t, x_i \) is the spatial distance, \( \mu \) is viscosity, the Boussinesq assumption is used and \( \mu_t \) is the eddy viscosity, \( g_i \) is the component of gravitational acceleration and \( S_i \) is an added source term.
(b) Turbine model

Two turbine models are used in this paper. Firstly, the uniform actuator disc model applies a constant resistance to the incoming flow. This resistance causes a thrust to act on the disc and this thrust should be similar to that acting on the turbine being simulated. From here on, this model is referred to as RANS + Disk. If the RANS equations are discretized using a finite volume approach, the actuator disc method must apply a momentum source term in a set of finite volumes, which represent the turbine region. In the region defined as ‘Water’ the standard Reynolds-averaged momentum equation will apply (equation (2.1)), while in the region defined as ‘Disc’ an additional source term will be added, which is shown in equation (2.3):

\[ S_i = \frac{K}{\Delta x} \rho \frac{U_i}{|U_i|}, \quad (2.3) \]

where \( \Delta x \) is the thickness of the actuator disc and \( K \) the resistance coefficient. The disc thickness is included to implement the resistance coefficient appropriately in the differential equation: as the rate of change of resistance over the disc thickness.

Secondly, the blade-element (BE) approach assumes that turbine forces are rotationally averaged, but vary with radius along the blade. This is referred to as RANS + BE. At each finite volume in the turbine domain, the forces are calculated based on incoming flow velocity and direction, the local coefficients of lift and drag of a turbine blade, the blade pitch and the rotational speed of the turbine. The force applied by the BE is rotation-averaged by applying the turbine solidity, and divided by the turbine domain thickness to produce a gradient term. Equations (2.4) and (2.5) show the stream-wise and azimuthal body force calculations in polar coordinates:

\[ S_x = \frac{\sigma r \rho}{2\Delta x} W^2 (C_L \cos \phi + C_D \sin \phi) \quad (2.4) \]

and

\[ S_\theta = \frac{\sigma r \rho}{2\Delta x} W^2 (C_L \sin \phi - C_D \cos \phi), \quad (2.5) \]

where \( S_x \) and \( S_\theta \) are the momentum sink in the stream-wise and azimuthal direction from the disc, respectively; \( W \) is the resultant velocity at the blade solved from equations (2.6) and (2.7); and \( C_L \) and \( C_D \) are the lift and drag coefficients at the apparent angle of attack \( \phi \).

\[ W = \sqrt{U_1^2 + \Omega r (1+a')^2} \quad (2.6) \]

and

\[ a' = \frac{\sqrt{U_2^2 + U_3^2}}{\Omega r}, \quad (2.7) \]

where indices 1, 2 and 3 represent the stream-wise, lateral and vertical directions, respectively.

Turbine blades introduce vortices and swirl to the flow which when modelled using RANS produce turbulence. When the turbine is represented as a uniform actuator disc no equivalent turbulence will be produced. Eddy-viscosity-based turbulence models produce turbulence kinetic energy in the flow at regions of high velocity gradients; so the BE disc will cause some increased turbulence production owing to velocity gradients along the blade.

An empirical function can be used to calculate the level of turbulence introduced by the turbine and to specify these sources of turbulence kinetic energy and turbulence eddy dissipation into the turbulence model equations (similar to the introduction of momentum source terms to the momentum equations to represent forces applied by the turbine).

An outline of methods for calculating turbulence characteristics in wind turbine wakes is provided in [23]. This includes an empirical function relating the maximum turbulence intensity in the wake \( (I_m) \) to the axial induction factor \( (\alpha) \) of the turbine:

\[ I_m = 0.725\alpha. \quad (2.8) \]

Using the relationship between axial flow induction factor and resistance coefficient from actuator disc momentum theory [24], maximum turbulence intensity may be approximated as.
a function of resistance coefficient, shown in equation (2.9):

$$\Delta I_m = 0.362 \left[ 1 - \frac{1}{\sqrt{1 - K(1 + K/4)^2}} \right].$$ (2.9)

3. Turbine geometry and validation data

(a) Turbine geometry

The comparisons presented in this paper are made from experiments for two different test facilities with the same turbine test rig and with both predictions based on the blade element momentum theory (BEMT). The three bladed turbine has a diameter of 0.8 m and blades derived from NACA 48XX sections. The chord ($c$), twist ($\theta$) and thickness ($t$) distributions are shown in figure 2. Full details of the experimental apparatus are presented in [21,22]. For the RANS + BE predictions of the 2D lift and drag coefficients along the blade length are required. These have been solved using XFOIL [25] for six thickness ratios, shown in figure 3. This dataset is used to interpolate for any given angle of attack ($\alpha$) along the length of the blade when solving equations (2.4) and (2.5).

(b) Turbine performance

To verify the predictions of performance based on the RANS-BE model, the simulations are compared with both predictions based on the standard BEMT and measurements in a towing tank. The BEMT simulations were performed using the code developed in [26] and solved using the same lift and drag dataset as used in the simulations.

Measurements in the towing tank were performed at a towing speed of 0.9 m s$^{-1}$ for a range of tip speed ratios (TSR = $\Omega R/U_1$) from approximately 4 to 7. The towing tank has a length of 60 m, breadth of 3.7 m and depth of 1.8 m. This corresponds to a blockage ratio of 7.5 per cent, as detailed in [26]. Further details of these experiments are presented in [22]. The inlet turbulence is assumed to be minimal. In order to compare with the BEMT these experiments and the presented simulations have been corrected for blockage as detailed in the appendix of Bahaj et al. [27].

Figure 2. Graphical representation of the blade shape particulars. (a) Chord distribution, (b) twist distribution and (c) thickness ratio.
Figure 3. Basic sectional 2D hydrofoil performance solved from XFOIL for NACA 48XX [25]. (Online version in colour.)

(c) Wake profiles

For the wake study, the simulations have been compared with measurements performed in the 18 m long circulating water channel at IFREMER, Boulogne-sur-Mer, France. This channel has a depth of 2 m and breadth of 4 m and the experiments were performed in a circulating water speed of around $0.8\,\text{m\,s}^{-1}$. This corresponds to a blockage ratio of 6.3 per cent. The wake measurements were carried out at a TSR of 8.4 and were performed using a combination of particle image velocimetry and an acoustic doppler velocimeter as detailed in [21].

4. Methodology

(a) Numerical method

The transport equations were solved using Ansys CFX. They are integrated over each discrete finite volume within the solution domain and linearized by using a combination of the finite volume and finite-element differencing schemes. The coupled system of linear equations is then solved in a matrix form with an iterative solver. A hybrid advection scheme was used for all transport equations which combines first- and second-order differencing methods to achieve a stable, accurate and computationally efficient solution [28].

Reynolds stresses were resolved with an eddy-viscosity approach, using the $k-\epsilon$ turbulence model. Recent studies which have adopted the eddy-viscosity approach have used one of the two main equations of turbulence models, either $k-\epsilon$ or $k-\omega$ SST (shear stress transport). Preliminary investigations to this study found that the blending functions used in the $k-\omega$ SST model caused it to under-predict eddy-viscosity in the turbine wake, and therefore under-predict the rate of wake recovery [29]. It was found that improved agreement could be achieved by using the $k-\epsilon$ model, and introducing turbulence sources at the disc to represent turbulence produced by the turbine.

In the present investigation, the flow details in the boundary layer were not of specific interest, and wall functions were used to define flow characteristics close to the wall. CFX implements an automatic wall function, which shifts from a low Reynolds number formulation in the sub-layer to a wall function based on the grid-spacing of the near-wall cell. This automatic wall
function significantly reduces constraints on near-wall spacing and allows the use of arbitrarily fine grids [30].

It was found in [29] that for a Froude number and blockage ratio similar to those tested (which were 0.18% and 6.3%, respectively) the change in free surface across the turbine is insignificant. The analytical expression derived in [31] predicts a head drop of 0.0003 per cent of the water depth at a Froude number of 0.17 and blockage ratio of 2 per cent, and it was found that this had an insignificant effect on the wake, when modelled using a volume of fluid-free surface model. Therefore, in the present work a free-slip wall was used which does not deform at the free surface and significantly improves computational time and convergence.

(b) Domain and boundary conditions

The computational domain was a copy of the dimensions of the experimental facilities used for the wake measurements [21], and is shown in figure 4. The inlet profiles for velocity and turbulence are shown in figure 5. While figure 5b shows that the turbulence intensity in the model is over-predicted 1D upstream of the turbine, figure 5c shows that it dissipates rapidly, and falls to a value of 5.9 per cent between 2D and 4D downstream of the disc. When modelling wake dissipation, it is important that the ambient turbulence in the free stream flow surrounding the wake is correct, and it was therefore considered that high values of turbulence intensity upstream of the turbine could be tolerated to achieve accurate values downstream around the wake.

(c) Mesh

A sample is shown in figure 6. Mesh parameters are shown in table 1. A mesh refinement ratio of 1.5 was used to achieve coarse, medium and fine meshes for a mesh independence study. Richardson extrapolation has been performed and extrapolated relative error (ERE) and the grid convergence index (GCI) were calculated. Details of these techniques are outlined in [32].

(d) Discretization error

Table 2 shows the discretization error for the centreline velocity and the coefficient of thrust. (Note: the ERE and GCI values are for the fine mesh.) The results show that convergence is oscillatory and the maximum GCI error is nine per cent which occurs on the centreline at 4D downstream. The ambient turbulence intensity (e.g. values at y/D < 0.5 and y/D > 2) is susceptible to changes in mesh density and this has a significant effect on the discretization error.
Figure 5. Inlet velocity and turbulence profiles. (a) Depth-wise velocity profile $1D$ upstream of the turbine. (b) Depth-wise turbulence intensity profile $1D$ upstream of the turbine. (c) Variation in turbulence intensity downstream of the inlet at $100$ mm from the bed. The disc is located at $x/D = 0$.

Figure 6. Mesh, view from inlet, disc at centre of image.

Table 1. Mesh parameters for Ifremer case.

<table>
<thead>
<tr>
<th>mesh</th>
<th>elements</th>
<th>nodes</th>
<th>ELR$^a$</th>
<th>EVR$^b$</th>
<th>max. FA$^c$</th>
<th>min. FA$^d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>coarse</td>
<td>$9.14 \times 10^5$</td>
<td>$9.34 \times 10^5$</td>
<td>$&lt;224.9$</td>
<td>$&lt;2.50$</td>
<td>$&lt;135.0^\circ$</td>
<td>$&gt;51.0^\circ$</td>
</tr>
<tr>
<td>medium</td>
<td>$3.41 \times 10^6$</td>
<td>$3.46 \times 10^6$</td>
<td>$&lt;233.7$</td>
<td>$&lt;2.66$</td>
<td>$&lt;135.0^\circ$</td>
<td>$&gt;48.8^\circ$</td>
</tr>
<tr>
<td>fine</td>
<td>$12.32 \times 10^6$</td>
<td>$12.44 \times 10^6$</td>
<td>$&lt;242.6$</td>
<td>$&lt;3.06$</td>
<td>$&lt;135.0^\circ$</td>
<td>$&gt;47.5^\circ$</td>
</tr>
</tbody>
</table>

$^a$Edge length ratio.
$^b$Element volume ratio.
$^c$Maximum face angle.
$^d$Minimum face angle.
Figure 7. Integrated performance calculated using RANS + BE (crosses), BEMT (solid line) and from experiments (diamonds). (a) Power coefficient, (b) thrust coefficient.

Table 2. Analysis of discretization error at a tip speed ratio of 8.4.

<table>
<thead>
<tr>
<th>parameter</th>
<th>C_t</th>
<th>normalized centreline velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>4D</td>
</tr>
<tr>
<td>fine</td>
<td>1.115</td>
<td>0.546</td>
</tr>
<tr>
<td>medium</td>
<td>1.102</td>
<td>0.471</td>
</tr>
<tr>
<td>coarse</td>
<td>1.109</td>
<td>0.696</td>
</tr>
<tr>
<td>extrapolated</td>
<td>1.131</td>
<td>0.586</td>
</tr>
<tr>
<td>ERE (% fine)</td>
<td>1.3</td>
<td>6.7</td>
</tr>
<tr>
<td>GCI (% fine)</td>
<td>1.7</td>
<td>9.0</td>
</tr>
</tbody>
</table>

(e) Iterative convergence and computational time

Over the final 50 iterations normalized parameters varied by less than 0.16 per cent on the fine mesh. Computational time and convergence errors in the mass and momentum equations are shown in table 3.

5. Results and discussion

(a) Turbine performance

The accuracy of the RANS + BE model to predict turbine performance over a range of TSR is presented in figure 7. The figure shows the integrated power and thrust coefficients for the turbine for RANS + BE simulations, BEMT predictions and experimental data after correction for blockage. The data for the RANS + BE model are shown for a low ambient turbulence case (close to 0% turbulence intensity at the rotor), to match the assumptions of BEMT theory as closely as possible (BEMT assumes no turbulence and one-dimensional flow) and the conditions in a towing tank.
Figure 8. Comparison between RANS + Disk and RANS + BE models; centreline velocity and turbulence intensity profiles (solid line, RANS + BE model; dashed line, RANS + Disk; crosses, experimental data [21]). (Online version in colour.)

Table 3. Convergence errors in mass and momentum equations.

<table>
<thead>
<tr>
<th>mesh</th>
<th>max. r.m.s.e.</th>
<th>max. errors</th>
<th>procs.</th>
<th>max. time per proc</th>
</tr>
</thead>
<tbody>
<tr>
<td>coarse</td>
<td>$5.70 \times 10^{-5}$</td>
<td>$3.1 \times 10^{-3}$</td>
<td>8</td>
<td>00.24.23</td>
</tr>
<tr>
<td>medium</td>
<td>$1.98 \times 10^{-5}$</td>
<td>$1.9 \times 10^{-3}$</td>
<td>16</td>
<td>01.02.15</td>
</tr>
<tr>
<td>fine</td>
<td>$9.2 \times 10^{-6}$</td>
<td>$1.4 \times 10^{-3}$</td>
<td>16</td>
<td>03.42.15</td>
</tr>
</tbody>
</table>

Figure 7a shows that the RANS + BE model has a slight tendency to under-predict the power coefficient. At TSR > 6.5 there is a deviation between the BEMT predictions and both the experiments and CFD simulations, suggesting limitations with the BEMT model. Figure 7b shows the RANS + BE model over-predicts the thrust coefficient of the turbine, compared with experimental results and BEMT. In general, the model appears to be conservative since it under-predicts how much of the energy exerted on the turbine will be converted into rotation on the blades and useful power, but over-predicts the thrust and loadings acting on the turbine.

(b) Wake

The accuracy of both the RANS + Disk and RANS + BE models to predict the wake is presented in figures 8 and 9. On the centreline of the wake figure 8 shows that the uniform disc generates higher near-wake turbulence intensities than occur with the BE model. This is caused by the turbulence source terms used with the uniform disc model. In this study, turbulence source terms were not used with the BE model, since the velocity shear in the wake (caused by varying radial source terms) generates turbulence. It is observed that higher near-wake turbulence causes the wake to recover more quickly, as expected [15].
To assess the agreement on the centreline the coefficient of determination \([33]\) has been calculated for two different actuator turbine models. This approach has previously been used for comparing between wind turbine modelling data and experiments \([34]\). The coefficient of determination was 0.94 for the RANS + BE model and 0.92 for the RANS + Disk model (where 1 indicates perfect agreement). Comparisons of centreline turbulence intensity found a coefficient of determination of 0.67 for the RANS + BE model, and 0.07 for the uniform disc model (which over-predicted turbulence between 0\(D\) and 7\(D\) downstream). Figure 9b shows that after 8\(D\) the centreline velocity and turbulence intensity profiles through the water column (on the wake centreline) are similar between the two models.

A similar trend is also observed in the centreline velocity profiles (figure 8). The velocities and turbulence intensities are higher in the wake using the uniform model with turbulence sources, indicating that the wake is recovering faster. Figure 8 also shows the influence of modelling the free surface as a free-slip boundary condition, as the velocity at the free surface is constantly over-predicted as the boundary layer is not correctly modelled.

Both figures 10 and 11 demonstrate the difference in velocities and turbulence intensities between the two turbine models in more detail. The variation in power extraction with turbine radius in the BE model causes a variation in near-wake velocity (figure 10a). The velocity gradients cause turbulence kinetic energy to be generated, and increase the turbulence intensity in the near wake (figure 11a). In the near wake of the uniform model velocities are near constant with radius (figure 10b). Similarly the uniform model turbulence source at the disc results in uniform turbulence intensities in the wake (figure 11b).

6. Conclusions

This study has demonstrated that both models accurately predict the time averaged velocities and turbulence intensities in the wake of a turbine. The results provide a conservative prediction of the performance and loadings on a turbine, at a blockage ratio of 6 per cent and Froude number of 0.18. These parameters are similar to those expected at full-scale tidal sites.
Figure 10. Contours of velocity magnitude (m s$^{-1}$) on the $xy$ centre plane. (a) RANS + BE, (b) uniform disc. (Online version in colour.)

Figure 11. Contours of turbulence intensity on the $xy$ centre. (a) RANS + BE, (b) uniform disc. (Online version in colour.)
Over a range of tip speed ratios the turbine power coefficient predicted using the RANS + BE model was found to quantitatively compare well with experimental data. Although the model has a tendency to over-predict the thrust coefficient, this is not a significant problem as this technique is only suitable for assessing interactions within a farm of tidal turbines not absolute forces on an individual turbine blade.

In the wake, the high levels of the coefficient of determination for velocity demonstrate good confidence in the prediction of the far-wake velocities. The good agreement with turbulence intensity is achieved only after 8D. To further improve the quality of the validation, further work is required to model the friction component of the free surface.

The RANS + BE model has generally been shown to be preferable to the uniform disc approach for modelling farms of tidal stream turbines as: (i) the wake measurements show a higher level of agreement; (ii) the power of the turbine can be estimated; and (iii) it removes the requirements for empirical turbulence source terms at the turbine location.

References


29. Harrison ME. 2011 The accuracy of the actuator disc-RANS model for predicting the performance and far wake of a horizontal axis tidal stream turbine. PhD dissertation, Faculty of Engineering and the Environment, University of Southampton, UK.


