CFD simulations of a full-scale tidal turbine: comparison of LES and RANS with field data

Citation for published version (APA):

Published in:
host publication

Citing this paper
Please note that where the full-text provided on Manchester Research Explorer is the Author Accepted Manuscript or Proof version this may differ from the final Published version. If citing, it is advised that you check and use the publisher’s definitive version.

General rights
Copyright and moral rights for the publications made accessible in the Research Explorer are retained by the authors and/or other copyright owners and it is a condition of accessing publications that users recognise and abide by the legal requirements associated with these rights.

Takedown policy
If you believe that this document breaches copyright please refer to the University of Manchester’s Takedown Procedures [http://man.ac.uk/04Y6Bo] or contact uml.scholarlycommunications@manchester.ac.uk providing relevant details, so we can investigate your claim.
CFD Simulations of a Full-Scale Tidal Turbine: Comparison of LES and RANS with Field Data

Umair Ahmed¹, Imran Afgan², David D. Apsley³, Tim Stallard⁴, Peter K. Stansby⁵

School of MACE, University of Manchester, England
¹umair.ahmed@manchester.ac.uk
²imran.afgan@manchester.ac.uk
³david.d.apsley@manchester.ac.uk
⁴tim.stallard@manchester.ac.uk
⁵peter.k.stansby@manchester.ac.uk

Abstract— CFD simulations have been performed for a geometry-resolved full-scale tidal-stream turbine and compared with experimental data from the EMEC test site in the Orkney Isles. The mesh comprises two regions: a rotating part, containing the turbine, and a stationary outer part, including the support tower. A sliding-mesh interface couples the two parts.

Initially, Reynolds-averaged Navier-Stokes and large-eddy simulations were performed using an inflow velocity profile representative of the test site but low inflow turbulence, yielding satisfactory mean power coefficients. LES with synthetic turbulence prescribed at inlet was then employed to try to predict realistic load fluctuations. Load fluctuations (power, thrust and blade bending moments) may arise from onset mean velocity shear, influence of the support tower, blade-generated turbulence, approach-flow turbulence and waves. Inflow statistics were prescribed to match the vertical distribution of mean velocity, Reynolds stresses and length scales determined from a channel-flow simulation, with additional factoring of stresses and length scales to match as far as possible those measured on-site. LES simulations with synthetic turbulence at inflow satisfactorily reproduce the spectral distribution of blade bending moments provided that spectra are normalised by variance to reflect the relatively small number of rotations computed.

Keywords— Tidal-stream turbine; computational fluid dynamics; large-eddy simulation; synthetic-eddy modelling; load spectra.

1. INTRODUCTION

The production of energy from tidal stream turbines (TSTs) presents an attractive opportunity, with significant potential resources around the coast of Britain [1]. Advantages include predictability and large energy density, no impounding of waters and limited visual impact. Disadvantages or challenges include the demanding environment, interaction with marine wildlife and obstruction of narrow shipping lanes. Recent awards for full-scale deployment have been made by the French government to two consortia, DCNS/EDF and Alstom/GDF Suez, whilst similar government-funded undertakings have been initiated in Canada and Japan (IHI/Toshiba). Full-scale field tests have been conducted around the world at various sites, notably at the EMEC test site in the Orkney Isles and the FORCE site in the Bay of Fundy, Nova Scotia.

Laboratory studies of TSTs have been conducted in laboratory flumes or towing tanks, including the effects of cavitation [2], waves [3] and turbulence [4]. An important consideration is the interaction of multiple turbines in an array [5], where questions include both degree of effective blockage and downstream wake recovery.

The cost of field or laboratory trials means that much initial design work is undertaken by numerical simulation. The mainstay of industrial design methods is Blade Element Momentum (BEM) Theory, originally developed for wind energy [6], where the aerodynamic load coefficients of individual aerofoil blade sections inform an overall control-volume momentum balance. For tidal stream applications this has been implemented, for example, in the code Tidal Bladed of DNV-GL. Although such models have been successful in replicating power coefficients under steady-flow design conditions, and are well integrated with structural dynamics and electrical powertrain models, they incorporate, at best, very empirical procedures for dealing with some of the challenges in the field: for example, approach-flow velocity shear and turbulence. Computational Fluid Dynamics (CFD) is capable of addressing these, either with blade-resolved calculations [7] or with blades represented as rotating momentum sources or actuator lines [8].

Fluctuating loads on tidal turbines affect both operational performance and design life. Fluctuations arise due to fixed-frequency cyclic effects (onset mean-velocity shear; influence of support tower and surrounding boundaries) and the full spectra of eddies due to blade-generated turbulence (high-frequency), approach-flow turbulence (low to mid frequencies) and waves (low frequencies). Whilst the cyclic fluctuations can be simulated with Reynolds-averaged Navier Stokes (RANS) solvers, the full spectra of turbulent fluctuations can only be addressed by large-eddy simulations (LES), with a representative spectrum of turbulence in the onset flow.

This paper reports blade-resolved RANS and LES of a full-scale tidal turbine currently being deployed for testing at the EMEC site by Alstom. The simulations use the open-source CFD program Code_Saturne of EDF. A geometrically-accurate representation of the turbine rotor is imbedded in a rotating inner region of cells, coupled to a stationary outer region by a sliding interface [9]. Initially, both RANS and LES were used to provide baseline simulations with
(nominally) zero inflow turbulence. Then LES was used to investigate the effects of realistic levels of inflow turbulence. A synthetic-eddy method (SEM) provided representative turbulence at inflow, the profiles of mean velocity, Reynolds stresses and turbulent length scales having been obtained from a separate channel-flow simulation (at much lower Reynolds number), with some rescaling of length scales and Reynolds stresses for the conditions met on site. More detail of the validation and verification of the methods used for RANS and LES can be found in references [9] and [7] respectively.

The structure of the rest of the paper is as follows. Section II describes the geometry, boundary conditions and meshing and defines the relevant load parameters. Section III details the numerical method, including the choice of inflow conditions and the method for synthesising inlet turbulence. Section IV presents results of simulations defined by conditions at the EMEC site, with emphasis on blade-load fluctuations. Section V summarises the main findings and outlines ongoing research priorities.

II. GEOMETRY AND LOAD PARAMETERS

A. Turbine Geometry and Mesh

The rotating element is a 3-bladed turbine rotor with swept diameter $D = 18.3$ m. The geometry of the blades and nacelle was provided by Alstom, based on a 1 MW turbine being tested at the EMEC site in the Orkneys. Only a single blade pitch was simulated. The design conditions simulated here used nominal hub-height velocities of $1.8 \text{– } 2.7 \text{ m s}^{-1}$, with tip-speed ratios of $5 \text{– } 6$.

A block-structured mesh was produced using ICEM, part of the ANSYS Fluent suite. The basic geometry and domain dimensions are shown in Fig. 1. To permit rotation an inner cylindrical region of cells (diameter $1.09D$), containing the turbine rotor, rotates inside a stationary outer domain, which includes the turbine support tower. The two domains are coupled by a sliding interface [9], which provides internal Dirichlet boundary conditions for all flow variables. Meshes of 8.4 million and 17.6 million cells were used for RANS and LES calculations, respectively. In the LES mesh about 10 million cells are used in the rotating region. A similar level of detail in the wake region would, however, be computationally impractical. Fig. 2 and Fig. 3 show details of the mesh.

![Fig. 1 Geometry and domain dimensions.](image1)

B. Load Parameters

To cover a wide variety of inflow conditions results are presented in non-dimensional form. The main performance-related parameters are defined below. Here, $R$ is the tip radius, $A$ is the rotor swept area and $\Omega$ is the angular velocity of rotation. $U_0$ is a suitable approach-flow reference velocity, here taken as the hub-height velocity to reflect local measurements on site. (In the CFD calculations this is taken one diameter upstream of the turbine rather than at the inflow.

![Fig. 2 Surface mesh on the rotor.](image2)

![Fig. 3 Cross-stream mesh showing the interface region.](image3)
Two particular moment axes are considered for blade bending moments. Fig. 4 defines the axes for flapwise (about chord line) and edgewise (about pitch axis) moments. Only the former is reported here.

Experimental loading data was measured and provided by Alstom and is to be reported separately at this conference.

![Fig. 4 Definition of axes for blade bending moments.](image)

### III. COMPUTATIONAL DETAILS

Calculations were undertaken with the open-source CFD solver Code_Saturne [10]. The code benefits from extensive parallelisation (simulations here used typically 4096 processor cores on EDF’s Blue Gene Q supercomputer) and the ability to include user routines – in this case, to implement a sliding-mesh interface. Typical computation times for LES were about 1 week per turbine rotation.

#### A. Turbulence Modelling

Two levels of turbulence modelling were undertaken: RANS calculations using the SST k-ω model of [11], and LES calculations with the dynamic subgrid-scale model of [12], as modified by the popular least-squares formulation of [13]. In the latter, the subgrid-scale eddy viscosity is given by:

\[
\frac{v_{s,SGS}}{\rho} = C\Delta [\mathbf{S}] ,
\]

where

\[
||S|| = \sqrt{2S_y S_y^\prime} ,
\]

\[
S_y = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right)
\]

Here, the result of minimising the squared difference between unresolved stress and strain on two different scales \( \Delta \) and \( \hat{\Delta} \) is that \( C \) is not constant, but given locally by

\[
C = \frac{L_y M_{yy}}{M_y \hat{M}_{yy}}
\]

where

\[
L_y = -(<u_i u_j> - <u_i> <u_j>)
\]

and

\[
M_{yy} = 2\hat{\Delta} - \mathbf{S} > <S_y> - 2\hat{\Delta} <||\mathbf{S}||S_y>
\]

\(< > \) denotes a spatial average of the \( \Delta \)-resolved velocities (i.e. those in the computation) over the larger filter width \( \hat{\Delta} \). In Code_Saturne this corresponds to the “extended neighbourhood” of a cell, or all cells sharing a common vertex). For stability reasons \( C \) was constrained to lie between 0 and 0.13². In particular, the case \( C < 0 \) (“backscatter”) was not permitted.

Resolving full viscous boundary layers at such high Reynolds numbers would have been prohibitively expensive, and both RANS and LES calculations used standard wall functions on all solid surfaces. On the LES mesh typical blade-tip \( y^+ \) values were about 300.

#### B. Inflow Conditions

Two types of inflow conditions were considered: nominally zero turbulence (with a prescribed mean-velocity profile based on a representative curve fit to the flood-tide currents at the EMEC site, [14]) and a deep turbulent boundary layer (with mean and turbulence profiles determined from a separate channel-flow simulation).

With RANS simulations any turbulence supplied at inflow was not maintained by bed-generated turbulence and was largely dissipated by the time it reached the rotor, so only the zero-turbulence inflow was considered. For LES a separate fully-developed channel-flow simulation was undertaken (at the much lower \( Re_\#: 9300 \) – a friction Reynolds number based on the full-scale turbine would have been about 630000) to provide non-dimensional profiles of mean velocity, Reynolds stresses and turbulent length scales; these could then be scaled to the desired bulk velocity \( U_b \) and depth \( h \). The mean-velocity profiles employed in zero-turbulence and representative-turbulence simulations are shown in Fig. 5, whilst the Reynolds stresses and length scales determined by fully-developed channel-flow simulations are shown in Fig. 6 and Fig. 7 respectively. Subsequent analysis of data from the EMEC site suggested smaller length scales and greater Reynolds stresses. Accordingly, a further set of simulations were conducted, in which the turbulent length scales and Reynolds stresses derived from channel-flow simulations were multiplied by constant factors 0.5 and 1.8 respectively, to bring them into line with measured data at hub height.
Synthetic eddy modelling (SEM) was used to supply LES with a time-varying inlet velocity field having the desired statistical distribution of Reynolds stresses and turbulent length scales. In the implementation here, based on the work of [15], fluctuating velocities are generated from eddies advected through a virtual box (volume $V_B$) containing the nominal inlet plane. As each eddy leaves the box another eddy is generated at a random location on the opposite side of the box. The velocity fluctuations are given by

$$u'(\mathbf{x}) = \frac{1}{\sqrt{N}} \sum_{e} a_{ij} \epsilon_{ij} f_L(\mathbf{r}_e)$$

where $N$ is the number of eddies $e$ in the box, $\mathbf{r}_e = \mathbf{x} - \mathbf{x}_e$ is the displacement relative to the eddy centre, $a_{ij}$ are the Lund coefficients (Cholesky decomposition $\mathbf{a}^T \mathbf{a}$ of symmetric tensor $\mathbf{u} \mathbf{u}$), $\epsilon_{ij}$ are a set of random numbers with mean 0 and variance 1, and $f_L$ is a shape function depending on integral length scales $L_x, L_y, L_z$ (which are different for each velocity component). Full details can be found in [15].

A summary of the cases undertaken is given in Table I.

<table>
<thead>
<tr>
<th>Case</th>
<th>Turbulence closure</th>
<th>Inlet mean velocity</th>
<th>Inlet turbulence</th>
<th>$U_{hub}$ (m s$^{-1}$)</th>
<th>TSR</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>RANS (SST $k-\omega$)</td>
<td>Flood-tide profile</td>
<td>Zero (nominal)</td>
<td>1.84</td>
<td>5.86</td>
</tr>
<tr>
<td>B</td>
<td>LES</td>
<td>Flood-tide profile</td>
<td>Zero</td>
<td>1.85</td>
<td>5.86</td>
</tr>
<tr>
<td>C</td>
<td>LES</td>
<td>Channel flow</td>
<td>SEM based on channel flow</td>
<td>1.73</td>
<td>5.07</td>
</tr>
<tr>
<td>D</td>
<td>LES</td>
<td>Channel flow</td>
<td>SEM, with factored length scales and stresses</td>
<td>2.48</td>
<td>5.07</td>
</tr>
</tbody>
</table>

IV. RESULTS

A: Flow Field

Fig. 8 shows instantaneous views of the approach flow and near wake for LES simulations, via shaded plots of the streamwise velocity component. Mean inflow velocities are largely maintained up to the point where the effect of the rotor is felt, about $\frac{1}{2}$ to 1 diameters upstream. The SEM calculations show the advection of turbulent eddies with a streamwise length comparable to the water depth. Unlike the flow behind a bluff body, there is a relatively narrow and sharply-defined wake, with velocities dropping to about half their approach-flow value immediately downstream of the rotor. The near wake spreads comparatively little radially beyond the rotor disc, with slightly greater spreading rate in the higher turbulence cases. This interacts with quite a significant wake behind the support tower.
Fig. 8 Instantaneous LES flow field showing streamwise mean velocity: (a) zero inflow turbulence (Case B); (b) SEM based on channel flow (Case C); (c) SEM based on channel flow, with factored length scales and stresses (Case D).

Fig. 9 shows the vortical structures in the near wake. The plots are isosurfaces of streamwise vorticity, coloured by the vorticity indicator $Q$, where

$$Q = \frac{1}{2} (S_{ij}S_{ij} - \Omega_{ij}\Omega_{ij})$$

and $S_{ij}$ and $\Omega_{ij}$ are the mean strain and vorticity.

The main vortical structures are shed from the blade tips, with additional vorticity developed on the support tower. The tip vortices serve to delineate and constrain the wake as they are swept downstream. For the low-turbulence inflow the vortices are sharp and advected downstream with only gradual dissipation. In the simulations with significant inflow turbulence they become more significantly sheared and tend to break up rapidly.

B: Thrust and Power Coefficients

Fig 10 and Fig. 11 show the phase-averaged (over about 8 rotations) thrust and power coefficients respectively. Both
graphs show the signature of the support tower, with three blades passing it during each rotation and consequently experiencing an elevated downstream pressure at these points in the cycle. For low inflow turbulence a minimum in both thrust and power coincides almost exactly with the passing of a blade in front of the support tower. For the low-turbulence-inflow cases, RANS (k-ω SST) and LES predict similar phase-averaged thrust and power coefficients. For the latter coefficient all computational results are slightly less than the values obtained in the field (about 0.43 – 0.44). The various LES simulations suggest only small effects of blade-generated or inflow turbulence on phase-averaged load coefficients: more significant intracycle variations and frequency spectra are indicated below.

For low inflow turbulence a minimum in both thrust and power coincides almost exactly with the passing of a blade in front of the support tower. For the low-turbulence-inflow cases, RANS (k-ω SST) and LES predict similar phase-averaged thrust and power coefficients. For the latter coefficient all computational results are slightly less than the values obtained in the field (about 0.43 – 0.44). The various LES simulations suggest only small effects of blade-generated or inflow turbulence on phase-averaged load coefficients: more significant intracycle variations and frequency spectra are indicated below.

The degree of variation in loading for the whole rotor is deceptively small, especially when representing the sum of loads from three blades and phase-averaged over a number of cycles. By contrast, Fig. 12 shows the variation in power coefficient for one blade during a single rotation. The coefficient has been normalised by the cycle average. In contrast to the whole rotor, where summed contributions from the three blades tend to smooth out variation in power coefficient, the impact of the support tower (and a smaller effect of velocity shear) leads to power variation for a single blade of about ±13%, with results from RANS and LES calculations with zero turbulence at inflow being very similar. Approach-flow turbulence introduces substantially greater fluctuations in load, with significant implications for fatigue damage to the blade root. Fig. 12 also shows the expected behaviour when the turbulent lengthscales are halved at inlet, the corresponding reduction in turbulent timescales inducing more rapid fluctuations in load.

C: Blade Pressure Coefficient

Differences between RANS and LES simulations (with nominally zero turbulence at inflow) are examined by plotting the pressure coefficient on one blade at various radii. Pressure is here normalised to the local azimuthal speed ΩR rather than the approach-flow velocity in order to provide comparable range over the blade radius. The negative pressure coefficient is plotted for clarity and to emphasise lift. Fig 13 shows an instantaneous snapshot, whilst Fig 14 shows c_p values based on average pressure over several cycles. Differences between average pressures using RANS and LES are relatively small, mainly being confined to the suction surface downstream of peak suction; however, the instantaneous snapshot illustrates the large time-varying fluctuations beyond about 50% chord and 50% tip radius that are resolved by LES, even without inflow turbulence.
**D: Load Spectra**

A major objective of the present work was to compare fluctuations in load with measurements from instrumented blades on site. Data for this purpose was supplied by Alstom Ocean Energy and is reported separately at this conference.

Fig. 15 shows energy spectra of flapwise bending moment near blade root, comparing experiments, LES with zero inflow turbulence and LES with synthetic inflow turbulence. The frequency scale is $f/f_0$, where $f_0$ is the tower-passing frequency (i.e. whole rotation) of an individual blade (about 0.2 Hz). Experimental measurements were taken at 50 Hz, so the local spectral peak at $f/f_0 = 100$, which corresponds to a frequency of 20 Hz, is unexplained, but may correspond to a natural frequency of the instrumentation or the blade.

Comparing LES simulations in Fig. 15, when there is zero turbulence at inlet there is much less energy at frequencies intermediate between the tower-passing frequency ($f/f_0 = 1$) and the high frequencies typical of blade-generated turbulence ($f/f_0 > 20$). When onset turbulence is included there is much more energy in the intermediate range.

Because a relatively small number of rotations has been sampled a more appropriate comparison with experiment is made in Fig. 16, where spectral energy has been normalised by variance. Only the LES with representative onset turbulence is able to replicate the energy distribution throughout the whole frequency range.
V. CONCLUSIONS AND FURTHER WORK

Reynolds-averaged and large-eddy simulations have been performed for a geometry-resolved full-scale tidal-stream turbine. Mean and fluctuating load data has been compared with experimental measurements at the EMEC test site. Whilst phase-averaged loads (including the influence of the support tower) are satisfactorily reproduced by the popular SST k-ω model, only LES with representative synthetic turbulence at inflow is able to reproduce the full frequency range of load fluctuations on individual blades.

Future work is under way to characterise the near-wake structure of the flow, so that this can be input to simulations with a second downstream rotor. This is a large step in understanding the likely behaviour of tidal stream turbines operating in arrays. There remains a more open-ended aspiration to investigate the load fluctuations on turbines with additional onset-flow variations due to waves.

ACKNOWLEDGEMENTS

The ReDAPT project was commissioned by the ETI, with additional financial and computational support by EDF. Much code-development work was done by Dr James McNaughton during his PhD research. Experimental data was supplied by the University of Edinburgh (flow) and Alstom (load).

REFERENCES