Influence of turbulence on the wake of a marine current turbine simulator

T. Blackmore, W. M. J. Batten and A. S. Bahaj

Sustainable Energy Research Group, Energy and Climate Change Division, Faculty of Engineering and the Environment, University of Southampton, Southampton, UK

Marine current turbine commercial prototypes have now been deployed and arrays of multiple turbines under design. The tidal flows in which they operate are highly turbulent, but the characteristics of the inflow turbulence have not being considered in present design methods. This work considers the effects of inflow turbulence on the wake behind an actuator disc representation of a marine current turbine. Different turbulence intensities and integral length scales were generated in a large eddy simulation using a gridInlet, which produces turbulence from a grid pattern on the inlet boundary. The results highlight the significance of turbulence on the wake profile, with a different flow regime occurring for the zero turbulence case. Increasing the turbulence intensity reduced the velocity deficit and shifted the maximum deficit closer to the turbine. Increasing the integral length scale increased the velocity deficit close to the turbine due to an increased production of turbulent energy. However, the wake recovery was increased due to the higher rate of turbulent mixing causing the wake to expand. The implication of this work is that marine current turbine arrays could be further optimized, increasing the energy yield of the array when the site-specific turbulence characteristics are considered.

1. Introduction

Marine current turbine commercial prototypes have now been deployed and significant progress has been made in recent years, with many devices now in development [1–3]. The next phase of development will include the design and planning of multiple device arrays [2–4]. Estimating the extractable power from an array of turbines in a tidal channel may
be estimated using simple analytical expressions [5–7]. A turbine extracts energy from the flow causing a drop in velocity across the turbine rotor. This lower velocity region downstream of the turbine is known as the wake. Turbulent mixing spreads the wake and mixes in higher momentum fluid that did not pass through the turbine. Far downstream, the upstream velocity profile is once again observed, albeit with a slightly lower overall magnitude due to the energy extraction [8]. Factors that may limit the power extraction have been identified as blockage (the ratio of turbine area to channel cross-sectional area), drag from the support structure and mixing between the wake and surrounding flow [6]. For maximum power extraction, turbines would be arranged in rows forming fences that completely block the channel. However, for a combination of navigational, ecological and environmental reasons, tidal fences are unlikely and smaller arrays of turbines that only partially block the channel are envisaged. Maximizing the power extraction of a partially blocked channel array will require numerical or experimental models due to the complex dynamics and interactions between the constrained tidal flow and the turbines [6].

Small-scale laboratory experiments using porous disc rotor simulators [4,9] or numerical simulations using computational fluid dynamics (CFD) [10–14] have been used for array design. It has been shown that tidal flows are highly turbulent with intensities of around 10% and a broad range of length scales, or turbulent eddy sizes. The spectrum of length scales, or turbulent eddy sizes, are dependent on the specific site [15,16]. However, little attention has been given to the inflow characteristics used in these numerical or physical models, yet the turbulence levels in marine currents identified as issues to be addressed [1]. This work aims to investigate the effects of both turbulence intensity and length scale on the wake of an actuator disc. It is found that significant variations in the wake profile exist with different turbulence conditions.

Recent studies have shown how turbulence significantly affects the mean thrust coefficients of porous disc rotor simulators by up to 20% [17]. These variations in thrust coefficient are likely to be a result of changes in the wake profile. In this study, the turbulence was characterized as a turbulence intensity (equation (1.1)) and integral length scale (equation (1.2)). The integral length scale can be thought of as the size of the largest energy containing turbulent eddies.

\[ I = \frac{u'}{U} \]  

(1.1)

where \( I \) is the turbulence intensity, \( U \) is the mean velocity and \( u' \) is the RMS of the resolved velocity fluctuations.

The integral length scale, \( \ell_x \), can be calculated, using Taylor’s hypothesis, as the product of the mean velocity and the integral of the autocorrelation function, \( R_N(s) \), of the resolved fluctuations [18]

\[ \ell_x \approx U \int_0^\infty R_N(s) \, ds. \]  

(1.2)

It has previously been found that increasing turbulence intensity speeds up the wake recovery [19–21], and more recently that downstream turbines in an array have a faster wake recovery due to the added turbulence from the upstream turbine [4]. Wake velocities are often presented as a non-dimensional velocity deficit, \( U_D \), defined as

\[ U_D = \frac{U_\infty - U_w}{U_\infty}, \]  

(1.3)

where \( U_w \) is the velocity in the wake and \( U_\infty \) is the mean freestream velocity. Comparisons have been made between the measured centreline velocity deficit behind model turbine rotors and porous disc simulators with thrust coefficients of approximately 0.9 and with different turbulence intensities, as shown in figure 1. Note, the integral length scales were not considered in these studies, apart from in [24]. It can be seen for each individual study that increasing the turbulence intensity reduces the velocity deficit in the wake, particularly for [23,24]. However, cross-comparing between the different studies shows a large overall variation in the measured velocity deficits. At \( x/D = 4 \), the variation in velocity deficit is from \( \approx 0.1 \) [23] to \( \approx 0.6 \) [24] highlighting the sensitivity of the wake to different flow conditions.
Figure 1. Centreline velocity deficit from experiments with model rotors and porous disc rotor simulators with thrust coefficients of approximately 0.9 and turbulence intensities of approximately 5–15%, turbulent length scales unknown. (a) Chamorro & Porté-Agel [22], (b) Mycek et al. [23], (c) Myers et al. [24], (d) Krogstad & Eriksen [25], (e) Myers & Bahaj [9]. (Online version in colour.)

Considering the two cases of [22,23] from a wind tunnel and circulating water flume, respectively, and turbulence intensities of ≈ 5–15%, one would expect to see little difference in the wake profiles. But the velocity deficit reported by Mycek et al. [23] is significantly lower than reported by Chamorro & Porté-Agel [22]. The integral length scales are not reported, but it seems likely that the high-intensity case of [23] will have large length scales, whereas the low-intensity case will have small length scales due to the flow straighteners used to achieve low intensities. The wind tunnel experiments of [22] are likely to have small integral length scales, as commonly found in wind tunnels. Other factors may also influence the wake profile such as the support structure drag being included in the thrust coefficient calculation. For example, Mycek et al. [23] do include the drag contribution from the support structure and hence the rotor thrust may be lower, which could, in part, explain the lower velocity deficits. Owing to the limited information presented, it is not possible to draw a firm conclusion, but it seems likely that the wake recovery is sensitive to the integral length scale of the ambient flow.

CFD simulations with actuator discs are commonly used to investigate turbine wakes as they can provide information on the whole flow field at reduced cost compared with experiments [14,20,21,26,27]. An actuator disc represents the forces applied to the surrounding flow field by a turbine rotor as a momentum sink term in the computational domain. Therefore, the turbine rotor does not need to be resolved, reducing the computational requirements [10]. Reynolds averaged Navier–Stokes (RANS) turbulence models are often used to produce a steady-state solution with reasonable computational cost. It has been shown that a turbulence source is required at the actuator disc to account for the turbulence generated by the turbine [13,28]. RANS actuator disc simulations were used in [21] to investigate the effects of free stream turbulence and blade-induced turbulence on the thrust and power of a tidal turbine. The results show that the wake recovery is faster with high ambient turbulence, which is in agreement with [23,24,28]. However, steady-state RANS models cannot capture transient effects of turbulence. A large eddy simulation (LES) model has therefore been used where the largest energy containing eddies are fully resolved, so transient effects may be captured and with fewer modelling assumptions compared to RANS.

As found for actuator disc RANS models, a turbulence source is required at the actuator disc to account for the turbulence generated by a turbine. While the far wake velocity profiles compare well with experiments it was found that actuator disc LES models under predict the
turbulence intensity in the near wake and the mean velocity is over predicted [11]. An extension to the uniform actuator disc model was developed to generate turbulence for LES models [29]. The method, called a gridded-actuator disc, applies the momentum sink terms on a grid pattern within the actuator disc. This generates turbulence from the shear between the grid bars, akin to grid generated turbulence. Further, the generation of turbulence at the inlet to LES models is also problematic [30]. A new, and simple method has been created where a grid pattern is projected onto the inlet boundary producing grid-generated turbulence that develops naturally within the LES domain, called a gridInlet [31]. The generated turbulence then decays such that different turbulence intensities and length scales are produced at different distances downstream from the inlet.

The problem of understanding the effects of turbulence on a marine current turbine has therefore been simplified to investigate the effects of grid-generated turbulence on an actuator disc. Using this method, the integral length scales and turbulence intensities are controlled by changing the grid size and downstream distance from the inlet, as described in [31]. Grid-generated turbulence is approximately isotropic and uniform profiles of velocity are produced, eliminating effects from boundary layers and other shear gradients.

This work considers the effects of turbulence intensity and integral length scale on the wake behind a gridded actuator disc using an LES model with a gridInlet to produce grid-generated turbulence. The numerical method is first described before the general effects of turbulence are discussed along with the influence of turbulence on the centreline velocity profile. The isolated effects of turbulence intensity and integral length scale are then discussed and compared with published experimental data before the conclusions and implications of this work are made.

2. Numerical domain and method

The numerical domain for the analysis is shown in figure 2. The domain has a low blockage ratio of less than 1.5%, where the blockage ratio is defined as the ratio of actuator disc area to total cross-sectional area. The domain was split into a mesh of hexahedral cells using the OpenFOAM native meshing software BLOCKMESH and SNAPPYHEXMESH [32]. Simulations were performed with an actuator disc on different mesh densities refined from one cell across the actuator disc to 11 cells across the actuator disc. Figure 3 shows very little change in the wake velocity at 2D with different mesh densities. However, the thrust and power coefficients will be more sensitive as are functions of velocity squared, and cubed, respectively. It was found that a mesh density with a minimum of four cells across the disc thickness was required for mesh independence, as shown in figure 4. This results in a mesh with a cell size of 0.5 mm in the disc region. The mesh expands to 2 mm cell size up to 6D downstream and 4 mm cell size in the far wake. The mesh requirements at the inlet were based on the findings presented in [31] for a gridInlet. Table 1 details the overall number of cells and the turbulence characteristics for the five different cases used.

(a) Inlet grid-generated turbulence using a gridInlet

Two different inlets were used. The first was a uniform inlet with velocity of \( U_\infty = 0.3 \text{ m s}^{-1} \) and zero turbulence. The second a gridInlet [31] that develops turbulence in the computational domain by projecting a grid pattern of solid patches onto the inlet boundary. Flow enters the domain between the grid bars of the solid patch and turbulence generated from the resulting shear [31]. The grid turbulence decays such that different turbulence intensities and integral length scales are found at different downstream locations, as shown in figure 5. The inlet flow velocity is set to maintain flow rate with an average velocity of \( U_\infty = 0.3 \text{ m s}^{-1} \). Two different grid patterns were used with spacings of \( M = 100 \text{ mm} \) and \( M = 300 \text{ mm} \) to produce small-scale turbulence \( (\ell < D/3) \) and body scale turbulence \( (\ell > D/2) \), respectively. The downstream location of the actuator disc from the gridInlet is changed to produce different turbulence intensities, as detailed in table 1. The walls, bottom and surface were set as free slip due to the uniform velocity
Figure 2. Numerical domain with a gridInlet [31] to generate turbulence at the inlet, and a gridded actuator disc to generate turbulence within the actuator disc representing the turbulence generated by a turbine.

Figure 3. Velocity normalized with the free stream at 2D downstream of actuator discs with different mesh densities. Note, the x-axis starts from the centre of the disc. (Online version in colour.)

Figure 4. Actuator disc thrust and power coefficients with increasing mesh density or cells across the disc thickness. (Adapted from [29].)
profiles generated behind the grids. Therefore, boundary layer effects were not considered. Finally, the outlet was set to zero gradient.

(b) Numerical method

Simulations were run using a steady-state RANS simulation to obtain an initial condition for the LES model to reduce simulation time. A time step of $\Delta t = 0.001$ s was used to achieve solution stability for the mesh density used. Simulations were run until pseudo steady-state conditions were achieved. Results were then sampled from this time until the time-averaged turbulence statistics converged to a steady value, approximately 40 s, using the method described in [31]. The open source code OpenFOAM 2.0.1 was used to solve the incompressible finite volume discretization of the RANS and LES equations [32]. The RANS solution was used as the initial condition for the LES model to reduce computational time and the same mesh used for both RANS and LES.

(i) Reynolds averaged Navier–Stokes

The incompressible finite volume discretization of the RANS equations are [32]

$$\frac{\partial U_i}{\partial x_i} = 0$$

and

$$U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu + \nu_T \right) \left( \frac{\partial U_i}{\partial x_j} \right) + F_{Bi}$$
where \( U \) is the time averaged velocity, \( x \) is displacement, \( \rho \) is density, \( p \) is pressure, \( v \) is kinematic viscosity, \( \nu_T \) is the turbulent eddy viscosity and \( F_B \) is the momentum sink term representing the forces a turbine rotor would exert on the flow. The turbulent viscosity, \( \nu_T \), is resolved using the \( k-\epsilon \) turbulence model and run using the simpleFoam solver which uses the SIMPLE algorithm [33] for pressure–velocity coupling. Discretization was performed using the second-order central differencing method.

(ii) Large eddy simulation

The incompressible finite volume discretization of the filtered Navier–Stokes equations are [32]

\[
\frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu + \nu_{SGS} \frac{\partial \bar{u}_i}{\partial x_j} \right) + F_B, \tag{2.4}
\]

where \( u_i \) is velocity, \( t \) is time, \( \nu_{SGS} \) is the sub-grid scale viscosity and the over-bar represents filtered, grid-scale (GS) components. The filter function is taken as a simple box filter with filter width equal to the cube root of the cell volume. This splits the flow into GS and sub-grid scale (SGS) components. The effect of filtering introduces residual stresses that are modelled, using the Boussinesq approximation, as a SGS eddy viscosity using the Smagorinsky model [18]

\[
\nu_{SGS} = (C_s \Delta_g)^2 \sqrt{2\bar{S}_{ij}\bar{S}_{ij}}, \tag{2.5}
\]

where \( C_s \) is the Smagorinsky coefficient, \( \Delta_g \) is the grid size and \( \bar{S}_{ij} \) is the filtered strain tensor.

The equations were spatially discretized using second-order central differencing and temporally discretized using the first-order backward differencing method. The pimpleFoam solver was used which is a merged SIMPLE and PISO algorithm for pressure–velocity coupling and is a transient solver that is more stable for larger time steps [32].

(c) Performance coefficients and the gridded actuator disc

A gridded actuator disc was used to generate turbulence at the actuator disc to account for the turbulence generated by a turbine, as described in [29]. The thrust and power coefficients of a turbine are defined as [8]

\[
C_T = \frac{T}{(1/2)\rho U_{\infty}^2 A}, \tag{2.6}
\]

and

\[
C_P = \frac{P}{(1/2)\rho U_{\infty}^3 A}, \tag{2.7}
\]

where \( T \) is the thrust force, \( P \) is power output, \( U_{\infty} \) is the free-stream velocity and \( A \) is the turbine rotor area. The velocity at the turbine, \( U_t \) is reduced from the free-stream value due to the extraction of kinetic energy from the flow. The amount the velocity is retarded at the turbine is described by the axial induction factor, \( a \), which is defined as [8]

\[
a = 1 - \frac{U_t}{U_{\infty}} = 1 - \frac{C_P}{C_T}. \tag{2.8}
\]

The thrust coefficient may be related to the axial induction factor as,

\[
C_T = 4a(1 - a) \tag{2.9}
\]

The thrust force exerted by a turbine can be found by relating equations (2.6) and (2.9). Dividing by the total disc volume, \( V \), and density, for an incompressible fluid yields the
Figure 6. Distribution of momentum sink terms for the gridded actuator disc. (Adapted from [29].)

The momentum sink term is calculated at the start of the simulation using prescribed values of $U_\infty$ and $a$, and then held constant throughout the simulation. Therefore, this method does not enforce a specified velocity at the disc so the velocity, and hence thrust, can vary with different inflow conditions. This allows the effects of inflow turbulence on an actuator disc turbine representation to be investigated. For a uniform actuator disc, this body force is applied uniformly throughout the volume occupied by the turbine rotor. However, this does not take into account the turbulence that would be generated by a turbine rotor. For the gridded actuator disc method, the momentum sink term is split into a maximum and minimum value and applied on a grid pattern as shown in figure 6, such that the average body force is equivalent to the uniform case.

The maximum and minimum momentum sink terms are calculated by multiplying $F_B$ by the factors $\alpha$ and $\beta$, where $\alpha$ and $\beta$ are the maximum and minimum body force factors, respectively.

$$F_{B\text{(max.)}} = \alpha F_B$$

and

$$F_{B\text{(min.)}} = \beta F_B.$$ 

For a grid with porosity of 0.5 (equal areas of maximum and minimum momentum sink terms) $(\alpha + \beta)/2 = 1$.

The body force ratio, $r$, is defined as

$$r = \frac{\alpha}{\beta}.$$ 

Changing the bar width, $b_d$, and the body force ratio, $r$, controls the characteristics of the disc-generated turbulence, akin to grid-generated turbulence used in wind tunnels. A gridded disc was used with set axial induction factor of $a = 0.3$, body force ratio of $r = 2$, a bar width of $b_d = 0.01$ m and porosity of 0.5 as found from [29].
3. Results and discussion

The following results sections first consider the general effects of turbulence on a marine current turbine simulator. Centreline profiles are discussed before the specific consideration of how turbulence intensity and integral length scale affect the wake behind a marine current turbine simulator.

(a) Effects of turbulence on the wake of an actuator disc

Figure 7 shows the mean velocity deficit for the three cases with zero, low and high levels of turbulence and the edge of the wake shown as a white line. The resolved turbulent structure is shown as 3D contours of vorticity magnitude on the right-hand side of figure 7. Comparing the zero-turbulence case (figure 7a) with the low-turbulence case (figure 7b), it can be seen that the velocity deficit in the wake is reduced. However, the width of the wake is not significantly affected as seen by the white contour line defining the edge of the wake where \( U_w = 0.95U_\infty \) or \( U_D = 0.05 \).

Comparing now with the high-turbulence case (figure 7c), it can be seen that the velocity deficit initially increases close to the disc. However, the width of the wake is increased, over the low- or zero-turbulence case, resulting in a faster wake recovery with lower velocity deficit further downstream. This can also be seen by the vorticity contours breaking up and spreading with larger scale turbulence (figure 7c) over the zero-turbulence case (figure 7a).

(i) Centreline velocity deficit for all cases

Figure 8 shows the centreline profiles of velocity deficit for cases 1–5 (see table 1 for case details). It can be seen that the greatest velocity deficit, \( U_D > 1 \) at 2.7D, is recorded for case 1 with zero ambient turbulence. Note, a velocity deficit greater than 1 indicates flow reversal. In all other cases with ambient turbulence, the maximum velocity deficit is less than 1 and the wake recovery is much faster. This shows that tests performed with very low or zero turbulence, such as in a towing tank, may produce significantly different flow regimes and wake profiles to those expected in a turbulent flow. This can also be seen in the spreading of vortices behind the gridded actuator disc in figure 7b,c when compared with the zero-turbulence case in figure 7a.

Increasing the turbulence intensity from \( I = 5.6\% \) to 11.7\% (cases 3 and 2, respectively) reduces the velocity deficit and shifts the point of maximum deficit closer to the gridded actuator disc. While increasing the integral length scale from \( \ell/D = 0.25 \) to 0.55 (cases 2 and 4, respectively) increases the maximum velocity deficit, and further shifts the point of maximum velocity deficit closer to the actuator disc. However, increasing the integral length scale speeds up the wake recovery such that further downstream, the velocity deficit is lower for larger integral length scales.

These results are consistent with the findings of Harrison [34] where higher power outputs were recorded in an array of turbines when the ambient turbulence intensity was increased, suggesting the wake recovery was faster. The implications of this result is that arrays of multiple turbines may be further optimized when the specific turbulence characteristics are considered. Turbines could be located closer together when operating in a turbulent flow, so the power generated per unit area may be increased.

(ii) Effects of turbulence intensity, cases 1–3

Section 3a(i) showed that increasing the turbulence intensity reduced the velocity deficit and shifted the position of maximum deficit closer to the disc. This section further considers the effects of turbulence intensity using case 1 with zero turbulence, and cases 2 and 3 with similar length scales but different turbulence intensities.

Figure 9 shows the vertical profiles of axial velocity at 2, 4, 6 and 10D downstream of the disc for cases 1–3 with \( \approx 0.5, 10\% \) turbulence intensity, respectively. It can be seen that, at all locations downstream, increasing the turbulence intensity reduces the velocity deficit.
(a) zero turbulence

(b) small scale ($\ell_x/D = 0.27$), low intensity ($I = 5.6\%$)

(c) large scale ($\ell_x/D = 0.55$), high intensity ($I = 11.3\%$)

Figure 7. Contours of mean velocity deficit and 3D contours of vorticity magnitude for (a) zero turbulence, (b) small length-scale low-intensity turbulence and (c) high-intensity large length scales. The wake edge, defined as $U_D = 0.05$, is also shown as a white line. (Online version in colour.)

Figure 8. Velocity deficit downstream of a gridded actuator disc with different turbulence conditions. (Online version in colour.)
Considering the Reynolds stresses, which are defined for a constant density fluid as [18]:

\[ R_S = [u'u', v'v', w'w', u'v', u'w', v'w'], \]

where \( u'u' \), \( v'v' \), \( w'w' \) are the time average of the product of velocity fluctuations in the streamwise, transverse and vertical directions, respectively. \( u'v' \), \( v'w' \), \( v'w' \) are the time average of the cross products of velocity fluctuations. 

Figure 10 shows that \( u'u' \) is initially larger across the centre of the disc for higher ambient turbulence intensity, but further downstream \( u'u' \) becomes lower for the higher turbulence intensity case. Profiles of \( v'v' \) and \( w'w' \) showed similar trends due to the isotropic turbulence. While \( u'v' \) and \( v'w' \) were approximately zero, the profiles of \( u'w' \) shows similar trends to the profile of \( u'u' \) as shown in figure 11. Initially, \( u'w' \) is comparable for both cases with ambient turbulence, but further downstream \( u'w' \) becomes lower for the higher ambient turbulence intensity case. A faster wake recovery is observed for higher turbulence intensities due to the larger Reynolds stress components that are responsible for mixing the high-momentum fluid of the free stream with the low-momentum fluid in the wake. Figure 12 shows the position of the wake edge, \( e_W \), defined where \( U_w = 0.95U_\infty \) or \( U_D = 0.05 \). It can be seen that increasing turbulence intensity has little effect on the width of the wake, although the wake width is slightly increased over the zero-turbulence case. This is consistent with the RANS investigation of [21].

Overall, it has been shown that increasing the turbulence intensity reduces the velocity deficit in the wake due to larger Reynolds stress components, resulting in faster wake mixing with the high-momentum, free-stream flow. However, the wake width is unaffected by turbulence.
intensity. The implication of this result is that only considering the turbulence intensity of a flow, realistic wake expansion may not be observed. Section 3a(iii) considers how the integral length scale affects the wake profile.

(iii) Effects of integral length scale, cases 1,2,4

It was shown in §3a(i) that increasing the integral length scale increased the maximum velocity deficit and shifted the point of maximum deficit closer to the disc. This section further considers the effects of integral length scale using case 1 with zero turbulence and cases 2 and 4 with similar turbulence intensities but different integral length scales.

Figure 13 shows the vertical profiles of axial velocity at 2, 4, 6 and 10D downstream of the disc. It can be seen for case 1 with zero turbulence that the wake persists far downstream with minimal spreading. The wake recovery is faster with larger scale turbulence, at 2D Ud = 0.7 for $\ell_x/D = 0.55$ and $U_D = 0.6$ for $\ell_x/D = 0.25$, but further downstream the velocity deficit is lower for larger length scales.

Considering the Reynolds stresses, it can be seen in figure 14 that the component $u'w'$ is greatest for $\ell_x/D = 0.55$. By 10D, the profile is almost uniform for $\ell_x/D = 0.55$, whereas the wake profile is still visible for $\ell_x/D = 0.25$, with small-scale turbulence and still very pronounced for
case 1 with zero turbulence. It can also be seen that the profile of $u'u'$ spreads more rapidly for $\ell_x/D = 0.55$ than the case with small scale or zero turbulence. Further, the increase in $u'u'$ is greater for larger scale turbulence, approximately four times the increase in $u'u'$ over ambient conditions, at 2D downstream, for case 4 with large-scale turbulence over case 2 with small-scale turbulence. Similar trends are seen in the profiles of $v'v'$ and $w'w'$ due to the isotropic structure of the turbulence. As $u'u'$ is equivalent to twice the axial component of turbulent kinetic energy, this shows that a greater proportion of the flows energy, or momentum, is being converted in to turbulence with larger length scales in the ambient flow. This explains why the velocity deficit is greater for flows with larger integral length scales, as a greater proportion of the fluids momentum is being converted to turbulent energy.

Considering the cross component Reynolds stresses, $u'v'$ and $v'w'$ were approximately zero, but it can be seen in figure 15 that $u'w'$ for $\ell_x/D = 0.55$ is over three times that of $\ell_x/D = 0.25$, while significantly larger than the zero-turbulence case. This shows that the rate of mixing of high-momentum fluid from the ambient flow with the low-momentum fluid in the wake is much higher with larger scale turbulence. Considering the edge of the wake, $e_W$, it can be seen in figure 16 that the larger length scales cause the wake to expand to a greater width than the case with smaller scale turbulence. This further demonstrates the increase in mixing between the low-momentum wake and high-momentum free-stream flow with larger length-scale turbulence.

Overall, it has been shown that larger scales cause a greater increase in turbulent energy across the disc, hence the velocity deficit is increased close to the disc. This is due to the increased generation of turbulent energy further reducing the momentum of the fluid in the wake. However, owing to the increase in mixing with the high-momentum fluid in the surrounding flow with the
low-momentum fluid of the wake, the wake width is increased and the wake recovery occurs at a faster rate with larger scales of turbulence. Therefore, the velocity deficit is lower further downstream with larger scale turbulent flows.

(b) Comparison with published data from turbine rotors and porous discs

Comparisons have been made with the published centreline velocity deficits from different studies using turbine rotors and porous discs. It can be seen in figure 17a that the results with turbulence intensities of approximately 5% compare well with the published experimental data of the earlier studies [22–24]. While the velocity deficit is larger in the near wake, from 3D downstream the wake profiles from this model fall within the range of measured experimental data. Further, figure 17b shows comparisons of centreline velocity deficits with high levels of ambient turbulence intensity, $I \approx 10-15\%$. As before the model produces larger velocity deficits in the near wake, but beyond 3D the model compares well with the published experimental data and the results from this model fall within the experimental range.

The centreline velocity profiles obtained from this model with different length scales of turbulence fall within the range of published experimental data for both low- and high-intensity turbulent conditions. This demonstrates the significance of the integral length scale on the wake profile and could explain some of the variation in the published data. Other factors could influence the experimental variation, such as whether the support structure is included in the
Figure 17. Comparison of centreline velocity deficit with published experimental data with $C_T \approx 0.9$ and (a) low intensities, (b) high intensities. (a) Chamorro & Porté-Agel [22], (b) Mycek et al. [23], (c) Myers et al. [24], (d) this study: (i) small scales and (ii) large scales. (Online version in colour.)

thrust calculation. Overall, a detailed assessment is not possible as the inflow conditions have not been accurately reported from the experiments. Future modelling may therefore be improved by better reporting of turbulence inflow conditions. Further work is also required to confirm these trends on a turbine rotor. However, this work highlights the significance of the turbulence length scales and intensity on the wake profile. For realistic predictions of array performance and optimization of multiple turbine arrays, accurate reproduction of site-specific turbulence characteristics are required.

4. Conclusion

The inflow turbulence characteristics of small-scale experiments and numerical models of marine current turbines are often poorly reported. This work considers the effects of the inflow turbulence characteristics on the wake profile of an actuator disc representation of a marine current turbine. Grid-generated turbulence was developed with different turbulence intensities and integral length scales in an LES model using a gridInlet technique. Turbulence intensities of 5–10% and integral length scales of $\ell_x/D = 0.25–0.8$ were generated and their effects on the wake of a gridded actuator disc investigated. The findings of this study are summarized below:

— increasing the turbulence intensity of the ambient flow reduces the velocity deficit and shifts the location of maximum deficit closer to the disc (figure 8);
— turbulence intensity has a limited effect on the wake width (figure 12);
— increasing the integral length scale increases the velocity deficit in the near wake, shifts the location of maximum deficit closer to the disc and speeds up wake recovery (figure 8);
— increasing the integral length scale increases the wake width (figure 16);
— the results from this model compare well with published experimental data beyond 3D downstream, and the observed variation in centreline velocity profile may be attributed to differences in the inflow turbulence length scales (figure 17).

The main implication of this work relates to the design of multiple turbines arrays. If the site-specific turbulence characteristics are not considered, then sub-optimal arrays will be designed. For example, if arrays are designed in low-turbulence conditions the wake expansion and recovery rate will be lower than when operating in a highly turbulent flow. Turbines could therefore have been positioned closer together and the energy output per unit area of the array...
would be lower than optimum. However, if the turbulence characteristics are considered, the array may be optimized for maximum energy extraction per unit area and the economic viability of the array improved.

Further work is required on the characterization and reproduction of site-specific turbulence in small-scale experiments and numerical simulations. Further experimental work is currently being undertaken by the authors to fully characterize the inflow conditions to confirm these results on a turbine rotor, as well as addressing how turbulence affects the turbine itself from a loading and reliability perspective.

Acknowledgements. The authors acknowledge the use of the IRIDIS High Performance Computing Facility at the University of Southampton, in the completion of this work. This work is part of the marine energy activities at the University of Southampton, conducted and supported by the Energy and Climate Change Division & the Sustainable Energy Research Group within the Faculty of Engineering and the Environment (www.energy.soton.ac.uk).

Funding statement. This work was, in part, funded by the EPSRC and supported by the Energy and Climate Change Division & the Sustainable Energy Research Group within the Faculty of Engineering and the Environment at the University of Southampton (www.energy.soton.ac.uk).

References


34. Harrison ME. 2011 The accuracy of the actuator disc-RANS approach for modelling performance and wake characteristics of a horizontal axis tidal stream turbine. PhD, University of Southampton, Southampton, UK.