Large Eddy Simulation of the meandering of a wind turbine wake with stochastically generated boundary conditions

Yann-Aël Muller¹, Christian Masson² and Sandrine Aubrun¹

¹ Laboratoire PRISME, University of Orleans, 8 rue Léonard de Vinci 45072, Orléans, France
² École de Technologie Supérieure, 1100 rue Notre-Dame Ouest, Montréal (Québec) H3C1K3, Canada

E-mail: yann-ael.muller@laposte.net

Abstract.

Wind turbine wakes are known to be affected by the large atmospheric turbulent scales, which can cause trajectory variations within a wide frequency band. This phenomenon, called meandering, is suspected to be a cause of premature wear on turbines located inside wind farms. This work proposes a method to generate and apply synthetic turbulent velocity series as boundary conditions in a Large Eddy Simulation of an actuator disk in a flow with realistic turbulence characteristics. The stochastic generation method relies on the inverse Short-Time Fourier Transform (STFT) of a random vector field correlated in Fourier space according to the covariance tensor calculated from the homogeneous isotropic spectral tensor. In contrast with a single Fourier transform, the STFT allows the generation of arbitrarily large velocity fields. The generated series are used as boundary values on the inlet as well as on the lateral boundaries of the domain. This allows for sustained turbulent forcing on the whole length of the domain which is especially useful for a small computational domain relative to the size of the dominant turbulent scales.

1. Introduction

The study of the wake meandering represents a challenge in that although the general mechanism seems to be understood, the conditions for its existence are quite difficult to reproduce. It has been conjectured by Trujillo [1] that the meandering is a direct result of the large scale atmospheric turbulence. Specifically large atmospheric eddies, with scales significantly larger than that of the turbine can be assimilated to random transient drift conditions imposed on the turbine. This hypothesis has been tested on full scale measurements [2] as well as wind tunnel tests [3].

Several works deal with the numerical simulation of the wake meandering. Since the meandering is an essentially transient phenomenon, it is appropriately simulated with an unsteady simulation technique, such as unsteady RANS (Reynolds Averaged Navier Stokes) or LES. The task is computationally demanding, and some compromises must generally be made between the modelling of the rotor flow and near wake or the modelling of the large scale atmospheric turbulence and the far wake.

An approach commonly employed to ensure that the turbulence of the simulated atmospheric
flow is adequate is to use a stochastic method to generate turbulent velocity series which are then introduced inside the computational domain by mean of unsteady source terms [4].

The present work proposes a variation of this methodology by applying the stochastically generated series directly at the boundaries of the simulation. The stochastic generation of turbulent velocity series is a subject that has been approached by various authors [5, 6, 7], but the most commonly employed in the field of wind turbine simulation is probably that of Mann [8]. In comparison to the Mann method the generator used in this work simplifies the calculation of the spectral tensor by only considering the case where the turbulent field is isotropic. It is proposed to use a form of short-time Fourier transform to generate very long turbulent velocity time series.

2. Simulation setup

2.1. Objectives

As an initial attempt to test the methodology, a case simulation an uniform homogeneous isotropic turbulent flow with a 5m.s\(^{-1}\) average velocity is devised. The addition of a 40m diameter porous actuator disk inside the domain is also investigated. The turbulent velocity is prescribed at the boundaries according to the stochastic generation method.

The setup differs from purely decaying turbulence in that the turbulent velocities are applied not only at the entrance of the domain, but also on the lateral, top and bottom faces. The case is not entirely representative of a real atmospheric boundary layer flow, since the modelling of a sheared velocity profile is not considered. Additionally, the isotropic turbulence hypothesis is generally not strictly applicable to boundary layer turbulence. However the model is designed to be simple while still presenting the most relevant characteristics of the atmospheric turbulence for the simulation of the meandering of a wind turbine wake.

To this end, the integral scale and turbulent intensity of the generated velocity field are chosen to be representative of the conditions of a moderately rough atmospheric boundary layer. Specifically, the values are chosen to be similar to one of the cases presented by España in [9], where the integral scale is \(L_{uw} = 400\) m, the turbulence intensity is \(I_u = 0.126\) at hub height for an actuator disk of diameter \(D = 40\) m and induction factor \(a = 0.19\). The integral scale can be described as a characteristic coherence length of the turbulence. It can be calculated in two ways: one is to take the integral of the normalized autocorrelation curve of the velocity, the other is to find the wavelength of the maximum of \(F_{uu}(k_x) \ast k_x\), with \(F_{uu}(k_x)\) the one-dimensional power spectral density of \(u\) in the longitudinal direction.

2.2. Stochastic velocity series generation

The turbulent velocity series in wavenumber space \(z_i(k)\) are generated by multiplying a normally distributed random complex three-dimensional vector field \(n_i(k)\) by the spectral velocity covariance matrix \(C_{ij}(k)\) (4), which can be calculated using the Cholesky decomposition of the incompressible homogeneous isotropic spectral tensor \(\phi_{ij}(k)\) (1). \(z_i\) is then transformed to the spatial domain thanks to the inverse Fourier transform (5).

This is essentially equivalent to the Mann method when the shearing factor is \(\Gamma = 0\), which results in a field with no anisotropy [8]. This parameter of the Mann method does not appear in the development presented here, since only the isotropic case is considered.
\[ \phi_{ij}(k) = \frac{E(k)}{4\pi k^4} (\delta_{ij} k^2 - k_i k_j) \]  
(1)

\[ E(k) = AL^{5/3} \frac{(Lk)^4}{(1 + (Lk)^2)^{17/4}} \]  
(2)

\[ C_{ik}^*(k) C_{jk}(k) = \phi_{ij}(k) \]  
(3)

\[ z_i(k) = C_{ij}(k) * n_i(k) \]  
(4)

\[ u_i(x) = \int z_i(k) e^{ikx} dk \]  
(5)

- With \( k \) the wavenumber domain and \( x \) the spatial domain (both are three-dimensional fields).
- \( E(k) \) is the Von Kármán isotropic energy spectrum.
- \( A \) a constant related to the variance.
- \( L \) is the three-dimensional turbulent scale which is related to the integral scale. From [8], \( L = L_{ux}/0.816 \).

The result is a three-dimensional velocity field, which presents adequate coherence and spectral characteristics in all directions, for all velocity components. The required parameters for the generator are the mesh size and resolution, which directly affect the resolved spectral domain, as well as the variance and turbulent scale. The resulting field can be seen as an instantaneous snapshot of an homogeneous isotropic turbulence field, similar to what can be measured in a case of decaying grid turbulence, minus the average velocity.

The continuity equation for an incompressible flow in the spectral domain (7) is trivially obtained from the spatial form (6) by applying the property of the Fourier transform which linearizes the spatial differential operator. (7) is always verified for any field generated by the present method given the form of \( \phi_{ij} \) (1), which was originally derived in the incompressible case by J. Kampé de Feriet (1948), an explanation of which can also be found in [10] (chapter 2.4).

\[ \frac{\partial u_i}{\partial x_i} = 0 \]  
\[ k_i z_i = 0 \]  
(6)

When numerically calculating the divergence of one generated field however, the result may be found to be locally different from 0, even though it should always be 0 when averaged on a sufficiently large number of points. This can be caused by the differencing scheme used in the computation which may commonly not capture the high frequencies present in the synthetic field which extend up to the Nyquist frequency. Nevertheless, this error is assumed small enough that no correction is employed.

Since no assumption is made about the average velocity field or the time dynamic of the turbulence field, it is necessary to apply the Taylor hypothesis of frozen turbulence to the generated field in order to use the velocities as boundary values in an unsteady simulation.

The velocity values needed at every faces in the simulation are retrieved from the stochastically generated field by interpolation. At the subsequent time-step, the advection of the turbulent field by the average velocity is accounted for by interpolating in the stochastic field at a slightly displaced position.
Since the field calculated by the inverse Fourier transform has a fully defined coherence in the spatial domain, it must be fully computed in one step before the simulation is run. As a consequence the generated field cannot be of any arbitrary length and resolution. Indeed there is a practical limit to the number of points that can be transformed with reasonable computational resources, since even the fast Fourier transform algorithm has a computational cost proportional to $n \times \log(n)$, with $n$ the number of points.

However, flow regions separated by a very large distance need not be correlated, since the correlation levels for distances much larger than the integral scales will tend to 0. This property allows arbitrarily large turbulent velocity fields to be reconstructed by joining several fields individually generated with a Fourier transform as in (5), denoted from here on as $u_{ft}(x, \xi_i)$, where $\xi_i$ is the discrete variable which expresses the locality of any given field (the position around which it is centered). These sort of transformations are called Short-Time Fourier Transforms. The parameter $\xi_i$ is also known as the “slow time” of the transform because it has the same dimensionality as $x$ (or $t$ for time-frequency transforms). In this work there is no difference in the spectral characteristics of the fields, only the random vectors are chosen independently for different $\xi_i$.

To ensure continuity, the fields can be blended together with a windowing function. Most commonly, the windowing function is Gaussian, the specific appellation of the transform in this case is the Gabor transform. For the purpose of generating random velocity fields however, it is possible to use a truncated sinus cardinal windowing function $w(x)$ (8), which is narrowly bounded and has the nice property of preserving the variance of the interpolated fields, even when the overlap between windows is kept low. The overlap chosen in this work is 50%.

\[
  w(x) = \begin{cases} 
  \frac{\sin(\pi x)}{x} & \text{if } x \in [-a, a] \\
  0 & \text{otherwise}
  \end{cases} 
\]  

Or, in continuous form:

\[
  u(x) = \sum_{i, \text{overlapping windows}} w(x - \xi_i)u_{ft}(x, \xi_i) 
\]

\[
  u(x) = \int w(x - \xi)u_{ft}(x, \xi)d\xi
\]

With $a$ the width of the windowing function.

The advantage of using a field generated in this manner in contrast to data that may be obtained from a precursor simulation is that the desired scale and turbulent intensity can be directly prescribed as a parameter, instead of being defined indirectly by the setup of a precursor simulation.

2.3. Numerical simulation setup

The simulation software used is OpenFoam[11], more specifically the incompressible, short time-step (PISO) solver distributed with the OpenFoam numerical simulation framework. The traditional Smagorinsky turbulence model is used, with the constant $C_s = 0.167$. The time advancement scheme is backward Euler. The spatial interpolation schemes are all second order linear except for the divergence where the linear scheme may be completed with up to 20% of a first order upwind scheme to dampen eventual local oscillations of the solution (the filteredLinear scheme in OpenFoam).

The geometry is a box of dimension $lx = 500m$, $ly = 240m$, $lz = 140m$. $ly$ is chosen larger than $lz$ so as to reduce the blockage in the direction which the meandering is measured in.

The meshes are generated as nearly isotropic uniform Cartesian grids. The choice of a uniform mesh is desirable because it allows reasoning on the scales of the turbulence without
worrying about transitions between regions of different resolutions. Also, due to the meandering phenomenon, the wake of the porous disk is displaced significantly inside the computational domain and thus the choice of an area to refine is not obvious.

An uniform mesh however imposes limitations on the size and resolution of the computational mesh, which are made acceptable with the special treatment of the boundaries proposed in this paper.

Different resolutions were tried with the following cell counts for each direction: R1: 103x49x29, R2: 143x69x40, R3: 195x93x54. Comments on the grid convergence of some of the measured variables are made in section 3. The meshes were kept relatively coarse in order to be able to compute many time-steps, the total simulated time is 2000 seconds for all cases. The time-step are dynamically adjusted in order to keep the maximum Courant number under 1 with the additional constraint of having a fixed time-step every 1 second in order to record regularly sampled velocity values. The initial values inside the domain can be considered completely swept outside the domain after 120 seconds of simulated time since the average velocity is $U_\infty = 5 \, m.s^{-1}$, these values are used for the calculation of the statistics presented in section 3.

The boundary conditions are specified as follows:

- The turbulent viscosity is set as a zero gradient on all boundaries.
- The pressure is set as a zero gradient on all boundaries, while a fixed reference point is assigned inside the domain.
- Velocity:
  - On the inlet boundary the velocity is set as time varying non uniform fixed value as obtained from the generator.
  - On the outlet boundary, a zero gradient condition is used.
  - On the lateral boundaries (including top/bottom) the values of the stochastic generator are also used, albeit in a different fashion. The solution retained is to employ pure Dirichlet boundary conditions for inflow regions, and to treat outflow regions by applying pure Neumann boundary conditions to the tangential components of the velocity only, while still enforcing the normal component of the velocity from the stochastic generator with a pure Dirichlet boundary condition.

Although the average velocity is tangential to the lateral boundaries, significant turbulent exchanges take place through these boundaries, due to the combination of the high turbulence level, large integral scales and relatively small size of the domain. The particular kind of conditions used on the lateral boundaries for the velocity effectively enforces the flux through the boundary according to the stochastic velocity field, which is divergence free, while still managing to completely eliminate the oscillations of the velocity near outflow regions.

First attempts employed Dirichlet type boundary conditions, but it was found that this caused severe oscillations of the velocity near the regions of the boundaries where the flow exits the domain. This led to the use of a hybrid (Dirichlet/Neumann) boundary condition, where the coefficient $f$ governing the proportion of each condition in the final result is based on the direction of the velocity at the boundary. Flow entering the domain leads to a pure Dirichlet boundary condition, while flow leaving the domain leads to a pure Neumann condition. While this solved the problem of parasitic oscillations of the velocity near outflows, this allowed too much flow to exit the domain through the lateral boundaries, thus causing a slow down of the mean velocity inside the domain.

Hybrid boundary conditions with a fixed coefficient $f$ were also tried, but exhibit both limitations, that is insufficient chanelling of the main flow and oscillations near outflows, while solving neither completely.

The average velocity is not expected to have an influence on the dimensionless results, since the induction factor is fixed and the flow should be Reynolds independant.
• Individual fields $u_{fig}$ are cubes with a side length of 1200m and an uniform resolution of 15m. These fields are then interpolated to retrieve the values at the CFD boundary faces.

• The chosen integral scale $L_{ux}$ of the turbulence is 400m. The length scale $L$ can be expressed as $L = L_{ux}/0.816 = 490$ m, from [8].

• The turbulent intensity of the generated fields is set slightly higher than the desired value in the CFD ($I_u = 0.126$), since the turbulent intensity drops slightly when the velocities are introduced in the simulation, in all likeliness due to interpolation error.

The turbine is modelled by mean of a porous actuator disk with a diameter of $D = 40$ m and induction factor $a = 0.19$ at a position $x_{fan} = 160$m, which removes momentum from the flow as described by Froude’s theory. In practice this is implemented in the simulation as an internal boundary with a pressure jump but continuous velocity. Since the porous disk is circular and the mesh is Cartesian, it is inevitable that the disk’s internal boundary presents jagged edges. On the coarser mesh the shape of the internal boundary is especially rough. However the amount of momentum taken from the flow is fairly constant between mesh resolutions, as shown by the results in part 3.2.

The pressure jump $\Delta p$ is defined as a function of the local velocity within the disk’s plane $U_d$ and of the induction factor $a$ according to Froude’s actuator disk theory (13):

$$\Delta p = \frac{1}{2} \rho (U_0^2 - U_1^2) \quad (11)$$

$$U_1 = (1 - 2a)U_0 \quad (12)$$

$$U_d = \frac{U_0 + U_1}{2} \quad (13)$$

With $\rho$ the air density, $U_0$ and $U_1$ the axial upstream and downstream velocities.

Given these relations it is possible to give the pressure jump as a function of $U_d$, the velocity on the disk’s plane:

$$\Delta p = 2 \rho (a + a^2) U_d^2 \quad (14)$$

3. Results

3.1. Free flow, no actuator disk

Analysis of the results of the simulations seems to indicate that although average velocities are well converged, proper statistical convergence of local second order moments is not attained. The large turbulent scales found in the flow would require equivalently long simulation times, however this has proven challenging to achieve, particularly with the finest mesh. The coherence
time of the flow, with an integral scale of 400m and a mean advection velocity of 5m.s\(^{-1}\) is 80 seconds which is rather large compared to the 1880 seconds of simulated time that are used for the statistics. However it is worth mentioning that the synthetic turbulent velocity field is exactly the same for all meshes and cases presented here, so comparisons between them are still relevant.

The streamwise turbulent kinetic energy profile on figure 2 shows that the turbulent energy content within the flow stays within 10% of its initial value. Since there is no shear in the average inflow velocity profile, it could be expected that the turbulence would only decay inside the domain. The inflow velocity however contains very large turbulent structures, with an integral scale larger than the width of the domain. Since these large structures create significant shear inside the domain, there is a also a sustained production of smaller scale turbulence. The subgrid turbulent kinetic energy in the domain amounts to 2.0% of the total kinetic energy for R2 and 2.6% for R1, which is not unexpected given the large size of the integral scale relative to the mesh cell size.

![Figure 2](image.png)

**Figure 2.** Average streamwise profile of the resolved and total streamwise turbulent kinetic energy, for mesh R2.

The streamwise power spectra of the longitudinal velocity in the CFD domain and in the synthetic turbulence field are shown on figure 3. The turbulence measured inside the CFD domain is very close to that of the synthetic field in the production and inertial ranges, however it also shows some differences close to the sampling frequency of the synthetic field. It appears that the high frequency turbulence is the lowest close to the inlet, while it progressively increases along the longitudinal axis, finally approaching the level of the synthetic turbulence. This discrepancy may be a consequence of the mismatch between the resolved frequencies in the synthetic field and in the simulation, or it may also be due to interpolation and local continuity errors as discussed in part 2.3.

The average resolved turbulent intensity inside the domain for R1 is \(I_u = 0.121\), the subgrid turbulent intensity is \(I_{usgs} = 0.014\). For R2 \(I_u = 0.122\) and \(I_{usgs} = 0.012\). It was found that it is difficult to predict the exact turbulent intensity obtained inside the CFD domain from a given stochastic input field, which may possibly be explained by any of the reasons previously discussed.

Trials with different synthetic field resolutions were not considered practical for the case...
presented here because even with the short-time Fourier transform approach used, each individual fields must still be large enough to cover several times the prescribed integral scale, and each doubling of the number of points in each direction translates to an eight-fold \((2^3)\) increase of the total number of points required for each transform. The practical limit for the total simultaneous number of points used however is constrained by the available random access memory. Additionally, changing any of the synthetic field generation parameters will lead to a completely different outcome as the values supplied by the pseudo-random number generator, although strictly ordered and identical, will end up being assigned to different spectral coefficients thus making comparison between cases impossible without full statistical convergence of the results.

![Energy spectra](image1)

**Figure 3.** Energy spectra

### 3.2. With the actuator disk

![Average transverse profiles](image2)

**Figure 4.** Average transverse profiles of \(U_x\) four diameters downstream of the disk for R3
Due to the low number of cells around the actuator disk, the annular shearing region in the near wake behind the disk is not very well resolved. The average transverse profiles are shown on figure 4. We compare the recovery of the wake four diameters downstream of the disk for the different meshes in table 3.2. Unfortunately proper grid convergence is not attained, although the error itself is contained. It is possible that the varying ratios of the maximum resolved scales in the CFD and in the synthetic field is preventing convergence when increasing the mesh size.

| mesh | $U(x_{fan} + 4D)/U_{\infty}$ | R1   | R2   | R3   |
|------|-------------------------------|------|------|------|
|      |                               | 0.819| 0.827| 0.839|

**Table 1.** Velocity recovery in the wake of the actuator disk

In order to deduce an instantaneous wake position from the transverse velocity profiles, the local longitudinal velocity deficit for each point of the profile $\Delta u_x(y)$ is measured and used to compute a ”gravity centre” of the velocity deficit on the profile (15). The weighting is empirically chosen to be exponential so as to give more weight to extreme velocity deficits, while still accounting for the distribution of the deficit over the profile. This method gives time series of the wake position on the transverse axis. This same procedure has also been tested on wind tunnel data [3]. Another approach which was tried is to fit a Gaussian curve to the velocity profile to characterize the deficit, however it was found that the instantaneous velocity profiles can differ significantly from a smooth Gaussian profile given the turbulence level in the flow. The non linear optimization procedure required to retrieve the parameters of the Gaussian curve can be problematic in those conditions, this is avoided with the present method.

$$y_{\text{wake}} = y_{\text{fan}} + \frac{y_{\text{fan}} + D}{\sum_{y_i = y_{\text{fan}} - D} e^{\Delta u_x(y_i)} y_i}$$  \hspace{1cm} (15)

**Figure 5.** Time series of the transverse position of the wake centreline four diameters downstream of the actuator disk

It would also have been possible to locate the wake on a full two dimensional plane orthogonal to the main flow with a similar method, however since there is no wall in this simulation the results would not have been representative of the behaviour of a real wake in the vertical direction.

The standard deviation of the position of the wake four diameters downstream of the disk on the y axis is calculated for each mesh. The value is low pass filtered with a cutoff frequency...
Figure 6. Instantaneous absolute velocity inside the CFD domain in the (x,y) median cutting plane showing an extreme event of the wake meandering.

Equivalent to the average advection time over two diameters of the disk, which is not expected to affect the meandering frequencies [3], while diminishing the influence of the mesh resolution on the data. The filtered value for R1 it is $\theta_{\text{wake}} = 0.25D$, for R2 it is $\theta_{\text{wake}} = 0.28D$, for R3 it is $\theta_{\text{wake}} = 0.30D$. España in [9] gives a value of around $\theta_{\text{wake}} = 0.28D$ for the experimental data with similar turbulent conditions, while Aubrun in [12] gives for full scale data with a very different integral length scale $L_{ux} = 160m$ and turbulent intensity $I_u = 0.129$, a value of $\theta_{\text{wake}} = 0.11D$. It is to be noted that the methodologies for measuring the wake position in these references differs from the one used this work.

4. Conclusion
The methodology presented in this work has been specifically developed to simulate wind turbine wake meandering and gives interesting results even though no attempt has been made to simulate the atmospheric boundary layer yet. The results obtained support the assumption that the inflow low frequency content and integral scale are indeed the most important parameters when simulating the meandering.

The observed meandering is similar to the one which is measured in comparable experimental conditions. In contrast with experimental data, the LES simulation can potentially provide more detailed informations, especially time resolved spatial data which is difficult to obtain with experimental methods.

Bibliography
[1] Trujillo J J, Bingöl F, Larsen G C, Mann J and Kühn M 2011 Wind Energy 14 61–75 ISSN 10954244
[2] Bingöl F, Mann J and Larsen G C 2010 Wind Energy 13 51–61 ISSN 10954244, 10991824
[3] Muller Y A, Aubrun S, Loyer S and Masson C 2013 EWEA Proceedings
[4] Keck R E, Mikkelsen R, Troldborg N, de Maré M and Hansen K S 2013 Wind Energy n/a–n/a ISSN 10954244
[5] Davidson L and Billson M 2006 International Journal of Heat and Fluid Flow 27 1028–1042 ISSN 0142727X
[6] Klein M, Sadiki A and Janicka J 2003 Journal of Computational Physics 186 652–665 ISSN 00219991
[7] Lee S, Lele S K and Moin P 1992 Physics of Fluids A: Fluid Dynamics 4 1521 ISSN 08998213
[8] Mann J 1994 Journal of Fluid Mechanics 273 141–168
[9] España G, Aubrun S, Loyer S and Devinant P 2011 Wind Energy 923–937 ISSN 10954244
[10] Batchelor G K 1953 The theory of homogeneous turbulence
[11] OpenFOAM Foundation URL http://www.openfoam.org/
[12] Aubrun S, Tchouaké F, España g, Gunnar L, Mann J and Bingöl F Proceedings of the iTi conference in turbulence 2010