Intercomparison of terrain-following coordinate transformation and immersed boundary methods in large-eddy simulation of wind fields over complex terrain

Jiannong Fang and Fernando Porté-Agel
Wind Engineering and Renewable Energy Laboratory (WIRE), École Polytechnique Fédérale de Lausanne (EPFL), 1015 Lausanne, Switzerland
E-mail: jiannong.fang@epfl.ch

Abstract. Accurate modeling of complex terrain, especially steep terrain, in the simulation of wind fields remains a challenge. It is well known that the terrain-following coordinate transformation method (TFCT) generally used in atmospheric flow simulations is restricted to non-steep terrain with slope angles less than 45 degrees. Due to the advantage of keeping the basic computational grids and numerical schemes unchanged, the immersed boundary method (IBM) has been widely implemented in various numerical codes to handle arbitrary domain geometry including steep terrain. However, IBM could introduce considerable implementation errors in wall modeling through various interpolations because an immersed boundary is generally not co-located with a grid line. In this paper, we perform an intercomparison of TFCT and IBM in large-eddy simulation of a turbulent wind field over a three-dimensional (3D) hill for the purpose of evaluating the implementation errors in IBM. The slopes of the three-dimensional hill are not steep and, therefore, TFCT can be applied. Since TFCT is free from interpolation-induced implementation errors in wall modeling, its results can serve as a reference for the evaluation so that the influence of errors from wall models themselves can be excluded. For TFCT, a new algorithm for solving the pressure Poisson equation in the transformed coordinate system is proposed and first validated for a laminar flow over periodic two-dimensional hills by comparing with a benchmark solution. For the turbulent flow over the 3D hill, the wind-tunnel measurements used for validation contain both vertical and horizontal profiles of mean velocities and variances, thus allowing an in-depth comparison of the numerical models. In this case, TFCT is expected to be preferable to IBM. This is confirmed by the presented results of comparison. It is shown that the implementation errors in IBM lead to large discrepancies between the results obtained by TFCT and IBM near the surface. The effects of different schemes used to implement wall boundary conditions in IBM are studied. The source of errors and possible ways to improve the IBM implementation are discussed.

1. Introduction
Wind energy, as a safe and renewable source of energy, has been undergoing a large-scale development worldwide in the past decade. The ability to accurately predict turbulent wind fields plays an important role in the assessment of wind power potentials, short-term wind energy forecasting and optimal micro-siting of wind turbines. Large-eddy simulation (LES) has become an important tool for simulating turbulent flows, largely due to the fact that LES is superior
to the conventional Reynolds-averaged Navier-Stokes approach in predicting unsteady turbulent behaviors, and computationally more efficient than direct numerical simulation. Recently the development and application of LES-based numerical models for wind energy applications is on the rise. So far satisfactory results have been reported mainly for cases where the terrain is flat [1]. Complex terrain, especially steep terrain, remains a challenge for wind flow modeling in general.

For a long time, the terrain-following coordinate transformation method (TFCT) has been used in both mesoscale models and LES to represent topography effects [2–7]. TFCT allows an irregular bottom boundary to be mapped to a Cartesian grid so that application of bottom boundary conditions is simplified. While this aspect is advantageous, TFCT has the disadvantage of introducing additional terms in the governing equations, and some corresponding numerical errors in the presence of any slope. For slopes lower than 30 degrees those errors are negligible, but at higher slopes model errors become large and can cause stability problems. Recently, the immersed boundary method (IBM) has become popular in numerical simulation of atmospheric flows over complex terrain in urban and mountainous areas [8–16]. The advantage of IBM is that it offers a simple strategy to use a regular computational grid while solving flow problems with complex geometries. Interpolation methods are used in IBM to represent the effect of the boundary on the flow [17–19] (i.e. to satisfy boundary conditions). In addition to the no-slip boundary condition for velocity, in practice, a wall stress model based on the equilibrium stress balance assumption or the Monin-Obukhov similarity theory is usually adopted when applying IBM to LES [20–22], due to impractical refinement of the whole regular grid in order to resolve the near-wall region, and the lack of an accurate wall model applicable for rough surfaces [23]. However, implementation of such a wall model in the context of IBM is not as straightforward as in TFCT due to the fact that an immersed boundary in IBM is generally not co-located with a grid line, and commonly used approaches such as smearing and linear interpolation [11, 21] could introduce non-negligible errors.

We here perform an intercomparison of TFCT and IBM in large-eddy simulation of a turbulent wind field over a three-dimensional (3D) hill, for which wind-tunnel measurements are available for validation [24]. The main purpose of this study is to evaluate and improve the implementation of wall models in IBM. The slopes of the 3D hill are not steep, so the applicability of TFCT is ensured. Since TFCT is free from interpolation-induced implementation errors in wall modeling, the results from it can then be used as a reference for the evaluation so that the implementation errors in IBM can be separated from the modeling errors in wall models themselves.

2. Governing Equations and Numerical Methods

The governing equations for LES are the three-dimensional filtered Navier-Stokes equations, which can be written in rotational form as

\[
\frac{\partial \tilde{u}_i}{\partial t} + \tilde{u}_j \left( \frac{\partial \tilde{u}_i}{\partial x_j} - \frac{\partial \tilde{u}_j}{\partial x_i} \right) = -\frac{\partial \tilde{p}^*}{\partial x_i} + \nu \frac{\partial^2 \tilde{u}_i}{\partial x_j \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j} + F_i, \tag{2}
\]

where \(t\) denotes time, \(\tilde{u}_i\) is the \(i\)-th component of the instantaneous resolved velocity, \(x_i\) is the \(i\)-th component of the position vector, \(\tilde{p}^* = \tilde{p}/\rho + \frac{1}{2} \tilde{u}_i \tilde{u}_i\) is the modified pressure divided by density \(\rho\) and including the resolved kinetic energy, \(\nu\) denotes the kinematic viscosity, \(\tau_{ij}\) is the sub-grid scale (SGS) stress tensor, and \(F_i\) is the component of a body force for IBM. Buoyancy effects are not considered (i.e. neutral stability is assumed). The SGS stresses are calculated using the new-generation Lagrangian-averaged scale-dependent dynamic model [25].
Equations (1) and (2) are solved in time with two fractional steps. At the first step the intermediate velocity is computed as follows:

\[ \tilde{u}_i^n = \tilde{u}_i^n + \frac{3}{2} \Delta t R_i^n - \frac{1}{2} \Delta t \left( R_i^{n-1} - \frac{\partial \tilde{p}_n}{\partial x_i} \right), \]  
(3)

where

\[ R_i^n = -\tilde{u}_j^n \left( \frac{\partial \tilde{u}_i^n}{\partial x_j} - \frac{\partial \tilde{u}_j^n}{\partial x_i} \right) + \nu \frac{\partial^2 \tilde{u}_i^n}{\partial x_j \partial x_j} - \frac{\partial r_{ij}^n}{\partial x_j}. \]

Eq. (3) is obtained by integrating the momentum equation forward in time using the second-order Adams-Bashforth scheme without the pressure gradient term at the \( n \)-th level. Then, at the second step, the updated velocity after a full time step is calculated as

\[ \tilde{u}_i^{n+1} = \tilde{u}_i^n - \frac{3}{2} \Delta t \frac{\partial \tilde{p}_n}{\partial x_i}, \]  
(4)

where the pressure gradient is added. To satisfy the incompressibility constraint, the pressure is determined by the Poisson equation

\[ \frac{\partial^2 \tilde{p}_n}{\partial x_i \partial x_i} = \frac{2}{3 \Delta t} \frac{\partial \tilde{u}_i^n}{\partial x_i}. \]  
(5)

The spatial derivatives appearing in the above equations are computed by the pseudo-spectral method in the horizontal \( x \) and \( y \) directions, while in the vertical \( z \) direction they are computed with the second-order centered finite difference scheme in a staggered grid formulation. The nonlinear terms are de-aliased in Fourier space by the 3/2 rule.

In simulation of turbulent flows over complex terrains (e.g., hills, mountains, urban area), the complexity of the domain geometry leads to challenges in discretizing the computational domain. The immersed boundary method has been adopted by many researchers to handle the problem because it has the advantage that no change has to be made to the conventional Cartesian grid. In the present study, to model the topography effects on the flow, the immersed boundary method using the discrete-time momentum forcing [26] is adopted. A discrete force is introduced into the discretized momentum equation to stop the flow inside the solid and enforce desired velocities near the solid boundary by respecting the no-slip condition. The approach is implemented by modifying Eq. (4) to

\[ \tilde{u}_i^{n+1} = \tilde{u}_i^n - \frac{3}{2} \Delta t \frac{\partial \tilde{p}_n}{\partial x_i} + \Delta t f_i^n, \]  
(6)

where the discrete force \( f_i \) only acts on grid points inside or near the surface of the obstacle and drives the fluid velocities to desired values on these points. Zero is a practical choice for the desired value inside the solid. For grid points near the solid surface, the desired values for the velocities are obtained by interpolations between the immersed points on the solid surface and the auxiliary points inside the fluid (see [17, 18] for more details). In general, \( f_i \) is obtained as

\[ f_i^n(x) = \begin{cases} 
(3/2)(\partial \tilde{p}_n/\partial x_i) - (\tilde{u}_i^n - \tilde{u}_i)^{\text{desired}}/\Delta t & \text{if } \varphi(x) \leq \delta, \\
0 & \text{if } \varphi(x) > \delta.
\end{cases} \]  
(7)

where \( \varphi \) is a signed distance function used to identify the points (i.e., \( \varphi \leq 0/\varphi > 0 \) indicates the point is inside/outside the solid). Accordingly, the Poisson equation for pressure becomes

\[ \frac{\partial^2 \tilde{p}_n}{\partial x_i \partial x_i} = \frac{2}{3 \Delta t} \frac{\partial \tilde{u}_i^n}{\partial x_i} + \frac{2}{3} \frac{\partial f_i^n}{\partial x_i}. \]  
(8)
Equations (6)-(8) are coupled, and can be solved iteratively till the numerical solution for $\tilde{p}_n$ and $f_n^i$ converges to enforce both no-slip and divergence-free conditions [19].

Additional wall treatment of turbulent stresses is needed for large-eddy simulation. We here follow the approach of Chester et al. (2007) [21]. First, the wall shear stress at each immersed point is calculated using the log-law assumption as

$$
\tau_w = -\rho \left[ \frac{\kappa |V_t|^2}{\ln(1 + d/z_0)} \right]
$$

in a local coordinate system defined by the local tangential flow direction and the wall normal direction. In Eq. (9), $\kappa$ is the von Kármán constant, $z_0$ is the roughness length, and $V_t$ is the tangential velocity interpolated at a point of a normal distance $d$ to the solid surface. Other components of the stress tensor at the wall are assumed to be zero. The stresses are then transformed back to the original coordinate system. Next, for each grid point in a layer above the solid surface ($0 \leq \varphi(x) \leq \delta$), the stresses are interpolated between the stresses on the wall and the stresses outside the layer (i.e., the linear approach) or assumed equal to the wall stresses (i.e., the smeared approach). Furthermore, a second layer is defined under the surface, in which the stresses are extrapolated to ensure a smooth boundary transition. For the remaining points further inside, the stress profiles are smoothed in order to be compatible with the spectral differentiation used in LES. It is worth mentioning that the normal derivative of an expected logarithmic mean profile diverges at the wall, causing some errors in the numerical differentiations. Hence, a normal derivative correction (NDC) is implemented for the computation of the velocity gradient near the solid surface. To do this, the velocity gradient calculated using the adopted numerical scheme is first rotated to the local coordinate system, then the component corresponding to the normal derivative of the tangential velocity is replaced by the corrected value resulting from the differentiation of the logarithmic profile, and finally the modified velocity gradient in the local coordinate system is rotated back to the x, y, z frame.

In the terrain-following coordinate transformation method, the physical domain is discretized using a body-fitted grid. With the coordinate transformation defined as

$$
\bar{x} = x, \quad \bar{y} = y, \quad \bar{z} = \frac{H z - z_s}{H - z_s},
$$

where $H$ is the maximum height of the domain and $z_s$ is the surface elevation, the grid becomes a regular one in the new coordinate system, so all calculations here can be done using the numerical schemes for a regular grid. For the derivative calculations in the $x$, $y$, $z$ frame, the chain rule is applied to derive the formulas as follows:

$$
\frac{\partial \phi}{\partial x} = \frac{\partial \phi}{\partial \bar{x}} + \frac{G^{13}}{\sqrt{G}} \frac{\partial \phi}{\partial \bar{z}},
$$

$$
\frac{\partial \phi}{\partial y} = \frac{\partial \phi}{\partial \bar{y}} + \frac{G^{23}}{\sqrt{G}} \frac{\partial \phi}{\partial \bar{z}},
$$

$$
\frac{\partial \phi}{\partial z} = \frac{1}{\sqrt{G}} \frac{\partial \phi}{\partial \bar{z}},
$$

where $G^{13} = (\bar{z}/H - 1) \frac{\partial \bar{z}}{\partial x}$, $G^{23} = (\bar{z}/H - 1) \frac{\partial \bar{z}}{\partial y}$, and $\sqrt{G} = 1 - z_s/H$. Accordingly, the Poisson equation for pressure should be rewritten for the new coordinate system. It becomes rather complex if all the terms are expanded using the formulas above. So, instead of solving the equation directly, which leads to a large system of linear algebraic equations, we here propose an
iterative approach. In the new coordinate system, the divergence-free condition can be expressed in the following form:

$$\text{div} = \frac{\partial}{\partial x} (\sqrt{G} u) + \frac{\partial}{\partial y} (\sqrt{G} v) + \frac{\partial}{\partial z} (\sqrt{G} \omega) = 0,$$

(14)

where $\omega = (G^{13}u + G^{23}v + w)/\sqrt{G}$. Replacing the velocity components in Eq. (14) by those determined by Eq. (4) and applying the coordinate transformation formulas to the derivatives involved lead to the same equation for pressure as the Poisson equation written in the transformed coordinate system. So, solving the equation for pressure is equivalent to finding the pressure to satisfy Eq. (14). The latter can be done by iteratively solving the following equation:

$$\frac{\partial^2 \tilde{p}_{k+1}}{\partial x_i \partial x_i} = \text{div}(\tilde{p}_k) + \frac{\partial^2 \tilde{p}_k}{\partial x_i \partial x_i},$$

(15)

where $k$ denotes the iteration step. When converged, with the final pressure values obtained, the divergence is close to zero. At each iteration, we solve a standard Poisson equation in the $\bar{x}$, $\bar{y}$, $\bar{z}$ frame, using the Fourier method in the horizontal plane and the finite difference method in the vertical direction with a tridiagonal solver for the coupled Fourier coefficients. The advantage of the TFCT method is that it facilitates the implementation of the wall boundary conditions since we have the grid points exactly on the wall. The disadvantage of this method is that it only works for topography with low to medium slope.

### 3. Results

To test the proposed iterative pressure solver in the TFCT method, direct numerical simulation of a laminar flow over a wavy wall is performed. The flow is driven by a pressure gradient applied in the $x$ direction. The Reynolds number calculated based on the domain height and the maximum velocity is around 100. Figures 1 and 2 show the results of the velocity components $u$ and $w$ predicted by the TFCT method and the differences compared with those obtained by a well-tested meshless finite point method [27]. The differences are rather small indicating an excellent agreement between the results obtained by the two methods.

To evaluate the performance of TFCT and IBM in large eddy simulation of flows over hilly topography, turbulent flow over a three-dimensional hill is simulated by the two methods, and

![Figure 1](image-url). Contour of the velocity component $u$ for a laminar flow over a wavy wall predicted by the TFCT method (shown on the left) and the difference relative to those obtained by a meshless method (shown on the right).
simulation results are compared with the data from a wind-tunnel experiment conducted by Ishihara et al. in 1999 [24]. Many previous comparisons between LES and wind-tunnel data focus exclusively on two-dimensional hills and vertical profiles. The main advantage of the Ishihara dataset is that comparisons can also be made along the horizontal plane. The shape of the hill is described by the equation

\[
z_s(x, y) = h \cos^2 \left( \frac{\pi \sqrt{x^2 + y^2}}{2l} \right)
\]

with \( h = 4 \text{ cm} \) and \( l = 10 \text{ cm} \). The flow measured without the hill corresponded to a neutrally stratified atmospheric boundary layer. For more details on the experiment, we refer the reader to the paper mentioned above. The test-section floor was covered with plywood whose roughness length was estimated to be \( z_0 = 0.001 \text{ cm} \). Since the model hill was machined from wood, its roughness length is set to a higher value of \( 0.002 \text{ cm} \) in the simulations. The simulation domain sizes in the \( x \), \( y \), \( z \) directions are set to 128 cm, 64 cm, and 32 cm respectively. The domain is discretized into \( 512 \times 256 \times 256 \) points, resulting in a resolution of \( dx = dy = 0.25 \text{ cm} \) and \( dz = 0.125 \text{ cm} \). Periodic boundary condition is applied in the \( y \) direction, which is automatically implemented by the spectral method adopted in the horizontal directions. The upper boundary condition is specified as stress-free. Since the domain sizes in the \( y \) and \( z \) directions are much larger than the size of the hill, those boundary conditions do not have significant impacts on the flow in the vicinity of the hill. Turbulent inflow boundary condition is imposed in the \( x \) direction. The inflow boundary condition is generated from a precursor simulation, in which no hill is present and the flow is forced through the domain using a constant mean pressure gradient along the \( x \) direction. The mean pressure gradient is set in such a way that the resulting velocity profile near the surface matches the measured velocity profile which is adequately represented by a logarithmic law with \( u_* = 0.212 \text{ m/s} \) and \( z_0 = 0.001 \text{ cm} \). To eliminate the periodic effect on the imposed inflow condition caused by the spectral method applied in the \( x \) direction, a buffer zone is introduced in the first eighth of the domain to smoothly force the velocities back to the stored velocities from the precursor simulation. The LES results are averaged in time over about 8 s and compared with the experimental values.

Figure 3 shows the vertical profiles of the mean streamwise velocity obtained by the simulations and experiment. It is found that the simulation results obtained by the TFCT
method are in good quantitative agreement with the data obtained by the wind-tunnel measurement and close to the results simulated recently by Liu et al. (2016) [28] using the commercial CFD code ANSYS/Fluent6.3, while the performance of IBM is inferior to the performance of TFCT in terms of matching the experimental data. Here the smeared approach is used in IBM for the implementation of the wall model. Discrepancies between the results obtained by TFCT and IBM are significant near the surface. The simulation using TFCT is able to reproduce the main features of the mean streamwise velocity observed experimentally, i.e., the increase in velocity near the top of the hill, the slight decrease in velocity at the upwind hill foot, the flow separation at the lee side of the hill, and the wake recovery in the downstream of the hill. The simulation using IBM is able to capture those features qualitatively. Quantitatively, it predicts a larger recirculation zone behind the hill, hence lower velocity values near the surface behind the hill.

![Comparison of the vertical profiles of the mean streamwise velocity normalized by $U_h = 4.4$ m/s.](image)

The results of the comparison between the wind-tunnel measurements and the large-eddy simulations for the vertical profiles of the streamwise velocity variance are presented in Fig. 4. It shows that the profiles predicted by both TFCT and IBM have a close agreement with those obtained by the wind-tunnel experiment. On the lee side of the hill, the streamwise velocity variance overshoots around the regions of maximum shear in the streamwise velocity profiles, and the peak values are far in excess of those that occur in the undisturbed boundary layer. This indicates that highly turbulent motions are generated by the wake behind the hill. In the near-wake region, the peak value of the vertical profile decays gradually with distance behind the hill.

Spanwise distributions of the mean streamwise velocity at two different heights for a fixed streamwise position after the hill are shown in Fig. 5. Results are shown in only half of the domain, assuming that the mean flow field is symmetrical with respect to the central plane. Again, the results of TFCT agree well with the experimental results, while the LES with IBM underpredicts the mean streamwise velocity near the ground in the wake of the hill. Overall, the comparison between the IBM results and the measurements becomes more favourable at greater distances from the wall as seen from both the horizontal profiles shown here and the vertical profiles shown before.
Figure 4. Comparison of the vertical profiles of the streamwise velocity variance normalized by $U_h = 4.4$ m/s.

Figure 5. Spanwise distribution of the normalized mean streamwise velocity behind the hill at (a) $x/h = 3.75$, $z/h = 0.125$ and (b) $x/h = 3.75$, $z/h = 1$.

Figure 6 shows the spanwise distribution of the normalized streamwise velocity variance at the same locations as in Fig. 5. Like the vertical profiles for the same variable, the results obtained by both TFCT and IBM are consistent with the experimental results. At the elevation of $z/h = 1$, the variance monotonically decays from the maximum value at the central position to a constant value at positions far away from the central plane. This behavior can be explained by the flow dynamics in the near-wake region. At the elevation of $z/h = 0.125$, the spanwise variation of the variance is more involved, displaying a broad peak at a clearly off-center position which corresponds to maximum shear in the profile of the streamwise velocity. The result here implies that spanwise coherent motions exist in the near-wall region behind the hill.

For the IBM with the smeared approach, the mismatch is mainly found in the mean streamwise velocity and at locations near the ground behind the hill. Considering the fact that the LES with TFCT using the same SGS and wall models is able to reproduce the
Figure 6. Spanwise distribution of the normalized streamwise velocity variance behind the hill at (a) $x/h = 3.75$, $z/h = 0.125$ and (b) $x/h = 3.75$, $z/h = 1$.

Figure 7. Comparison of the vertical profiles of the normalized mean streamwise velocity obtained by the LES with various IBM options. Here, IBM1 denotes the IBM with the smeared approach, IBM2 denotes the IBM with the linear approach, IBM3 denotes the IBM with the smeared approach and without NDC, and IBM4 denotes the IBM with the smeared approach and the second-order interpolation for the near-wall velocity reconstruction.

experimental results, it is reasonable to infer that the main source of errors in IBM comes from the interpolation-based implementation of the wall model which obviously cannot represent the topography effects as accurately as the TFCT does. Those errors could lead to the predicted location of the separation point deviating from the true one, which is most likely the main reason of the mismatch. Some attempts are made here to improve the IBM results by varying the implementation scheme and control parameters.

Figure 7 shows the vertical profiles of the mean streamwise velocity obtained by the IBM simulations with different implementation options. It turns out that, among the variants IBM1, IBM2 and IBM3, the differences are negligible, while the variant IBM4 performs better than the others do. Not shown are the vertical profiles of the mean streamwise velocity obtained by the
IBM using the smeared approach with different values of $\delta$ which controls the thickness of the outer layer in which the wall model is implemented. The differences between the various results are marginal. It is worth mentioning that enforcing a logarithmic velocity profile near the solid surface through the IBM formulation described in Sec. 2 led to the divergence of the simulation.

4. Conclusion
A new algorithm for solving the pressure Poisson equation in the transformed coordinate system is proposed and first validated for a laminar flow over periodic two-dimensional hills. An intercomparison of TFCT and IBM in large-eddy simulation of a turbulent wind field over a three-dimensional hill is performed. Both vertical and horizontal profiles of mean velocity components and turbulent intensity are compared between the numerical models and wind tunnel measurements. With the same wall model, results from TFCT agree well with the measurements, while results from IBM deteriorate near the surface. This observation allows us to have an unambiguous evaluation on the implementation errors in IBM. The effects of different schemes used to implement wall boundary conditions in IBM are studied. In general, IBM tends to predict a larger separation bubble behind the hill. Most likely, this mismatch is caused by the interpolation-based implementation of the boundary conditions on the hill surface, which leads to an incorrect prediction of the separation point location. Most of the implementation options attempted in this study, except for the one using the second-order interpolation for the near-wall velocity reconstruction, failed to bring significant improvements to the IBM results. Therefore, IBM must be used with caution for large-eddy simulation of turbulent flows over complex topography, for which, the separation point is difficult to predict. Further developments of the wall modeling in the context of IBM are needed.

Acknowledgements
This research was supported by the Swiss National Science Foundation (grant IZERZ0-142168) and the Swiss Innovation and Technology Committee (CTI) in the frame of the SCCER-FURIES project. Computing resources were provided by the Swiss National Supercomputing Centre (CSCS) under project ID s599.

References
[1] Porte-Agel F, Wu Y T, Lu H and Conzemius R J 2011 J Wind Eng Ind Aerodyn 99 154–168
[2] Clark T 1977 J Comput Phys 24 186–215
[3] Schär C, Leuenberger D, Fuhrer O, Lüthi D and Girard C 2002 Mon Weather Rev 130 2459–2480
[4] Wan F, Porte-Agel F and Stoll R 2007 Atmos Environ 41 2719–2728
[5] Wan F and Porté-Agel F 2011 Boundary-Layer Meteorol 141 201–217
[6] Mirocha J, Kosović B and Kirkil G 2014 Mon Weather Rev 142 806–831
[7] Nottrott A, Kleissl J and Keeling R 2014 Atmos Environ 82 172–182
[8] Tseng Y H, Meneveau C and Parlange M 2006 Environ Sci Technol 40 2653–2662
[9] Bou-Zeid E, Overney J, Rogers B D and Parlange M B 2009 Boundary-Layer Meteorol 132 415–436
[10] Lundquist K, Chow F and Lundquist J 2010 Mon Weather Rev 138 796–817
[11] Diebold M, Higgins C, Fang J, Bechmann A and Parlange M B 2013 Boundary-Layer Meteorol 148 177–194
[12] Lundquist K, Chow F and Lundquist J 2012 Mon Weather Rev 140 3936–3955
[13] Jafari S, Chokani N and Ahhari R 2012 J Sol Energ-T ASME 134 011006
[14] Cheng W C and Porte-Agel F 2013 Int J Heat Fluid Fl 44 301–311
[15] Cheng W C and Porté-Agel F 2015 Boundary-Layer Meteorol 155 249–270
[16] Cheng W C and Porté-Agel F 2016 Int J Heat Fluid Fl 60 47–60
[17] Fadlun E A, Verzicco R, Orlandi P and Mohd-Yusof J 2000 J Comput Phys 161 35–60
[18] Tseng Y H and Ferziger J H 2003 J Comput Phys 192 593–623
[19] Fang J, Diebold M, Higgins C and Parlange M B 2011 J Comput Phys 230 8179–8191
[20] Cristallo A and Verzicco R 2006 Flow Turbul Combust 77 3–26
[21] Chester S, Meneveau C and Parlange M B 2007 J Comput Phys 225 427–448
[22] Roman F, Armenio V and Fröhlich J 2009 Phys Fluids 21 101701
[23] Yang X, Sadique J, Mittal R and Meneveau C 2015 Phys Fluids 27 025112
[24] Ishihara T, Hibi K and Oikawa S 1999 J Wind Eng Ind Aerodyn 83 95–107
[25] Stoll R and Porté-Agel F 2006 Water Resour Res 42 W01409
[26] Mohd-Yusof J 1997 CTR Annual Research Briefs (Stanford Univ.) pp 317–327
[27] Fang J and Parriaux A 2008 J Comput Phys 227 8894–8908
[28] Liu Z, Ishihara T, Tanaka T and He X 2016 J Wind Eng Ind Aerodyn 153 1–12