Chapter 2
Computational Fluid Dynamics Methods for Wind Turbines Performance Analysis

Navid Goudarzi

2.1 Introduction

2.1.1 Wind Turbines

The importance of reducing greenhouse gases leads to developing sustainable and efficient technologies. Wind power as a free, abundant, and globally available energy source is one of the most promising energy resources for green electricity generation. Figure 2.1 shows the world's total cumulative installed wind power capacity between 1991 and 2016 (Goudarzi and Zhu 2013). The average annual growth in the total installed wind power capacity in the last 10 years is more than 25% per year; it is anticipated that 12% of the world's electricity consumption will be provided by wind power by 2020 (Goudarzi and Zhu 2013).

A wind turbine converts the captured kinetic energy in the wind to electrical energy. Rankine-Froude momentum or actuator disk model is known as the first estimation for wind turbine efficiency (Mikkelsen 2003). Betz law shows the maximum ideal captured power by a horizontal axis turbine cannot exceed 59.3% of the kinetic energy in wind (Manwell et al. 2010). Current commercial wind turbines work at 75–80% of the Betz limit (Mikkelsen 2003). While more wind turbines are installed in large wind farms, reduced power production (8% for onshore farms and 12% for offshore farms) due to wake velocity deficits and increased dynamic loads on turbine blades due to higher turbulent flow levels should be studied for determining wind turbines aerodynamic characteristics. Understanding the aerodynamic performance of wind turbines and in particular turbine blades

N. Goudarzi (✉)
Mechanical Engineering (ETCM) Program, University of North Carolina at Charlotte, Charlotte, NC, USA
e-mail: navid.goudarzi@uncc.edu

W. Hu (ed.), *Advanced Wind Turbine Technology*,
https://doi.org/10.1007/978-3-319-78166-2_2

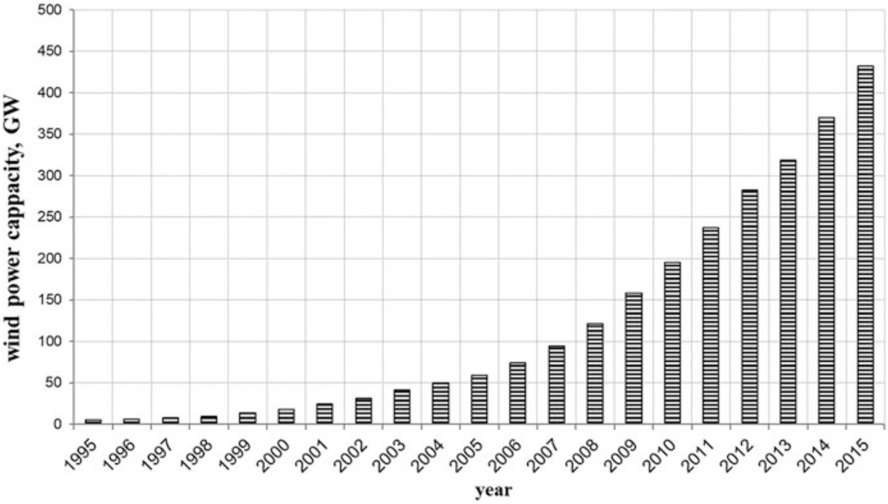

Fig. 2.1 World's cumulative installed wind power capacity during 1995–2015

at different wind speeds can help to improve the accuracy in the prediction of wind turbine performance and facilitate an optimized design of turbine blades for a desired performance goal.

There has been a significant number of researchers studying aerodynamic performance of turbine blades (Martinez Tossas and Leonardi 2013; Lee and Wu 2011). They have developed computational methods to analyze the global flow field around the turbine blades. These methods got further improved to reduce the numerical results uncertainties by comparing them with experimental tests. This chapter provides a brief review on the developed numerical techniques with emphasis on computational fluid dynamics (CFD) methods used for wind turbine applications: airfoil design, blade design, and load calculations.

2.1.2 Aerodynamics Characteristics of Wind Turbines

The turbine blade efficiency, power coefficient C_P, is an important factor in determining the aerodynamic performance of a wind turbine:

$$C_p = \frac{P_m}{P_w} = \frac{T_m * \omega}{1/2 \rho A_r V^3} \tag{2.1}$$

where P_m is the mechanical power, P_w is the wind power, T_m is the mechanical torque, ω is the rotational speed, ρ is the air density, A_r is the rotor area, and V is the wind speed. The fraction of the year the turbine generator is operating at rated

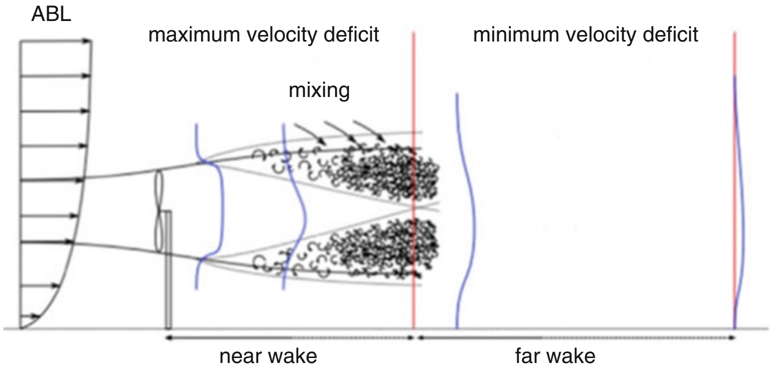

Fig. 2.2 Velocity profile behind the wind turbine (Shakoor et al. 2016)

power is defined by the capacity factor (CF). It is based on both turbine and site characteristics. Aerodynamic characteristics of wind turbines are directly related to the airfoil designs used in turbine blades.

Airfoils such as NACA44xx, NACA230xx, and NACA63xxx with high maximum lift coefficient, low minimum drag, and low pitching moment are among popular airfoils used for wind turbine blades. Flow separation and stall phenomenon are among the main concerns in designing airfoils. Stall phenomenon occurs either at high angles of attack (generally at angles more than 15°) or at low tip-speed ratios (TSR) at a given wind speed. During stall, the airfoil lift decreases significantly, and the draft force increases.

The wake flow behind the wind turbine (near wake or far wake) can have a significant impact on the turbine blade performance (Fig. 2.2). The near wake is within 1–2 rotor diameters downstream behind the turbine rotor which is affected directly by the turbine geometry, the presence of tip vortices, and mixing flows. Hence, the wake analysis should include both axial- and tangential-induced velocities from formed vortices and the blade shear layers. The far wake is affected indirectly by the turbine geometry; it has a reduced axial velocity and an increased turbulence intensity. The far wake analysis would be needed in the case of wind farm design. Generally, three types of turbulence exist in the far wake: atmospheric turbulence due to surface roughness and thermal effects, mechanical turbulences due to the wind turbine, and wake turbulence from vortex breakdown. The wake flow applies a swirl velocity component to the air in the opposite direction to the turning of the blades. This wake behind the turbine slows down the airflow going through the rotor and changes the local angle of attack of blades which directly impacts the aerodynamic forces. Far downstream, a Gaussian and axisymmetric velocity field is observed (Hu 2016).

The future trends for wind turbine technology developments include performance improvements and cost reductions. Wind turbines with novel designs, higher towers, larger blades, and an improved reliability and availability, at a reduced weight and an expanded installation in offshore sites, can further reduce the cost of energy from wind and make it more competitive on the energy market. While airfoil aerodynamics characteristics are well-known (Abbot and Doenhoff 1949), understanding and accurate estimation of turbine blade interactions due to different forces in a rotating frame such as centrifugal forces as well as the wake behind the turbine and modifying the blade aerodynamic characteristics facilitate achieving the future goals of this industry.

2.2 Numerical Techniques

Blade element momentum (BEM) theory and CFD methods are widely used to predict wind turbine performance. BEM methods predict the aerodynamic performance and obtain the optimal blade design by integrating linear and angular air mass momentum changes with the toque and axial forces acting on blades. CFD methods provide visualization capabilities of flow behavior on the blade surface and in the wake region. Hence, BEM is used for designing the wind turbine rotor geometry, and CFD is used for validating and evaluating the design and its performance.

2.2.1 Blade Element Method (BEM)

BEM predicts the turbine blade aerodynamic characteristics based on linear momentum theory. It assumes that (1) the flow is steady state and incompressible, (2) there are an infinite number of blades, (3) there is no rotating wake behind the turbine blade, (4) there is a uniform thrust over the actuator disk, and (5) there is no frictional drag. It only considers the 2D lift and drag forces on airfoils that do not have aerodynamic interactions with each other. Basic BEM methods underpredict torque, as they compute the aerodynamics of each airfoil section along the blades independently of neighboring sections. It will result in neglecting spanwise flow and other potential 3D effects. Such effects might be significantly important especially near wind turbine blade roots. Correction equations such as the Prandtl tip loss factor (Manwell et al. 2010), the stall delay model (Martinez Tossas and Leonardi 2013), the Viterna-Corrigan stall model (Lee and Wu 2011), and the Spera's correction (Spera 2009) are used to improve the prediction accuracy and consider the 3D effects in basic BEM methods.

2.2.2 Computational Fluid Dynamics (CFD)

CFD is a powerful tool to estimate the aerodynamic performance characteristics and to visualize the flow behavior around wind turbine components. In recent years, commercial CFD software products such as ANSYS Fluent, COMSOL, Star

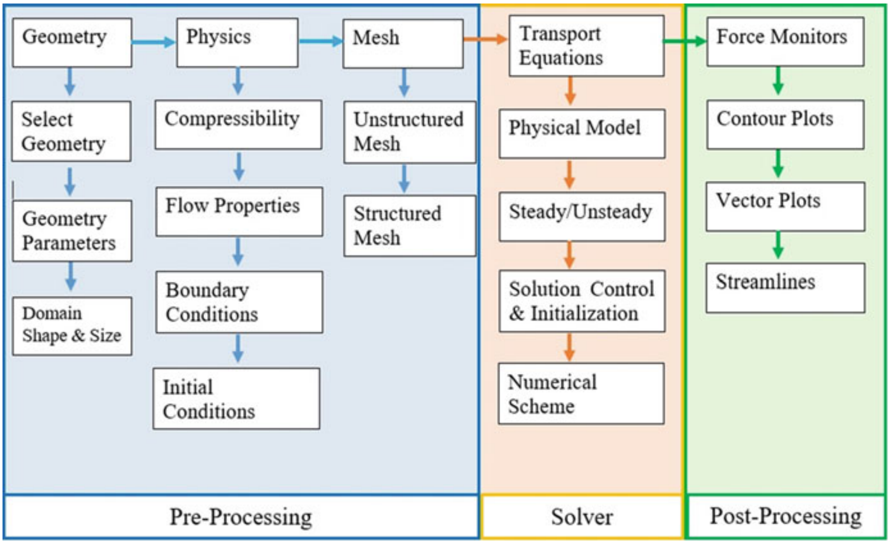

Fig. 2.3 Three main steps in CFD

CCM+, EllipSys3D, Open Foam, etc. are widely used by engineers. Figure 2.3 illustrates three common main steps in all of these tools: preprocessing, solver, and post-processing. The geometry creation, mesh generation, and boundary conditions definitions are conducted in the preprocessing step. A very important step in solving the partial differential Navier-Stokes (NS) equations is to use and develop stable, consistent, and accurate algebraic replacements for the NS equations, called discretization, where the physics and inherent structure of the problem are retained. Generally, three types of numerical discretization schemes including finite volume method (FVM), finite element method (FEM), and finite difference method (FDM) are currently in use. These schemes transform the infinite-dimensional NS equations into finite-dimensional algebraic equations. The FVM methods are flux conserving construction based on the approximation of conservation laws. Compared to the FVM methods, FEMs have a more flexible discretization but with fewer quality constraints. FDMs use a completely different approach compared to the preceding two and are limited to structured grids. Within the solver step, the NS equations are solved for a time-dependent velocity field, appropriate turbulence models are selected, and the solver settings such as solution control and initialization are performed. Finally, the post-processing will obtain the forces (such as normal force, thrust, and torque) and visualization of flow using contour plots, vector plots, and streamlines.

CFD techniques (turbulence models are more elaborated in Sect. 2.2.2.2) can be classified in four broad categories:

1. Actuator blade methods that provide physics-based characterization of wind turbine wakes at a reduced computational cost

2. Hybrid Reynolds-averaged Navier-Stokes (RANS)/large eddy simulation (LES) methods that provide an improved estimation of unsteady and separated flows
3. The overset methods (Chimera methods) to treat the relative motion between rotor and its support structures
4. Combined CFD-computational structure dynamics (CSD) methods to model the aeroelastic response of the rotor blades

A combination of level of understanding of the problem (flow physics), computational cost, and ease of using a specific CFD technique, range of applicability, and the level of required accuracy will determine the desired CFD technique for an application.

2.2.2.1 Mesh and Boundary Conditions

Design and construction of a quality grid are crucial to the success of the CFD analysis. There are three general approaches that can be used to simulate the turbine blade rotation (Cabezon et al. 2009):

1. Moving reference frame (MRF): it simulates the aerodynamic performance of a single turbine blade using a periodic boundary condition in steady-state flow conditions.
2. Sliding mesh model: it simulates the aerodynamic performance of a full-scale model in transient flow with two distinct domains that have a relative motion and a nonmatching grid.
3. Dynamic mesh model: it has a high computational cost and is useful for modeling relative motions between different components.

A trade-off between accuracy, computational time, and the objective of the problem can determine the most appropriate discretization. The structured and unstructured grid generation can be conducted in a number of mesh tools such as Gambit, Pointwise, and ICEM CFD. The structured grids are widely used along the blade boundary layer. Based on the geometry complexity, flow field, and solver-supported cell types, other computational domains including upstream and downstream of the boundary can be meshed with either structured or unstructured grids. The grid quality can be determined based on three measures: skewness (zero is the best, and one is the worst), smoothness (change in size), and aspect ratio (one is the ideal value for an equilateral triangle or a square). Accurate and fast converging solutions require a high grid quality: skewness does not exceed 0.85–0.90, local cell size variations have to be gradual (the maximum change in grid spacing should be less than 20%), and the aspect ratio has to be defined based on the pertinent flow features. While more cells can offer higher accuracy, it significantly increases the computational cost. Cell counts in the order of 10^4–10^7 are common for small- to large-size problems. Higher cell counts should be avoided if possible; otherwise, different techniques such as using multiple CPUs should be employed.

In order to ensure adequate grid resolution on turbine blades, solution-based grid adoptions can be employed based on gradients of flow, boundary cells, volume changes, and adjacent to wall cells (nondimensional wall distant y^+). The near-wall region is generally modeled by either the wall function or the near-wall function (viscous sub-layer and buffer layer). The wall function approach solves the attached flow on turbine blade surface at a large initial y^+ value with a low computational cost. However, an accurate prediction of the stall point for separation flows requires a fine mesh along the inboard blade section. In recent years, near-wall functions use standard or modified turbulence models (based on the y^+) to obtain accurate solutions for the laminar sub-layer region.

One of the ongoing challenges in CFD simulation techniques is the boundary condition definition, especially when comparing with experimental data and when the model solves for the wind turbine wakes. While uniform and laminar inflow profiles have been widely used in early CFD simulations, the LES simulations showed the presence of both shear inflow profile and turbulence in the incoming flow; these have significant impact on the flow field behind the rotor. To address velocity components and turbulence quantities independent of time, Monin-Obukhov similarity theory can be used for RANS simulations (Martinez Tossas and Leonardi 2013).

2.2.2.2 Turbulence Model

Inherent turbulent characteristics of the atmospheric airflow make wind turbine rotors' operation impacted by the turbulent fluctuations. There are a number of approaches such as RANS, LES, and detached eddy simulation (DES) for modeling turbulent flows and determining the velocity fluctuations on turbine blades. The focus of this section is on RANS models. Solving the RANS equations for the flow field around a turbine blade requires a proper turbulent model. RANS models provide a statistical description of the flow and describe the turbulent flow as a random variation around a mean value. The RANS equations can be written as

$$\rho \frac{\partial U_i}{\partial t} + \rho U_j \frac{\partial U_i}{\partial x_j} = -\frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_j}\left(2\mu S_{ij} - \rho\overline{u_j^{'} u_i^{'}}\right) \tag{2.2}$$

with the time-averaged mass conservation to be as

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{2.3}$$

where U_i is the time-averaged velocity, u_i' is the fluctuating velocity, μ is the molecular viscosity, and S_{ij} is the deformation tensor. Note that the $\rho\overline{u_j^{'} u_i^{'}} = \rho\tau_{ij}$ is known as the Reynolds stress tensor and two main approaches of either turbulent-viscosity models or Reynolds stress models (RSM) are used to model it. The turbulent-viscosity models are more suitable for simple turbulent shear flows such as boundary layers, channel flows, and mixing flows. The RMS models solve transport equations for individual Reynolds stress terms and for the dissipation rate ε or specific dissipation rate ω. They are advantageous to more complex turbulent flows such as swirling flows and the effects of large streamlines curvature. In RANS equations, there are 10 unknowns (pressure, 1; velocity components, 3; and Reynolds stress tensor components, 6) and four equations. Hence, more equations should be introduced to solve for the unknowns. There are different models introduced such as (Sanderse et al. 2011):

1. One-equation models (Spalart-Allmaras turbulence model): they solve transport equations for the turbulent viscosity ($\mu = f(\tilde{\upsilon})$). They are less sensitive in the near wall and used mainly for aerodynamic applications with mild separation.
2. Two-equation models (k-ε and k-ω turbulence models): they solve transport equations for two turbulence quantities, turbulent kinetic energy k, and turbulent dissipation rate ε. Two-equation eddy k-ε models ($\mu = f(\rho k^2/\varepsilon)$) are rarely used in wind turbine studies, as they do not offer good results for flows with large pressure gradients and strong separation. For such flows, the renormalization group k-ε models and realizable k-ε models offer better and superior performance, respectively (Jones and Launder 1972). Two-equation eddy viscosity k-ω models ($\mu = f(\rho k/\omega)$ where $\omega = k/\varepsilon$ is the specific dissipation rate) are currently popular for turbine blades aerodynamic forces simulation analysis (Wilcox 1988). The transition k-ω SST turbulence models conduct 3D aerodynamic analysis of turbine blades and show strong agreements with the experimental results (Menter 1994).
3. Hybrid turbulence models: these models address multiple different flow behaviors. For example, DES combines the accuracy of LES within separation region (solving for smallest, subgrid-scale (SGS) eddies) and efficiency of RANS inside a boundary layer to address high-level separation flows with high-level transient properties and vortex shedding. There are a number of successful RANS-LES hybrid models such as very large eddy simulation (VLES), detached eddy simulation (DES), and partially averaged Navier-Stokes (PANS).

Note that there is literature on alternative numerical methods to solve complex CFD problem. One example (Pasquali 2016) is the implication of lattice Boltzmann method for computational domain discretization (grid generation) and near-wall turbulent flow analysis (boundary treatment for turbulent flows).

2.3 Numerical Simulation of Wind Turbine Wakes

RANS and LES methods are largely used for studying the influences of atmospheric and wake turbulence around wind turbines. The LES methods with abilities such as handling unsteady, anisotropic turbulent flows have drawing more attention in recent years. However, the LES computational cost is much higher than RANS models. The actuator disk model (ADM) and actuator line model (ALM) predict blade forces

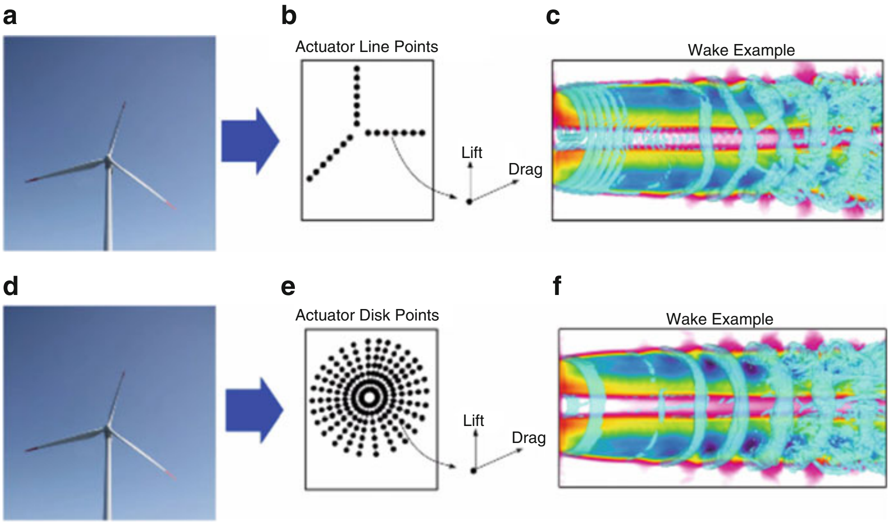

Fig. 2.4 ADM/ALM for a real turbine wake flow visualization (Martinez Tossas et al. 2014). (**a**) Commercial scale turbine. (**b**) Actuator line representation. (**c**) Flow visualization. (**d**) Commercial scale turbine. (**e**) Actuator disk representation. (**f**) Flow visualization

(as body forces) based on the local fluid velocity at each actuator element, without taking care of the full blade geometry (Fig. 2.4). The ADM simulates a wind turbine as a distributed force (both axial and tangential forces) on the rotor disk. The ALM simulates the turbine blade by a distributed force from the hub to the blade tip. It can predict the vortex structures and instabilities formation for both near and far wake. Several computational and experimental works have shown that both ADM and ALM can successfully simulate the wind turbine aerodynamic characteristics as well as the effects of local and regional turbulent fluxes of momentum and heat in wind farms (Martinez Tossas and Leonardi 2013).

The CFD-BEM approach uses the flow simulation around a wind turbine to estimate power curves, forces, and moments. There are a number of literature comparing the use of either of those methods or a combined approach (Lynch 2011). It is shown that this approach offers a lower computational cost at a high degree of accuracy. The accuracy of numerical approaches requires a good understanding about the physics of the problem. The BEM does not provide a reliable aerodynamic load simulation on the turbine blade especially at stall flow conditions. Different corrections such as hub loss correction, tip loss correction, Glauert, skewed wake correction, buhl empirical corrections, and 3D corrections can improve the BEM results. Most RANS methods have a stable and robust approach with using second-order accurate finite volume schemes on structured grids, with upwind discretization of convective terms and central discretization of diffusive terms. In LES methods, spatial convective terms require more careful discretization; central and spectral schemes are preferred to upwind schemes as the latter one introduces numerical dissipation in spatial discretization.

2.4 Conclusions and Future Recommendations

This chapter provided numerical approaches with focus on CFD methods used in studying the aerodynamic performance of wind turbine blades. Some conclusions include:

- BEM methods provide a rapid aerodynamic characteristics and performance of a turbine blade, using 2D airfoil data. Correction methods should be used to address the stall and 3D effects and improving the prediction accuracy.
- CFD methods provide flow visualization capabilities and accurate aerodynamic characteristic estimations for turbine blades. Turbulence model selection and discretization techniques play key roles in simulation accuracy. It showed that the LES with ADM and ALM models provide reliable estimations in turbine blades wake. However, more work should be done to study the impact of wind turbine tower and nacelle on estimated turbine performance and wake profiles.
- The CFD-BEM approach offers an accurate prediction model at a low computational cost.
- Further research can focus on quantifying computational uncertainties from discretization, from turbulence modeling, as well as from the inflow description, terrain geometry, and rotor geometry. This improves CFD results for comparing with experimental data that can further enhance the accuracy of wind turbine aerodynamic characteristics estimations. In a bigger picture, a combination of CFD analysis together with experimental methods such as field measurements, wind tunnel measurements, and particle image velocimetry (PIV) techniques improves the development and evaluation of wind harnessing machines. For instance, the most comprehensive wake measurements are determined experimentally, using PIV systems. Hence, the added value of combining experimental methods in addressing the fundamental and practical gaps in wind energy development should be further explored.

References

Abbot IH, Doenhoff AE (1949) Theory of wings sections including a summary of airfoil data. McGraw-Hill, New York

Cabezón D, Sanz J, Marti I (2009) Crespo A CFD modeling of the interaction between the surface boundary layer and rotor wake. Comparison of results obtained with different turbulence models and mesh strategies. In: European wind energy conference and exhibition, Marseille

Goudarzi N, Zhu W (2013) A review of wind turbine generators across the world. Int J Dyn Control 1:192–202
Hu B (2016) Design of a simple wake model for the wind farm layout optimization considering the wake meandering effect. MSc Dissertation, TuDelft University
Jones W, Launder B (1972) The prediction of laminarization with a two-equation model of turbulence. Int J Heat Mass Transf 15(2):301–314
Lee HM, Wu Y (2011) An experimental study of stall delay on the blade of a horizontal axis wind turbine using tomographic particle image velocimetry. J Wind Eng Ind Aerodyn 123(A):56–68
Lynch E (2011) Advanced CFD methods for wind turbine analysis. Aerospace Engineering Department, Georgia Institute of Technology, Atlanta
Manwell JF, McGowan JG, Rogers AL (2010) Wind energy explained: theory, design, and application. Wiley, New York
Martinez Tossas LA, Leonardi S (2013) Wind turbine modeling for computational fluid dynamics. National Renewable Energy Laboratory, Golden, Technical report no NREL/SR-5000-55-54
Martinez Tossas LA, Churchfield MJ, Leonardi S (2014) Large eddy simulation of the flow past wind turbines: actuator line and disk modeling. Wind Energy J 18:1047–1060
Menter FR (1994) Two-equation eddy-viscosity turbulence models for engineering applications. AIAA J 32(8):1598–1605
Mikkelsen R (2003) Actuator disc methods applied to wind turbines. Mechanical Engineering, Technical University of Denmark, Copenhagen
Pasquali A (2016) Enabling the cumulant lattice Boltzmann method for complex CFD engineering problems. In: Architecture, civil engineering and environmental sciences. Technical University of Braunschweig, Braunschweig
Sanderse B, Pijl SP, Koren B (2011) Review of computational fluid dynamics for wind turbine wake aerodynamics. Wind Energy J 14:799–819
Shakoor R, Hassan MY, Raheem A, Wu YK (2016) Wake effect modeling: a review of wind farm layout optimization using Jensen's model. Renew Sustain Energy Rev J 58:1048–1059
Spera DA (2009) Wind turbine technology: fundamental concepts in wind turbine engineering. American Society of Mechanical Engineers (ASME), New York
Wilcox D (1988) Reassessment of the scale determining equation for advanced turbulent models. AIAA J 19(2):248–819