Computers & Fluids 92 (2014) 138–150

Contents lists available at ScienceDirect

Computers & Fluids

journal homepage: www.elsevier.com/locate/compfluid

An efficient actuating blade model for unsteady rotating system wake simulations

C.E. Lynch [1], D.T. Prosser *,[1], M.J. Smith [1]

School of Aerospace Engineering, The Georgia Institute of Technology, Atlanta, GA 30332-0150, USA

ARTICLE INFO

Article history:
Received 19 March 2013
Received in revised form 3 December 2013
Accepted 10 December 2013
Available online 27 December 2013

Keywords:
Rotating system
Wind turbine
CFD
Actuator blade
Wake

ABSTRACT

This paper describes an innovative, efficient actuating blade model to capture the unsteady motion of a rotating system within Computational Fluid Dynamics (CFD) methods, with application to wind turbine blades. Each blade planform is modeled via a cloud of sources that move independently during the simulation to provide rotation of the blade as well as optional motion such as blade flexibility (aeroelasticity) and active controls (flaps, morphing, adaptive shapes). The model can be implemented into structured or unstructured methods that span the gamut from full potential to Large Eddy Simulations (LES), and it does not require the use of overset grids. A key feature of this model is the development of a highly efficient parallelized kd-tree algorithm to determine the interactions between actuator sources and grid nodes. Computational evaluation of the method successfully demonstrates its capability to predict root and tip vortex location and strength compared to an overset Navier–Stokes methodology on an identical background grid, and further improvements in the solution are shown by the use of grid adaptation.

1. Introduction

Rotating systems are abundant in engineering fluid dynamics applications. A few examples include wind turbines, propellers, rotors, compressors, and jet engines. Wind turbines represent one type of rotating system that has gained incredible interest in recent years due to the push towards alternative energy sources. Despite their relatively simple appearance, horizontal-axis wind turbines (HAWTs) operate in an aerodynamic environment that is challenging to model. Due to the atmospheric boundary layer, there can be considerable variation in wind speed between the top and bottom of the rotor disc. If the turbine is not facing directly into the wind, there will be yaw error and the blade loads will vary cyclically as they rotate. All wind turbines have some kind of yaw control, but the wind direction can vary too quickly for the controller to maintain zero yaw. At high wind speeds, even in axial flow, the blades may be stalled.

Though Computational Fluid Dynamics (CFD) has made significant inroads as a research tool in wind turbine aerodynamics [1–3], simple, inexpensive methods are still the workhorses in design and aeroelasticity applications [4]. These can range from blade element momentum (BEM) theory methods [5] to more accurate but still inexpensive vortex methods [6]. BEM methods provide basic insight into turbine flows, but only under the simplest conditions: constant wind speed with zero yaw error. BEM methods operate under the *independence principle*, in which the aerodynamics of each airfoil section along the blades are computed independently of neighboring sections [7]. As a result BEM methods completely neglect spanwise flow and other three dimensional (3-D) effects, which have been shown to result in significant lift augmentation and stall delay, especially near the blade roots [8]. Vortex methods such as prescribed wake models [9] can capture some unsteady effects, but like BEM methods, lack the ability to handle 3-D effects. As a result, both typically underpredict torque, even when they incorporate a 3-D correction [10]. Designs based on such simulations can result in structures that succumb to fatigue sooner that expected [11,12].

CFD techniques can mitigate many of the inaccuracies from the simplifying assumptions for wind turbine analysis methods. There are currently four broad classes of recent CFD improvements that can be applied to improve wind turbine design and analysis tools. The first class is based on hybrid Reynolds-Averaged Navier–Stokes (RANS)–Large Eddy Simulation (LES) methods, where improved turbulence models can improve the prediction of unsteady, separated flows. RANS and LES usually apply the concept of overset grids, which introduce the ability to treat the relative motion between rotor and its support structure. Source-based methods, including actuator disks and actuator lines, can provide physics-based characterizations of wind turbine wakes while reducing computational expense associated with modeling the blades, and do not require overset grids. As wind turbine blades continue to

* Corresponding author.
E-mail address: dtprosser@gatech.edu (D.T. Prosser).
[1] Research is supported by the National Research Foundation under Grant number CBET-0731034.

0045-7930/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.compfluid.2013.12.014

increase in size, coupling between CFD and computational structural dynamics (CSD) methodologies to capture the aeroelastic response of the rotor blades becomes increasingly important.

The momentum source technique is one approach to managing complexity and computational expense in simulations of wind turbines. Actuator disc methods, based on momentum theory, seek to model the *effects* of a rotor on the surrounding flowfield without the need to model the physical rotor. Since the flow over the rotor blades produces lift and drag forces on the blades, there must be another force equal in magnitude and opposite in direction acting on the flow. This reaction force can be included in the underlying CFD model in two ways. It can be implemented in a manner similar to a boundary condition. A special permeable boundary surface is embedded in the mesh, with the flow velocity constrained to be continuous through the surface while the pressure is discontinuous. Alternatively, those forces can be included as body forces in the Navier–Stokes equations. The body force approach offers additional flexibility over the pressure discontinuity. Since no surface is needed in the grid, the position of the disc can change, allowing different choices of the tip path plane on the same grid. In either case, the blades themselves are neglected. The pressure discontinuity or body forces provide the same effects on the flowfield as rotating blades, but in a time-averaged sense. Without modeling the blades directly, CFD simulations can either use fewer grid points, speeding their execution compared to more detailed methods, or place more points in the turbine wake gaining greater modeling fidelity.

CFD with actuator discs has seen widespread use in both the rotorcraft and wind industries [13–16]. For a good review of actuator disc techniques, largely from the perspective of structured-grid CFD, see Ref. [17]. O'Brien implemented unsteady actuator disc and blade techniques in the unstructured solver FUN3D and compared those against fully overset techniques on several geometries, demonstrating that an actuator disc can significantly improve predictions of helicopter fuselage loads [18,19]. While actuator disc methods provide a good approximation of the influence of the rotor, the fact that they lack discrete blades means that they model the rotor in an azimuthally averaged fashion. This makes them unsuitable for cases with yaw error since the rotor wake varies azimuthally, as well as radially. O'Brien demonstrated that an actuator disc acts as a elliptic wing and generates the roll-up of two tip vortices in the near wake, but they cannot capture the helical vortex wake associated with rotating blade systems, which an actuator-blade method was able to predict [19].

To address this issue for wind turbines, Sørensen et al. applied a variant of the actuator-blade method, known as an actuator-line method, whereby the blades are modeled by filaments or lines of sources along which body forces act [20,21]. These body forces are typically derived from a BEM method that uses tabulated airfoil data. In this approach, the unsteady effect on the flowfield by individual blades can be included in the analysis while still avoiding direct CFD modeling of the blade surfaces and their associated boundary layers. Sørensen et al. applied this actuator line method to a 500 kW Nordtank turbine and achieved good agreement with the experimental power curve for pre-stall wind speeds. Above about 12 m/s, where that particular rotor is stalled, they over-predicted power because using airfoil data implicitly ignores the three-dimensional effects present in the stalled regime.

The actuator line method has also been applied with good results by Mikkelsen, who compared it against an axisymmetric actuator disc approach and used it to validate some of the fundamental assumptions of traditional BEM methods [22]. Whereas Sørensen et al. used a code that solves the Navier–Stokes equations in vorticity–velocity form, Mikkelsen employed the incompressible code EllipSys3D [23,24], which solves in pressure–velocity form. This latter method permits the inclusion of solid boundaries like the turbine tower.

The EllipSys3D code was also used by Ivanell et al., who evaluated the vortical wake structures produced by the actuator line method [25]. In that work, the actuator lines were fixed in the grid, and the effects of blade rotation were applied via a boundary condition, which allowed them to use an efficient steady-state formulation. While this significantly decreases computational expense, it renders the method unsuitable for yawed cases. In addition, the azimuthal boundary condition was also periodic, so a N-bladed rotor could be modeled in only $1/N$ revolutions. Again, this precludes the application of the method to any case that does not have a periodic solution, which are the cases primary configurations of interest.

O'Brien's actuator blade method was originally applied to helicopter rotor-fuselage interaction problems using an unstructured CFD code [18,19]. The key difference between actuator line and actuator blade methods is that in the former, each blade consists of a single line of sources. To avoid discontinuities, each source's loading is distributed over multiple grid points using a "regularization function" that makes a source's influence at a distance r away from it scale with e^{r^2}. Conversely, the actuator blade method of O'Brien uses a rectangular array of sources for each blade, providing a continuous influence without the need for a regularization function (though smaller discontinuities in loading do still exist at the outlines of the blade). A rectangular array of sources also provides the means to vary the local angle of attack (a key input to the underlying blade element model) with chord. It should be noted, however, that the results presented here use a local angle of attack that is constant along the chord but still varying along the span.

Though they are undoubtably easier to apply to complex geometries, source cloud actuator methods like that of O'Brien [19] suffer from one significant drawback. In order to apply body forces from the actuator sources to the flow, it is necessary to know which grid node (or cell centroid for a cell-centered code) is closest to each actuator source. This entails some sort of nearest neighbor search procedure. In actuator disc cases or actuator line/blade cases in which the sources are fixed in the grid (perhaps because the equations are being solved in a rotating frame), this search can be performed as a one-time pre-processing step. In this instance, the cost of the search algorithm is not of paramount importance. However, if the sources move in relation to the inertial grid, the search must be repeated at each time step.

In a structured grid, there is a regular structure in memory that mimics planar spatial structures, making it readily efficient to search through a range of grid indices. In an unstructured grid, a much more general search procedure is required. With the exception of O'Brien's work, the actuator methods discussed thus far have been implemented within structured CFD methods. O'Brien's implementation for unstructured grids performs an exhaustive nearest-neighbor search by looping over all the grid nodes to find the node associated with a single source, and then repeating that loop for all the other sources. The exhaustive search represents another significant expense in addition to the usual computational overhead associated with storing and accessing connectivity information in an unstructured grid, so a better search algorithm is necessary for practical engineering applications.

In this paper, an efficient method for modeling rotating systems in engineering fluid dynamics applications is presented. The method is demonstrated using a wind turbine model for validations. This paper documents the adaption of the actuator blade model using clouds of sources that relocate in an inertial background grid that can include other stationary objects such towers and other turbines (with or without moving blades). In particular, details of a parallel search algorithm are included, as this implementation