Renewable Energy 85 (2016) 419–435

Contents lists available at ScienceDirect

Renewable Energy

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

Critical issues in the CFD simulation of Darrieus wind turbines

Francesco Balduzzi [a], Alessandro Bianchini [a], Riccardo Maleci [a], Giovanni Ferrara [a], Lorenzo Ferrari [b, *]

[a] Department of Industrial Engineering, University of Florence, Via di Santa Marta 3, 50139 Firenze, Italy
[b] CNR-ICCOM, National Research Council of Italy, Via Madonna del Piano 10, 50019 Sesto Fiorentino, Italy

ARTICLE INFO

Article history:
Received 4 December 2013
Received in revised form
3 April 2015
Accepted 16 June 2015
Available online xxx

Keywords:
CFD
Darrieus
Wind turbine
Simulation
Unsteady
Sensitivity analysis

ABSTRACT

Computational Fluid Dynamics is thought to provide in the near future an essential contribution to the development of vertical-axis wind turbines, helping this technology to rise towards a more mature industrial diffusion. The unsteady flow past rotating blades is, however, one of the most challenging applications for a numerical simulation and some critical issues have not been settled yet.

In this work, an extended analysis is presented which has been carried out with the final aim of identifying the most effective simulation settings to ensure a reliable fully-unsteady, two-dimensional simulation of an H-type Darrieus turbine.

Moving from an extended literature survey, the main analysis parameters have been selected and their influence has been analyzed together with the mutual influences between them; the benefits and drawbacks of the proposed approach are also discussed.

The selected settings were applied to simulate the geometry of a real rotor which was tested in the wind tunnel, obtaining notable agreement between numerical estimations and experimental data. Moreover, the proposed approach was further validated by means of two other sets of simulations, based on literature study-cases.

1. Introduction

Darrieus Vertical-Axis Wind Turbines (VAWTs) are receiving increasing interest in the wind energy scenario, as this turbine typology is thought to represent the most suitable solution in non-conventional installation areas, due to the reduced variations of the power coefficient even in turbulent and unstructured flows (Refs. from Ref. [1–7]), with low noise emissions and high reliability. Moreover, this technology is also gaining popularity for large-size floating off-shore installations (e.g. Ref. [8]).

The design and development of these rotors have been historically carried out with relatively simple computational tools based on the BEM (Blade Element Momentum) theory [9–12]. This kind of approach can still provide some advantages in many cases, especially concerning the preliminary design of a machine (e.g. overall dimensions and attended power), as it is generally quite reliable and with very reduced computational cost [9]. In addition, some more advanced techniques are presently available like wake models, vortex models or the Actuator Cylinder flow model [13].

As discussed by several authors (e.g. Refs. [14,15]), however, an accurate modeling of these machines cannot disregard anymore the recent developments in CFD simulations, as they can significantly contribute to the technological improvement in designing the rotors, needed to rise the technology towards a well-established industrial production. On this basis, one can easily argue that the goal of assessing a reliable approach to CFD simulation of Darrieus turbines is thought to represent one of the most challenging prospects for the future wind energy research.

Some of the most complex and less understood phenomena in the field of numerical simulations are involved in the analysis of the flow past rotating blades [9]. With particular reference to Darrieus wind turbines, the problems to be solved to correctly describe the flow field developing around the turbine are increased by the constant variation of the incidence angle with the azimuthal position of the blade and the strong interaction between the upwind and the downwind halves of the rotor ([9] and [12]). Moreover, a major aspect of the unsteady aerodynamics of Darrieus rotors is represented by dynamic stall, which often occurs at low tip-speed

* Corresponding author.
E-mail addresses: balduzzi@vega.de.unifi.it (F. Balduzzi), bianchini@vega.de.unifi.it (A. Bianchini), maleci@vega.de.unifi.it (R. Maleci), giovanni.ferrara@unifi.it (G. Ferrara), lorenzo.ferrari@iccom.cnr.it (L. Ferrari).

http://dx.doi.org/10.1016/j.renene.2015.06.048
0960-1481/

ratios (TSRs), where the range of variation of the incidence angle on the airfoils is larger ([9] and [16,17]).

Within this scenario, a relevant aspect which has not been often discussed in sufficient detail in the technical literature is the "philosophical" approach to CFD simulations, i.e. the goal of the simulations themselves and the most suitable tools to achieve it. In detail, if one would go through the problem with a logical approach, two main observations can be promptly made:

- The functioning principle of vertical-axis wind turbines, where the flow conditions seen by the blades changes instant by instant as a function of the position occupied in the revolution trajectory, made any typology of simulation definitely ineffective, with the only exception of a fully unsteady approach (i.e. the only able to catch the real interactions between the blades).
- Based on the above, the circumferential symmetry cannot be exploited like in many other turbomachinery applications: the full revolution of the blade must hence be described, leading to very heavy simulations in terms of mesh size and computational time.

Moving forward in the analysis, although the three-dimensional approach is the only able to really describe the flow field around the turbine (i.e. also the real performance), some considerations are here proposed to focus the attention on benefits, drawbacks and requirements of the a 2D or a 3D approach. In particular:

- A 3D approach is needed in case the simulations are deemed to provide the attended power output of the rotor. In these cases, the influence of spanwise velocity components, tip effects and interactions with the "parasitic" components (e.g. struts, tower, etc.) cannot indeed be neglected ([9] [12], and [18]).
- By doing so, enormous computational resources are generally needed [19] and in some cases (e.g. Ref. [20]) authors have proposed to apply different settings to three-dimensional simulations with respect to what has been defined for the "lighter" two-dimensional calculations.
- Some applications, however, do not indeed require an exact estimation of the overall performance of the rotor. In particular, if properly assessed, a 2D approach could be successfully applied to the analysis of many relevant issues connected to the functioning of Darrieus rotors, like the dynamic stall, the flow curvature effects and the wake interaction with the downwind half of the revolution [9]. Moreover, a reliable 2D simulation, coupled with simplified models to account for the main secondary effects [12], could also provide a first estimation of the overall performance of the rotor, to be compared and integrated with the results of the BEM codes conventionally exploited by industrial manufacturers.

Based on the above and observing that no agreement was found between the most accredited literature sources, in this work an extended analysis on the critical issues to properly perform a 2D simulation of a Darrieus rotor is presented and discussed. The assessment of a reliable setting for this type of approach can provide a very useful tool to more in depth analyze the real functioning of the turbines; contemporarily, it could represent the basis for a future extension of the analysis to full 3D models.

2. Literature review

The ongoing evolution of CFD solvers is providing new opportunities for wind turbine designers to enhance the comprehension of the real blades-flow interaction; in addition, the diffusion of commercial codes is thought to guarantee in the next future reliable tools for the development of new machines, with noticeable cost and time savings. On the other hand, the great advantages of CFD simulations in such complex phenomena, like those connected to rotating blades, can be nullified if unsuitable settings are implemented by the user. In particular, a proper definition of the computational parameters can be obtained only by means of a thorough sensitivity analysis on the influence of these variables on the results accuracy, possibly coupled with a validation study based on experimental data.

Within the present study, a detailed review on the state-of-the-art of numerical approaches for the simulation of Darrieus turbines was first carried out by the authors. Upon examination of the literature, several works were identified ([15] and [20–39]), all published in the past five years; as one may then notice, the topic is still quite new and there is a lack of extensive studies for the definition of practical guidelines to properly model the flow around a Darrieus turbine's blade.

As a result, even though the considered studies are all focused on the evaluation of the average power coefficient as a function of the TSR and/or the instantaneous power coefficient as a function of the azimuthal position of the blade, poor agreement was found on the most suitable settings to be adopted for the simulation. An effective convergence was indeed found only on the bases of the simulating approach, i.e.:

- The unsteady approach ([[15] and [20–34]) is largely preferred to the steady-state ([35–37]) or to the multiple reference frame ([38] and [39]) approaches. In the unsteady approach, the rotating machine is simulated with two distinct sub-grids: a circular zone containing the turbine geometry, and rotating with its angular velocity, and a fixed outer zone (with a rectangular shape in most cases), which defines the boundaries of the overall calculation domain. The two regions communicate by means of a sliding interface.
- Boundary Conditions: as widely accepted for similar simulations, a velocity inlet and a pressure outlet are used in the mainstream direction, whereas lateral boundaries are threated either as solid walls or with the symmetry condition ([15] [20–22] [24], [27,-28] [30], [33], [35,36]).
- A fully 3D approach is rarely adopted ([33,34], and [38]). In all other cases, a 2D approach is basically used, sometimes compared with a 3D attempt with very raw settings due to the enormous calculation costs ([20,21,23,30], and [37]).
- The simulations are mainly performed with the commercial code ANSYS® Fluent® ([15] [21–24] [26–29], [31], and [33–36]).
- The accuracy of the numerical results is usually checked by means of experimental data derived from wind tunnel measurements ([15] [21–26], [28,-29]).

Since the present study was conceived in view of a 2D unsteady approach, a more extensive analysis of the studies [15] and [20–32] is given in Table 1; the goal of this comparative analysis was in fact to highlight whether some general tendencies could have been found among the considered cases.

In particular, the benchmark was focused on the followings parameters:

- Turbulence modeling approach and models
- Numerical settings (solution algorithm and methods for the discretization of the N–S equations)
- Time-dependent solution settings (angular discretization and global duration of the calculation)
- Distance of the domain boundaries (inlet, outlet, lateral and sliding interface) from the turbine