J. Wind Eng. Ind. Aerodyn. 147 (2015) 77–84

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

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

3D URANS analysis of a vertical axis wind turbine in skewed flows

A. Orlandi [a,b], M. Collu [a,*], S. Zanforlin [b], A. Shires [c]

[a] Offshore Renewable Energy Engineering Centre, Cranfield University, Cranfield, MK43 0AL, UK
[b] Dipartimento di Ingegneria dell'Energia, dei Sistemi, del Territorio e delle Costruzioni, Universita di Pisa, Pisa 56122, Italy
[c] School of Mechanical Engineering, University of Leeds, Leeds LS2 9JT, UK

ARTICLE INFO

Article history:
Received 24 May 2015
Received in revised form
12 September 2015
Accepted 20 September 2015

Keywords:
Vertical axis wind turbine
Offshore
Skewed flow
Dynamic stall
CFD

ABSTRACT

The paper demonstrates the potential of an unsteady RANS 3D approach to predict the effects of skewed winds on the performance of an H-type vertical-axis wind turbine (VAWT). The approach is validated through a comparison between numerical and experimental results for a full-scale Darrieus turbine, demonstrating an improved prediction ability of 3D CFD with respect to both 2D CFD and semi-empirical models based on the double multiple stream tubes method. A 3D URANS approach is then adopted to investigate the power increase observed for a straight-bladed small-scale turbine in a wind tunnel when the rotational axis is inclined from 0° to 15° from the vertical. The main advantage of this approach is a more realistic description of complex three-dimensional flow characteristics, such as dynamic stall, and the opportunity to derive local blade flow conditions on any blade portion during upwind and downwind paths. Consequently, in addition to deriving the turbine overall performance in terms of power coefficient, a better insight into the temporal and spatial evolution of the physical mechanisms is obtained. Our principal finding is that the power gain in skewed flows is obtained during the downwind phase of the revolution as the end part of the blade is less disturbed by the wake generated during the upwind phase.

1. Introduction

The pursuit of reducing the cost of offshore wind energy in deep waters has led to a re-emerging interest in vertical-axis wind turbines (VAWTs) for floating applications due to apparent advantages over conventional horizontal-axis wind turbines (HAWTs) (Borg et al., 2014; Borg and Collu, 2015; Giorgetti et al., 2015). In parallel, there is a resurgence of interest in VAWTs as a promising alternative to HAWTs also for small-scale electric power in urban areas (Mertens, 2002).

VAWTs are characterized by some significant advantages: the ability to capture wind from any direction without a yaw control mechanism, low noise, compact design, simpler access, installation, maintenance and repair (since the gearbox and drive train components can be located at ground level rather than at the top of the tower as for HAWTs).

Furthermore, although VAWTs generally have a lower aerodynamic efficiency, there is some evidence that VAWTs can be positioned closer together in a wind farm giving a higher power density due to lower wake interference. Furthermore, counter-rotating VAWTs in close proximity have been shown experimentally (Kinzel et al., 2012) and numerically (Giorgetti et al., 2015) to have a mutually beneficial effect on power production.

It is perhaps the offshore environment that has attracted the greatest interest for VAWTs because of several inherent attributes that offer advantages with respect to HAWTs, particularly the scalability and low over-turning moments with better accessibility to drive train components (Shires, 2013; Borg et al., 2014).

A correct prediction of the turbine performance in skewed flows is very important for both micro generation in the built environment and large-scale offshore applications (Collu et al., 2014), since urban winds have noticeable vertical components (micro generation), and waves and unsteady wind speeds will induce pitch/roll motions to the turbine axis and therefore a periodic tilt angle with respect to the vertical (offshore floating wind turbine). HAWTs and VAWTs exhibit completely different behaviours in skewed flows. Theoretical studies and experimental measurements have shown that the power output in a skewed flow is reduced for an HAWT (Tongchitpakdee et al., 2005). This trend is mainly due to a reduction in the effective swept area (i.e. the area perpendicular to the oncoming wind direction). Consequently, initial studies have indicated that HAWTs can suffer severe performance losses when installed on floating support structures (Bekiropoulos and Rie, 2012). On the other hand for VAWTs, depending on the configuration, the performance degradation due to skewed flow is generally lower and for H-VAWT (VAWT with two vertical blades connected to the central tower through one/more arms) configurations the power coefficient may even be enhanced.

* Corresponding author: Tel.: +44 (0)1234 75 4779.
E-mail address: maurizio.collu@cranfield.ac.uk (M. Collu).

http://dx.doi.org/10.1016/j.jweia.2015.09.010
0167-6105/

Experimental investigations on H-VAWT turbines have demonstrated an enhanced performance for relatively small tilt angles (up to 25–30°) depending on the shape of the rotor, whereas a reduced performance is produced by higher skew values (Mertens et al., 2003). This benefit might be explained by the fact that VAWT blades sweep out a cylindrical surface, as opposed to a planar surface for HAWTs. As a consequence, during misaligned flow operations, the swept area of the turbine is increased.

To account for skewed flow effects on VAWT performance, the efforts of researchers have focused on the implementation of semi-empirical corrections in blade element momentum models (Bianchini et al., 2012; Madsen et al., 2014). To the authors' knowledge, there are no studies in the literature presenting a 3D CFD (Computational Fluid Dynamics) study on inclined VAWT performance. The present study is aimed at contributing to a better understanding of the physical processes that result in the measured performance enhancement reported for VAWTs in skewed flows.

2. Numerical approach

The two wind turbine configurations analysed in this work are:

- the SANDIA 17 m-diameter Darrieus-Type VAWT – used to validate the CFD modelling approach that has been adopted. It has an aspect ratio (height to diameter ratio) equal to 1.02, two blades with a NACA 0015 aerofoil section and a constant chord of 0.612 m. Rotational speed is fixed at 38.7 RPM (rotational speed, in revolutions per minute) for aerodynamic force measurements and 42.2 RPM for c_p measurements (Akins, 1989);
- a two bladed H-Darrieus VAWT with NACA 0018 aerofoil sections and constant 0.08 m chord. The rotor height is 0.5 m and diameter is 0.755 m. This configuration was used in wind tunnel tests to study the influence of skewed flows (Mertens, 2002) allowing direct comparison with CFD predictions.

For all the simulations, two different grid levels have been adopted: a fixed sub-grid with the external dimensions of the flow domain, and a dynamic sub-grid that includes the VAWT geometry and allows a relative motion with respect to the fixed grid. This grid arrangement utilizes the sliding mesh technique (Ansys, 2009) and allows the simulation of the rotating motion of the wind turbine with a steady RANS (Reynolds-averaged Navier-Stokes equations) or URANS (unsteady RANS) analysis.

The CFD process was validated for the SANDIA National Labs configuration using both 2D and 3D grids. The 2D mesh adopted is a hybrid structured–unstructured mesh with around 460 000 elements. The flow domain external dimensions were 37D x 25D, and the wall distance from the first layer of cells is set at $1.6{*}10-5c$ with $y+$ (dimensionless wall distance) <1, where D is the diameter of the turbine and c is the blade chord length. The 3D grid contains around 2 800 000 elements with flow domain external dimensions of $15D\times 5D\times 4D$ and a wall distance equal to $1.9{*}10-4c$, resulting in $y+<5$.

For the analysis of the rotor in a skewed flow, only a 3D approach was considered and the mesh contains around 1 800 000 elements. The external dimensions are $14D\times 30D\times 12D$ and the wall distance is $1{*}10-4c$, necessary for an averaged $y+<5$, following a mesh sensitivity analysis performed in order to verify if the grids adopted were sufficiently fine to resolve the primary flow features. Fig. 2 shows the analysis for the 2D model for the 17 m Darrieus VAWT and the 3D model for the small H-Darrieus VAWT; the 3D model of the SANDIA turbine has been excluded from this analysis due to the high number of cells of the domain, but the mesh adopted is similar to that used by Howell et al. (2010) and Zhang et al. (2013).

2.1. 2D and 3D mesh sensitivity analyses

For all the cases presented, both for the 2D and the 3D models, mesh sensitivity analyses have been conducted, to assess the adequate mesh element number able to resolve the flow. As regards the 2D model, three meshes were analysed (coarse, medium, fine) as shown in Table 1, keeping the wall distance fixed at $1.6{*}10^{-5}$ with $y+<1$.

The 3D mesh independence study was performed only for the skewed flow case. The same philosophy was then applied to the SANDIA case. For the 3D model of the turbine in skewed flow, two grids have been considered (4 and 5), as shown in Table 2. In general a larger number cases are analysed in order to conduct a sensitivity analysis, but in this case we considered the previous work done in this area (see cited papers). All the 3D meshes were built considering a wall distance equal to $1.9{*}10-4c$, resulting in $y+<5$.

The results of these analyses are shown in Fig. 1, where the Ct (tangential force coefficient) values are compared.

As regard the 2D analysis, the difference between mesh 2 and mesh 3 is minor, and mesh 2 was adopted, in order to save time and computational resources without substantially affecting accuracy. The same conclusion can be derived from the second graphs, showing a small gap between the two grids. So that MESH 4 was used for the 3D analysis. Furthermore, this meshing philosophy was subsequently adopted for the SANDIA turbine case.

2.2. CFD solver approach

The CFD software used for the present analyses is FLUENT v.15, developed by Ansys Inc. This study is based on the URANS implicit model, and turbulence is modelled using the k–ω SST (Shear Stress Transport) model. This turbulence scheme was adopted because of its aptitude in cases involving high adverse pressure gradients and therefore smooth surface separations (Menter, 1993). The air was considered as incompressible since the different cases studied did not exceed a local Mach number greater than 0.3.

The setup settings for the simulations are shown in Table 3. The simulations are done using a First Order Implicit scheme for all the variables of the spatial discretization in the first revolutions, after that increase the order of the schemes considered to prevent the model from numerical diffusion errors. The convergence criteria were set at $1{*}10^{-5}$ for all residuals.

Table 1
Details of the 2D grids.

Mesh	No. of nodes on blade	No. of elements	Wall distance
1	620	301 163	1.6 E−05
2	1240	464 845	1.6 E−05
3	1920	700 292	1.6 E−05

Table 2
Details of the 3D grids for H-Darrieus turbine.

Mesh	No. of elements	Nodes around blades	Nodes spanwise	Wall distance
4	1 724 930	84	200	1.6 E−04 chords
5	2 672 993	186	300	1.6 E−04 chords