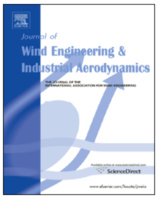
J. Wind Eng. Ind. Aerodyn. 154 (2016) 1–9

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

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



Assessment of blockage effects in wind tunnel testing of wind turbines

S. Zaghi [a,*], R. Muscari [a], A. Di Mascio [b]

[a] CNR-INSEAN, Istituto Nazionale per Studi ed Esperienze di Architettura Navale, Via di Vallerano 139, Rome 00128, Italy
[b] CNR-IAC, Istituto per le Applicazioni del Calcolo "M. Picone", Via dei Taurini 19, Rome 00185, Italy

ARTICLE INFO

Article history:
Received 3 August 2015
Received in revised form
11 March 2016
Accepted 31 March 2016
Available online 16 April 2016

Keywords:
Wind energy
Wind turbine
Wind turbine wake
Blockage
CFD

ABSTRACT

The aim of the present paper is the analysis of simplified boundary conditions to be used in numerical simulations, to take into account blockage effects for wind tunnel experiments of large scale wind turbines. The goal is the development of an efficient and reliable tool to be used to correct data obtained from experiments where the blockage coefficient is high and/or the turbine is highly loaded, for which traditional correction coefficients (derived from the Glauert theory or its more recent versions) fail.

Numerical simulations of the flow around a three-bladed model-scale wind turbine with horizontal axis are reported; in all test cases, the turbine diameter is comparable with test section dimensions, and therefore blockage effects are significant. The actual experiments were approximated numerically with a simplified wind tunnel geometry, that retains the symmetries of the isolated turbine simulation in a rotating frame and therefore allows steady state computations. To this end, two circular wind tunnel were tested: for the first, the radius was chosen to retain the same cross-section as the actual wind tunnel; in the second, its was set to be equal to half of the smallest cross-section dimension.

The aerodynamic performances of the turbine, in terms of power and thrust coefficients, are analyzed and compared with available experimental data. Detailed analysis of the flow in the wake is also reported. Analogous simulations in an unbounded domain are also reported.

1. Introduction

Nowadays, research on renewable energy is strongly increasing because of both the increased awareness of the need to preserve the environment and the decreased availability of fossil fuels. In the course of time, wind turbines evolved from rather primitive and small windmills to technologically advanced and huge mechanical systems. Their efficiency has also fostered the deployment of large Wind Turbine Farms (WTF). A historical review of wind energy conversion, together with a review of aerodynamic modeling and optimization of wind turbines, can be found in Sørensen (2011).

Of course, the larger the turbine, the more demanding is the design of the apparatus, as many aerodynamic, structural and environmental problems arises. Moreover, in large wind farms, the downstream turbine always operate in off-design conditions, and therefore the accurate prediction of the Horizontal-Axis Wind Turbine (HAWT) wake is of paramount importance.

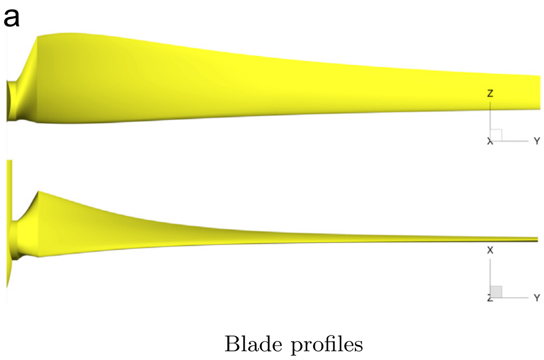
The increasing importance of wind energy is naturally accompanied by an intense design activity by numerical simulation, in particular for new concept designs. Of course, the development of new apparatuses requires accurate experimental confirmation, that fostered many experimental campaigns about turbine performances. The most comprehensive and recent measurements were carried out in the large wind tunnel test at NASA-Ames in the framework of the NREL Phase VI Wind Tunnel project (Hand et al., 2001), in the experimental campaign in the framework of the international cooperation project "Mexico" (Schepers and Snel, 2007; Snel et al., 2007) and in the experiments performed at the Norwegian University of Science and Technology (NTNU) (Krogstad and Adaramola, 2012; Adaramola and Krogstad, 2011). The wind tunnels adopted in the first two experiments were very large when compared with turbine diameter, and therefore blockage effects were estimated negligible; in the third one, instead, the presence of the side walls could not be simply neglected, because the ratio of the rotor area to the test section area is about 12.4% (Krogstad and Adaramola, 2012).

There are many methods to assess blockage effects, that must be properly estimated to extract the required information in the design phase. They range from the traditional methods based on the momentum theory (like the one developer by Glauert, 1983 or its recent versions Segalini and Inghels, 2014; Werle, 2010a, 2010b) to detailed unsteady Navier–Stokes simulations that take into account the actual wind tunnel geometry. The first kind of corrections are very useful when the blockage coefficient is small

---

[*] Corresponding author.
E-mail addresses: stefano.zaghi@cnr.it (S. Zaghi), roberto.muscari@cnr.it (R. Muscari), a.dimascio@iac.cnr.it (A.D. Mascio).

http://dx.doi.org/10.1016/j.jweia.2016.03.012
0167-6105/© 2016 Elsevier Ltd. All rights reserved.



Blade profiles

b

Main characteristics

Number of blades	3
Rotor diameter $[m]$	0.9000
Nacelle diameter $[m]$	0.0900
Root chord $[m]$	0.0495
Tip chord $[m]$	0.0259
Root twist angle $[deg]$	38.00
Tip twist angle $[deg]$	−0.72
Profile thickness $[\%chord]$	14.0

Fig. 1. Model-scale blade geometry. (a) Blade profiles. (b) Main characteristics.

and when the turbine is lightly or mildly loaded. On the contrary, when studying large (when compared to tunnel size) turbines, possibly highly loaded, a trustworthy answer about blockage effects can be obtained only by full Navier–Stokes simulations. This last approach, although reliable, is nevertheless expensive, because the flow cannot be steady, even when simplifying the problem by considering an isolated turbine in the rotating frame, as in general tunnel walls are not axisymmetric.

In the present paper, we investigated the possibility to replace the actual geometry of the wind tunnel with a circular section, in order to perform steady state computations in the rotating frame, while retaining flow details around the actual turbine geometry in the full range of loading conditions. This approach is much faster than the full unsteady Navier–Stokes simulation with the actual geometry, and therefore allows to widen the range of test cases for fixed computational resources. Of course, the choice of the equivalent wind tunnel section is all but trivial, as the only parameter at disposal (the section radius) must be such that the outcome of the simulations match both global and local quantities with the actual experiments. To this purpose, two test sections were considered: in the first, the radius was chosen so that the fictitious section has the same area as the actual section; in the second, the radius was set equal to the half of the smallest dimension of the cross-section. Numerical results were checked against the data obtained in the NTNU experiment (Krogstad and Eriksen, 2013; Krogstad and Adaramola, 2012; Krogstad and Lund, 2012).

The paper is organized as follows: a brief description of the model-scale wind turbine is reported in Section 2; mathematical and numerical models are briefly recalled in Section 3; details of the numerical setups are explained in Section 4; discussion on the numerical results is contained in Section 5; concluding remarks can be found in Section 6.

2. Wind turbine geometry

The wind turbine considered in the present paper is the three-bladed horizontal axis turbine, with NREL S826 section profile (Somers, 2005), designed and manufactured at the NTNU (Krogstad and Adaramola, 2012). The profile has 14% thickness, designed to optimize the performances for a Reynolds number of 2×10^6. The blades are made of aluminum and mounted on a small hub (nacelle) with a diameter $d = 0.09$ m. Between the blade and the nacelle there is a transition surface, i.e. a linear interpolation between the S826 profile and the circular profile on the nacelle surface. Fig. 1(a) shows two side-views, whereas Fig. 1(b) summarizes its main characteristics. Note that the blade terminates with a sharp cut.

More details about the turbine geometry (tabulated coordinates of blade profiles, twist angle and chord length) can be found in Krogstad and Adaramola (2012), where we also took all experimental data used to validate our numerical results.

Table 1
Inflow conditions and reference values.

(a) Inflow conditions	
$U_{inf}\ \left[\frac{m}{s}\right]$	10.00
$\rho_{inf}\ \left[\frac{kg}{m^3}\right]$	1.20
$\nu_{inf}\ \left[\frac{m^2}{s}\right]$	1.50×10^{-5}
(b) Reference values	
U_0	$0.7\lambda U_{inf} = 0.7\omega R$
$\rho_0\ \left[\frac{kg}{m^3}\right]$	$1.200 = \rho_{inf}$
$\nu_0\ \left[\frac{m^2}{s}\right]$	$1.500 \cdot 10^{-5} = \nu_{inf}$
L_0 [m]	$25.926 \cdot 10^{-3} = Chord_{tip}$

Table 2
Tests matrix.

λ	$\omega\ \left[\frac{rad}{s}\right]$	Re
1	22.222	1.210×10^4
2	44.444	2.420×10^4
3	66.667	3.630×10^4
4	88.889	4.840×10^4
5	111.111	6.049×10^4
6	133.333	7.259×10^4
7	155.556	8.469×10^4
8	177.778	9.679×10^4
9	200.000	1.089×10^5
10	222.222	1.210×10^5
11	244.444	1.331×10^5
12	266.667	1.452×10^5

Table 3
Numerical grids dimensions.

Region	Number of blocks	Number of finite volumes
Background	142	3.52 M
Single blade	78 ($\times 3 = 234$)	2.50 M ($\times 3 = 7.50$ M)
Nacelle	40	1.40 M
Total	416	12.42 M

3. Mathematical and numerical models

The presented simulations have been performed by means an in-house software developed and validated for aero/hydrodynamics