Applied Ocean Research 69 (2017) 138–147

Contents lists available at ScienceDirect

Applied Ocean Research

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

Flow dynamics inside the rotor of a three straight bladed cross-flow turbine

M. Somoano, F.J. Huera-Huarte *

Department of Mechanical Engineering, Universitat Rovira i Virgili (URV), 43007 Tarragona, Spain

ARTICLE INFO

Article history:
Received 14 June 2017
Received in revised form 5 September 2017
Accepted 20 October 2017

Keywords:
Cross-flow turbines
Darrieus rotors
H-rotors
Straight-bladed turbines
Vertical axis wind turbines

ABSTRACT

In this work we study experimentally the flow dynamics inside the rotor of a three straight-bladed Cross-Flow Turbine (CFT). The CFT model used in the experiments is based on symmetric NACA-0015 profiles, with a chord to rotor diameter ratio of 0.16. The turbine model was designed in order to quantify the flow inside and around the rotor using planar Digital Particle Image Velocimetry (DPIV). Tests were made by forcing the rotation of the turbine with a DC motor, which provided precise control of the Tip Speed Ratio (TSR), while being towed in a still-water tank at a constant turbine diameter Reynolds number of 6.1×10^4. The range of TSRs covered in the experiments went from 0.7 to 2.3.

The focus is given to the analysis of the blade-wake interactions inside the rotor. The investigation has allowed us to relate the interactions with the performance differences in this type of turbines, as a function of the operational tip speed ratio.

1. Introduction

In the last decade there has been an increased interest for Cross-Flow Turbines (CFTs) and their capabilities for energy harvesting, whatever the origin of the stream is [1,2]. Cross-flow turbines are also referred to as Vertical Axis Wind Turbines (VAWTs) or Darrieus rotors, and compared to Axial Flow Turbines (AFTs), usually designated as Horizontal Axis Wind Turbines (HAWTs), straight-bladed CFTs have several advantages: omni-directionality, lower manufacturing cost, less noise emittance, better mechanical performance at high velocities and greater efficiency in turbulent flows [3–5]. Furthermore, flow energy utilization for ideal Darrieus rotors [6] can be in theory as high as 0.72 [7], well over the limit of 0.59 imposed by Betz's law [8]. Lifting devices are designed for a Tip Speed Ratio (TSR) greater than one [9]. The tip speed ratio is defined as the ratio between the tangential speed of the blade due to its rotation and the free stream velocity, $\lambda = \omega D/2u_\infty$, with D being the rotor diameter, ω the angular velocity of the rotor and u_∞ the free stream velocity. Nevertheless, the performance of CFTs is far from the ideal values, fact that could be in part attributed to a lack of understanding of the complex flow phenomena inside the rotor.

Kjellin et al. [10] performed experimental tests on a straight-bladed CFT in order to obtain its power coefficient (C_P) as function of λ. The C_P is a dimensionless quantity for the power output of the turbine, defined as $2M\omega/\rho L D u_\infty^3$, where M is the torque on the shaft, ρ is the fluid density and L is the blade span. The rotational speed of the generator used in the experiments was controlled in order to keep the turbine speed constant, while the wind speed was systematically changed in their facility. Bachant and Wosnik [11,12] did experiments in a water tank in which they towed the turbine model at a constant speed, leading to a constant Reynolds number based on the rotor diameter, $Re_D = u_\infty D/\nu$ with ν being the fluid kinematic viscosity, while the turbine shaft was loaded with an AC servo motor that provided a precise control of TSR. In these experiments, the authors used the same towing tank and the same CFT model. It consisted of a three straight bladed turbine based on a constant NACA-0020 profile, with a chord to diameter ratio $c/D = 0.14$. The main difference was that in Bachant and Wosnik [11] the authors studied the turbine performance at a $Re_D = 1.0 \times 10^6$ whilst Bachant and Wosnik [12] did it for Re_D in the range that goes from 3.0×10^5 to 1.3×10^6.

Bravo et al. [13] and Fiedler and Tullis [14] used a straight-bladed CFT to perform experiments at two different wind-tunnel facilities, obtaining similar $C_P(\lambda)$ curves. Their models were three-bladed based on constant NACA-0015 profiles with $c/D = 0.16$. Bravo et al. [13] carried out the experiments for Reynolds numbers in the range $Re_D = 1.0 \times 10^6$–2.7×10^6 and Fiedler and Tullis [14] in the range $Re_D = 8.3 \times 10^5$–1.8×10^6. Takao et al. [15] presented as well, the efficiency of a three straight bladed turbine based on a constant NACA-4518 profile with a chord to diameter ratio

* Corresponding author.
E-mail address: francisco.huera@urv.cat (F.J. Huera-Huarte).

https://doi.org/10.1016/j.apor.2017.10.007
0141-1187/

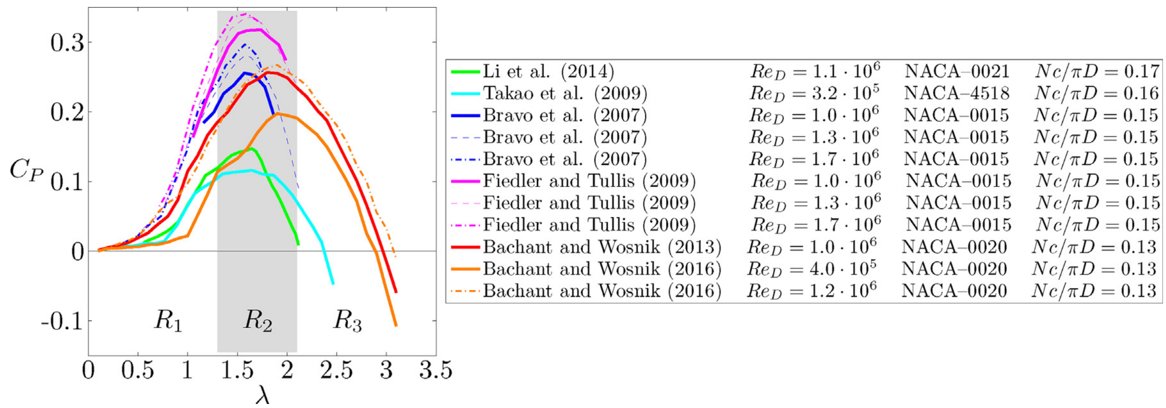

Fig. 1. Dimensionless performance curves associated to different studies with turbines of similar solidity in works reported by some authors, at several Re_D.

$c/D = 0.17$, in wind-tunnel experiments at a $Re_D = 3.2 \times 10^5$. Li et al. [16] compared the performance obtained by different measuring instruments on a four-bladed CFT at a wind-tunnel facility at a $Re_D = 1.1 \times 10^6$. His wind turbine was based on constant NACA-0021 blades with $c/D = 0.13$. Fig. 1 compiles some of the performance curves obtained experimentally by all these authors for comparison, for turbine solidities $\sigma = Nc/\pi D$ in the range 0.13 to 0.17.

As it can be seen in the figure, the performance curves for this type of turbines show three clearly distinct and well defined regions in the λ axis. The first region (R_1) is for the lowest tip speed ratios (due to the highest shaft resisting torques) where C_P grows monotonically. In the second region (R_2) for intermediate tip speed ratios, the C_P exhibits the highest values with its maximum. Finally, in the third region (R_3) the coefficient drops to very low values that tend to 0, point at which the turbine is not loaded and therefore the resisting torque becomes 0. In Fig. 1 the second region or R_2, in which the performance is higher, appears shaded for reference. The intention here is not to define these three regions in a closed manner, i.e. describing the λ at which each region starts and ends, but to emphasize from a qualitative perspective, how for sufficiently low or high λ, the performance of this type of turbines is considerably lower than that typical in the second region. It is also well known that decreasing the turbine solidity yields an increase in the value of λ that maximizes the performance of the turbine [17–19]. Moreover, by varying the velocity of the incoming flow, i.e. Re_D, the value of the maximum power coefficient is modified but the ranges of TSRs at which these three different regions take place, remain practically the same.

Bachant and Wosnik [20] showed that near wake statistics such as mean velocity, turbulence intensity and Reynolds stress, varied very weakly over the range $Re_D = 3.0 \times 10^5$–1.3×10^6. In the work by Parker and Leftwich [21], the authors reported that the general structure of the time-averaged stream-wise velocity in the wake was very similar over a wide range of Re_D (from 6.0×10^4 to 1.8×10^5). The turbine they used was a three straight bladed model with a c/D ratio of 0.16. These authors conducted their experiments with a model that was not flow-driven but motor-driven because of the low Re at which the facility was limited, arguing that the dynamics in the wake were the same in both situations. The aim was to study and compare the results obtained in the laboratory with a motor-driven turbine, with those of a flow-driven full scale machine, that operated at considerably higher Re. The comparison between a flow and a motor driven turbine was in fact first investigated by Araya and Dabiri [22], who found a very weak dependence of the wake mean velocities on Reynolds number, but a very strong dependence on TSR over the range of Re_D from 3.7×10^4 to 7.8×10^4. The authors reported measurements in the near wake of their turbine, for comparison between a flow-driven and a motor-driven CFT model, using Digital Particle Image Velocimetry (DPIV). They found nearly identical stream-wise and lateral mean velocity profiles, lateral velocity fluctuation power spectra, shaft torque, and circulation in the wake in both situations, flow-driven and motor-driven, over a TSR range of $1.2 < \lambda < 1.7$. They concluded that the kinematics of the spinning turbine, not the method by which it is set in motion, determines the wake dynamics. Based on their assumption it appears obvious that a motor-driven turbine can be used in order to study the flow dynamics and the blade interactions in turbines, even if Re typical of flow-driven machines are not achieved.

Although there is a large number of articles devoted to the analysis of the performance of cross-flow turbines, there are only a few that deal with the fluid dynamics of the rotor and in particular with the flow interactions between the blades. Most of the papers that deal with the fluid dynamics of CFTs focus their attention on the near wake region downstream of the turbine or in time averaged measurements [21,23,24], but not inside the rotor, because of the complexity of having reliable measurements or computations in that area. To study dynamic stall, Fujisawa and Shibuya [25] and Simão Ferreira et al. [26] carried out DPIV in a water and in a wind tunnel respectively, of the flow field around a rotating airfoil. In a recent numerical work by Ferrer and Willden [27], the authors defined what they called blade-wake interaction limits as the tips speed ratios that change the types of interaction in the rotor of their turbine. The regions that they defined in the λ axis are dependant on how the wake of the blades that are in the upstream part of the rotor, interact with the ones on the downstream part. This dependence is established based on an axial flow speed induction factor, that they use to quantify the velocity deficit inside the rotor and how vortices generated in the upstream part, are convected and interact with the downstream ones.

The objective of this article is the study of the flow dynamics and the interaction between the wakes shed by the blades of a cross-flow turbine. The experimental investigation presented here is based on a three bladed CFT, designed to match the main parameters in the flow-driven experiments reported in Bravo et al. [13] and Fiedler and Tullis [14], with the same number of blades and chord to diameter ratio, i.e. the same turbine solidity, as well as with the same blade profiles. The performance curves in these experiments exhibited the three previously commented regions (R_1, R_2 and R_3) in the tip speed ratio axis, independently of the Reynolds number. The aim is to study the blade-wake interactions inside the rotor of CFTs, in order to better understand the flow physics associated to each region in their performance curves. Experiments were carried out at different λ, with a turbine model specifically designed to quantify the flow using DPIV. The rotation of the turbine was prescribed as in Araya and Dabiri [22] and Parker and Leftwich [21], in order