Energy 35 (2010) 3349–3363

Contents lists available at ScienceDirect

Energy

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

# Unsteady Aerodynamics of a Savonius wind rotor: a new computational approach for the simulation of energy performance

V. D'Alessandro*, S. Montelpare, R. Ricci, A. Secchiaroli

*Università Politecnica delle Marche, Dipartimento di Energetica, Via Brecce Bianche 1, 60131 Ancona, Italy*

ARTICLE INFO

*Article history:*
Received 27 October 2009
Received in revised form
8 March 2010
Accepted 10 April 2010
Available online 15 May 2010

*Keywords:*
Savonius
Vertical axis wind turbine
Computational fluid-dynamics
Wind tunnel testing

ABSTRACT

When compared with of other wind turbine the Savonius wind rotor offers lower performance in terms of power coefficient, on the other hand it offers a number of advantages as it is extremely simple to built, it is self-starting and it has no need to be oriented in the wind direction. Although it is well suited to be integrated in urban environment as mini or micro wind turbine it is inappropriate when high power is requested. For this reason several studies have been carried-out in recent years in order to improve its aerodynamic performance. The aim of this research is to gain an insight into the complex flow field developing around a Savonius wind rotor and to evaluate its performance. A mathematical model of the interaction between the flow field and the rotor blades was developed and validated by comparing its results with data obtained at Environmental Wind Tunnel (EWT) laboratory of the "Polytechnic University of Marche".

## 1. Introduction

Wind turbines are usually classified in two main classes: lift driven wind turbines and drag driven wind turbines. In the former case the aerodynamic lift is the force responsible for the rotation whereas in the latter case the aerodynamic drag is the force which makes the turbine to spin.

The Savonius wind turbine is a vertical axis wind turbine (VAWT) created for the first time by the Finnish engineer Sigurd Savonius in 1925. As this rotor is classified as a drag driven device, aerodynamic theories developed in order to analyze wind turbines driven by lift force, cannot be applied.

In the field of horizontal axis wind turbine as well as lift driven vertical axis turbine such as the Darrieus rotor, BEM (Blade Element Momentum) theory [1] finds its best application.

From what above observed, follows that wind tunnel tests and computational techniques are the only tools available for studying Savonius wind rotors.

The aim of this work is to gain an insight into the complex flow field developing around a Savonius (split-type) wind rotor and to evaluate its performance.

Several numerical analyses have been performed on Savonius rotor aerodynamic using both DVM (Discrete Vortex Method) and CFD methods. Fujisawa [2] and Fernando and Modi [3] used DVM in the prediction of the rotor performance. It is well known that this method does not yield a quantitatively accurate prediction of the rotor performance (in comparison with measured data) but it reproduces the main features of the performance curves and flow field as reported in [2,3]. CFD analyses were conducted by Shinohara and Ishimatsu [4] and Redchyts and Prykhodko [5]. In particular an unstructured finite volume method was used in [5]. In this analysis Reynolds Averaged Navier–Stokes (RANS) equations were solved for the computation of the turbulent flow using a third order TVD (Total Variation Diminishing) and a fifth order Roe flux scheme for the discretization of the convective terms, while diffusive terms were discretized by a central difference scheme; Spalart–Allmaras turbulence model [6] was used. Other numerical studies about Savonius rotor aerodynamic performance are available in literature: these simulations were conducted in static conditions, varying the rotor angular position relative to the wind direction as in [7].

A crucial issue in the analysis of the flow field around the Savonius rotor is the treatment of the fluid–solid coupling and its modelling. Modelling the fluid-structure interaction (FSI) is a problem in many industrial applications. Generally a distinction is made between three categories of fluid–solid coupling [8]: a one-way solid to fluid reaction (solid motion influences fluid pattern but the fluid field does not affects the solid), one-way fluid to solid reaction (fluid field moves the solid but the latter does not modify the fluid pattern) and the two-way coupled interaction where reciprocal influences are modelled. Obviously, in these simulations the accuracy of the model and its computational cost is heavily

---

* Corresponding author. Tel.: +39 071 220 4359; fax: +39 071 220 4770.
*E-mail address:* v.dalessandro@univpm.it (V. D'Alessandro).

0360-5442/$ – see front matter © 2010 Elsevier Ltd. All rights reserved.
doi:10.1016/j.energy.2010.04.021

**Nomenclature**

| | |
|---|---|
| $\mathbf{u}$ | velocity vector [m/s] |
| $\mathbf{x}$ | position vector [m] |
| $\mathbf{x}_G$ | rotation center [m] |
| $P$ | Power [W] |
| $C$ | dimensionless coefficient |
| $\overline{\phi}$ | generic time-averaged variable |
| $\langle\phi\rangle$ | generic angular averaged variable |
| $R$ | rotor radius [m] |
| $R_{mg}$ | radius of the rotating domain [m] |
| $N$ | number of cells sliding at each time-step |
| $S$ | stage number of Runge–Kutta method |
| $H$ | polynomial order of resistant torque |
| $Z$ | Number of time-steps for every round |
| $\mathbf{e}_k$ | unit vector normal to the fluid flow panel |
| $M$ | torque [Nm] |
| $I$ | moment of inertia [kg m$^2$] |
| $p$ | pressure [Pa] |
| $k$ | turbulent kinetic energy [m$^2$/s$^2$] |
| $\mathbf{T}$ | stress tensor [Pa] |
| $\bar{\bar{\mathbf{I}}}$ | identity tensor |
| $\bar{\bar{\bar{C}}}_p$ | Power coefficient |
| $C_m$ | Torque coefficient |
| $u_\infty$ | wind velocity [m/s] |
| $f$ | elliptic relaxation factor [1/s] |
| $v^2$ | Reynolds stress normal to the wall [m$^2$/s$^2$] |
| $R^2$ | coefficient of determination |
| $\Delta t$ | time-step size |

*Greek Symbols*

| | |
|---|---|
| $\Gamma$ | domain boundary |
| $\Omega$ | generic calculus domain |
| $\rho$ | fluid density [kg/m$^3$] |
| $\nu$ | kinematic viscosity [m$^2$/s] |
| $\mu$ | dynamic viscosity [Pa s] |
| $\lambda$ | Tip-speed ratio |
| $\omega$ | angular velocity [rad/s] |
| $\varepsilon$ | rate of dissipation of TKE [m$^2$/s$^3$] |
| $\beta$ | blockage factor |
| $\eta$ | Kolmogorov length scale [m] |
| $\phi$ | generic fluid-dynamic variable |
| $\varepsilon_{\mathrm{ijk}}$ | Ricci-Cubastro tensor |

*Superscript*

| | |
|---|---|
| $(n)$ | time-step number |
| $(k)$ | generic rotor round |
| $T$ | transpose |

*Subscripts*

| | |
|---|---|
| $R$ | resistant |
| $F$ | fluid |
| $S$ | solid |
| $a$ | aerodynamics |
| in | domain inlet |
| out | domain outlet |
| $w$ | wall |
| $G$ | rotation center |
| $mg$ | moving grid |
| $i$ | Runge–Kutta method substep |
| $g$ | grid |
| ^ | unit vector |
| $k$ | summation index |
| $m$ | torque |
| $p$ | power |

influenced by the assumptions made on the nature of the fluid (viscous-inviscid) and on the structure (rigid-deformable). The model treated in this work is a two-way coupling type; the structure (rotor) was treated as a rigid body while the fluid was modelled as incompressible and viscous. The main problem in the fluid-structure interaction modelling is the procedure used to take into account the motion of the solid body in the solution of the fluid-dynamics equations. The strategy to solve this problem used in this work was a SMM (Sliding Mesh Model) approach while Navier–Stokes equations were solved using the finite volume code FLUENT. The solid body motion was treated solving the second cardinal equation of dynamics by means of a custom MatLab numerical algorithm able to import CFD data, calculate the rotor angular velocity and export this variable as input to the CFD code. Time marching of the solution of the second cardinal equation has been executed using an Euler method in the initial steps and a four stage Runge–Kutta (or an Adams–Bashfort) scheme in the following steps.

## 2. Experimental analysis

Experimental measurements on a full scale Savonius wind turbine were carried-out in the Environmental Wind Tunnel (EWT) of the Polytechnic University of Marche, (Fig. 1). The EWT test chamber consists of three main sections: the first is used for aerodynamic tests requiring a uniform velocity distribution and a low turbulence level; the second is used to test reciprocal interference effects between slender bodies; the latter is used to test wind effects over buildings, structures, orography models which are subjected to fully developed environmental boundary layers. A schematic representation of the test section is reported in Fig. 2. The wind tunnel is supplied by a fan having a constant rotational speed of 975 RPM, consisting of 16 blades with an adjustable pitch that ensure a regulated wind velocity in the test section between 6 m/s and 40 m/s. Constant Temperature Hot Wire Anemometer (CTA HWA) measurements showed a lack of flow uniformity less

**Fig. 1.** The Environmental Wind Tunnel of the “Polytechnic University of Marche”.