European Association for the
Development of Renewable Energies,
Environment and Power Quality

International Conference on Renewable Energies and Power Quality (ICREPQ'09)

Valencia (Spain), 15th to 17th April, 2009

Simulation of the Aerodynamic Behaviour of a Micro Wind Turbine

J. M. M. Monteiro[1], J. C. Páscoa[1] and F. M R. P. Brójo[2]

[1] Department of Electromechanical Engineering
[2]Department of Aerospace Sciences and Technology
University of Beira Interior
R. Marquês D'Avila e Bolama – 6201-001 Covilhã (Portugal)
Fax number:+00351 275 329762, e-mail: monteiro@ubi.pt, pascoa@ubi.pt, brojo@ubi.pt

Extended Abstract.

In this paper the Computational Fluid Dynamics (CFD) simulation of a wind turbine with a rotor diameter of 2.2 meters is performed using the Fluent commercial code. Renewable energies are only effective if implemented in a large scale basis. Given this fact, this CFD study is part of the effort that is currently being made at the University of Beira Interior, Portugal, in order to develop a family of complete micro wind systems suitable for partial production by the final users. The target objectives of the project are to provide a quality, open source, small wind turbine project for the masses, with emphasis on low cost, reliability and ease construction. In a first phase battery charging systems are being developed, to be followed later by grid connected ones. The micro turbines being developed follow the mainstream configuration for this size class, and so are of the horizontal axis type, have three blades, a tip speed ratio of 6, constant pitch and a passive wind orientation system with side furling. The generators are of the axial flux permanent magnet type. The blades have a NACA 4415 wing section in their entire span which is, at present, judged as a good compromise between performance and structural ruggedness given that it is a relatively thick profile. A 3D CAD model of the rotor was modelled with the Rhinoceros 3D program. In order to support the physical development of the wind turbine CFD simulations were performed. In this work results will be presented for turbulent flow computations around the wind turbine. The flow was computed by solving the Navier-Stokes equations with $k-\varepsilon$ turbulence model. A detailed analysis of the pressure distribution for three sections along the blade span is also presented. Load distribution and boundary layer separation is studied. Finally, a series of computations are performed in order to obtain the wind turbine torque and efficiency signature as a function of rotation. The obtained results will be, in the near future, compared to full-scale field tests. These will be carried out on a mobile test rig currently under development at University of Beira Interior, Portugal.

Key words

wind turbines, turbulence, CFD, low-Reynolds number

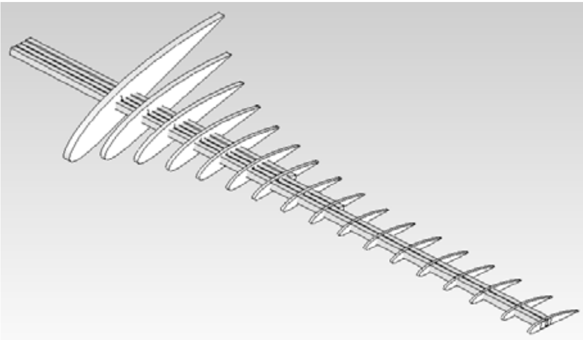

Fig. 1. 3D modelling of the internal structure of the wind turbine blade.

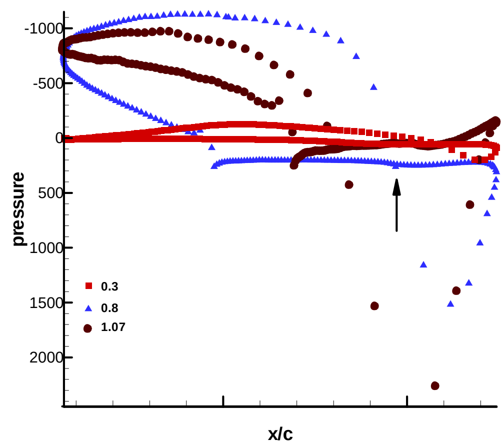

Fig. 5. Pressure distribution, in SI units, around a blade section of the wind turbine. Results are presented for a section near hub (0.3 m from rotating axis), for a section at midspan (0.8 m) and for a near tip section (1.07 m)