STUDY OF THE FLOW FIELD AROUND A WIND TURBINE

Author: Babadjanova Aygul Kamiljanovna Supervisors: Óscar López Pouso (USC), José Luis Ferrín González (USC), José Carlos Araujo Martín (Iberdrola Renovables)

ABSTRACT

A wind turbine is a device for extracting kinetic energy from the wind. By removing some of its kinetic energy the wind must slow down but only that mass of air which passes through the rotor disc is affected. So behind the rotor disc the nacelle wind speed U_{nac} differs from the free stream wind speed U_∞. For managing and optimizing the wind farm power production an accurate calculation of nacelle wind speed is demanded.

This study was concerned with performing simulation of airflow using CFD software named FLUENT so as to visualize the flow behavior around and within a rotating wind turbine. To achieve that, the realizable k-ε turbulence model was used by taking advantage of moving mesh method to simulate the rotation of wind turbine and the consequent results were obtained through the sequential process which ensured accuracy of the computations. During the simulation the flow field past the blade of a Horizontal Axis Wind Turbine (HAWT) has been modeled with 3-D Reynolds averaged Navier-Stokes (RANS) equations approach and we consider that the turbulence flow is steady. The tip speed ratio is a very important parameter for wind turbine design.

$$\lambda = \frac{\Omega R}{U_\infty}, \qquad (1)$$

where Ω is the angular velocity, R the radius of the blades and U_∞ the undistributed wind speed. Usually the value of tip speed ratio for three blades turbines is between 6 and 8. The rotational speed is 2.8 rad/sec with a free stream velocity of 16 m/s. Hence, the TSR (tip speed ratio) is equal to 7 (see Equation 1).

Different size mesh generation has been presented taking into account the different range of geometric scales.

Resulting of using the 3D RANS solver method indicates that:

- The pressure contour plots show the pressure drop across the rotor and the lower pressure zone is behind the rotor such that the around the rotation center (see Fig.1). And we have seen that the lowest pressure region on the nacelle gives approximately the same range of the velocity magnitude as in the free-stream. This information can be used to look for the optimum location of the anemometer (see Fig.2) .
- The rotation of the wake is opposite to the rotation of the rotor as it is predicted in the rotor disc theory.
- The visualization of the path-lines behind and under the nacelle shows that there is turbulence which can hurt the turbine and reduce its operational life (see Fig.3).

RESULTS

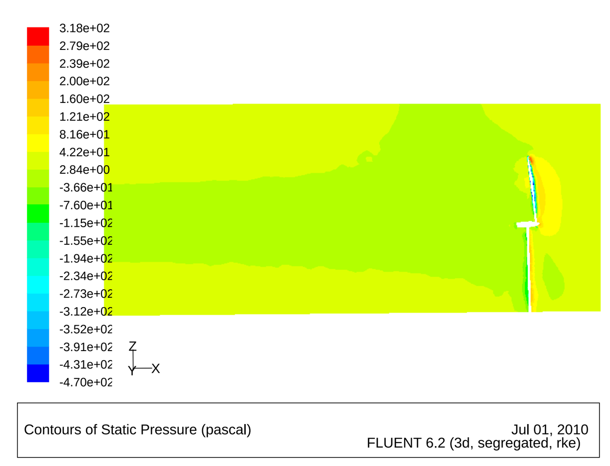

Fig. 1: Pressure drop across the rotor.

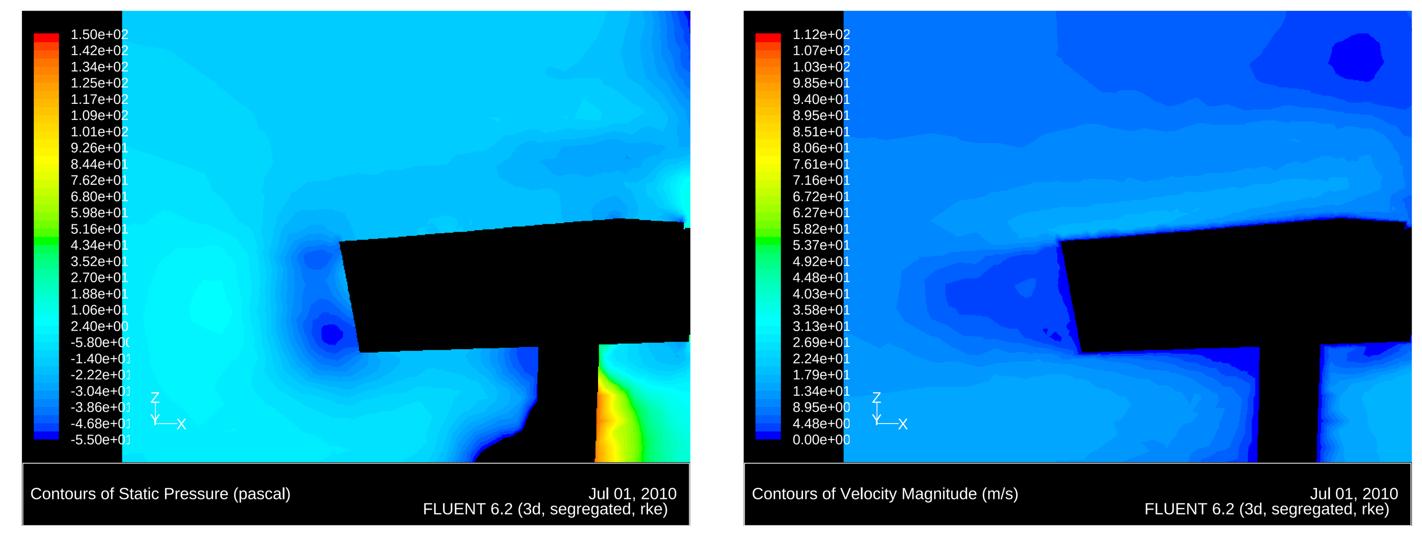

Fig. 2: Pressure and velocity on the nacelle.

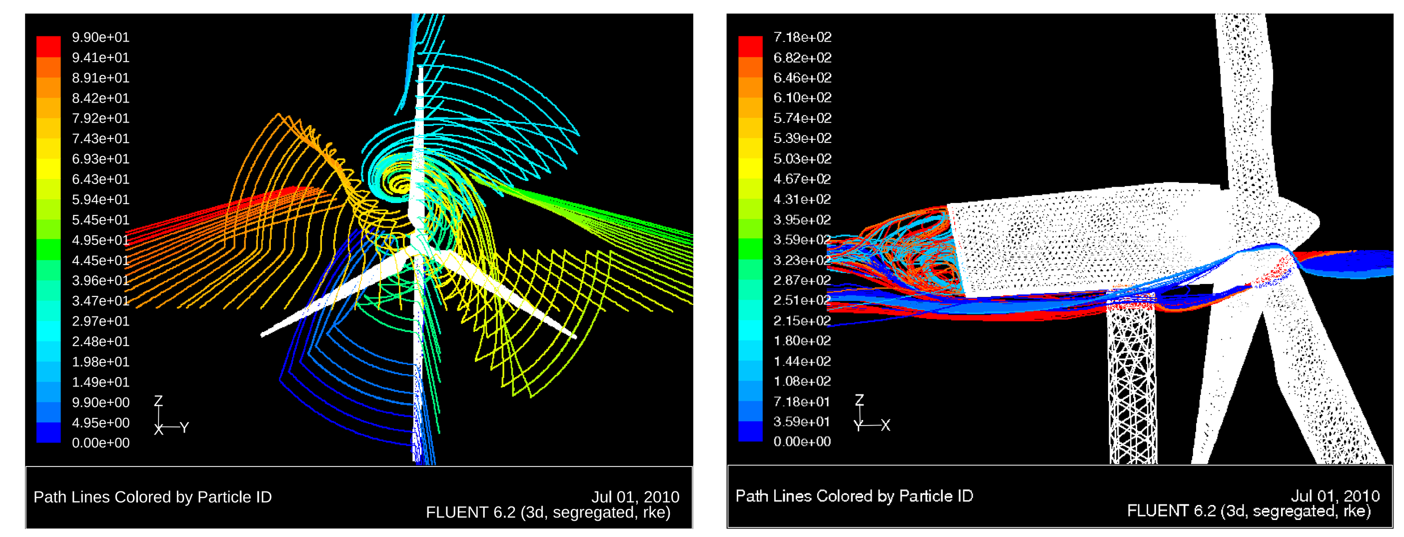

Fig. 3: Pathlines of particles and turbulence around the nacelle.

REFERENCES

[1] Carcangiu, C. E.; Sørensen, J. N.; Cambuli, F. and Natalino, M. 2007 *CFD-RANS Analysis of the Rotational Effects on the Boundary Layer of Wind Turbine Blades.* J. Phys: Conf. ser **75**.

[2] Masson, C. and Smaïli, A. 2005 *Numerical Study of Turbulent Flow around a Wind Turbine Nacelle.* Wind Energy **9**, issue 3, pp. 281-298.

UNIVERSIDADE DE VIGO USC UNIVERSIDADE DE SANTIAGO DE COMPOSTELA UNIVERSIDADE DA CORUÑA

Proxecto de Máster. Curso 2009-10
Máster Oficial en Enxeñaría Matemática

IBERDROLA RENOVABLES