11th World Congress on Computational Mechanics (WCCM XI)
5th European Conference on Computational Mechanics (ECCM V)
6th European Conference on Computational Fluid Dynamics (ECFD VI)
July 20 - 25, 2014, Barcelona, Spain

COMPUTATIONAL FLUID DYNAMICS (CFD) SIMULATIONS OF AN H-DARRIEUS ROTOR WITH DIFFERENT TURBULENCE MODELS

László Daróczy[1]*, Gábor Janiga and Dominique Thévenin

[1]University of Magdeburg "Otto von Guericke", Universitätsplatz 2, D-39106 Magdeburg,
laszlo.daroczy@gmail.com, http://www.uni-magdeburg.de/isut/LSS/

Key Words: *Computational Fluid Dynamics, Turbulence models, Wind energy, Darrieus*

Wind energy represents nowadays an increasingly important area of energy industry. Due to the world's ever growing hunger for clean and reliable energy the gross production of wind energy in the EU27 increased from 80 PJ (2000) to 537 PJ (2010) just over a period of ten years [1], and an increase to 2300 PJ is prognosed for 2030 [2]. Although Horizontal Axis Wind Turbines (HAWTs) are most common, there is an increased interest towards VAWTs (Vertical Axis Wind Turbines), especially the Darrieus-concept. These rotors are particularly well fitted for lower wind speeds, urban environments and for private households and are of very simple design.

While Horizontal Axis Wind Turbines (HAWTs) can be simulated in a quasi-steady way, in contrast, VAWTs are highly transient, with significantly varying angle of attack (AOA) (and thus aerodynamic characteristics) even during a single rotation. As a result, flow phenomena, such as stall, are difficult to be precisely simulated even with methods such as LES (Large Eddy Simulation) or DNS (Direct Numerical Simulation). Moreover, it is impossible to apply these technics with the available computing resources for optimizations. As a result, many different methods were developed to address these turbines, such as Blade Element Momentum theory or vortice lattice methods. Lately, due to the improved availability of computing resources many researchers turned to CFD (Computational Fluid Dynamics) techniques to evaluate the performance, but the different publications yielded to completely different proposed models, mesh sizes and solver settings [3-5], there is still no agreement.

The goal of the current publication is to analyse a single configuration with different turbulence models, and to choose the most precise model, while keeping the run-time still appropriate for optimizations. All simulations are carried out in the commercial CFD software, ANSYS Fluent 14.0. The simulations are prepared using in-house parameterization, design-of-experiment and optimization software of the institute, OPAL++ [6]. In the first step a mesh independency test is performed using 5 different mesh sizes for the chosen geometry for a single TSR value with different turbulence models, such as e.g., the k-ε Realizable model with Enhanced Wall Treatment, the Scale-Adaptive Simulations method (SAS) or the k-ω SST model. As a result, models are chosen, which produce grid-independent solution at an acceptable mesh size.

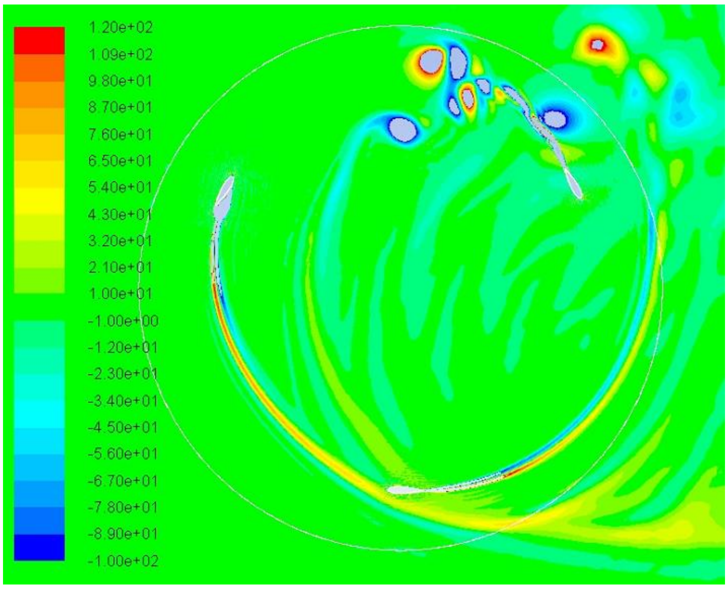

Figure 1: Vorticity contours around an H-Darrius rotor with Scale Adaptive Simulations method

In the second step of the analysis the characteristic curve of the rotor is evaluated at four different TSR values, and compared against the measurement. During the comparison 3D effects are neglected, but strut losses are compensated based on a semi-analytical model. Finally, conclusions are drawn and turbulence model is proposed for later studies.

REFERENCES

[1] European Commission, *ENERGY - Country Factsheets V.1.3*. 2012.

[2] European Commission - Directorate-General for Energy, *EU energy trends to 2030*. 2009.

[3] M.R. Castelli, A. Englaro and E. Benini, The Darrieus wind turbine: Proposal for a new performance prediction model based on CFD. *Energy*, Vol. **36**, pp. 4919–4934, 2011.

[4] T. Maître, E. Amet and C. Pellone, Modeling of the flow in a Darrieus water turbine: Wall grid refinement analysis and comparison with experiments. *Renewable Energy*, Vol. **51**, pp. 497–512, 2013.

[5] K.M. Almohamaddi, D.B. Ingham, L. Ma and M. Pourkashan, Computational fluid dynamics (CFD) mesh independency techniques for a straight blade vertical axis wind turbine. *Energy*, Vol. **58**, pp. 483–493, 2013.

[6] L. Daróczy, G. Janiga and D. Thévenin, Systematic analysis of the heat exchanger arrangement problem using multi-objective genetic optimization. Energy, in press, 2013. doi: http://dx.doi.org/10.1016/j.energy.2013.11.035