6th European Conference on Computational Mechanics (ECCM 6)
7th European Conference on Computational Fluid Dynamics (ECFD 7)
11 – 15 June 2018, Glasgow, UK

# Numerical modelling of Savonius wind turbine using RANS turbulence modelling approach

**T. Krysiński[1], Z. Buliński[2], A.J. Nowak [3]**

[1] Institute of Thermal Technology, Silesian University of Technology, Konarskiego 22, 44-100 Gliwice, Poland. tomasz.krysinski@polsl.pl, http://www.itc.polsl.pl/krysinski
[2] Institute of Thermal Technology, Silesian University of Technology, Konarskiego 22, 44-100 Gliwice, Poland. zbigniew.bulinski@polsl.pl, http://www.itc.polsl.pl/bulinski
[3] Institute of Thermal Technology, Silesian University of Technology, Konarskiego 22, 44-100 Gliwice, Poland. andrzej.j.nowak@polsl.pl, http://www.itc.polsl.pl/nowak

**Key Words:** *wind turbine, computational fluid dynamics (CFD), vertical axis wind turbine, turbulence.*

According to present world energy policy, a carbon dioxide reduction is a priority. This issue affects a strongly growing interest in the efficient use of renewable energy sources such as wind, sun or ocean tides. In this paper, comparison study of Reynolds Averaged Navier-Stokes type turbulence models performance in case of drag-based vertical axis wind turbine is presented.
The presented research covers numerical modelling of a typical Savonius type wind turbine. The Savonius wind turbine is vertical axis wind turbine proposed by Finnish engineer S.J. Savonius around 1920 [1]. The exact geometry of modelled turbine rotor was based on experimental rig presented in the paper of Kamoji et al. [2]. Geometries, numerical domain and all simulation parameters like boundary conditions, turbulence models, discretisation schemes where defined using commercial Computational Fluid Dynamics (CFD) software package *Ansys*. Transient calculations of the turbine rotor movement were carried out with use of the Sliding-Mesh approach. The Sliding Mesh method allows us to obtain an accurate time-dependent solution of interactions between rotating and stationary objects by resolving exactly rotation of the turbine rotor. The fluid flow around vertical axis wind turbine under operation is highly complicated mainly due to several phenomena that occur, like boundary layer separation, vortex shedding or dynamically changing of turbine blade curvature from the fluid point of view. Moreover, fluid flow around performing wind turbine rotor is highly turbulent. The turbulence that occurs in flow around wind turbine has a great impact on overall turbine efficiency and work conditions. For comparison and validation purposes, computations were performed using wide range of Reynolds Averaged Navier-Stokes turbulence models, starting from the simplest two-equation models and finishing on the most advanced Reynolds Stress Models with quadratic pressure-strain interaction. Influence of turbulence models on the flow field simulation and wind turbine performance is presented. Finally, obtained computational results were validated against experimental results presented in the literature.

## REFERENCES

[1] S.J. Savonius, The S-rotor and its applications, *Mechanical Engineering*, Vol. **53**, pp. 333-337, 1931.
[2] M.A. Kamoji, S.B. Kedarel, S.V. Prabhu, Experimental investigations on single stage, two stage and three stage conventional Savonius rotor, *International Journal of Energy Research*, Vol. **35**, pp.877-895, 2008