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

EFFECTS OF TURBULENCE MODELLING ON THE PREDICTIONS OF THE PRESSURE DISTRIBUTION AROUND THE WING OF A SMALL SCALE VERTICAL AXIS WIND TURBINE

Mohamed M. Elsakka[1,2], Derek B. Ingham[3], Lin Ma[4]*, and Mohamed Pourkashanian[5]

[1] Energy2050, University of Sheffield, Sheffield S10 2TN, UK, mmmel-sakka1@sheffield.ac.uk
[2] Faculty of Engineering, Port Said University, Port Said, Egypt
[3] Energy2050, University of Sheffield, Sheffield S10 2TN, UK, d.ingham@sheffield.ac.uk
[4] Energy2050, University of Sheffield, Sheffield S10 2TN, UK, lin.ma@sheffield.ac.uk
[5] Energy2050, University of Sheffield, Sheffield S10 2TN, UK, m.pourkashanian@sheffield.ac.uk

Key Words: *VAWT, CFD, Turbulence model, Moderate Reynolds Number.*

The aerodynamics of the flow around Vertical Axis Wind Turbines (VAWTs) has become a very important research topic. The combination of the blade rotational motion and the free stream wind leads to an oscillating Angle of Attack (AOA) around the turbine blade. The small-scale VAWTs typically operate at low and moderate Reynolds numbers and low Tip Speed Ratios (TSRs) and this leads to a higher amplitude in the oscillating AOA and complex aerodynamics that includes boundary layer flow transition, separation, and dynamic stall. This directly contributes to the pressure distribution around the blade and the flow turbulence plays an important role. The aim of this paper is to compare two different turbulence models in a 2D Computational Fluid Dynamics (CFD) prediction of the pressure distribution, and the overall turbine power generation of a small scale VAWT, and compare against the experimental data that is available in the literature. The turbine investigated is a small two-bladed VAWT operating at an average Reynolds number of approximately $2.5*10^5$. A commercial solver is used for the CFD calculations, while the average non-dimensional wall distance is maintained at about $y^+=1$ in order to resolve the details of the boundary layer. The SST transition and *k-ω* SST turbulence models are implemented and their results are compared to the experimental data taken at the mid-span of the turbine at different azimuthal locations. For the turbine investigated, the CFD results, based on the two turbulence models, in general show a good agreement with the experimental data and in particular in the upstream section of the turbine. The *k-ω* SST model gives reasonable predictions to the pressure distribution over the blade at almost all blade locations from low to high angles of attack. The agreement with experimental data is very good for the suction side of the blade. The SST transition model is able to capture the small laminar separation bubbles that occur near to the leading edge of the aerofoil and predict well the pressure distribution on the suction side when the boundary layer flow is attached. However, at high AOAs where stall occurs, the SST transition results appear to overestimate the flow separation, while the *k-ω* SST results fit better with the experimental data. In terms of the instantaneous power coefficients, both models can predict reasonably well the measurement data although the SST transition model appears to predict results which are slightly closer to the experimental data than the *k-ω* SST model when the blade is moving across the upwind side of the turbine where majority of the power is generated. It is expected that further investigations are needed on the prediction of flow at stalled flow conditions and the blade wake interactions in order to accurately predict the performance of a VAWT.