CFD simulations of the MEXICO rotor

The wake behind a wind turbine model is investigated using Computational Fluid Dynamics (CFD), and results are compared with measurements. The turbine investigated is the three-bladed test rotor (D = 4.5 m) used in the Model Experiments in Controlled Conditions (MEXICO) wind tunnel experiment. During the MEXICO experiment, particle image velocimetry measurements of the induction upstream and downstream of the rotor were performed for different operating conditions, giving a unique dataset to verify theoretical models and CFD models. The present paper first describes the efforts in reproducing the experimental results using the Reynold-Averaged Navier-Stokes method. Second, three-dimensional airfoil characteristics are extracted that allow simulations with simpler wake models. Copyright © 2011 John Wiley & Sons, Ltd.