NUMERICAL SIMULATIONS ON THE FLOW AROUND A ROTATING WIND TURBINE

XINRU WANG[1,2], XINLIANG TIAN[1,2], LONGFEI XIAO[1,2], GANG CHEN[1]

[1]*State Key Laboratory of Ocean Engineering, Shanghai Jiao Tong University, Shanghai, China, 200240*

[2]*Collaborative Innovation Center for Advanced Ship and Deep-Sea Exploration, Shanghai, China, 200240*

The vortex–body interaction problem, which characterizes the wake meandering around a quiescent offshore wind turbine, is investigated by numerical simulations. When the blade length of a horizontal-axis wind turbine is limited, blade rotating leads to the vortices, which distributing as the spiral line downstream of the wind wheel. Vortex wake vortices exists in two main areas: the edge of the wind turbine wake and the linear path of the rotation axis (Lee, K.H., 2005). All the computations are carried out using large-eddy simulation (LES) method, based on the open source software-OpenFOAM, and the features of the wake is investigated by the typical flow variables (pressure coefficient, thrust coefficient, torque coefficient, velocity, limiting streamlines of the suction side, vertical structure and FFT) of the instantaneous flow field, all geometrical details of the full-scale turbine structure taken into account. Furthermore, the flow field around a rotating wind turbine is compared with the quiescent one to make out the difference on the tip and hub vortices evolution. Analysis of the calculated instantaneous flow field around a quiescent wind turbine reveals a detailed flow fields with complex geometry, and the vortex in the wake of the turbine behaves quite diversely, where the vortex core is relatively coherent in the wake nearby the turbine blade. A transparent distinction of flow fields is predicted between a rotating wind turbine and a quiescent one, which can be reflected by a detailed streamlined diagram and the charts on some typical flow variables parameters. The results identify for a visual description of the flow field around the offshore wind turbine with micromesh, explore the influence of the blade rotating, and then, summarize the features of the flow fields under the various motion and load status, preparing for further research.