Flow Simulations of a Rotating Mid-Sized Rim Driven Wind Turbine

Bryan E. Kaiser¹, Andrew B. Porteous, Dr. Svetlana V. Poroseva³, Dr. Rob O. Hovsapian⁴

¹UNM Department of Mechanical Engineering, University of New Mexico, Albuquerque, NM 87131
³Idaho National Laboratory, Idaho Falls, ID, 83415

E-mail: poroseva@unm.edu

Introduction

Conventional horizontal axis wind turbines (HAWTs) require relatively high free stream wind velocities that limit the geographic areas suitable for wind energy harvesting. To overcome this limitation, small to mid-sized wind turbine designs capable of power generation at low wind speeds have started to received a renewed attention from industry and consumers. The Keuka rim-driven wind turbine (RDWT) (U.S. Patent 7399162) developed by Keuka Energy LLC is one of the new designs currently in production and testing. The objective of our study is to predict its performance using simulations for a range of wind speeds characteristic of states with a low average wind speed, such as, for example, Florida, New Mexico, and Texas. The Keuka RDWT is a drag driven wind turbine designed for wind energy extraction in locations of wind class three or below. The design is passive-stall-controlled for simplicity and lower capital expense. It features high solidity (16 blades) and is power rated at 15kW. The turbine diameter considered in the current study is approximately 7.5 m. The results of the current study will be used to investigate whether RDWT generates wakes less destructive than those of the conventional three-bladed HAWTs and thus, can be implemented in larger numbers in wind farms.

Numerical Simulations

Computations were conducted using commercial CFD software STAR CCM+. The complex structure of a flow around RDWT presents many challenges for conducting accurate numerical simulations. The presence of dynamic stall, turbine operation over a wide range of Reynolds numbers, and interaction with atmospheric turbulence are just a few of such challenges. Significant requirements for an appropriate computational grid resolution render Direct Numerical Simulations and Large Eddy Simulations computationally unfeasible for such simulations. Typically, Reynolds-Averaged Navier-Stokes (RANS) turbulence models are used for simulating a flow around and behind a wind turbine. A sensitivity analysis was conducted to determine models most suitable for the problem. It was found that the Menter’s SST version of k-ω model and the realizable k-ε turbulence model implemented in STAR CCM+ produce the most reliable results.

Structured and unstructured polyhedral computational grids were used in this study. The grids around the turbine are shown in Figure 1. The domain size was 10Dx10Dx10D (D is the turbine diameter). The mesh size was up to 3,500,000 cells for unstructured and 1,500,210 cells for structured grids. The turbine was located at 2D from the inlet plane. The uniform inlet velocity equal to the free stream wind speed was assigned as the inlet condition. Its value was varied in the range of inlet velocities from 1 ms⁻¹ to 12 ms⁻¹ corresponding to a range of Reynolds numbers (based on turbine diameter) from 480,000 to 5,700,000. Angular velocities of the turbine were specified to correspond to experimental values obtained for an RDWT test prototype.
Results

Figure 2 shows the mean velocity contours in the near wake of RDWT at the freestream velocity of 1 m/s and the power coefficient values obtained with the two turbulence models for the whole turbine at the free-stream wind velocity of 1, 3, 6, 9, and 12 m/s.

Sensitivity study was performed to determine how the size of the computational domain influences simulation results in the far wake behind the turbine. Figure 3 shows the velocity magnitude fields when the domain was increased in the vertical direction (15Dx10Dx10D), in the spanwise direction (10Dx15Dx10D), and in the streamwise direction (10Dx10Dx15D).

Conclusions

Preliminary results show that war-wake simulations are most sensitive to the computational domain size in the streamwise direction. Simulations conducted in the 10Dx10Dx10D domain are computationally the least expensive and reliably capture the near-wake flow structure.

Publications


Acknowledgements

This work was supported in part by the New Mexico Consortium and the Center for Advanced Power Systems (CAPS) at the Florida State University. The UNM CARC supported the research in a form of access to HPC, and CD-adapco provided Star-CCM+ for academic purposes.