CFD Simulations of Two Wind Turbines Operating In Line

Maokun Ye
Department of Ocean Engineering, Texas A&M University
College Station, Texas, USA

Hamn-Ching Chen
Zachry Department of Civil Engineering and Department of Ocean Engineering, Texas A&M University
College Station, Texas, USA

Arjen Koop
MARIN – Maritime Research Institute Netherlands
Wageningen, The Netherlands

ABSTRACT

Generally, the wake behind a wind turbine is characterized as a reduction in wind velocity and an increase in turbulence level compared to the free stream condition. In wind farms where wind turbines are grouped in arrays, under unfavorable conditions, downstream wind turbines will operate in the wakes of upstream turbines, and thus will harm the overall efficiency of wind farms. Accurately predicting the performance of downstream turbines and the interactions between multiple turbine wakes are crucial to the design of more efficient wind farms because it forms the cornerstone of wind farm layout optimization algorithms.

In the present study, we perform CFD simulations for the NTNU Blind Test 2 experiment in which two turbines were placed in a closed-loop wind tunnel and operating in line. The Reynolds-Averaged Navier Stokes (RANS) equations with the k-ω SST turbulence model are adopted in the simulations. For each of the two wind turbines, geometries including the blades, hub, nacelle, and tower are fully resolved. The Moving-Grid-Formulation (MVG) approach with a sliding interface technique is leveraged to handle the relative motion between the rotating and stationary portions of the wind turbines. In the simulations, the values of tip-speed ratio (TSR) for the upstream and downstream turbines are 4 and 6, respectively. The CFD-predicted thrust and power coefficients are obtained under an inlet velocity of 10 m/s and are compared against the experiment results. In addition, the wake structures of the two wind turbines are also visualized and discussed.

KEY WORDS: CFD; Wind Turbine; Wake Interactions.

INTRODUCTION

The wake generated by a horizontal-axis wind turbine (HAWT) is characterized by a decrease in wind velocity and an increase in the turbulence level compared to the free stream condition. Grouped in clusters in modern onshore and offshore wind farms, wind turbines will unavoidably be operating in the wake of upstream turbines. Therefore, the power generation efficiency of the downstream wind turbines in a wind farm will decline, and as a result, the overall power generated by a wind farm will be affected (Vanderwende et al., 2016). Researchers estimated that the overall power loss of a large wind farm is 10% - 25% (Wu and Porté-Agel, 2015). To fulfill the potential of wind power as a major source of clean energy in the future, higher-efficiency wind turbines and wind farms need to be designed. Therefore, as the premise of the wind farm layout optimization algorithms, accurate prediction of the wind turbine wakes and wake interactions is of great importance.

Computational Fluid Dynamics (CFD) tools with varying fidelity have been successfully applied to wind energy applications in the past two decades. However, due to the multiscale nature of the flow in a wind farm and the limitation of computational resources, it is still challenging to resolve the turbine boundary layers and the turbine wake simultaneously. One commonly used approach is to leverage the blade element method (BEM) based methods, i.e. the actuator disc (AD) (Sørensen and Myken, 1992; Sørensen et al., 2020) and actuator line (AL) (Sørensen and Shen, 2002; Shen et al., 2005) methods, to model the rotating turbine as forces in the fluid domain. Those methods are then coupled with the Reynolds-Averaged Navier–Stokes (RANS) or Large Eddy Simulation (LES) solvers to calculate the fluid field around wind turbines. This approach has been extensively used in wind turbine wake simulations (Trolldborg, 2009; Mehta et al., 2014; Breton et al., 2014) and wind farm simulations (Xie and Archer, 2017; Cortina, 2020) where multiple wind turbines are present.

However, the credibility of the BEM-based methods may be questioned in real-world applications because they usually assume that the airflow on the blade surface is two-dimensional (Duque et al., 1999; Tran and Kim, 2016). In particular, the calculations of the airflow around a floating offshore wind turbine (FOWT), the flow near the FOWT may be highly three-dimensional (Liu et al., 2017). Under certain conditions, the turbine will even operate in its own wake (Tran and Kim, 2016).