Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 94 (2016) 269 – 277

Energy Procedia

13th Deep Sea Offshore Wind R&D Conference, EERA DeepWind'2016, 20-22 January 2016, Trondheim, Norway

The actuator disc concept in PHOENICS

N. Simisiroglou[a,b,*], M. Karatsioris[b], K. Nilsson[b], S.P. Breton[b], S. Ivanell[b]

[a] *WindSim AS, Fjordgaten 15, N-3125 Tønsberg , Norway*

[b] *Department of Earth Sciences, Wind Energy Section, Uppsala University Campus Gotland, Cramérgatan 3, 621 67 Visby, Sweden*

Abstract

This study presents two models to simulate a wind turbine. This is done by employing the 1D momentum actuator disc theory in PHOENICS, a general purpose computational fluid dynamics software. To test the general applicability of these models, single wind turbine simulations are conducted using eight different wind turbine models from two manufacturers. The simulations are performed by imposing sheared inflow with hub height wind speeds ranging from 3 m/s up to 25 m/s. A range of computational parameters are investigated, including the resolution of the domain, the thickness of the actuator disc and the iterative convergence criteria. To investigate the wake development produced by these methods, a comparison study is performed with the more complex large-eddy simulation software EllipSys3D using an actuator disc approach for validation purposes. The resulting wind turbine thrust and power outputs from PHOENICS are compared with the experimental power curves and thrust values provided by the manufacturers for each wind turbine. The results show that actuator disc methods are able to provide a reasonable estimation of the conventional wind turbine power and thrust output with low computational effort. Moreover, the results from the preliminary comparison of the wake produced from these two rotor models compare well with the wake produced by the actuator disc in EllipSys3D.

Peer-review under responsibility of SINTEF Energi AS

Keywords: Wind energy ; Computational Fluid Dynamics (CFD) ; Reynolds averaged Navier-Stokes (RANS) ; Actuator Disc (ACD) ; Large-eddy simulation (LES)

1. Introduction

The ability to model the flow within a modern wind farm is an area of growing importance as it has been shown that wind turbine wakes may account for a decrease on average of 10% to 20% of the annual power production of a modern wind farm [1]. In recent years as computational resources have vastly increased, computational fluid dynamic (CFD) methods that previously were numerically prohibitive are now accessible not only for academic research but for industrial purposes as well. To utilize CFD methods to estimate the flow within a wind farm one needs at first to describe the flow using the appropriate Euler or Navier Stokes equations and then to represent the effects of the wind turbine on the flow field. In this study we will focus primarily on the latter, and will present initial results

* Corresponding author. Tel.: +46-072-903-8821.
E-mail address: nikolaos.simisiroglou@geo.uu.se

Peer-review under responsibility of SINTEF Energi AS
doi:10.1016/j.egypro.2016.09.182

on the wake development within the flow by means of a comparison with a widely used research code. Frequently used methods to represent the effects of a wind turbine on the flow field are the actuator disc (ACD), actuator line (ACL) and the actuator surface model. Within these models the presence of the wind turbine is represented in the momentum equation by means of an additional force term. The ACD model which enjoys a substantial advantage in its relative simplicity was first formulated by Froude in 1889 [2] for naval applications. Numerical computations using the ACD concept for wind turbine applications have later been performed by researchers such as Sørensen and Myken [3], for the Nibe wind turbine and Mikkelsen [4] who further investigated a generalized actuator disc based on the blade element momentum method. Continuing with models of increasing complexity, Sørensen and Shen in 2002 [5] proposed the ACL concept where the forces are distributed along lines that represent the blades of an actual wind turbine. Subsequent development along these lines was introduced by Shen and Sørensen [6] with the actuator surface. The distribution of the forces with the actuator surface model is not only along a line as in the ACL but in the chordwise direction as well, representing thus the blade more accurately. A comprehensive review of analytical wake modelling methods and CFD methods is provided by Vermeer *et al.*[7] and Sanderse *et al.*[8].

The tendency towards ever more complicated models to represent a wind turbine in a numerical domain, although usually improving the accuracy of the models, also increases the required computational resources, hence making these models currently unattractive for industrial application. The aim of this study is to bridge the gap between academic research and industrial application. Accordingly, the general applicability and accuracy of two novel implementations of the ACD model in the commercial CFD software PHOENICS [9] will be studied, where PHOENICS is the core code used within the commercial wind assessment software WindSim [10]. These models are developed with the goal to provide accurate estimations of conventional wind turbine power and thrust outputs, as well as to simulate the effect from wind turbine rotors on the flow, with low computational effort. To this end, single wind turbine simulations are conducted using eight different wind turbine models from two manufacturers, five Enercon and three Siemens wind turbines. The impact of a range of computational parameters on the simulation output is investigated. The results of the simulations are then compared with the experimental power curves and thrust values provided by the manufacturers for each wind turbine. In addition a preliminary comparison study is performed with the ACD in Ellipsys3D [11–13] to test the wake development predicted by using these models. Although in the present study only single wind turbine simulations are conducted, these models are developed with the intention to be applicable to arrays of wind turbines i.e. wind farm simulations.

The paper unfolds as follows; section 2 presents the theoretical background and computational settings of the study. Section 3 introduces the performed sensitivity study. Section 4 includes the results and discussion. Lastly, in section 5 the main conclusions of this study are presented.

2. Actuator disc and numerical model

2.1. Actuator disc theory

To model the effect of the wind turbine's rotor on the incoming flow in a computational simulation, the ACD concept based on the 1D momentum theory is employed. In this set up, the wind turbine blade geometry is approximated by a disc and the blade forces are smeared over the surface of the disc. In the literature numerous methods are proposed for estimating the magnitude and distribution of the forces applied to the disc, leading thus to different ACD implementations. In this study the thrust forces are imposed by two models that are described below. In a pre-processing step a modified thrust coefficient curve is created including the thrust coefficient values given by the manufacturer and the wind speed at the disc. This modified thrust coefficient curve is generated by replacing the undisturbed wind speed values from the manufacturer's thrust coefficient curve with the wind speed values at the disc. This transformation is possible by the following steps. Using the definition of the thrust coefficient $C_T \equiv 4\alpha(1-\alpha)$ and solving for the axial induction factor yields

$$\alpha = \frac{1}{2}\left(1 - \sqrt{1 - C_T}\right). \tag{1}$$

Continuing, the axial induction factor is by definition equal to

$$\alpha \equiv \frac{U_\infty - U_1}{U_\infty}. \tag{2}$$