Research outputs:

**Numerical investigations into the idealized diurnal cycle of atmospheric boundary layer and its impact on wind turbine's power performance**

The power generated by a wind turbine largely depends on the properties of wind resource. In this work, wind characteristics in different regimes occurring throughout the idealized diurnal cycle and its impact on wind turbine's power performance are investigated systematically by means of large-eddy simulation (LES), and blade element momentum method (BEM), respectively. Through a precursor simulation of the atmospheric boundary layer (ABL) over a homogenous surface throughout a day, it is found that the resulting shapes of wind profiles (including wind speed, wind direction and turbulence level) vary significantly at different time periods, induced by distinct stabilities of the atmosphere. The simulated wind field data are then applied to a NREL 5 MW wind turbine for its power evaluation. Due to variabilities in wind shear and turbulence, the equivalent (disk-averaged) wind speed is introduced for power prediction. It is found that the magnitude and fluctuation of turbine's diurnal power are closely related to the atmospheric stability. In general, the average power production is higher under convective conditions during the day than under stable conditions at night, with a difference approaching 24.4%. This indicates that wind energy resource assessment will close to reality and benefit from increased accuracy if atmospheric stability impacts are considered for turbine's power predictions.
Advanced flow and noise simulation method for wind farm assessment in complex terrain

A new wind farm during its development phase requires a technical evaluation of its annual power prediction and noise issue in the area in and around the wind farm. In this paper, for the first time, an advanced numerical method for wind farm assessment is developed, which consists of a Reynolds averaged Navier-Stokes - Actuator Disc model for the flow modelling, a semi-engineering model for the noise source modelling, and a parabolic wave equation model for the sound propagation modelling. The developed method can evaluate both annual energy production of wind farm and noise emission at receivers nearby. The wind farm considered in this study is located in a typical mountainous area. The energy production of the wind turbines is simulated and compared with measured data. The flow simulations over the complex terrain are performed using the in-house developed Navier-Stokes solver. A wind farm noise map is created by solving the parabolic wave equation. The obtained flow results are the inputs to the parabolic wave equation solver for sound propagation. The numerical computations are performed on a standard high-performance computer cluster. The developed numerical method provides a reliable assessment method for wind farm about its energy efficiency and noise features.

Special Issue on Wind Turbine Aerodynamics

In order to reach the goal of 100% renewable energy in energy systems, wind energy, as a pioneer of renewable energy, is developing very quickly all over the world. To reduce the levelized cost of energy (LCOE), the size of a single wind turbine has been increased to 12 MW nowadays and it will increase further in the near future. Big wind turbines and their associated wind farms have many advantages but also challenges in aerodynamics, aero-elasticity and aero acoustics. The typical effects are mainly related to the increase in Reynolds number and blade flexibility. This Special Issue collects some important works addressing the aerodynamic challenges appearing in such development. Aerodynamics of wind turbines is a classic concept and is the key for wind energy development as all other parts rely on the accuracy of its aerodynamic models. There are numerous books and articles dealing with wind turbine aerodynamic problems and models. As a good example, the wind energy handbook by Burton et al. [1] gives an overview of wind turbine aerodynamics and its related problems. There are also several special issues on wind turbine aerodynamics. This author edited a special issue on aerodynamics of offshore wind energy systems and wakes [2] in 2014, which collected state-of-the-art research articles on the development of offshore wind energy.
Wind turbine noise propagation in flat terrain for wind farm layout optimization frameworks
This paper describes the wind turbine noise propagation in flat terrain for use in wind farm layout optimization frameworks. Large-eddy simulations of a single wind turbine in flat terrain at varying wind speeds, shear and turbulence levels are performed. The wind turbine is modeled using an actuator line approach, while the wind turbine noise propagation is computed using the Technical University of Denmark's WindSTAR-Pro (Wind turbine Simulation Tool for AeRodynamic noise Propagation). The wind turbine noise propagation is computed in a quasi-three-dimensional manner by using a two-dimensional (2D) parabolic equation (PE) model at numerous 2D-planes around the wind turbine. In this study, the 2D, wide-angle, Crank-Nicholson PE model is used. The relative sound pressure level obtained from WindSTAR-Pro around the wind turbine is computed for varying wind speeds, shear and turbulence levels. Wind flow velocity and relative sound pressure levels for selected wind conditions are shown. The end goal is to create a noise propagation database that can be used as a lookup table in wind farm layout optimization frameworks to mitigate the noise impact in the development of new wind farms.

An optimization framework for wind farm design in complex terrain
Designing wind farms in complex terrain is an important task, especially for countries with a large portion of complex terrain territory. To tackle this task, an optimization framework is developed in this study, which combines the solution from a wind resource assessment tool, an engineering wake model adapted for complex terrain, and an advanced wind farm layout optimization algorithm. Various realistic constraints are modelled and considered, such as the inclusive and exclusive boundaries, minimal distances between turbines, and specific requirements on wind resource and terrain conditions. The default objective function in this framework is the total net annual energy production (AEP) of the wind farm, and the Random Search algorithm is employed to solve the optimization problem. A new algorithm called Heuristic Fill is also developed in this study to find good initial layouts for optimizing wind farms in complex terrain. The ability of the framework is demonstrated in a case study based on a real wind farm with 25 turbines in complex terrain. Results show that the framework can find a better design, with 2.70% higher net AEP than the original design, while keeping the occupied area and minimal distance between turbines at the same level. Comparison with two popular algorithms (Particle Swarm Optimization and Genetic Algorithm) also shows the superiority of the Random Search algorithm.
Aero-structural optimization of wind turbine blades using a reduced set of design load cases including turbulence

Modern wind turbine aero-structural blade design codes generally use a smaller fraction of the full design load base (DLB) or neglect turbulent inflow as defined by the International Electrotechnical Commission standards. The current article describes an automated blade design optimization method based on surrogate modeling that includes a very large number of design load cases (DLCs) including turbulence. In the present work, 325 DLCs representative of the full DLB are selected based on the message-passing-interface (MPI) limitations in Matlab. Other methods are currently being investigated, e.g., a Python MPI implementation, to overcome the limitations in Matlab MPI and ultimately achieve a full DLB optimization framework. The reduced DLB and the annual energy production are computed using the state-of-the-art aero-servo-elastic tool HAWC2. Furthermore, some of the interior dimensions of the blade structure are optimized using the finite-element based cross-sectional analysis tool BECAS. The optimization framework is applied to redesign the NREL 5 MW wind turbine blade to obtain improvements in rotor performance and blade weight.
Analysis of winglets and sweep on wind turbine blades using a lifting line vortex particle method in complex inflow conditions: Paper

An in-house aero-elastic vortex code, called MIRAS, is used to investigate the aerodynamic performance of winglets and sweep on horizontal-axis wind turbine (HAWT) blades in simple and complex inflow conditions. Previous studies using vortex codes applied to study winglets and blade sweep on HAWTs have typically not considered complex inflow conditions such as turbulent wind and shear. The reasons may include the absence of modeling capability, the computational cost associated with simulating long turbulent time series, and/or the computational cost associated with resolving the blade tips to a very fine level. A preliminary study is performed here, where the MIRAS code is applied on the NREL 5MW wind turbine with an arbitrary winglet shape and blade sweep. Results indicate that wind turbine blades with sweep or winglets might be better in performance compared to their straight blade counterparts.

An integrated numerical method for wind turbine flow simulation, sound generation and propagation

The scope of the paper is to present an efficient numerical method that predicts: (a) wind turbine aerodynamic loads and power; (b) wind turbine noise source; (c) long distance wind turbine noise source propagation. The numerical methods involved in this study are a combination of Computational Fluid Dynamics (CFD) and wind turbine aeroacoustic methods. The results from the CFD simulation provide necessary information of wind turbine power and thrust etc. The 2D Actuator Disc (AD) theory is applied for such a purpose. The computational efficiency becomes very high while using a steady 2D CFD approach. The flow geometry at each blade element is required for wind turbine noise source calculations. The predicted wind turbine noise source is the starting field for long distance noise propagation model which is based on solving the Parabolic Equations (PE) in the frequency domain. Results showed that the integrated wind turbine flow-aerodynamic prediction method is capable of calculating wind turbine aerodynamic, aerodynamic noise source and long range sound propagation.
Assessment of inflow boundary conditions for RANS simulations of neutral ABL and wind turbine wake flow

It is known that wind turbines actually operate in the lower part of the atmospheric boundary layer (ABL), the modelling of this ABL flow is an important precondition for the simulations of wind turbine wakes. So, the capabilities of various inlet boundary conditions with related modelling methodologies in constructing equilibrium ABL are assessed firstly through cases of ABL flows over uniformly rough terrain, to ascertain that there are no substantial changes in the prescribed profiles throughout the whole computational domain. In this process, six popular turbulence inflow profiles, including four uniform and two non-uniform ones, are considered and investigated. Then, sensitivity studies on inflow profiles for predicting wind turbine wake development are carried out. Through comparing with the Sexbierum field experimental data, in terms of wake velocity and turbulence intensity along the cross-wind direction at several downstream positions, this study finds out that the shape and magnitude of wake velocity and wake turbulence profiles are significantly affected by different inflow profiles. Possible reasons for this sensibility are discussed, and accordingly, some suggestions are given to improve the accuracy of wind turbine wake simulation.

Atmospheric stability and topography effects on wind turbine performance and wake properties in complex terrain

This paper evaluates the influence of atmospheric stability and topography on wind turbine performance and wake properties in complex terrain. To assess atmospheric stability effects on wind turbine performance, an equivalent wind speed calculated with the power output and the manufacture power curve is proposed and calibrated with the mast hub-height wind speed. After estimating the thrust coefficient and turbulence dissipation, this paper examines wind turbine performance curves and wake profiles segregated by atmospheric stability. Results show that the equivalent wind speed at a given mast wind speed can increase by 2% under stable conditions and decrease by 5% under unstable conditions as
compared with that under neutral conditions, yielding about 16% reductions of power output and thrust coefficient from stable conditions to unstable conditions. Due to the lower thrust coefficient and the enhanced turbulence, the wind turbine wakes are found to recover faster under unstable conditions than under other stability conditions. Differences in wind turbine performance and asymmetric wake profiles due to topographic effects are also observed. Results suggest that atmospheric stability and topography have significant influences on wind turbine performance and wake properties. Considering effects of atmospheric stability and topography will benefit the wind resource assessment in complex terrain.

**General information**
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Hohai University
Contributors: Han, X., Liu, D., Xu, C., Shen, W. Z.
Pages: 640-651
Publication date: 2018
Peer-reviewed: Yes

**Publication information**
Journal: Renewable Energy
Volume: 126
ISSN (Print): 0960-1481
Ratings:
- BFI (2018): BFI-level 1
- Scopus rating (2018): CiteScore 6.19 SJR 1.889 SNIP 2.075
- Web of Science (2018): Impact factor 5.439
Keywords: Atmospheric stability, Complex terrain, Thrust measurements, Wind turbine performance, Wind turbine wakes
DOIs:
- 10.1016/j.renene.2018.03.048
Source: FindIt
Source-ID: 2397699557
Research output: Contribution to journal › Journal article – Annual report year: 2018 › Research › peer-review

---

This article describes Computational Fluid Dynamics (CFD) simulations of flows in a wind farm in complex terrain in Shaanxi, China and the comparisons of the computational results with utility scale field measurements. The CFD simulations performed in the study are using either a Reynolds-Averaged Navier–Stokes (RANS) or Large-Eddy Simulation (LES) solver. The RANS method together with an Actuator Disc (AD) approach is employed to predict the performance of the 25 wind turbines in the farm, while the LES and Actuator Line (AL) technique is used to obtain a detailed description of the flow field around a specific wind turbine #14 near two met masts. The AD-RANS simulation results are compared with the mean values of power obtained from field measurements. Furthermore, the AL-LES results are compared with the mean values of power, rotor speed, and wind speed measured from the wind turbine and its nearby two masts. Results from the simulations indicate that both AD-RANS and AL-LES methods can reasonably predict the performance of the wind farm and wind turbine #14, respectively, in complex terrain in Shaanxi. The mean percent difference obtained for power in the AD-RANS simulations was approximately 20%. Percent differences obtained for power and rotor RPM in the AL-LES varied between 0.08% and 11.6%. The mean percent differences in the AL-LES for power and rotor RPM are approximately 7% and 1%, respectively.

**General information**
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Aerodynamic design, Yangzhou University
Number of pages: 21
Publication date: 2018
Peer-reviewed: Yes

**Publication information**
Journal: Applied Sciences
Volume: 8
Issue number: 5
Article number: 788
Ratings:
- BFI (2018): BFI-level 1
- Scopus rating (2018): CiteScore 2.52
- Web of Science (2018): Impact factor 2.217
Development of a CFD-Based Wind Turbine Rotor Optimization Tool in Considering Wake Effects

In the present study, a computational fluid dynamic (CFD)-based blade optimization algorithm is introduced for designing single or multiple wind turbine rotors. It is shown that the CFD methods provide more detailed aerodynamics features during the design process. Because high computational cost limits the conventional CFD applications in particular for rotor optimization purposes, in the current paper, a CFD-based 2D Actuator Disc (AD) model is used to represent turbulent flows over wind turbine rotors. With the ideal case of axisymmetric flows, the simulation time is significantly reduced with the 2D method. The design variables are the shape parameters comprising the chord, twist, and relative thickness of the wind turbine rotor blades as well as the rotational speed. Due to the wake effects, the optimized blade shapes are different for the upstream and downstream turbines. The comparative aerodynamic performance is analyzed between the original and optimized reference wind turbine rotor. The results show that the present numerical optimization algorithm for multiple turbines is efficient and more advanced than conventional methods. The current method achieves the same accuracy as 3D CFD simulations, and the computational efficiency is not significantly higher than the Blade Element Momentum (BEM) theory. The paper shows that CFD for rotor design is possible using a high-performance single personal computer with multiple cores.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Yangzhou University, Nanjing University of Aeronautics and Astronautics
Contributors: Cao, J., Zhu, W., Shen, W., Sørensen, J. N., Wang, T.
Publication date: 2018
Peer-reviewed: Yes

Publication information
Journal: Applied Sciences
Volume: 8
Issue number: 7
Article number: 1056
Ratings:
BFI (2018): BFI-level 1
Scopus rating (2018): CiteScore 2.52
Web of Science (2018): Impact factor 2.217
Web of Science (2018): Indexed yes
Original language: English
Keywords: wind turbine blade optimization, computational fluid dynamic, actuator disc, wake effect, Non-dominated Sorting Genetic Algorithm (NSGA-II)
Electronic versions:
applsci_08_01056_1_.pdf
DOIs:
10.3390/app8071056
Source: FindIt
Source-ID: 2437579188
Research output: Contribution to journal › Journal article – Annual report year: 2018 › Research › peer-review

Development of an efficient numerical method for wind turbine flow, sound generation, and propagation under multi-wake conditions

The propagation of aerodynamic noise from multi-wind turbines is studied. An efficient hybrid method is developed to jointly predict the aerodynamic and aeroacoustics performances of wind turbines, such as blade loading, rotor power, rotor aerodynamic noise sources, and propagation of noise. This numerical method combined the simulations of wind turbine flow, noise source and its propagation which is solved for long propagation path and under complex flow environment. The results from computational fluid dynamics (CFD) calculations not only provide wind turbine power and thrust information, but also provide detailed wake flow. The wake flow is computed with a 2D Actuator Disc (AD) method that is based on the axisymmetric flow assumption. The relative inflow velocity and angle of attack (AOA) of each blade element form input data to the noise source model. The noise source is also the initial condition for the wave equation that solves long distance noise propagation in frequency domain. Simulations were conducted under different atmospheric conditions which showed that wake flow is an important part that has to be included in wind turbine noise propagation.
Evaluation of different methods for determining the angle of attack on wind turbine blades with CFD results under axial inflow conditions

This work presents an investigation on different methods for the calculation of the angle of attack and the underlying induced velocity on wind turbine blades using data obtained from three-dimensional Computational Fluid Dynamics (CFD). Several methods are examined and their advantages, as well as shortcomings, are presented. The investigations are performed for two 10 MW reference wind turbines under axial inflow conditions, namely the turbines designed in the EU AVATAR and INNWIND.EU projects. The results show that the evaluated methods are in good agreement with each other at the mid-span, though some deviations are observed at the root and tip regions of the blades. This indicates that CFD results can be used for the calibration of induction modeling for Blade Element Momentum (BEM) tools. Moreover, using any of the proposed methods, it is possible to obtain airfoil characteristics for lift and drag coefficients as a function of the angle of attack.
Evaluation of different methods of determining the angle of attack on wind turbine blades under yawed inflow conditions

As part of the AVATAR and Mexnext projects, this study compares several methods used to derive lifting line variables from CFD simulations of the MEXICO rotor in yawed inflow. The results from six partners within the AVATAR/Mexnext consortium using five different methods of extraction were compared. Overall comparison of the induced velocities at the mid and tip parts of blade shows fairly good agreement between the tested methods, where the derived angle of attack differs within 1°, within the linear range this accounts to < 10% uncertainty on the aerodynamic forces. The presented comparison shows inadequate agreement between the methods for application towards the root.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, ECN, University of Oldenburg, University of Malta, Delft University of Technology, University of Stuttgart
Number of pages: 10
Publication date: 2018
Peer-reviewed: Yes

Evaluation of the power-law wind-speed extrapolation method with atmospheric stability classification methods for flows over different terrain types

The atmospheric stability and ground topography play an important role in shaping wind-speed profiles. However, the commonly used power-law wind-speed extrapolation method is usually applied, ignoring atmospheric stability effects. In the present work, a new power-law wind-speed extrapolation method based on atmospheric stability classification is proposed and evaluated for flows over different types of terrain. The method uses the wind shear exponent estimated in different stability conditions rather than its average value in all stability conditions. Four stability classification methods, namely the Richardson Gradient (RG) method, the Wind Direction Standard Deviation (WDSD) method, the Wind Speed Ratio (WSR) method and the Monin-Obukhov (MO) method are applied in the wind speed extrapolation method for three different types of terrain. Applicability is analyzed by comparing the errors between the measured data and the calculated results at the hub height. It is indicated that the WSR classification method is effective for all the terrains investigated while the WDSD method is more suitable in plain areas. Moreover, the RG and MO methods perform better in complex terrains than the other methods, if two-level temperature data are available.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Hohai University, Naval Aeronautical University
LES simulation and experimental validation of the unsteady aerodynamics of blunt wind turbine airfoils

In order to investigate the unsteady performance of blunt wind turbine airfoils caused by boundary layer separation and wake eddies, this paper studies the aerodynamic performance by large eddy simulation (LES) and wind tunnel experiment at a Reynolds number of 2.62×10^5. The blunt airfoils are obtained by enlarging the trailing edge of the DU 91-W2-250 airfoil to 6% and 10% chords symmetrically on both pressure and suction sides of the airfoil. The simulation was carried out with the incompressible finite-volume Navier-Stokes code EllipSys3D; and, the experiment was done in a wind tunnel with a cross-section of 0.5m × 0.5m by measuring the surface pressure and wake velocities using ESP-64HD pressure scanner and TSI hot-wire anemometer. The unsteady wake was captured by hot-wire in the wind tunnel, and LES with EllipSys3D. Both experiment and LES show that the spectrum of aerodynamic forces has a broadband nature which is in coincidence with the wake eddies, implying that the unsteady Kármán vortex sheet is the driving mechanism of the force fluctuation. Moreover, the trailing edge size affects the separation bubbles and transition process in the boundary layer. It shows that the boundary layer near the leading edge is unstable in the spanwise direction, which is characterized by low frequency waves.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Liaoning University of Petroleum and Chemical Technology, Chinese Academy of Sciences
Contributors: Wang, G., Zhang, L., Shen, W. Z.
Number of pages: 13
Pages: 911-923
Publication date: 2018
Peer-reviewed: Yes

Publication information
Journal: Energy
Volume: 158
ISSN (Print): 0360-5442
Ratings:
BFI (2018): BFI-level 2
Scopus rating (2018): CiteScore 6.2 SJR 2.048 SNIP 1.822
Web of Science (2018): Impact factor 5.537
Web of Science (2018): Indexed yes
Original language: English
Keywords: Blunt airfoil, Wake, Unsteady, Transition, Large eddy simulation, Experiment
DOIs:
10.1016/j.energy.2018.06.093
Source: FindIt
Source-ID: 2435582657

LES simulation and experimental validation of the unsteady aerodynamics of blunt wind turbine airfoils

In order to investigate the unsteady performance of blunt wind turbine airfoils caused by boundary layer separation and wake eddies, this paper studies the aerodynamic performance by large eddy simulation (LES) and wind tunnel experiment at a Reynolds number of 2.62×10^5. The blunt airfoils are obtained by enlarging the trailing edge of the DU 91-W2-250 airfoil to 6% and 10% chords symmetrically on both pressure and suction sides of the airfoil. The simulation was carried out with the incompressible finite-volume Navier-Stokes code EllipSys3D; and, the experiment was done in a wind tunnel with a cross-section of 0.5m × 0.5m by measuring the surface pressure and wake velocities using ESP-64HD pressure scanner and TSI hot-wire anemometer. The unsteady wake was captured by hot-wire in the wind tunnel, and LES with EllipSys3D. Both experiment and LES show that the spectrum of aerodynamic forces has a broadband nature which is in coincidence with the wake eddies, implying that the unsteady Kármán vortex sheet is the driving mechanism of the force fluctuation. Moreover, the trailing edge size affects the separation bubbles and transition process in the boundary layer. It shows that the boundary layer near the leading edge is unstable in the spanwise direction, which is characterized by low frequency waves.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Liaoning University of Petroleum and Chemical Technology, Chinese Academy of Sciences
Contributors: Wang, G., Zhang, L., Shen, W. Z.
Number of pages: 13
Pages: 911-923
Publication date: 2018
Peer-reviewed: Yes

Publication information
Journal: Energy
Volume: 158
ISSN (Print): 0360-5442
Ratings:
BFI (2018): BFI-level 2
Scopus rating (2018): CiteScore 6.2 SJR 2.048 SNIP 1.822
Web of Science (2018): Impact factor 5.537
Web of Science (2018): Indexed yes
Original language: English
Keywords: Blunt airfoil, Wake, Unsteady, Transition, Large eddy simulation, Experiment
DOIs:
10.1016/j.energy.2018.06.093
Source: FindIt
Source-ID: 2435582657
Numerical Fluid-Structure Interaction Study on the NREL 5MW HAWT

The development of reliable Fluid-Structure Interaction (FSI) simulation tools and models for the wind turbines is a critical step in the design procedure towards achieving optimized large wind turbine structures. Such approach will mitigate the aeroelastic instabilities like: torsional flutter, stall flutter and edgewise instability that introduce extra stresses to the turbine structure leading to reduced life time and substantial failures. In this study, FSI simulations were held using the commercial package Ansys v18.2 solvers as a preliminary step towards our on-going development of a reliable Open-Source solver. These simulations were applied to the full-scale rotor blades of the NREL 5MW reference horizontal axis wind turbine. The aerodynamic loads and structural responses computations were carried out using a steady-state FSI analysis. The computations were run on the Kyushu University multi-core Linux cluster using the public domain openMPI implementation of the standard message passing interface (MPI). Finally, the results were validated against the Technical University of Denmark’s (DTU) MIRAS aeroelastic code results as well as the widely used FLEX5-Q3UIC and FAST codes in different cases showing reasonable agreement.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Kyushu University
Contributors: Halawa, A. M., Sessarego, M., Shen, W. Z., Yoshida, S.
Number of pages: 9
Publication date: 2018
Peer-reviewed: Yes

Publication information
Journal: Journal of Physics: Conference Series
Volume: 1037
Issue number: 2
Article number: 022026
ISSN (Print): 1742-6596
Ratings:
BFI (2018): BFI-level 1
Scopus rating (2018): CiteScore 0.51 SJR 0.221 SNIP 0.454
Original language: English
Electronic versions:
Halawa_2018_J._Phys._3A_Conf._Ser._1037_022026_1_.pdf
DOIs:
10.1088/1742-6596/1037/2/022026

Bibliographical note
Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Source: FindIt
Source-ID: 2435911208

Parametric Model of Vortex Generators of Wind Turbine Considering Inter-effect of Winglets

The problem of large amounts of girds caused by solid boundary of vortex generators (VGs) in computation fluid dynamic modeling can be resolved by employing parametric model. Considering inter-effect between the wings, based on the theories of lift-line and delta-wing, a parametric model of an array of VGs which are composed of anti-rotating delta wings, is proposed to simulate the effect of shedding vortex on the aerodynamic of lift blade. The maximum circulation is computed from the model and added into the equation of N-S as a source term. The circulation of blade with VGs are calculated employing the VG-arrays model and the single-wing model without considering the inter-effect, the results are compared with the experimental results from literature, the comparison shows: when $0^\circ \leq \alpha \leq 18^\circ$, the maximum error of the array model is only 4.8%; when $\alpha=24^\circ$, the error reaches 13.2% which is still much lower than the result of the single-wing model (49.4%). The performance of DU91-W2-250 blade section with five pairs of VGs is modeled; the comparison with the experimental results show the array type model has higher agreement with the results of experiment. The simulation results of fluid field employing two models, real structure model and parametric model of array VGs, have high agreement.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Hohai University, Nanjing University of Aeronautics and Astronautics
Contributors: Zhao, Z., Yan, C., Wang, T., Shen, W., Zheng, Y., Xu, B.
Pages: 201-208
Variability of wind turbine noise over a diurnal cycle
The diurnal variation of atmospheric conditions over land has a significant effect on the wind and temperature distributions which greatly influence the generation and propagation of wind turbine aerodynamic sound. In this paper, a fully consistent unsteady approach is used to study wind turbine noise such that large eddy simulation with a rotational actuator disk wind turbine model is used to model the wind and temperature around a mega-watt scale wind turbine over a diurnal cycle, and time dependent flow and temperature fields are used as input to the coupled wind turbine noise generation-propagation model. Computations are carried out for four different 10 min datasets selected at certain periods of a day for a same hub height wind speed. It is observed that the time dependent as well as the time averaged sound pressure levels in near field do not show large variations during the day. However, as we move away from the turbine, the propagation effects take over and downwind of the turbine the night time levels exceed the day time levels (at 3600 m the averaged difference reaches 6.5 dBA).

Variable pitch approach for performance improving of straight-bladed VAWT at rated tip speed ratio
This paper presents a new variable pitch (VP) approach to increase the peak power coefficient of the straight-bladed vertical-axis wind turbine (VAWT), by widening the azimuthal angle band of the blade with the highest aerodynamic torque, instead of increasing the highest torque. The new VP-approach provides a curve of pitch angle designed for the blade operating at the rated tip speed ratio (TSR) corresponding to the peak power coefficient of the fixed pitch (FP)-VAWT. The effects of the new approach are exploited by using the double multiple stream tubes (DMST) model and
Prandtl's mathematics to evaluate the blade tip loss. The research describes the effects from six aspects, including the lift, drag, angle of attack (AoA), resultant velocity, torque, and power output, through a comparison between VP-VAWTs and FP-VAWTs working at four TSRs: 4, 4.5, 5, and 5.5. Compared with the FP-blade, the VP-blade has a wider azimuthal zone with the maximum AoA, lift, drag, and torque in the upwind half-cycle, and yields the two new larger maximum values in the downwind half-cycle. The power distribution in the swept area of the turbine changes from an arched shape of the FP-VAWT into the rectangular shape of the VP-VAWT. The new VP-approach markedly widens the highest-performance zone of the blade in a revolution, and ultimately achieves an 18.9% growth of the peak power coefficient of the VAWT at the optimum TSR. Besides achieving this growth, the new pitching method will enhance the performance at TSRs that are higher than current optimal values, and an increase of torque is also generated.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Hohai University, Nanjing University of Aeronautics and Astronautics
Publication date: 2018
Peer-reviewed: Yes

Publication information
Journal: Applied Sciences (switzerland)
Volume: 8
Issue number: 6
Article number: 957
ISSN (Print): 2076-3417
Ratings:
Scopus rating (2018): CiteScore 2.52
Web of Science (2018): Impact factor 2.217
Web of Science (2018): Indexed yes
Original language: English
Electronic versions:
applsci_08_00957_1_.pdf
DOIs:
10.3390/app8060957
Source: FindIt
Source-ID: 2435451297
Research output: Contribution to journal › Journal article – Annual report year: 2018 › Research › peer-review

Wind Turbine Acoustic Day 2018
The bi-annual event entitled Wind Turbine Acoustic Day dealing with wind turbine noise issues organized by DTU Wind Energy took place on May, 17th 2018 as its third edition. The abstracts and slides for the presentations are reported.

General information
Publication status: Published
Number of pages: 136
Publication date: 2018

Publication information
Publisher: DTU Wind Energy
ISBN (Electronic): 978-87-93549-36-4
Original language: English
(DTU Wind Energy E; No. 0168).
Electronic versions:
report_AcousticDay18_E_0168.pdf
Research output: Book/Report › Report – Annual report year: 2018 › Research › peer-review

Wind Turbine Acoustic Day 2018 - Summary of the 3rd edition
The bi-annual event entitled Wind Turbine Acoustic Day dealing with wind turbine noise issues organized by DTU Wind Energy took place on May, 17th 2018 as its third edition. The abstracts and slides for the presentations are reported.
Wind turbine noise generation and propagation modeling at DTU Wind Energy: A review

The present review paper provides a comprehensive overview of the research activities on wind turbine aeroacoustics at DTU over the last 20 years, as well as it gives the state-of-the-art of noise prediction models for wind turbines under complex inflow conditions. Various noise generation models developed at DTU are described and analyzed, including models based on the acoustic analogy, flow-acoustics splitting techniques, Amiet's model, and various engineering models. Some of the models are coupled to existing aero-elastic software and computational fluid mechanics models developed at DTU, and implemented in the simulation platform WindSTAR (Wind turbine Simulation Tool for AeRodymanic noise). This simulation platform consists of WindSTAR-Gen, dealing with models for generation of noise and design of low-noise wind turbines, and WindSTAR-Pro, which is developed to handle the modeling of long distance acoustic propagation. As specific features of the WindSTAR-Pro package, the rotation of the noise sources is modeled, the propagation simulations combine the so-called Parabolic Equations (PE) propagation model with numerical flow simulations to take into account effects from wind turbine wakes, atmospheric turbulence and wind shear.

Design optimization of offshore wind farms with multiple types of wind turbines

Most studies on offshore wind farm design assume a uniform wind farm, which consists of an identical type of wind turbines. In order to further reduce the cost of energy, we investigate the design of non-uniform offshore wind farms, i.e., wind farms with multiple types of wind turbines and hub-heights. Given a set of different types of wind turbines with a different default hub height for each type, we can specify the design of a wind farm by the types of turbines, number of turbines for each type, and turbine locations. We consider the optimization of such design to minimize the levelized cost of
energy, which is calculated using a capital cost model that covers the turbine cost and the balance of plant cost. An empirical wind turbine design cost and scaling model is utilized to model the cost of turbines with different sizes. Constraints on wind farm boundary, wind turbine proximity and total capacity are also included. We solve the problem with a newly developed extended random search algorithm and tested it in a realistic design optimization problem based on the Horns Rev 1 offshore wind farm in Denmark. The optimized non-uniform designs are compared with their uniform counterparts. We find that a non-uniform design can achieve a lower levelized cost of energy than its uniform counterparts, when the capital cost per MW is slightly lower for the smaller size turbine. Comparison with the mixed-discrete particle swarm optimization algorithm is also carried out for a non-uniform wind farm design problem with a fixed number of turbines, which shows the effectiveness and superiority of the proposed algorithm. Finally, the advantages and possible disadvantages of non-uniform design are also identified and discussed.

Validation of the actuator disc and actuator line techniques for yawed rotor flows using the New Mexico experimental data
Experimental data acquired in the New Mexico experiment on a yawed 4.5m diameter rotor model turbine are used here to validate the actuator line (AL) and actuator disc (AD) models implemented in the Large Eddy Simulation code EllipSys3D in terms of loading and velocity field. Even without modelling the geometry of the hub and nacelle, the AL and AD models produce similar results that are generally in good agreement with the experimental data under the various configurations considered. As expected, the AL model does better at capturing the induction effects from the individual blade tip vortices, while the AD model can reproduce the averaged features of the flow. The importance of using high quality airfoil data (including 3D corrections) as well as a fine grid resolution is highlighted by the results obtained. Overall, it is found that both models can satisfactorily predict the 3D velocity field and blade loading of the New Mexico rotor under yawed inflow.
Consistent modelling of wind turbine noise propagation from source to receiver
The unsteady nature of wind turbine noise is a major reason for annoyance. The variation of far-field sound pressure levels is not only caused by the continuous change in wind turbine noise source levels but also by the unsteady flow field and the ground characteristics between the turbine and receiver. To take these phenomena into account, a consistent numerical technique that models the sound propagation from the source to receiver is developed. Large eddy simulation with an actuator line technique is employed for the flow modelling and the corresponding flow fields are used to simulate sound generation and propagation. The local blade relative velocity, angle of attack, and turbulence characteristics are input to the sound generation model. Time-dependent blade locations and the velocity between the noise source and receiver are considered within a quasi-3D propagation model. Long-range noise propagation of a 5MW wind turbine is investigated. Sound pressure level time series evaluated at the source time are studied for varying wind speeds, surface roughness, and ground impedances within a 2000m radius from the turbine.

Cross-Cutting Activities 2016 on Wind Turbine Noise, Summary Report
The goal of this report is to summarize activities that took place in year 2016 as part of the Cross-Cutting Activity on Wind Turbine Noise, self-financed by DTU Wind Energy. A short description of the background behind this project (in particular Cross-Cutting Activities conducted in year 2015), the main objectives of the various studies and scientific achievements are reported in the introduction. Then, each Work Packages constituting this project are described in more details in the following sections.
Design of the OffWindChina 5 MW Wind Turbine Rotor
The current article describes the conceptual design of a rotor for a 5 MW machine situated at an offshore site in China (OffWindChina). The OffWindChina 5 MW rotor design work was divided into two parts between the Technical University of Denmark (DTU) and the Chong Qing University (CQU). The two parts consist of the aeroelastic and structural design phases. The aeroelastic part determines the optimal outer blade shape in terms of cost of energy (COE), while the structural part determines the internal laminate layup to achieve a minimum blade mass. Each part is performed sequentially using in-house optimization tools developed at DTU and CQU. The designed blade yields a high energy output while maintaining the structural feasibility with respect to international standards.

Development of an aeroelastic code based on three-dimensional viscous–inviscid method for wind turbine computations
Aerodynamic and structural dynamic performance analysis of modern wind turbines are routinely estimated in the wind energy field using computational tools known as aeroelastic codes. Most aeroelastic codes use the blade element momentum (BEM) technique to model the rotor aerodynamics and a modal, multi-body or the finite-element approach to model the turbine structural dynamics. The present work describes the development of a novel aeroelastic code that combines a three-dimensional viscous–inviscid interactive method, method for interactive rotor aerodynamic simulations (MIRAS), with the structural dynamics model used in the aeroelastic code FLEX5. The new code, called MIRAS-FLEX, is an improvement on standard aeroelastic codes because it uses a more advanced aerodynamic model than BEM. With the new aeroelastic code, more physical aerodynamic predictions than BEM can be obtained as BEM uses empirical relations, such as tip loss corrections, to determine the flow around a rotor. Although more costly than BEM, a small cluster is sufficient to run MIRAS-FLEX in a fast and easy way. MIRAS-FLEX is compared against the widely used FLEX5 and FAST, as well as the participant codes from the Offshore Code Comparison Collaboration Project. Simulation tests consist
Effects of wind turbine wake on atmospheric sound propagation

In this paper, we investigate the sound propagation from a wind turbine considering the effects of wake-induced velocity deficit and turbulence. In order to address this issue, an advanced approach was developed in which both scalar and vector parabolic equations in two dimensions are solved. Flow field input was obtained using the actuator line (AL) technique with Large Eddy Simulation (LES) to model the wind turbine and its wake and from an analytical wake model. The effect of incoming wind speed and atmospheric stability was investigated with the analytical wake input using a single point source. Unsteady acoustic simulations were carried out with the AL/LES input for three cases with different incoming turbulence intensity, and a moving source approach to mimic the rotating turbine blades. The results show a non-negligible effect of the wake on far-field noise prediction. Particularly under stable atmospheric conditions, SPL amplification reaches up to 7.5dB at the wake centre. Furthermore, it was observed that when the turbulence intensity level of the incoming flow is higher, the SPL difference between the moving and the steady source is lower.

General information
Description: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Resource Assessment Modelling
Pages: 51-61
Publication date: 2017
Peer-reviewed: Yes
Publication information
Journal: Applied Acoustics
Volume: 122
ISSN (Print): 0003-682X
Ratings:
BFI (2017): BFI-level 2
Scopus rating (2017): CiteScore 2.31 SJR 0.815 SNIP 1.543
Web of Science (2017): Impact factor 1.721
Web of Science (2017): Indexed yes
Original language: English
Electronic versions:
EBarlas_WakeEffectonSound_2_.pdf. Embargo ended: 28/02/2019
DOIs:
Flow field and load characteristics of the whole MEXICO wind turbine

CFD (Computational Fluid Dynamics) method was used to perform steady numerical simulation investigation on the flow field and load characteristics of MEXICO (Model EXperiment In Controlled cOnditions) wind turbine under non-yawed condition. Circumferentially-Averaged method was used to extract the calculated axial, radial and tangential components of velocity along the axial direction, then these components were compared with the experimental data, the compared results show that the computational components agree well with the experimental data and the computational results are reliable. The flow characteristics around the blade was analyzed and the points of flow separation were found along the blade, the results show that the points of flow separation move towards trailing edge with the increase of radius. The distribution of vorticity in the wake of MEXICO rotor was also analyzed. The distribution of vorticity in the wake of three blade passages is symmetrical approximately. The value of vorticity decreases gradually along the axial direction behind the rotor and the tower has limited effects on the wake when the CFD simulation is steady. Besides, the load distribution along the radial direction of rotor blades was analyzed and the distribution law of load along the blade was obtained. The obtained load characteristic can provide the basis for the analysis of aeroelasticity of wind turbines.

Improved fixed point iterative method for blade element momentum computations

The blade element momentum (BEM) theory is widely used in aerodynamic performance calculations and optimization applications for wind turbines. The fixed point iterative method is the most commonly utilized technique to solve the BEM equations. However, this method sometimes does not converge to the physical solution, especially for the locations near the blade tip and root where the failure rate of the iterative method is high. The stability and accuracy of aerodynamic calculations and optimizations are greatly reduced due to this problem. The intrinsic mechanisms leading to convergence problems are addressed through both theoretical analysis and numerical tests. A term from the BEM equations equals to zero at a critical inflow angle is the source of the convergence problems. When the initial inflow angle is set larger than the critical inflow angle and the relaxation methodology is adopted, the convergence ability of the iterative method will be greatly enhanced. Numerical tests have been performed under different combinations of local tip speed ratio, local solidity, local twist and airfoil aerodynamic data. Results show that the simple iterative methods have a good convergence ability which will improve the aerodynamic or structural design of wind turbines.
Modeling of wind turbine vortex generators in considering the inter-effects between arrays

Vortex generators (VGs) are commonly placed on wind turbine blades to delay flow separation in the boundary layer. VGs can be parametrically modeled in computational fluid dynamics for effective and efficient simulations of wind blade flow fields. Many researchers have studied the vortex circulation created by VGs and created various parametric models used with the Navier-Stokes equations, but most of them are based on a single winglet of VGs and do not include the inter-effects between the winglets. This paper proposes a parameterized VG array model based on counter-rotating VGs, which properly takes into account the inter-effects between winglets. Two cases, i.e., a plate with a pair of VGs and a DU-W2-250 blade segment with five pairs of VGs, are investigated to validate this model; the array type parametric model is in closer agreement with experimental data than traditional models. Compared to the solid VG model, the array type model has similar streamlines and surface pressure coefficients on the suction surface. The array type VG model can effectively reduce the number of grid points and yield highly accurate predictions of wind turbine blade aerodynamic characteristics.

General information

Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Hohai University, Nanjing University of Aeronautics and Astronautics
Contributors: Zhao, Z., Shen, W., Wang, R., Wang, T., Xu, B., Zheng, Y., Qian, S.
Number of pages: 18
Publication date: 2017
Peer-reviewed: Yes
Numerical Study of Wind Turbine Wake Modeling Based on a Actuator Surface Model

In the Actuator Surface Model (ALM), the turbine blades are represented by porous surfaces of velocity and pressure discontinuities to model the action of lifting surfaces on the flow. The numerical simulation is implemented on FLUENT platform combined with N-S equations. This model is improved on the basis of actuator line model(AM). By using ASM, the model of turbine can be simplified and the quantity of grids and computing time can be significantly reduced. A linear distribution model and a ASM Grid identification method are presented. This paper compares the ASM with ALM by computing both near and far wake of a Nibe A wind turbine, which combines wake velocity, turbulent intensity and vortex structure. Results show that ASM has better prediction accuracy and verify it's feasibility on numerical simulation of wind turbine wake.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Hohai University, Chinese Academy of Sciences
Contributors: Zhou, H., Xu, C., Han, X. X., Shen, W. Z., Zhang, M., Chen, X. Y.
Number of pages: 6
Pages: 535-540
Publication date: 2017
Peer-reviewed: Yes

Publication information
Journal: Gongcheng Rewuli Xuebao
Volume: 38
Issue number: 3
ISSN (Print): 0253-231X
Ratings:
Scopus rating (2017): CiteScore 0.23
Original language: English
Electronic versions: _-pdf
Source: FindIt
Source-ID: 2371424560
Research output: Contribution to journal › Journal article – Annual report year: 2017 › Research › peer-review

Prediction of multi-wake problems using an improved Jensen wake model

The improved analytical wake model named as 2D_k Jensen model (which was proposed to overcome some shortcomings in the classical Jensen wake model) is applied and validated in this work for wind turbine multi-wake predictions. Different from the original Jensen model, this newly developed 2D_k Jensen model uses a cosine shape instead of the top-hat shape for the velocity deficit in the wake, and the wake decay rate as a variable that is related to the ambient turbulence as well as the rotor generated turbulence. Coupled with four different multi-wake combination models, the 2D_k Jensen model is assessed through (1) simulating two wakes interaction under full wake and partial wake conditions and (2) predicting the power production in the Horns Rev wind farm for different wake sectors around two different wind directions. Through comparisons with field measurements, results from Large Eddy Simulations (LES) as well as results from other commercial codes, it is found that the predictions obtained with the 2D_k Jensen model exhibit good to excellent agreements with experimental and LES data.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Nanjing University of Aeronautics and Astronautics
Contributors: Tian, L., Zhu, W. J., Shen, W. Z., Song, Y., Zhao, N.
Number of pages: 13
Pages: 457-469
Publication date: 2017
Peer-reviewed: Yes

Publication information
Journal: Renewable Energy
Volume: 102
ISSN (Print): 0960-1481
Ratings:
BFI (2017): BFI-level 1
Scopus rating (2017): CiteScore 5.38 SJR 1.847 SNIP 2.008
Web of Science (2017): Impact factor 4.9
Web of Science (2017): Indexed yes
Original language: English
Study of integrated optimization design of wind farm in complex terrain

Aiming at the present stage the micro-site selection of wind farm in complex terrain and the wind turbine layout and other close relationship, and selecting more reasonable wind turbine layout, more online power and saving more investment as the goal, analyzing briefly the main factors influencing wind farm design in complex terrain and setting up integrated optimization mathematical model for micro-site selection, power lines and road maintenance design etc.. Based on the existing 1-year wind measurement data in the wind farm area, the genetic algorithm was used to optimize the micro-site selection. On the basis of location optimization of wind turbine, the optimization algorithms such as single-source shortest path algorithm and minimum spanning tree algorithm were used to optimize electric lines and maintenance roads. The practice shows that the research results can provide important process guidance for the actual preliminary work of wind farm construction.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Hohai University, Nanjing Nanrui Group Corporation
Contributors: Xu, C., Chen, D., Han, X., Shen, W., Wang, C., Shi, L.
Pages: 3368-3375
Publication date: 2017
Peer-reviewed: Yes

Publication information
Journal: Taiyang Neng Xuebao
Volume: 38
Issue number: 12
ISSN (Print): 0254-0096
Ratings:
Scopus rating (2017): CiteScore 0.25 SJR 0.17 SNIP 0.293
Original language: English
Source: FindIt
Source-ID: 2418392829
Research output: Contribution to journal › Journal article – Annual report year: 2018 › Research › peer-review

Study on variable pitch strategy in H-type wind turbine considering effect of small angle of attack

Variable-pitch (VP) technology is an effective approach to upgrade the aerodynamics of the blade of an H-type vertical-axis wind turbine (VAWT). At present, most of the research efforts are focused on the performance improvement of the azimuth angle owing to the large angle of attack (AoA). On the blade circular path of an H-type VAWT, there are two azimuth positions where torques are negative, and the performance is the poorest. The vicinity zones of the two azimuths also have low performance and greatly weaken the overall productivity of VAWT. In this paper, we propose a new technology that, unlike the traditional VP-technology, focuses mainly on the aerodynamics improvement of the azimuth position with small AoA. The purpose of this novel approach is to widen the band of azimuth positions with high performance and eventually enhance the power efficiency of the overall VAWT. The performance of the new VP-VAWT is predicted using the Double Multiple Streamtubes model and Prandtl's mathematics to evaluate the blade tip loss. Compared with the fixed-pitch (FP) blade, the VP-blade has a wider zone of the max AoA and tangential force in the upwind half-circle and yields the two new larger max values in the downwind half-circle. The new VP-strategy considerably narrows the two low-torque zones near the 0° and 180° azimuths and markedly widens the high-torque azimuth zone; the torque distribution appears in a trapezoidal shape in the upwind region and an M-like shape in the downwind region. The power distribution in the swept area of turbine changes from an arched shape of the FP-VAWT into a rectangular shape of the VP-VAWT. At last, an 18.9% growth in power efficiency is achieved. All of the above results confirm that the new VP-technology can effectively improve VAWT performance and also widens the highest performance tip speed ratio zone which makes the turbines capable of running with high efficiency in wider zones.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Hohai University, Nanjing University of Aeronautics and Astronautics
Contributors: Zhao, Z., Qian, S., Shen, W., Wang, T., Xu, B., Zheng, Y., Wang, R.
Number of pages: 21
Publication date: 2017
Verification of a novel innovative blade root design for wind turbines using a hybrid numerical method

To enhance the performance of horizontal axis wind turbines, it is proposed to place a cylindrical disc in front of the rotor in order to lead the incoming flow from the inner part to the outer part of the rotor blades. This is expected to increase the power output, as the kinetic energy is mainly captured at the outer part of the blades, where the relative wind speed is high. To assess the impact of this novel design idea, a hybrid numerical technique, based on solving the Reynolds-averaged Navier-Stokes equations, is utilized to determine the aerodynamic performance. The in-house developed EllipSys3D code, which is employed as basic numerical solver, is combined with an actuator disc representation of the wind turbine rotor and an immersed boundary technique for representing the upstream cylindrical disc. The impact of the disc on the rotor performance is assessed by systematically changing the size of the circular disc and its axial distance to the rotor. Based on a numerical study of a Megawatt size commercial wind turbine, it is found that up to 1.5% additional energy can be captured by placing a circular disc with a suitable diameter upstream of the rotor plane.

Wind farm design in complex terrain: the FarmOpt methodology

Designing wind farms in complex terrain is becoming more and more important, especially for countries like China, where a large portion of the territory is featured as complex terrain. Although potential richer wind resources could be expected at complex terrain sites (thanks to the terrain effects), they also expose many challenges for wind farm designers/developers. In this study, we present the FarmOpt methodology for designing wind farms in complex terrain, which combines the state-of-the-art wind resource assessment methods with engineering wake models adapted for complex terrain and advanced layout optimization algorithms. Various constraints are also modelled and considered in the design optimization.
problem for maximizing the annual energy production (AEP). A case study is presented to illustrate the effectiveness of the methodology. Further developments of the FarmOpt tool are also briefly introduced.

**General information**
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Test and Measurements, Resource Assessment Modelling, Wind turbine loads & control, Hohai University, Northwest Survey and Design Institute of China Hydropower Consulting Group, Aalborg University
Number of pages: 6
Publication date: 2017
Peer-reviewed: Yes
Electronic versions:
Wind_farm_design_in_complex_terrain_the_FarmOpt_methodology_full_paper_for_CWP2017.pdf
Source: PublicationPreSubmission
Source-ID: 141907808
Research output: Contribution to conference › Paper – Annual report year: 2017 › Research › peer-review

**Wind farm design in complex terrain: the FarmOpt methodology**
Designing wind farms in complex terrain is becoming more and more important, especially for countries like China, where a large portion of the territory is featured as complex terrain. Although potential richer wind resources could be expected at complex terrain sites (thanks to the terrain effects), they also expose many challenges for wind farm designers/developers. In this study, we present the FarmOpt methodology for designing wind farms in complex terrain, which combines the state-of-the-art wind resource assessment methods with engineering wake models adapted for complex terrain and advanced layout optimization algorithms. Various constraints are also modelled and considered in the design optimization problem for maximizing the annual energy production (AEP). A case study is presented to illustrate the effectiveness of the methodology. Further developments of the FarmOpt tool are also briefly introduced.

**Wind farm power production in the changing wind: Robustness quantification and layout optimization**
Wind farms operate often in the changing wind. The wind condition variations in a wide range of time scales lead to the variability of wind farms' power production. This imposes a major challenge to the power system operators who are facing a higher and higher penetration level of wind power. Thus, wind farm developers/owners need to take the variability into consideration in the designing/planning stage, in addition to the conventional main objective of maximizing the expected power output under a fixed wind distribution. In this study, we first propose a new metric to evaluate the variability of wind power based on the characteristics of the wind farm and its local wind conditions. Then a series of robustness metrics are proposed to quantify wind farm’s ability to produce power with high mean value and low variability under changing wind, considering both short-term and long-term wind condition variations. Based on these metrics, wind farm layout optimization is performed to maximize the robustness of a real offshore wind farm in Denmark. The results demonstrate that the robustness metrics are more flexible and complete than the conventional metrics for characterizing wind farm power production, such as mean power output or wind power variability alone, and it is feasible to design wind farms to produce power with high mean value and low variability.

**General information**
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Actuator disk model of wind farms based on the rotor average wind speed

Due to difficulty of estimating the reference wind speed for wake modeling in wind farm, this paper proposes a new method to calculate the momentum source based on the rotor average wind speed. The proposed model applies volume correction factor to reduce the influence of the mesh recognition of disk regions. The coefficient $C_4 \varepsilon$ of the turbulent source term is also discussed and modified to improve the simulation accuracy. To validate the model, results are presented for the Nibe-B wind turbine and Horns Rev I offshore wind farm and show a good agreement with the measurements.

Aeroacoustic calculations of a full scale Nordtank 500kW wind turbine

The Actuator Line/ Navier-stokes technique is used to compute the incompressible flow around a full scale Nordtank 500kW wind turbine under different complex flow conditions such as atmospheric turbulence and wind shear. The flow field is used as an input to aeroacoustic calculations based on; a semi empirical noise model; and a Navier-Stokes based computational aeroacoustic code (CAA). The Navier-Stokes based approach is solving acoustic perturbation equations and is capable of taking propagation and ground effects into account, but is limited to low frequency noise due to feasible mesh resolution, and due to the simplification in the actuator line method using body forces to represent the blade. Noise levels are compared to field measurements of a Nordtank 500kW wind turbine at different wind speeds and in flow profiles.
Aeroacoustic Calculations of Wind Turbine Noise with the Actuator Line/ Navier-Stokes Technique

Noise regulations in many countries are becoming extremely strict and wind turbine noise is thus becoming a barrier for further development of onshore wind turbines. Low noise wind turbine airfoil and blade design is an important technique for noise reduction. However, the ow situation of a wind turbine in wind farms is very complicated. In order to accurately model the noise generation and propagation from wind turbines in wind farms, it is urgent to develop a high-fidelity noise model to predict the noise features in complex situations. In the present study, we develop a flow-acoustic splitting technique where the wind turbine ow is calculated by using the in-house actuator line/LES/Navier-Stokes technique and the acoustics is obtained by solving the acoustic perturbation equations. In the ow solver, the wind turbine blades are modelled by rotating lines with body forces determined according to the local conditions and airfoil data. In the acoustic solver, the aeroacoustics is simulated by: (1) calculating the noise source using the improved engineering model (IBPM) based on the model developed by Brook, Pope and Marcolini (BPM); (2) introducing the noise source with an expected range of frequencies along the blade lines in the acoustic solver; (3) solving the acoustic perturbation equations with the introduced source and the source captured in the ow. The model can be used to study the prediction and propagation of low-frequency noise in complex situations. Noise generated by a wind turbine with and without yaw under wind shear and inflow turbulence will be presented in the paper.

Aerodynamic wind-turbine rotor design using surrogate modeling and three-dimensional viscous-inviscid interaction technique

In this paper, a surrogate optimization methodology using a three-dimensional viscous-inviscid interaction code for the aerodynamic design of wind-turbine rotors is presented. The framework presents a unique approach because it does not require the commonly-used blade element momentum (BEM) method. The three-dimensional viscous-inviscid interaction code used here is the accurate and fast MIRAS code developed at the Technical University of Denmark. In comparison with BEM, MIRAS is a higher-fidelity aerodynamic tool and thus more computationally expensive as well. Designing a
Extension of Goldstein's circulation function for optimal rotors with hub

The aerodynamic interaction or interference between rotor blades and hub body is usually very complicated, but some useful simplifications can be made by considering the hub with an infinite cylinder. Various attempts to find the optimum distribution of circulation by the lifting vortex lines method have been previously proposed to describe the blade interaction with the hub modeled by the infinite cylinder. In this case, the ideal distribution of bound circulation on the rotor blades is such that the shed vortex system in the hub-area is a set of helicoidal vortex sheets moving uniformly as if rigid, exactly as in the case where there is no influence of the streamtube deformations by the central hub-body. In the present investigation, we consider a more specific problem of the rotor-hub interaction where the initial flow streamtubes and the rotor slipstream submitted strong deformations at the nose-area of the semi-infinite hub.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Contributors: Okulov, V., Sørensen, J. N., Shen, W. Z.
Number of pages: 5
Publication date: 2016
Peer-reviewed: Yes

Publication information
Journal: Journal of Physics: Conference Series (Online)
Volume: 753
Article number: 022018
ISSN (Print): 1742-6596
Ratings:
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 0.45 SJR 0.24 SNIP 0.401
Web of Science (2016): Indexed yes
Original language: English
DOIs:
Improved blade element momentum theory for wind turbine aerodynamic computations

Blade element momentum (BEM) theory is widely used in aerodynamic performance predictions and design applications for wind turbines. However, the classic BEM method is not quite accurate which often tends to under-predict the aerodynamic forces near root and over-predict its performance near tip. The reliability of the aerodynamic calculations and design optimizations is greatly reduced due to this problem. To improve the momentum theory, in this paper the influence of pressure drop due to wake rotation and the effect of radial velocity at the rotor disc in the momentum theory are considered. Thus the axial induction factor in far downstream is not simply twice of the induction factor at disc. To calculate the performance of wind turbine rotors, the improved momentum theory is considered together with both Glaucert's tip correction and Shen's tip correction. Numerical tests have been performed for the MEXICO rotor. Results show that the improved BEM theory gives a better prediction than the classic BEM method, especially in the blade tip region, when comparing to the MEXICO measurements. (C) 2016 Elsevier Ltd. All rights reserved.

General information
Publication status: Published
Organisations: Department of Energy Engineering, Department of Mechanical Engineering, Department of Wind Energy, Fluid Mechanics, Chongqing University
Contributors: Sun, Z., Chen, J., Shen, W. Z., Zhu, W. J.
Pages: 824-831
Publication date: 2016
Peer-reviewed: Yes

Publication information
Journal: Renewable Energy
Volume: 96
ISSN (Print): 0960-1481
Ratings:
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 4.83 SJR 1.661 SNIP 2.05
Web of Science (2016): Impact factor 4.357
Web of Science (2016): Indexed yes
Original language: English
DOIs:
10.1016/j.renene.2016.05.035
Source: FindIt
Source-ID: 2304500548
Research output: Contribution to journal › Journal article – Annual report year: 2016 › Research › peer-review

Improvement of airfoil trailing edge bluntness noise model

In this article, airfoil trailing edge bluntness noise is investigated using both computational aero-acoustic and semi-empirical approach. For engineering purposes, one of the most commonly used prediction tools for trailing edge noise are based on semi-empirical approaches, for example, the Brooks, Pope, and Marcolini airfoil noise prediction model developed by Brooks, Pope, and Marcolini (NASA Reference Publication 1218, 1989). It was found in previous study that the Brooks, Pope, and Marcolini model tends to over-predict noise at high frequencies. Furthermore, it was observed that this was caused by a lack in the model to predict accurately noise from blunt trailing edges. For more physical understanding of bluntness noise generation, in this study, we also use an advanced in-house developed high-order computational aero-acoustic technique to investigate the details associated with trailing edge bluntness noise. The results from the numerical model form the basis for an improved Brooks, Pope, and Marcolini trailing edge bluntness noise model.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, University of Copenhagen
Contributors: Zhu, W. J., Shen, W. Z., Sørensen, J. N., Leloudas, G.
Number of pages: 12
Large Wind Turbine Rotor Design using an Aero-Elastic / Free-Wake Panel Coupling Code

Despite the advances in computing resources in the recent years, the majority of large wind-turbine rotor design problems still rely on aero-elastic codes that use blade element momentum (BEM) approaches to model the rotor aerodynamics. The present work describes an approach to wind-turbine rotor design by incorporating a higher-fidelity free-wake panel aero-elastic coupling code called MIRAS-FLEX. The optimization procedure includes a series of design load cases and a simple structural design code. Due to the heavy MIRAS-FLEX computations, a surrogate-modeling approach is applied to mitigate the overall computational cost of the optimization. Improvements in cost of energy, annual energy production, maximum a-p-wise root bending moment, and blade mass were obtained for the NREL 5MW baseline wind turbine.
LES tests on airfoil trailing edge serration
In the present study, a large number of acoustic simulations are carried out for a low noise airfoil with different Trailing Edge Serrations (TES). The Ffowcs Williams-Hawkings (FWH) acoustic analogy is used for noise prediction at trailing edge. The acoustic solver is running on the platform of our in-house incompressible flow solver EllipSys3D. The flow solution is first obtained from the Large Eddy Simulation (LES), the acoustic part is then carried out based on the instantaneous hydrodynamic pressure and velocity field. To obtain the time history data of sound pressure, the flow quantities are integrated around the airfoil surface through the FWH approach. For all the simulations, the chord based Reynolds number is around 1.5x10^6. In the test matrix, the effects from angle of attack, the TE flap angle, the length/width of the TES are investigated. Even though the airfoil under investigation is already optimized for low noise emission, most numerical simulations and wind tunnel experiments show that the noise level is further decreased by adding the TES device.

Multi-objective random search algorithm for simultaneously optimizing wind farm layout and number of turbines
A new algorithm for multi-objective wind farm layout optimization is presented. It formulates the wind turbine locations as continuous variables and is capable of optimizing the number of turbines and their locations in the wind farm simultaneously. Two objectives are considered. One is to maximize the total power production, which is calculated by considering the wake effects using the Jensen wake model combined with the local wind distribution. The other is to minimize the total electrical cable length. This length is assumed to be the total length of the minimal spanning tree that connects all turbines and is calculated by using Prim's algorithm. Constraints on wind farm boundary and wind turbine proximity are also considered. An ideal test case shows the proposed algorithm largely outperforms a famous multi-objective genetic algorithm (NSGA-II). In the real test case based on the Horn Rev 1 wind farm, the algorithm also obtains useful Pareto frontiers and provides a wide range of Pareto optimal layouts with different numbers of turbines for a real-life wind farm developer.
Noise model for serrated trailing edges compared to wind tunnel measurements
A new CFD RANS based method to predict the far field sound pressure emitted from an aerofoil with serrated trailing edge has been developed. The model was validated by comparison to measurements conducted in the Virginia Tech Stability Wind Tunnel. The model predicted 3 dB lower sound pressure levels, but the tendencies for the different configurations were predicted correctly. Therefore the model can be used to optimise the serration geometry. A disadvantage of the new model is that the computational costs are significantly higher than for the Amiet model for a straight trailing edge. However, it is by decades faster than LES methods.

The collection of the main issues for wind farm optimisation in complex terrain
The paper aims at establishing the collection of the main issues for wind farm optimisation in complex terrain. To make wind farm cost effective, this paper briefly analyses the main factors influencing wind farm design in complex terrain and sets up a series of mathematical model that includes micro-siting, collector circuits, access roads design for optimization problems. The paper relies on the existing one year wind data in the wind farm area and uses genetic algorithm to optimize the micro-siting problem. After optimization of the turbine layout, single-source shortest path algorithm and minimum spanning tree algorithm are used to optimize collector circuits and access roads. The obtained results can provide important guidance for wind farms construction.
Three-dimensional viscous-inviscid coupling method for wind turbine computations

In this paper, a computational model for predicting the aerodynamic behavior of wind turbine wakes and blades subjected to unsteady motions and viscous effects is presented. The model is based on a three-dimensional panel method using a surface distribution of quadrilateral sources and doublets, which is coupled to a viscous boundary layer solver. Unlike Navier-Stokes codes that need to solve the entire flow domain, the panel method solves the flow around a complex geometry by distributing singularity elements on the body surface, obtaining a faster solution and making this type of codes suitable for the design of wind turbines. A free-wake model has been employed to simulate the wake behind a wind turbine by using vortex filaments that carry the vorticity shed by the trailing edge of the blades. Viscous and rotational effects inside the boundary layer are taken into account via the transpiration velocity concept, applied using strip theory with the cross sectional angle of attack as coupling parameter. The transpiration velocity is obtained from the solution of the integral boundary layer equations with extension for rotational effects. It is found that viscosity plays a very important role in the predictions of blade aerodynamics and wake dynamics, especially at high angles of attack just before and after boundary layer separation takes place. The present code is validated in detail against the well-known MEXICO experiment and a set of non-rotating cases. Copyright © 2014 John Wiley & Sons, Ltd.
Wind turbine noise propagation modelling: An unsteady approach

Wind turbine sound generation and propagation phenomena are inherently time dependent, hence tools that incorporate the dynamic nature of these two issues are needed for accurate modelling. In this paper, we investigate the sound propagation from a wind turbine by considering the effects of unsteady flow around it and time dependent source characteristics. For the acoustics modelling we employ the Parabolic Equation (PE) method while Large Eddy Simulation (LES) as well as synthetically generated turbulence fields are used to generate the medium flow upon which sound propagates. Unsteady acoustic simulations are carried out for three incoming wind shear and various turbulence intensities, using a moving source approach to mimic the rotating turbine blades. The focus of the present paper is to study the near and far field amplitude modulation characteristics and time evolution of Sound Pressure Level (SPL).

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Number of pages: 11
Publication date: 2016
Peer-reviewed: Yes

Publication information
Journal: Journal of Physics: Conference Series (Online)
Volume: 753
Article number: 022003
ISSN (Print): 1742-6596
Ratings:
BFI (2016): BFI-level 1
Scopus rating (2016): CiteScore 0.45 SJR 0.24 SNIP 0.401
Web of Science (2016): Indexed yes
Original language: English
Electronic versions:
Wind_turbine_noise_propagation_modelling.pdf
DOIs:
10.1088/1742-6596/753/2/022003

Bibliographical note
Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Source: FindIt
Source-ID: 2346249231
Research output: Contribution to journal › Journal article – Annual report year: 2014 › Research › peer-review

Wind turbine wake measurement in complex terrain

SCADA data from a wind farm and high frequency time series measurements obtained with remote scanning systems have been analysed with focus on identification of wind turbine wake properties in complex terrain. The analysis indicates that within the flow regime characterized by medium to large downstream distances (more than 5 diameters) from the wake generating turbine, the wake changes according to local atmospheric conditions e.g. vertical wind speed. In very complex terrain the wake effects are often “overruled” by distortion effects due to the terrain complexity or topology.

General information
Publication status: Published
Number of pages: 10
Publication date: 2016
Peer-reviewed: Yes

Publication information
Journal: Journal of Physics: Conference Series (Online)
Volume: 753
Issue number: 3
Article number: 032013
Aerodynamic performance of wind turbine under different yaw angles

A typical dynamic characteristic of horizontal axis wind turbine shows up under yaw condition. Prediction accuracy is low for momentum-blade element theory and related engineering prediction model. In order to improve the prediction accuracy of dynamic load characteristics, the whole wind turbine models, based on the experiment about MEXICO (model experiments in controlled conditions) rotor in 2006, are established by three-dimensional software called Pro/E. under different yaw conditions, i.e. yaw angle of 0, 15, 30 and 45 degree. ICEM CFD (integrated computer engineering and manufacturing code for computational fluid dynamics) is applied to grid division. The rotating domain containing rotor part is meshed into hexahedral grids, and the static domain containing part of wheel hub, tower and outflow field is meshed into tetrahedral grids. When the grid size of the first layer of blade surface is set as 5×10^-6 m to ensure the first dimensionless size near the wall Y+<0.5 on the wall, the 2 numbers of grids are determined by the error of axial load on the airfoil in the 60% section of blades, which respectively are 6 572 451 and 2 961 385. The aerodynamic performance of models under rated condition is simulated by ANSYS CFX with the turbulence model of SST (shear stress transport), high resolution is chosen as advection scheme, and transient rotor stator as the domain interface method. The results are converted into data, processed and analyzed by MATLAB. Finally the following conclusions are drawn. The distributions of pressure coefficients along the airfoil chord in different blade sections calculated by CFD method are in good agreement with the experimental measurements, and the error on the suction surface of airfoil is mainly caused by stall separation occurring on the pressure surface of airfoil. With the increasing of yaw angle, the pressure coefficients of the suction side are increasing and the location of minimum pressure coefficient moves to airfoil trailing edge slightly. For the pressure side, the pressure coefficients increase at first and then decrease, and the location of maximum pressure coefficient moves to airfoil leading edge slightly. The axial load coefficients and tangential load coefficients of blades first decrease and then increase and then decrease again with the increase of the azimuthal angle. With the increase of the yaw angle, the axial and tangential load coefficients are both reduced. When the yaw angle is within 30°, the relative error of axial load coefficients is in the range of ±5% and the relative error of tangential load coefficients is in the range of ±15%. CFD method is higher than BEM (blade element momentum) method in forecasting accuracy of dynamic load calculation. Under yaw condition, the hysteresis characteristic of airfoil lift and drag in blade root is more remarkable than blade tip, while the variation range of the angle of attack in blade root is much less than that in blade tip. This characteristic must be considered when BEM method is used to predict wind turbine performance. For axial inflow condition, CFD method can well predict the average speed, but restricted by turbulence model and the wake model, CFD calculation did not show the velocity characteristics of rotating vortex shedding from wind turbine impeller under yaw condition. The study provides a data support to build up the forecast model on the engineering and provides the basis for wind turbine design under yaw condition.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Number of pages: 8
Pages: 78-85
Publication date: 2015
Peer-reviewed: Yes

Publication information
Journal: Nongye Gongcheng Xuebao
Volume: 31
Issue number: 16
ISSN (Print): 1002-6819
Ratings:
Scopus rating (2015): CiteScore 1.09 SJR 0.351 SNIP 0.837
Original language: Chinese
Keywords: Mechanical Engineering, Agricultural and Biological Sciences (all), Computational fluid dynamics, Dynamic stall, Models, Numerical analysis, The velocity distribution, The yaw angle, Wind turbines, Advection, Aerodynamic stalling,
Airfoil Trailing Edge Noise Generation and Its Surface Pressure Fluctuation

In the present work, Large Eddy Simulation (LES) of turbulent flows over a NACA 0015 airfoil is performed. The purpose of such numerical study is to relate the aerodynamic surface pressure with the noise generation. The results from LES are validated against detailed surface pressure measurements where the time history pressure data are recorded by the surface pressure microphones. After the flow-field is stabilized, the generated noise from the airfoil Trailing Edge (TE) is predicted using the acoustic analogy solver, where the results from LES are the input. It is found that there is a strong relation between TE noise and the aerodynamic pressure. The results of power spectrum density show that the fluctuation of aerodynamic pressure is responsible for noise generation.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Contributors: Zhu, W. J., Shen, W. Z.
Pages: 14-19
Publication date: 2015
Peer-reviewed: Yes

Publication information
Journal: Journal of Power and Energy Engineering
Volume: 3
ISSN (Print): 2327-588X
Original language: English
Keywords: Large Eddy Simulation, Trailing Edge Noise, Surface Pressure
Electronic versions:
Airfoil_Trailing_Edge
DOIs:
10.4236/jpee.2015.37003
Source: PublicationPreSubmission
Source-ID: 116365085
Research output: Contribution to journal › Journal article – Annual report year: 2015 › Research › peer-review

Denmark Wind Energy Programme

In this chapter, a summary of some ongoing wind energy projects in Denmark is given. The research topics comprise computational model development, wind turbine (WT) design, low-noise airfoil and blade design, control device development, wake modelling and wind farm layout optimization.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Contributors: Shen, W. Z.
Pages: 941-949
Publication date: 2015

Host publication information
Title of host publication: Renewable Energy in the Service of Mankind
Volume: 1
Publisher: Springer
ISBN (Print): 978-3-319-17776-2
ISBN (Electronic): 978-3-319-17777-9
Keywords: Wind turbine aerodynamics, Wind turbine aero-acoustics, Wind turbine design
Electronic versions:
Denmark_Wind_Energy_Programme.pdf
Design and Experimental Validation of Thick Airfoils for Large Wind Turbines

In this chapter, two new airfoils with thickness to chord ratios of 30 and 36% are presented, which were designed with an objective of good aerodynamic and structural features. Airfoil design is based on a direct method using shape perturbation function. The optimization algorithm is coupled with the viscous/inviscid flow solver XFOIL.

Development and validation of a new two-dimensional wake model for wind turbine wakes

A new two-dimensional (2D) wake model is developed and validated in this article to predict the velocity and turbulence distribution in the wake of a wind turbine. Based on the classical Jensen wake model, this model is further employing a cosine shape function to redistribute the spread of the wake deficit in the crosswind direction. Moreover, a variable wake decay rate is proposed to take into account both the ambient turbulence and the rotor generated turbulence, different from a constant wake decay rate used in the Jensen model. The obtained results are compared to field measurements, wind tunnel experiments, and results of an advanced k-ω turbulence model as well as large eddy simulations. From the comparisons, it is found that the proposed new wake model gives a good prediction in terms of both shape and velocity amplitude of the wake deficit, especially in the far wake which is the region of interest for wind farm development projects.
power, Wind tunnels, Wind turbines, Decay constants, Velocity deficits, Wake model, Wind farm development, Wakes

Electronic versions:
Developmentandvalidation.pdf
DOIs:
10.1016/j.jweia.2014.12.001

Bibliographical note
This work is supported by the National Basic Research Program of China (the "973" program. 2014CB046200), NSFC (grant 91130030) and the Danish Research Council for Strategic Research (DSF sagsnr. 10-094544).
Source: FindIt
Source-ID: 273757168
Research output: Contribution to journal › Journal article – Annual report year: 2015 › Research › peer-review

Development of a Fast Fluid-Structure Coupling Technique for Wind Turbine Computations
Fluid-structure interaction simulations are routinely used in the wind energy industry to evaluate the aerodynamic and structural dynamic performance of wind turbines. Most aero-elastic codes in modern times implement a blade element momentum technique to model the rotor aerodynamics and a modal, multi-body, or finite-element approach to model the turbine structural dynamics. The present paper describes a novel fluid-structure coupling technique which combines a threedimensional viscous-inviscid solver for horizontal-axis wind-turbine aerodynamics, called MIRAS, and the structural dynamics model used in the aero-elastic code FLEX5. The new code, MIRASFLEX, in general shows good agreement with the standard aero-elastic codes FLEX5 and FAST for various test cases. The structural model in MIRAS-FLEX acts to reduce the aerodynamic load computed by MIRAS, particularly near the tip and at high wind speeds.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Contributors: Sessarego, M., Ramos García, N., Shen, W. Z.
Pages: 1-6
Publication date: 2015
Peer-reviewed: Yes

Publication information
Journal: Journal of Power and Energy Engineering
Volume: 3
ISSN (Print): 2327-588X
Original language: English
Keywords: Fluid-Structure-Interaction, Wind-Turbine, Aero-Elasticity
Electronic versions:
JPEE_2015071716062193.pdf
DOIs:
10.4236/jpee.2015.37001
Research output: Contribution to journal › Journal article – Annual report year: 2015 › Research › peer-review

Development of a high-fidelity noise prediction and propagation model for noise generated from wind turbines
An approach to combine the actuator line technique with the improved Brooks, Pope, and Marcolini (IBPM) model for wind turbine noise calculation is presented. The IBPM needs Mach number, local angle of attack and blade position as an input. These can be calculated accurately with the actuator line technique for any kind of flow conditions. We investigated laminar/ turbulent inflow, as well as wind shear and yaw of the 2.3 MW NM80 wind turbine. The turbulent case shows higher noise levels than the laminar one. Yaw changes the directivity from a dipole characteristic to an oval shape, inclined by the yaw angle.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Contributors: Debertshäuser, H., Shen, W. Z., Zhu, W. J.
Number of pages: 7
Publication date: 2015

Host publication information
Title of host publication: Proceedings. 6th International Meeting on Wind Turbine Noise
Electronic versions:
Development_of_a_high_fidelity_noise.pdf
Source: PublicationPreSubmission
Source-ID: 118678370
Erratum: Validation of the actuator line method using near wake measurements of the MEXICO rotor

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Uppsala University, KTH - Royal Institute of Technology
Contributors: Nilsson, K., Shen, W. Z., Sørensen, J. N., Breton, S., Ivanell, S.
Number of pages: 1
Pages: 1683
Publication date: 2015
Peer-reviewed: Yes

Publication information
Journal: Wind Energy
Volume: 18
ISSN (Print): 1095-4244
Ratings:
BFI (2015): BFI-level 2
Scopus rating (2015): CiteScore 3.06 SJR 1.201 SNIP 2.231
Web of Science (2015): Impact factor 2.891
Web of Science (2015): Indexed yes
Original language: English
Keywords: MEXICO rotor, EllipSys3D, ACL method, CFD, MexNext
DOI: 10.1002/we.1864

Research output: Contribution to journal › Article in proceedings – Annual report year: 2015 › Research › peer-review

Fully Consistent SIMPLE-Like Algorithms on Collocated Grids

To increase the convergence rate of SIMPLE-like algorithms on collocated grids, a compatibility condition between mass flux interpolation methods and SIMPLE-like algorithms is presented. Results of unsteady flow computations show that the SIMPLECT algorithm, when obeying the compatibility condition, may obtain up to 35% higher convergence rate as compared to the standard SIMPLECT algorithm. Two new interpolation methods, fully compatible with the SIMPLECT algorithm, are presented and compared with some existing interpolation methods, including the standard methods of Choi [9] and Shen et al. [8]. Numerical results show that the time-step dependence of the standard methods may double the total discretization error at steady state. It is furthermore shown that the new methods are independent of time step and relaxation parameter at convergence. One of the new methods is shown to give a higher accuracy than the standard methods.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Aeroelastic Design
Contributors: Kolmogorov, D., Shen, W. Z., Sørensen, N. N., Sørensen, J. N.
Pages: 101–123
Publication date: 2015
Peer-reviewed: Yes

Publication information
Volume: 67
ISSN (Print): 1040-7790
Ratings:
BFI (2015): BFI-level 1
Scopus rating (2015): CiteScore 1.4 SJR 0.775 SNIP 1.151
Web of Science (2015): Impact factor 1.33
Web of Science (2015): Indexed yes
Original language: English
DOI: 10.1080/10407790.2014.949583

Research output: Contribution to journal › Article – Annual report year: 2015 › Research › peer-review
Modelling Wind for Wind Farm Layout Optimization Using Joint Distribution of Wind Speed and Wind Direction

Reliable wind modelling is of crucial importance for wind farm development. The common practice of using sector-wise Weibull distributions has been found inappropriate for wind farm layout optimization. In this study, we propose a simple and easily implementable method to construct joint distributions of wind speed and wind direction, which is based on the parameters of sector-wise Weibull distributions and interpolations between direction sectors. It is applied to the wind measurement data at Horns Rev and three different joint distributions are obtained, which all fit the measurement data quite well in terms of the coefficient of determination $R^2$. Then, the best of these joint distributions is used in the layout optimization of the Horns Rev 1 wind farm and the choice of bin sizes for wind speed and wind direction is also investigated. It is found that the choice of bin size for wind direction is especially critical for layout optimization and the recommended choice of bin sizes for wind speed and wind direction is finally presented.

Numerical simulation of airfoil trailing edge serration noise

In the present work, numerical simulations are carried out for a low noise airfoil with and without serrated Trailing Edge. The Ffowcs Williams-Hawkings acoustic analogy is implemented into the in-house incompressible flow solver EllipSys3D. The instantaneous hydrodynamic pressure and velocity field are obtained using Large Eddy Simulation. To obtain the time history data of sound pressure, the flow quantities are integrated around the airfoil surface through the FW-H approach. The extended length of the serration is about 16.7% of the airfoil chord and the geometric angle of the serration is 28 degrees. The chord based Reynolds number is around 1.5x10^6. Simulations are compared with existing wind tunnel experiments at various angles of attack. Even though the airfoil under investigation is already optimized for low noise emission, numerical simulations and wind tunnel experiments show that the noise level is further decreased by adding the TE serration device.
Numerical simulation of the aerodynamic field in complex terrain wind farm based on actuator disk model

Study on the aerodynamic field in complex terrain is significant to wind farm micro-siting and wind power prediction. This paper modeled the wind turbine through an actuator disk model, and solved the aerodynamic field by CFD to study the influence of meshing, boundary conditions and turbulence model on the calculation results. Comparison with the measured data of a wind farm was applied to find an appropriate method for simulating the aerodynamic field in the complex terrain wind farm. Related research can provide reference for wind farm micro-siting and wind power prediction.

Solving the wind farm layout optimization problem using random search algorithm

Wind farm (WF) layout optimization is to find the optimal positions of wind turbines (WTs) inside a WF, so as to maximize and/or minimize a single objective or multiple objectives, while satisfying certain constraints. In this work, a random search (RS) algorithm based on continuous formulation is presented, which starts from an initial feasible layout and then improves the layout iteratively in the feasible solution space. It was first proposed in our previous study and improved in this study by adding some adaptive mechanisms. It can serve both as a refinement tool to improve an initial design by expert guesses or other optimization methods, and as an optimization tool to find the optimal layout of WF with a certain number of WTs. A new strategy to evaluate layouts is also used, which can largely save the computation cost. This method is first applied to a widely studied ideal test problem, in which better results than the genetic algorithm (GA) and the old version of the RS algorithm are obtained. Second it is applied to the Horns Rev 1 WF, and the optimized layouts obtain a higher power production than its original layout, both for the real scenario and for two constructed scenarios. In this application, it is also found that in order to get consistent and reliable optimization results, up to 360 or more sectors for wind direction have to be used. Finally, considering the inevitable inter-annual variations in the wind conditions, the robustness of the optimized layouts against wind condition changes is analyzed, and the optimized layouts consistently show better performance in power production than the original layout, despite of considerable variations in wind direction and speed. © 2015 Elsevier Ltd. All rights reserved.
Study of wind turbine wake modeling based on a modified actuator disk model and extended k-ε turbulence model

This paper presented an improved computational fluid dynamics (CFD) model for simulating a horizontal-axis wind turbine wake. The model used the actuator disk model to simplify the wind turbine effect on the aerodynamic field by adding an extra momentum source and an improved term to correct the underestimation issue of the wind speed deficit when applying the STD k-ε model. In addition, the model also introduced a radial distribution function to assess the non-uniform load on the actuator disk and a coefficient C4ε of the turbulent source. To validate the model, the wind turbines of Nibe `B' and Dawin 180/23 were checked by different wake models with multiple entrance velocities. Results show that the improved wake model has better prediction accuracy with experimental data and can be used for wind turbine wake calculation.

Validation of the actuator line method using near wake measurements of the MEXICO rotor

The purpose of the present work is to validate the capability of the actuator line method to compute vortex structures in the near wake behind the MEXICO experimental wind turbine rotor. In the MEXICO project/MexNext Annex, particle image velocimetry measurements have made it possible to determine the exact position of each tip vortex core in a plane parallel to the flow direction. Determining center positions of the vortex cores makes it possible to determine the trajectory of the tip vortices, and thus the wake expansion in space, for the analyzed tip speed ratios. The corresponding cases, in terms of tip speed ratios, have been simulated by large-eddy simulations using a Navier–Stokes code combined with the actuator line method. The flow field is analyzed in terms of wake expansion, vortex core radius, circulation and axial and radial velocity distributions. Generally, the actuator line method generates significantly larger vortex cores than in the experimental cases, but predicts the expansion, the circulation and the velocity distributions with satisfying results. Additionally, the simulation and experimental data are used to test three different techniques to compute the average axial induction in the wake flow. These techniques are based on the helical pitch of the tip vortex structure, 1D momentum theory and wake expansion combined with mass conservation. The results from the different methods vary quite much, especially at high values of .
Validation of the Actuator Line Model for Simulating Flows past Yawed Wind Turbine Rotors
The Actuator Line/Navier-Stokes model is validated against wind tunnel measurements for flows past the yawed MEXICO rotor and past the yawed NREL Phase VI rotor. The MEXICO rotor is operated at a rotational speed of 424 rpm, a pitch angle of −2.3˚, wind speeds of 10, 15, 24 m/s and yaw angles of 15˚, 30˚ and 45˚. The computed loads as well as the velocity field behind the yawed MEXICO rotor are compared to the detailed pressure and PIV measurements which were carried out in the EU funded MEXICO project. For the NREL Phase VI rotor, computations were carried out at a rotational speed of 90.2 rpm, a pitch angle of 3˚, a wind speed of 5 m/s and yaw angles of 10˚ and 30˚. The computed loads are compared to the loads measured from pressure measurement.

Aerodynamic Analysis of Trailing Edge Enlarged Wind Turbine Airfoils
The aerodynamic performance of blunt trailing edge airfoils generated from the DU- 91-W2-250, DU-97-W-300 and DU-96-W-350 airfoils by enlarging the thickness of trailing edge symmetrically from the location of maximum thickness to chord to the trailing edge were analyzed by using CFD and RFOIL methods at a chord Reynolds number of 3 × 106. The goal of this study is to analyze the aerodynamic performance of blunt trailing edge airfoils with different thicknesses of trailing edge and maximum thicknesses to chord. The steady results calculated by the fully turbulent k-ω SST, transitional k-ω SST model and RFOIL all show that with the increase of thickness of trailing edge, the linear region of lift is extended and the maximum lift also increases, the increase rate and amount of lift become limited gradually at low angles of attack, while the drag increases dramatically. For thicker airfoils with larger maximum thickness to chord length, the increment of lift is larger than that of relatively thinner airfoils when the thickness of blunt trailing edge is increased from 5% to 10%
chord length. But too large lift can cause abrupt stall which is profitless for power output. The transient characteristics of blunt trailing edge airfoils are caused by blunt body vortices at low angles of attack, and by the combined effect of separation and blunt body vortices at large angles of attack. With the increase of thickness of blunt trailing edge, the vibration amplitudes of lift and drag curves increase. The transient calculations over-predict the lift at large angles of attack and drag at all angles of attack than the steady calculations which is likely to be caused by the artificial restriction of the flow in two dimensions.

**General information**
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Yangzhou University
Number of pages: 14
Publication date: 2014
Peer-reviewed: Yes

**Publication information**
Journal: Journal of Physics: Conference Series (Online)
Volume: 524
Issue number: 1
Article number: 012010
ISSN (Print): 1742-6596
Ratings:
BFI (2014): BFI-level 1
Scopus rating (2014): CiteScore 0.32 SJR 0.264 SNIP 0.361
Web of Science (2014): Indexed yes
Original language: English
Electronic versions: Aerodynamic_Analysis_of_Trailing_Edge.pdf
DOIs:
10.1088/1742-6596/524/1/012010

**Bibliographical note**
Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Published under licence by IOP Publishing Ltd
Source: FindIt
Source-ID: 268315543
Research output: Contribution to journal › Conference article – Annual report year: 2014 › Research › peer-review

**A quasi-3D viscous-inviscid interaction code: Q3UIC**
A computational model for predicting the aerodynamic behavior of wind turbine airfoils under rotation and subjected to steady and unsteady motions developed in [1] is presented herein. The model is based on a viscous-inviscid interaction technique using strong coupling between the viscous and inviscid parts. The rotational effects generated by centrifugal and Coriolis forces are introduced in Q3UIC via the streamwise and spanwise integral boundary layer momentum equations. A special inviscid version of the code has been developed to cope with massive separation. To check the ability of the code wind turbine airfoils in steady and unsteady conditions for a large range of angles of attack are considered here. Further, the new quasi-3D code Q3UIC is used to perform a parametric study of a wind turbine airfoil under rotation confined to its boundary layer.

**General information**
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Contributors: Ramos García, N., Sørensen, J. N., Shen, W. Z.
Number of pages: 12
Publication date: 2014
Peer-reviewed: Yes

**Publication information**
Journal: Journal of Physics: Conference Series (Online)
Volume: 555
Article number: 012041
ISSN (Print): 1742-6596
Ratings:
BFI (2014): BFI-level 1
A strong viscous–inviscid interaction model for rotating airfoils

Two-dimensional (2D) and quasi-three dimensional (3D), steady and unsteady, viscous–inviscid interactive codes capable of predicting the aerodynamic behavior of wind turbine airfoils are presented. The model is based on a viscous–inviscid interaction technique using strong coupling between the viscous and inviscid parts. The inviscid part is modeled by a 2D panel method, and the viscous part is modeled by solving the integral form of the laminar and turbulent boundary-layer equations with extension for 3D rotational effects. Laminar-to-turbulent transition is either forced by employing a boundary-layer trip or computed using an envelope transition method. Validation of the incompressible 2D version of the code is carried out against measurements and other numerical codes for different airfoil geometries at various Reynolds numbers, ranging from 0.9 · 10⁶ to 8.2 · 10⁶. In the quasi-3D version, a parametric study on rotational effects induced by the Coriolis and centrifugal forces in the boundary-layer equations shows that the effects of rotation are to decrease the growth of the boundary-layer and delay the onset of separation, hence increasing the lift coefficient slightly while decreasing the drag coefficient. Copyright © 2013 John Wiley & Sons, Ltd.
Design and validation of the high performance and low noise CQU-DTU-LN1 airfoils

This paper presents the design and validation of the high performance and low noise Chong Qing University and Technical University of Denmark LN1 (CQU-DTU-LN1) series of airfoils for wind turbine applications. The new design method uses target characteristics of wind turbine airfoils in the design objective, such as airfoil lift coefficient, drag coefficient and lift-drag ratio, and minimizes trailing edge noise as a constraint. To express airfoil shape, an analytical expression is used. One of the main advantages of the present design method is that it produces a highly smooth airfoil shape that can avoid the problem of curvature discontinuity. An airfoil profile with discontinuous curvature can produce a discontinuous pressure gradient (i.e., local flow acceleration or deceleration), which enhances flow separation and thus decreases the airfoil performance. By combining the design method with the blade element momentum theory, the viscous-inviscid XFOIL code and an airfoil self-noise prediction model, an optimization algorithm has been developed for designing the high performance and low noise CQU-DTU-LN1 series of airfoils with targets of maximum power coefficient and low noise emission. To validate the airfoil design, CQU-DTU-LN118 airfoil has been tested experimentally in the acoustic wind tunnel located at the Virginia Polytechnic Institute and State University (Virginia Tech), USA. To show the superiority of the CQU-DTU-LN1 airfoils, comparisons on aerodynamic performance and noise emission between the CQU-DTU-LN118 airfoil and the National Advisory Committee for Aeronautics (NACA) 64618 airfoil, which is used in modern wind turbine blades, are carried out. Copyright © 2013 John Wiley & Sons, Ltd.

Design of low noise wind turbine blades using Betz and Joukowski concepts

This paper presents the aerodynamic design of low noise wind turbine blades using Betz and Joukowski concepts. The aerodynamic model is based on Blade Element Momentum theory whereas the aeroacoustic prediction model is based on the BPM model. The investigation is started with a 3MW baseline/reference turbine rotor with a diameter of 80 m. To reduce the noise emission from the baseline rotor, the rotor is reconstructed with the low noise CQU-DTU-LN1 series of airfoils which has been tested in the acoustic wind tunnel located at Virginia Tech. Finally, 3MW low noise turbine rotors are designed using the concepts of Betz and Joukowski, and the CQU-DTU-LN1 series of airfoils. Performance analysis shows that the newly designed turbine rotors can achieve an overall noise reduction of 6 dB and 1.5 dB(A) with a similar power output as compared to the reference rotor.
Fully consistent CFD methods for incompressible flow computations

Nowadays collocated grid based CFD methods are one of the most efficient tools for computations of the flows past wind turbines. To ensure the robustness of the methods they require special attention to the well-known problem of pressure-velocity coupling. Many commercial codes to ensure the pressure-velocity coupling on collocated grids use the so-called momentum interpolation method of Rhie and Chow [1]. As known, the method and some of its widely spread modifications result in solutions, which are dependent of time step at convergence. In this paper the magnitude of the dependence is shown to contribute about 0.5% into the total error in a typical turbulent ow computation. Nevertheless if coarse grids are
used, the standard interpolation methods result in much higher non-consistent behavior. To overcome the problem, a recently developed interpolation method, which is independent of time step, is used. It is shown that in comparison to other time step independent method, the method may enhance the convergence rate of the SIMPLEC algorithm up to 25%. The method is verified using turbulent flow computations around a NACA 64618 airfoil and the roll-up of a shear layer, which may appear in wind turbine wake.

Hybrid wake model for free vortex viscous-inviscid simulations

Integrated airfoil and blade design method for large wind turbines

This paper presents an integrated method for designing airfoil families of large wind turbine blades. For a given rotor diameter and a tip speed ratio, optimal airfoils are designed based on the local speed ratios. To achieve a high power performance at low cost, the airfoils are designed with the objectives of high Cp and small chord length. When the airfoils are obtained, the optimum flow angle and rotor solidity are calculated which forms the basic input to the blade design. The new airfoils are designed based on a previous in-house designed airfoil family which was optimized at a Reynolds number of 3 million. A novel shape perturbation function is introduced to optimize the geometry based on the existing airfoils which simplifies the design procedure. The viscous/inviscid interactive code XFOIL is used as the aerodynamic tool for airfoil optimization at a Reynolds number of 16 million and a free-stream Mach number of 0.25 near the tip. Results show that the new airfoils achieve a high power coefficient in a wide range of angles of attack (AOA) and are extremely insensitive to surface roughness. Finally, a full blade analysis using computational fluid dynamics (CFD) and blade element momentum (BEM) technique proves the reliability of the integrated design method. © 2014 Elsevier Ltd.
Investigation of modified AD/RANS models for wind turbine wake predictions in large wind farm

Average power losses due to multiple wind turbine wakes in the large offshore wind farm is studied in this paper using properly modified k-ω SST turbulence models. The numerical simulations are carried out by the actuator disc methodology implemented in the flow solver EllipSys3D. In these simulations, the influence of different inflow conditions such as wind direction sectors are considered and discussed. Comparisons with measurements in terms of wake speed ratio and the corresponding power outputs show that the modified turbulence models had significant improvements; especially the SST-Csust model reflects the best ability in predicting the wake defect. The investigations of various inflow angles reveal that the agreement between predicted and measured data is improved for the wider sector case than the narrow case because of the wind direction uncertainty.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Nanjing University of Aeronautics and Astronautics
Contributors: Tian, L. L., Zhu, W. J., Shen, W. Z., Sørensen, J. N., Zhao, N.
Number of pages: 10
Publication date: 2014
Peer-reviewed: Yes

Publication information
Journal: Journal of Physics: Conference Series (Online)
Volume: 524
Issue number: 1
ISSN (Print): 1742-6596
Ratings:
BFI (2014): BFI-level 1
Scopus rating (2014): CiteScore 0.32 SJR 0.264 SNIP 0.361
Web of Science (2014): Indexed yes
Original language: English
Electronic versions:
Investigation_of_modified_AD_RANS.pdf
DOIs:
10.1088/1742-6596/524/1/012151

Bibliographical note
Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
Numerical Investigation of Flow Control Feasibility with a Trailing Edge Flap

This paper concerns a numerical study of employing an adaptive trailing edge flap to control the lift of an airfoil subject to unsteady inflow conditions. The periodically varying inflow is generated by two oscillating airfoils, which are located upstream of the controlled airfoil. To establish the control system, a standard PID controller is implemented in a finite volume based incompressible flow solver. An immersed boundary method is applied to treat the problem of simulating a deformable airfoil trailing edge. The flow field is solved using a 2D Reynolds averaged Navier-Stokes finite volume solver. In order to more accurately simulate wall bounded flows around the immersed boundary, a modified boundary condition is introduced in the k-ω turbulence model. As an example, turbulent flow over a NACA 64418 airfoil with a deformable trailing edge is investigated. Results from numerical simulations are convincing and may give some highlights for practical implementations of trailing edge flap to a wind turbine rotor blade.
Prediction of the wind turbine performance by using BEM with airfoil data extracted from CFD

Blade element momentum (BEM) theory with airfoil data is a widely used technique for prediction of wind turbine aerodynamic performance, but the reliability of the airfoil data is an important factor for the prediction accuracy of aerodynamic loads and power. The airfoil characteristics used in BEM codes are mostly based on 2D wind tunnel measurements of airfoils with constant span. Due to 3D effects, a BEM code using airfoil data obtained directly from 2D wind tunnel measurements will not yield the correct loading and power. As a consequence, 2D airfoil characteristics have to be corrected before they can be used in a BEM code. In this article, we consider the MEXICO (Model Experiments In Controlled cOntions) rotor where airfoil data are extracted from CFD (Computational Fluid Dynamics) results. The azimuthally averaged velocity is used as the sectional velocity to define the angle of attack and the coefficient of lift and drag is determined by the forces on the blade. The extracted airfoil data are put into a BEM code without further corrections, and the calculated axial and tangential forces are compared to both computations using BEM with Shen's tip loss correction model and experimental data. The comparisons show that the recalculated forces by using airfoil data extracted from CFD have good agreements with the experiment. © 2014 Elsevier Ltd.
Simulations of the Yawed MEXICO Rotor Using a Viscous-Inviscid Panel Method

In the present work the viscous-inviscid interactive model MIRAS is used to simulate flows past the MEXICO rotor in yawed conditions. The solver is based on an unsteady three-dimensional free wake panel method which uses a strong viscous-inviscid interaction technique to account for the viscous effects inside the boundary layer. Calculated wake velocities have been benchmarked against field PIV measurements, while computed blade aerodynamic characteristics are compared against the load calculated from pressure measurements at different locations along the blade span. Predicted and measured aerodynamic forces are in overall good agreement, however discrepancies appear in the root region which could be related to an underestimation of the rotational effects arising from Coriolis and centrifugal forces. The predicted wake velocities are generally in good agreement with measurements along the radial as well as the axial traverses performed during the experimental campaign.

Study of tip loss corrections using CFD rotor computations

Tip loss correction is known to play an important role for engineering prediction of wind turbine performance. There are two different types of tip loss corrections: tip corrections on momentum theory and tip corrections on airfoil data. In this paper, we study the latter using detailed CFD computations for wind turbines with sharp tip. Using the technique of determination of angle of attack and the CFD results for a NordTank 500 kW rotor, airfoil data are extracted and a new tip loss function on airfoil data is derived. To validate, BEM computations with the new tip loss function are carried out and compared with CFD results for the NordTank 500 kW turbine and the NREL 5 MW turbine. Comparisons show that BEM with the new tip loss function can predict correctly the loading near the blade tip.
Validation of a three-dimensional viscous-inviscid interactive solver for wind turbine rotors

MIRAS is a newly developed computational model that predicts the aerodynamic behavior of wind turbine blades and wakes subject to unsteady motions and viscous effects. The model is based on a three-dimensional panel method using a surface distribution of quadrilateral singularities with a Neumann no penetration condition. Viscous effects inside the boundary layer are taken into account through the coupling with the quasi-3D integral boundary layer solver Q3UIC. A free-wake model is employed to simulate the vorticity released by the blades in the wake. In this paper the new code is validated against measurements and/or CFD simulations for five wind turbine rotors, including three experimental model rotors [20-22], the 2.5 MW NM80 machine [23] and the NREL 5 MW virtual rotor [24]. Such a broad set of operational conditions and rotor sizes constitutes a very challenging validation matrix, with Reynolds numbers ranging from 5.0·10^4 to 1.2·10^7.

General information
Publication status: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Department of Wind Energy
Contributors: Ramos García, N., Sørensen, J. N., Shen, W. Z.
Pages: 78-92
Publication date: 2014
Peer-reviewed: Yes

Publication information
Journal: Renewable Energy
Volume: 70
ISSN (Print): 0960-1481
Ratings:
BFI (2014): BFI-level 1
Scopus rating (2014): CiteScore 4.51 SJR 1.925 SNIP 2.588
Web of Science (2014): Indexed yes
Original language: English
Keywords: Wind turbine, Panel method, Free wake, Viscous-inviscid interaction, Integral boundary layer equations
DOI: 10.1016/j.renene.2014.04.001

Bibliographical note
Special issue on aerodynamics of offshore wind energy systems and wakes
Research output: Contribution to journal › Journal article – Annual report year: 2014 › Research › peer-review

Validation of the CQU-DTU-LN1 series of airfoils

The CQU-DTU-LN1 series of airfoils were designed with an objective of high lift and low noise emission. In the design process, the aerodynamic performance is obtained using XFOIL while noise emission is obtained with the BPM model. In this paper we present some validations of the designed CQU-DTU-LN118 airfoil by using wind tunnel measurements in the acoustic wind tunnel located at Virginia Tech and numerical computations with the inhouse Q3uic and EllipSys 2D/3D codes. To show the superiority of the new airfoils, comparisons with a NACA64618 airfoil are made. For the aerodynamic features, the designed Cl and Cl/Cd agrees well with the experiment and are in general higher than those of the NACA airfoil. For the acoustic features, the noise emission of the LN118 airfoil is compared with the acoustic measurements and that of the NACA airfoil. Comparisons show that the BPM model can predict correctly the noise changes.
Wind farm layout optimization in complex terrain: A preliminary study on a Gaussian hill

One of the crucial problems for wind farm (WF) development is wind farm layout optimization. It seeks to find the optimal positions of wind turbines (WTs) inside a WF, so as to maximize and/or minimize a single objective or multiple objectives, while satisfying certain constraints. Although this problem for WFs in flat terrain or offshore has been investigated in many studies, it is still a challenging problem for WFs in complex terrain. In this preliminary study, the wind flow conditions of complex terrain without WTs are first obtained from computational fluid dynamics (CFD) simulation, then an adapted Jensen wake model is developed by considering the terrain features and taking the inflow conditions as input. Using this combined method, the wake effects of WF in complex terrain are properly modelled. Besides, a random search (RS) algorithm proposed in previous study is improved by adding some adaptive mechanisms and applied to solve the layout optimization problem of a WF on a Gaussian shape hill. The layout of the WF with a certain number of WTs is optimized to maximize the total power output, which obtained steady improvements over expert guess layouts.
Accurate wind turbine aero-acoustics by high-order schemes

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Contributors: Zhu, W. J., Shen, W. Z., Sørensen, J. N.
Publication date: 2013
Media of output: PowerPoint

Event information
Event: Danish Wind Power Research 2013
Location: Trinity, Fredericia, Denmark
Electronic versions:
Accurate_Wind_Turbine_Aero_acoustics_Wei_Jun_Zhu.pdf

Advanced Time Approach of FW-H Equations for Predicting Noise
An advanced time approach of Ffowcs Williams-Hawkings (FW-H) acoustic analogy is developed, and the integral equations and integral solution of FW-H acoustic analogy are derived. Compared with the retarded time approach, the transcendental equation need not to be solved in the advanced time approach, on the other hand, computational cost can be saved using the approach due to no demand of pre-storing lots of aerodynamic data. To further validate the efficiency of the advanced time approach for predicting noise, unsteady flow fields are firstly simulated for air around square cylinder and NACA0012 airfoil, then unsteady calculations are used as input for FW-H equations, and numerical predictions are made for noise induced by vortex shedding of square cylinder and NACA0012 airfoil using the advanced time approach. Finally, the retarded time approach and the advanced time approach are compared.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Nanjing University of Aeronautics and Astronautics, China Aerodynamics Research and Development Center
Contributors: Haiqing, S., Yan, S., Shen, W. Z., Xiaojun, W.
Number of pages: 6
Pages: 807-812
Publication date: 2013
Peer-reviewed: Yes

Publication information
Journal: Nanjing Hangkong Hangtian Daxue Xuebao
Volume: 45
Issue number: 6
ISSN (Print): 1005-2615
Ratings:
Scopus rating (2013): CiteScore 0.3 SJR 0.292 SNIP 0.615
ISI indexed (2013): ISI indexed no
Original language: English
Keywords: INTEGRAL equations, FW-H equations, Acoustic analogy method, Retarded time approach, Advanced time approach
Source: FindIt
Source-ID: 267038276

Analysis of turbulent wake behind a wind turbine
The aim of this study is to improve the classical analytical model for estimation of the rate of wake expansion and the decay of wake velocity deficit in the far wake region behind a wind turbine. The relations for a fully turbulent axisymmetric far wake were derived by applying the mass and momentum conservations, the selfsimilarity of mean velocity profile and the eddy viscosity closure. The theoretical approach is validated using the numerical results obtained from large eddy simulations with an actuator line technique at 0.1% and 3% ambient turbulence level and ambient wind velocity of 10 m/s, and 0.1% ambient turbulence level and ambient wind velocity of 7 m/s. The obtained results showed that neglecting the nonlinear term of velocity in the momentum equation in the far wake region cannot be a fair assumption, unlike what is
generally assumed in most of text books of fluid mechanics. Therefore the theoretical determination of the power law for the wake expansion and the decay of the wake velocity deficit may not be valid in the case of the wake generated behind a wind turbine with low ambient turbulence and high thrust coefficient. Although at higher ambient turbulence levels or lower ambient wind velocities (higher thrust coefficients), this trend may be improved due to the faster recovery of the wake and therefore closer values to the theoretical approach may be obtained. In addition, the assumption of self-similarity behavior of the mean velocity profile, when scaled with center line velocity deficit, could be correct in the far wake region of a wind turbine and low ambient turbulence levels.

General information
Publication status: Published
Organisations: Department of Mechanical Engineering, Thermal Energy, Department of Wind Energy, Fluid Mechanics
Contributors: Kermani, N. A., Andersen, S. J., Sørensen, J. N., Shen, W. Z.
Number of pages: 16
Publication date: 2013

Host publication information
Title of host publication: Proceedings of the 2013 International Conference on aerodynamics of Offshore Wind Energy Systems and wakes (ICOWES2013)
Publisher: Technical University of Denmark (DTU)
Editor: Shen, W.
Electronic versions:
Analysis_of_turbulent_wake_behind.pdf

A Research on Wind Farm Micro-sitting Optimization in Complex Terrain
Wind farm layout optimization in complex terrain is a pretty difficult issue for onshore wind farm. In this article, a novel optimization method is proposed to optimize the layout for wind farms in complex terrain. This method utilized Lissaman and Jensen wake models for taking the terrain height and the wake loss from the upstream turbines into the wind turbine power output calculation. Wind direction is divided into sixteen sections, and the wind speed is processed using the Weibull distribution. The objective is to maximize the total wind farm power output and the free design variables are the wind turbines’ park coordinates which subject to the boundary and minimum distance conditions between two wind turbines. A Cross Particle Swarm Optimization (CPSO) method is developed and applied to optimize the layout for a certain wind farm case. Compared with the uniform and experience method, results show that the CPSO method has a higher optimal value, and could be used to optimize the actual wind farm micro-sitting engineering projects.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Hohai University
Contributors: Xu, C., Yang, J., Li, C., Shen, W. Z., Zheng, Y., Liu, D.
Pages: 669-679
Publication date: 2013

Host publication information
Title of host publication: Proceedings of the 2013 International Conference on aerodynamics of Offshore Wind Energy Systems and wakes (ICOWES2013)
Publisher: Technical University of Denmark (DTU)
Editor: Shen, W.
Keywords: Wind farm micro-sitting, Complex terrain, Wake model, Probability density, Cross particle swarm optimization (CPSO)

Development of an AD/RANS model for predicting wind turbine wakes
The wake flow behind a single wind turbine is studied in this paper using the standard k-ω SST turbulence model with three different modifications: (1) adding sustain terms to maintain ambient turbulence level; (2) modifying the coefficients of the turbulence model; (3) combining the first two methods together. The simulations are performed by using the in-house flow solver EllipSys3D coupled with the actuator disc (AD) methodology. The main objective of the proposed model is to control the turbulence decay caused by the intrinsic property of the two-equation turbulence model and further increasing the predicted turbulence intensity through correcting the destruction terms. To validate the proposed method, comparisons are carried out between measurements and other numerical simulations for the Nibe B wind turbine.

Results show that the newly developed model can provide satisfactory predictions for the wind speed deficit and the turbulence intensity.
Development of a Three-Dimensional Viscous-Inviscid coupling Method for Wind Turbine Computations

MIRAS, a computational model for predicting the aerodynamic behavior of wind turbine blades and wakes subject to unsteady motions and viscous effects has been developed. The model is based on a three-dimensional panel method using a surface distribution of quadrilateral singularities with a Neumann no penetration condition. Viscous effects inside the boundary layer are taken into account through the coupling with the quasi-3D integral boundary layer solver Q^3UIC. A free-wake model is employed to simulate the vorticity released by the blades in the wake. In this paper simulations are presented in an effort to validate the code for three different rotor geometries, the MEXICO experiment rotor, the DELFT rotor and the NREL 5MW rotor.

Effect of non-uniform mean flow field on acoustic propagation problems in computational aeroacoustics

Acoustic propagation in the presence of a non-uniform mean flow is studied numerically by using two different acoustic propagating models, which solve linearized Euler equations (LEE) and acoustic perturbation equations (APE). As noise induced by turbulent flows often propagates from near field to far field in a non-quiescent medium, the effect of non-uniform mean flows on the propagation of acoustic waves is important to be considered. In this paper, 4 different non-uniform mean flows are considered. Results show that the non-uniform mean flows can modify both the amplitude and shape of the acoustic waves as compared to the uniform mean flow. From the numerical point of view, the APE model can predict almost the physical behaviors with a difference in amplitude which is dependent on location, as compared to LEE. On the other hand, APE solves one equation less and thus is computationally less expansive.
Hybrid immersed boundary method for airfoils with a trailing-edge flap

In this paper, a hybrid immersed boundary technique has been developed for simulating turbulent flows past airfoils with moving trailing-edge flaps. Over the main fixed part of the airfoil, the equations are solved using a standard body-fitted finite volume technique, whereas the moving trailing-edge flap is simulated using the immersed boundary method on a curvilinear mesh. An existing in-house-developed flow solver is employed to solve the incompressible Reynolds-Averaged Navier-Stokes equations together with the k-ω turbulence model. To achieve consistent wall boundary conditions at the immersed boundaries the k-ω turbulence model is modified and adapted to the local conditions associated with the immersed boundary method. The obtained results show that the hybrid approach is an efficient and accurate method for solving turbulent flows past airfoils with a trailing-edge flap and that flow control using an adjustable trailing-edge flap is an efficient way to regulate the aerodynamic loading on airfoils. Copyright © 2012 by the American Institute of Aeronautics and Astronautics, Inc. All rights reserved.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Contributors: Zhu, W. J., Behrens, T., Shen, W. Z., Sørensen, J. N.
Pages: 30-41
Publication date: 2013
Peer-reviewed: Yes

Integrated airfoil and blade design method for large wind turbines

This paper presents an integrated method for designing airfoil families of large wind turbine blades. For a given rotor diameter and tip speed ratio, the optimal airfoils are designed based on the local speed ratios. To achieve high power performance at low cost, the airfoils are designed with an objective of high Cp and small chord length. When the airfoils are obtained, the optimum flow angle and rotor solidity are calculated which forms the basic input to the blade design. The
new airfoils are designed based on the previous in-house airfoil family which were optimized at a Reynolds number of 3 million. A novel shape perturbation function is introduced to optimize the geometry on the existing airfoils and thus simplify the design procedure. The viscous/inviscid code Xfoil is used as the aerodynamic tool for airfoil optimization where the Reynolds number is set at 16 million with a free-stream Mach number of 0.25 at the blade tip. Results show that these new airfoils achieve high power coefficient in a wide range of angles of attack (AOA) and they are extremely insensitive to surface roughness.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Contributors: Zhu, W. J., Shen, W. Z.
Publication date: 2013

Host publication information
Title of host publication: Proceedings of the 2013 International Conference on aerodynamics of Offshore Wind Energy Systems and wakes (ICOWES2013)
Publisher: Technical University of Denmark (DTU)
Editor: Shen, W.
Electronic versions:
Integrated_airfoil_and_blade_design.pdf
Research output: Chapter in Book/Report/Conference proceeding - Article in proceedings – Annual report year: 2013 – peer-review

Multigrid technique and Optimized Schwarz method on block-structured grids with discontinuous interfaces
An Optimized Schwarz method using Robin boundary conditions for relaxation scheme is presented in the frame of Multigrid method on discontinuous grids. At each iteration the relaxation scheme is performed in two steps: one step with Dirichlet and another step with Robin boundary conditions at inner block boundaries. A Robin parameter that depends on grid geometry and grid discontinuity at block interfaces is introduced. The general solution algorithm is based on SIMPLE method and a conservative finite-volume scheme on block-structured grids with discontinuous interfaces. The multigrid method is used to obtain the solution of pressure-correction equation where an Incomplete Block LU factorization is used as the relaxation scheme. Solution on the coarsest grid is done with an Incomplete Block LU preconditioned Conjugate Gradient method. Results from computations of laminar ows around a circular cylinder on grids with nonmatching interfaces are presented.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Aeroelastic Design
Contributors: Kolmogorov, D., Sørensen, N. N., Shen, W. Z., Sørensen, J. N.
Number of pages: 15
Publication date: 2013

Host publication information
Publisher: American Institute of Aeronautics and Astronautics
DOI: 10.2514/6.2013-2578
Source: dtu
Source-ID: u::8063
Research output: Chapter in Book/Report/Conference proceeding - Article in proceedings – Annual report year: 2013 – peer-review

Optimization of Wind Farm Layout: A Refinement Method by Random Search
Wind farm layout optimization is to find the optimal positions of wind turbines inside a wind farm, so as to maximize and/or minimize a single objective or multiple objectives, while satisfying certain constraints. Most of the works in the literature divide the wind farm into cells in which turbines can be placed, hence, simplifying the problem from with continuous variables to with discrete variables. In this paper, a refinement method, based on continuous formulation, by using random search is proposed to improve the optimization results based on discrete formulations. Two sets of optimization results of a widely studied test case are refined using the proposed method. One set of the results is from a published work using GA based on discrete formulation, the other set is the improved results using authors’ own GA code. Steady improvements are obtained for both sets of results.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Optimization of Wind Farm Layout in Complex Terrain

Microscopic site selection for wind farms in complex terrain is a technological difficulty in the development of onshore wind farms. This paper presented a method for optimizing wind farm layout in complex terrain. This method employed Lissaman and Jensen wake models, took wind velocity distribution law and wake loss between different turbines into consideration and calculated the sheltering area effect of wake loss from upstream wind turbines on downstream wind turbines. Wind direction was divided into sixteen sections, and the wind speed was processed by the Weibull distribution. To calculate the output of each section, we used the wind speed distribution and its probability density as well as the wake loss between wind turbines for every section. The objective function is maximization of the whole wind farm's power output and the free variables are the wind turbines' coordinates which are subject to boundary conditions and minimum distance conditions. The improved genetic algorithm (GA) for real number coding was used to search the optimal result. Then the optimized result was compared to the result from the experienced layout method. Results show the advantages of the present method, and the limitations of the experienced method. © 2013 Chin. Soc. for Elec. Eng.
boundary condition and the outlet boundary condition. The boundaries of fan-shaped both sides are defined as rotationally periodic connection, and the freeze rotor model is applied at the interface of the rotating and stationary domains, which means the relative position of rotating and stationary domains is fixed when calculating the flow field. Speed no-slip conditions are applied to solid walls such as blades. In this paper, two different meshing methods are used to generate a hexahedral grid for the rotating domain and a tetrahedral grid for stationary domain, between which comparison of the deviation of axial force on 60% blade cross section under the design condition (\(V_{tun}=15\) m/s) leads to a clear decision of the better mesh method with less deviation. Taking the better mesh method into consideration, the final number of rotating domain grids is calculated according to verification of grid independence, with an amount of 2, 961, 385. The conclusion of this paper will be illustrated from the following points: first, the comparison of the calculated and the experimental angle of attack distribution along the span direction shows that the maximum relative errors of the attack angle calculated by BEM and CFD respectively are -0.402 and 0.099; it further illustrates that the experimental results are substantially between the results obtained by the two methods, and closer to the result of CFD at the blade tip. Meanwhile, the axial force on the blade increases with increasing radius, while the tangential force shows small change. All of the axial and tangential force in each section increases with increasing wind speed. Additionally, the maximum relative errors of axial force calculated by BEM and CFD respectively are -0.139 and -0.096. In a word, the experimental data are in good agreement with the results calculated by BEM and CFD, confirming the reliability of the MEXICO data. Second, the SST turbulence model can better capture the flow separation on the blade and has high aerodynamic performance prediction accuracy for a horizontal axis wind turbine in axial inflow conditions. Finally, the comparisons of the axial and tangential forces as well as the contrast of the angle of attack indicate that the prediction accuracy of BEM method is high when the blade is not in the stall condition. However, the airfoil characteristic becomes unstable in the stall condition, and the maximum relative error of tangential force calculated by BEM is -0.471. As a result, prediction accuracy of the BEM method needs to be further improved.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Contributors: Hong, Z., Yang, H., Xu, H., Shen, W. Z.
Pages: 67-74
Publication date: 2013
Peer-reviewed: Yes

Publication information
Journal: Nongye Gongcheng Xuebao
Volume: 29
Issue number: 18
ISSN (Print): 1002-6819
Ratings: Scopus rating (2013): CiteScore 0.98 SJR 0.334 SNIP 0.997
ISI indexed (2013): ISI indexed no
Original language: English
Keywords: Aerodynamic loads, Angle of attack, Axial flow, Boundary conditions, Boundary element method, Electrooxidation, Errors, Fans, Forecasting, MATLAB, Oceanography, Wind effects, Wind turbines, Computational fluid dynamics
DOIs: 10.3969/j.issn.1002-6819.2013.18.009

Bibliographical note
Transactions of the Chinese Society of Agricultural Engineering
Source: dtu
Source-ID: n:oai:DTIC-ART:compendex/409604859::33096
Research output: Contribution to journal › Journal article – Annual report year: 2013 › Research › peer-review

Prediction of the aerodynamic performance of the Mexico rotor by using airfoil data extracted from CFD
Blade Element Momentum (BEM) theory is a widely used technique for prediction of wind turbine aerodynamics performance, but the reliability of airfoil data is an important factor to improve the prediction accuracy of aerodynamic loads and power using a BEM code. The airfoil characteristics used in BEM codes are mostly based on 2D wind tunnel measurements of airfoils with constant span. However, a BEM code using airfoil data obtained directly from 2D wind tunnel measurements will not yield the correct loading and power. As a consequence, 2D airfoil characteristics have to be corrected by using some models before they can be used in a BEM code. In this article, the airfoil data for the MEXICO (Model EXperiments in Controlled cOnditions) rotor are extracted from CFD (Computational Fluid Dynamics) results. The azimuthally averaged velocity is used as the sectional velocity to define the angle of attack and the coefficient of lift and drag is determined by the forces on the blade. The extracted airfoil data are put into a BEM code without corrections of rotational or tip effects, and the calculated axial and tangential forces are compared to both computations using BEM with Shen’s tip loss correction models and experimental data. The comparisons show that the present method of determination of angle of attack is correct, and the re-calculated forces have good agreements with the experiment.
Prediction of wind energy distribution in complex terrain using CFD

Based on linear models, WAsP software predicts wind energy distribution, with a good accuracy for flat terrain, but with a large error under complicated topography. In this paper, numerical simulations are carried out using the FLUENT software on a mesh generated by the GAMBIT and ARGIS software to predict wind speed distribution in complex terrain. TECPLOT software post-processing is used to get the whole wind flow field, the wind speed distribution characteristics and distribution of wind energy. The obtained results are compared with the results of WAsP software and are also more accordance with the actual conditions.

Structural optimization study of composite wind turbine blade

In this paper the initial layout of a 2 MW composite wind turbine blade is designed first. The new airfoils families are selected to design a 2 MW wind turbine blade. The finite element parametric model for the blade is established. Based on the modified Blade Element Momentum theory, a new one-way fluid-structure interaction method is introduced. A procedure combining finite element analysis and particle swarm algorithm to optimize composite structures of the wind turbine blade is developed. The procedure proposed not only allows thickness variation but also permits the spar cap location variation over the structure. The results show that, compared to the initial blade, the mass of the optimized blades is reduced and especially for the scheme II (the location of blade spar cap is seen as one of the variables) which exhibit more mass saving. This present study has important significance for the structural design and optimization of wind turbine blades. © 2012.
The aerodynamics of wind turbines

In the paper we present state-of-the-art of research in wind turbine aerodynamics. We start by giving a brief historical review and a survey over aerodynamic research in wind energy. Next, we focus on some recent research results obtained by our wind energy group at Department of Mechanical Engineering at DTU. In particular, we show some new results on the classical problem of the ideal rotor and present a series of new results from an on-going research project dealing with the modelling and simulation of turbulent flow structures in the wake behind wind turbines.

General information
Publication status: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics
Contributors: Sørensen, J. N., Mikkelsen, R. F., Troldborg, N., Okulov, V., Shen, W. Z.
Pages: 231-247
Publication date: 2013

Host publication information
Title of host publication: Mechanics Down Under - Proceedings of the 22nd International Congress of Theoretical and Applied Mechanics, ICTAM 2008
Publisher: Springer Science+Business Media
ISBN (Print): 9789400759671
Keywords: Aerodynamics, Mechanics, Wind power, Wind turbines, Research
DOI:
10.1007/978-94-007-5968-8_15
Source: dtu
Source-ID: n:oai:DTIC-ART:compendex/391984430::31869

Wind turbine pitch control using ICPSO-PID algorithm

For the traditional simplified first-order pitch-control system model, it is difficult to describe a real dynamic characteristic of a variable pitch action system, thus a complete high order mathematical model has to be developed for the pitch control of wind turbine generation (WTG). In the paper, a pitch controller was designed based on power and wind speed and by considering the inertia and delay characteristics of a pitch-control system to achieve a constant power output when a wind speed was beyond the rated one. A novel ICPSO-PID control algorithm was proposed based on a combination of improved cooperative particle swarm optimization (ICPSO) and PID, subsequently, it was used to tune the pitch controller parameters; thus the difficulty in PID tuning was removed when a wind speed was above the rated speed. It was indicated that the proposed optimization algorithm can tune the pitch controller parameters quickly; and the feed-forward controller for wind speed can improve dynamics of a pitch-control system; additionally the power controller can allow a wind turbine to have a constant power output as a wind speed is over the rated one. Compared with a conventional PID, the controller with ICPSO-PID algorithm has a smaller overshoot, a shorter tuning time and better robustness. The design method proposed in the paper can be applied in a practical electro-hydraulic pitch control system for WTG.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Hohai University, NanJing Mennekes Electric Appliances Co., Ltd.
Actuator line/Navier–Stokes computations for the MEXICO rotor: comparison with detailed measurements

In the European collaborative MEXICO (Model Experiments in Controlled Conditions) project, a series of experiments was carried out on a 4.5 m diameter wind turbine rotor to validate numerical diagnostics tools. Here, some of the measured data are compared with computations of the combined actuator line/Navier–Stokes (AL/NS) model developed at the Technical University of Denmark. The AL/NS model was combined with a large eddy simulation technique and used to compute the flow past the MEXICO rotor in free air and in the DNW German-Dutch wind tunnel for three commonly defined test cases at wind speeds of 10, 15 and 24 m s⁻¹. Two sets of airfoil data were used. Comparisons of blade loadings showed that the AL/NS technique with the modified airfoil data is in better agreement with the measurements than with the original 2D airfoil data. Comparisons of detailed near-wake velocities showed good agreement with the measurements. Computations including the influence of the geometry of the wind tunnel showed that tunnel effects are not significant and the effect of the geometry of the wind tunnel only results in a speedup of 3% at a thrust coefficient of CT = 1. Copyright © 2011 John Wiley & Sons, Ltd.
Analysis of Mexico wind tunnel measurements: Final report of IEA Task 29, Mexnext (Phase 1)
This report describes the work performed within the first phase of IEA Task 29 Mexnext. In this IEA Task 29 a total of 20 organisations from 11 different countries collaborated in analysing the measurements which have been taken in the EU project 'Mexico'. Within this Mexico project 9 European institutes carried out a wind tunnel experiment in the Large Low Speed Facility (LLF) of the German DutchWind Facilities DNW on a rotor with a diameter of 4.5 m. Pressure distributions were measured at five location along the blade along with detailed flow field measurements around the rotor plane using stereo PIV.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics, Aeroelastic Design, Energy Research Centre of the Netherlands, Korea Aerospace Research Institute, National Renewable Energy Center, Kiel University of Applied Sciences, University of Stuttgart, ForWind, National Renewable Energy Laboratory, Delft University of Technology
Number of pages: 10
Publication date: 2012

Comparisons between LES and Wind Tunnel Hot-Wire Measurements
Large-eddy simulations (LES) are carried out for flows over a NACA 0015 airfoil at AoA = 8° and a chord based Reynolds number of 1.71 × 10^6. To accurately simulate the complex flow on the suction side of the airfoil, a reasonably large number of grid points is required. The computational mesh is constructed in a wind tunnel similar to the LM wind tunnel where the experiment for an NACA 0015 airfoil was carried out. The goal of this study is to validate the mixed scale SGS turbulence model against detailed measurements. Simulations are performed with the in-house EllipSys3D code on high performance computers. The stability and accuracy of the LES simulations are studied on various mesh configurations. The spanwise grid spacing was found important to produce correct flow disturbances along the airfoil span, which can affect the turbulent energy distribution.

General information
Publication status: Published
Organisations: Department of Wind Energy, Fluid Mechanics
Contributors: Zhu, W. J., Shen, W. Z., Sørensen, J. N.
Pages: 21-29
Publication date: 2012

Host publication information
Title of host publication: EUDP Project: Low Noise Airfoil - Final Report
Publisher: Wind Energy Department, Technical University of Denmark
Keywords: DTU-Wind-Energy-E-0004(EN), DTU-Wind-Energy-Report-E-0004
Electronic versions:
DTU_Wind_Energy_E_0004.pdf
Research output: Chapter in Book/Report/Conference proceeding → Report chapter – Annual report year: 2012 → Research

Comparisons between LES and wind tunnel hot-wire measurements of a NACA 0015 airfoil
Large-eddy simulations (LES) are carried out for flow over a NACA 0015 airfoil at AoA = 8° and chord based Reynolds number of 1.71106. To accurately simulate the complex flow at the suction side of the airfoil, a reasonably large number of grid points are required. The computational mesh is constructed in a wind tunnel similar to the condition where the experiments were carried out. The goal of this study is to validate the LES model against detailed measurements. The simulations are performed with in-house EllipSys3D code on high performance computers. Numerical study are focused on the stability and accuracy of the LES simulations on various mesh configurations. The spanwise grid spacing was found important to produce correct flow disturbance along the airfoil span, which further affects the turbulent energy distribution.

General information
Publication status: Published
Control of variable speed pitch-regulated wind turbines in strong wind conditions using a combined feedforward and feedback technique

Due to the increasing penetration of wind energy into power systems, it becomes critical to reduce the impact of wind energy on the stability and reliability of the overall power system. In precedent works, Shen and his co-workers developed a re-designed operation schema to run wind turbines in strong wind conditions based on optimization method and standard PI feedback control, which can prevent the typical shutdowns of wind turbines when reaching the cut-out wind speed. In this paper, a new control strategy combing the standard PI feedback control with feedforward controls using the optimization results is investigated for the operation of variable-speed pitch-regulated wind turbines in strong wind conditions. It is shown that the developed control strategy is capable of smoothening the power output of wind turbine and avoiding its sudden showdown at high wind speeds without worsening the loads on rotor and blades.

Fatigue distribution optimization for offshore wind farms using intelligent agent control

A novel control approach is proposed to optimize the fatigue distribution of wind turbines in a large-scale offshore wind farm on the basis of an intelligent agent theory. The turbine at the farm boundary communicates with its neighbouring downwind turbines and organizes them adaptively into a wind delivery group along the wind direction. The agent attributes and the event structure are designed on the basis of the intelligent agent theory by using the unified modelling language. The control strategy of the intelligent agent is studied using topology models. The reference power of an individual wind turbine from the wind farm controller is re-dispatched to balance the turbine fatigue in the power dispatch intervals. In the fatigue optimization, the goal function is to minimize the standard deviation of the fatigue coefficient for every wind turbine. The optimization is constrained such that the average fatigue for every turbine is smaller than what would be achieved by conventional dispatch and such that the total power loss of the wind farm is restricted to a few percent of the total power. This intelligent agent control approach is verified through the simulation of wind data from the Horns Rev offshore wind farm. The results illustrate that intelligent agent control is a feasible way to optimize fatigue distribution in wind farms, which may reduce the maintenance frequency and extend the service life of large-scale wind farms. Copyright © 2012 John Wiley & Sons, Ltd.
Investigation of Load Prediction on the Mexico Rotor Using the Technique of Determination of the Angle of Attack

Blade element moment (BEM) is a widely used technique for prediction of wind turbine aerodynamics performance, the reliability of airfoil data is an important factor to improve the prediction accuracy of aerodynamic loads and power using a BEM code. The method of determination of angle of attack on rotor blades developed by SHEN, et al is successfully used to extract airfoil data from experimental characteristics on the MEXICO (Model experiments in controlled conditions) rotor. Detailed surface pressure and particle image velocimetry (PIV) flow fields at different rotor azimuth positions are examined to determine the sectional airfoil data. The present technique uses simultaneously both PIV data and blade pressure data that include the actual flow conditions (for example, tunnel effects), therefore it is more advantageous than other techniques which only use the blade loading (pressure data). The extracted airfoil data are put into a BEM code, and the calculated axial and tangential forces are compared to both computations using BEM with Glauert's and SHEN's tip loss correction models and experimental data. The comparisons show that the present method of determination of angle of attack is correct, and the re-calculated forces have good agreements with the experiment.
Laminar-Turbulent transition on Wind Turbines
The present thesis deals with the study of the rotational effects on the laminar-turbulent transition on wind turbine blades. Linear stability theory is used to formulate the stability equations that include the effect of rotation. The mean flow required as an input to stability computations is obtained by a similarity transformation technique. This approach allows to transform the boundary layer equations that have included the effect of the Coriolis and centrifugal forces into a set of couple partial differential equations, that are more convenient to solve numerically. The solution have been parametrized and adapted to an wind turbine rotor geometry. The blade is resolved in radial sections along which calculations are performed. The obtained mean flow is classified according to the parameters used on the rotating configuration, geometry and operational conditions. The stability diagrams have been obtained by solving the stability equations as an eigenvalue problem. The Keller box Scheme that is second order accurate was used as a numerical method. Have found to be stable and effective in terms of computing time. The solution of the eigenvalue problem provide connection between the parameters used to define the resultant wave magnitude and direction. The propagation of disturbances in the boundary layers in three dimensional flows is relatively a complicated phenomena. The report discusses the available methods and techniques used to predict the transition location. Some common wind turbine airfoils are selected to performe parametrical studies with rotational effects. Finally a wind turbine rotor is used for comparison with transition experiments. The relative motion between the flow and the blade geometry defines the response of the flow to disturbances. Have been found that flow on the suction side of the blade has a stabilizing effect, while on the region from the stagnation point to the rotor plane has a destabilizing effect on the boundary layer. The tendency is that rotational effect stabilize the boundary layer on the wind turbine blade.

General information
Publication status: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics
Contributors: Martinez Hernandez, G. G., Sørensen, J. N., Shen, W. Z.
Number of pages: 133
Publication date: 2012

Operation Design of Wind Turbines in Strong Wind Conditions
In order to reduce the impact on the electrical grid from the shutdown of MW wind turbines at wind speeds higher than the cut-out wind speed of 25 m/s, we propose in this paper to run the turbines at high wind speeds up to 40 m/s. Two different operation designs are made for both constant speed and variable speed pitch regulated wind turbines. The variable speed design is more suitable for wind turbines to run at very high wind speeds which can help the turbine braking system to stop the turbine at the new "cut-out" wind speed. Reference power, rotational speed and pitch angle have been designed optimally. In order to reduce the possible increased loading, fatigue due to the wind gusts, control strategies have been considered for both constant speed and variable speed pitch regulated wind turbines. The control study shows that the designed controllers can reduce the standard deviations efficiently for wind turbines at some selected wind high speeds.

General information
Publication status: Published
Organisations: Department of Wind Energy, Department of Informatics and Mathematical Modeling, Mathematical Statistics, Department of Electrical Engineering, Automation and Control, Technical University of Denmark, KK-Electronic A/S
Contributors: Shen, W. Z., Montes, M. B., Odgaard, P. F., Poulsen, N. K., Niemann, H. H.
Number of pages: 10
Publication date: 2012
Validations and improvements of airfoil trailing-edge noise prediction models using detailed experimental data

This paper describes an extensive assessment and a step by step validation of different turbulent boundary-layer trailing-edge noise prediction schemes developed within the European Union funded wind energy project UpWind. To validate prediction models, measurements of turbulent boundary-layer properties such as two-point turbulent velocity correlations, the spectra of the associated wall pressure fluctuations and the emitted trailing-edge far-field noise were performed in the laminar wind tunnel of the Institute of Aerodynamics and Gas Dynamics, University of Stuttgart. The measurements were carried out for a NACA643-418 airfoil, at Re = 2.5 ×10^6, angle of attack of −6° to 6°. Numerical results of different prediction schemes are extensively validated and discussed elaborately. The investigations on the TNO-Blake noise prediction model show that the numerical wall pressure fluctuation and far-field radiated noise models capture well the measured peak amplitude level as well as the peak position if the turbulence noise source parameters are estimated properly including turbulence anisotropy effects. Large eddy simulation based computational aeroacoustic computations show good agreements with measurements in the frequency region higher than 1 kHz, whereas they over-predict the sound pressure level in the low-frequency region. Copyright © 2011 John Wiley & Sons, Ltd.

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering, Aeroelastic Design, Wind Energy Division, Risø National Laboratory for Sustainable Energy, University of Stuttgart
Pages: 45-61
Publication date: 2012
Peer-reviewed: Yes

Publication information
Journal: Wind Energy
Volume: 15
Issue number: 1
ISSN (Print): 1095-4244
Ratings:
BFI (2012): BFI-level 2
Scopus rating (2012): CiteScore 2.36 SJR 1.062 SNIP 2.326
Web of Science (2012): Impact factor 1.436
ISI indexed (2012): ISI indexed yes
Web of Science (2012): Indexed yes
Original language: English
Keywords: Two-point correlation, Wall pressure fluctuation, Trailing-edge noise, Far-field noise spectrum, Turbulent boundary-layer, Aeroelastic design methods
DOIs:
10.1002/we.505
Source: orbit
Source-ID: 312266
Research output: Contribution to journal › Journal article – Annual report year: 2011 › Research › peer-review

Actuator Line/Navier-Stokes Computations for Flows past the Yawed MEXICO Rotor

In the paper the Actuator Line/Navier-Stokes model has been used to simulate flows past the yawed MEXICO rotor. The computed loads as well as the velocity field behind the yawed rotor are compared to detailed pressure and PIV measurements which were carried out in the EU funded MEXICO project. The computed loading follows in general the experimental counterpart in a period of rotation with a slight overprediction on the mean loading which probably is caused by the inaccuracy of the airfoil data. The predicted wake velocity agrees well with the experiments in the near wake region. Computations with the DNW wind tunnel for the yawed rotor are also performed and show that the tunnel effects are very small in the loading and in the near wake field behind the rotor whereas in the far wake region (>1D) the influence becomes important.

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering, Yangzhou University
Contributors: Shen, W. Z., Sørensen, J. N., Yang, H.
Publication date: 2011

Host publication information
Title of host publication: Proceedings of the Wake Conference
Extraction of airfoil data using PIV and pressure measurements

A newly developed technique for determining the angle of attack (AOA) on a rotating blade is used to extract AOA and airfoil data from measurements obtained during the MEXICO (Model rotor EXperiments in COntrolled conditions) rotor experiment. Detailed surface pressure and Particle Image Velocimetry (PIV) flow fields at different rotor azimuth positions are examined for determining sectional airfoil data. The AOA is derived locally by determining the local circulation on the blade from pressure data and subtracting the induction of the bound circulation from the local velocity. The derived airfoil data are compared to 2D data from wind tunnel experiments and XFOIL computations. The comparison suggests that the rotor is subject to severe 3D effects originating from the geometry of the rotor, and explains why the Blade Element Momentum technique with 2D airfoil data over-predicts the loading of the rotor. The extraction technique is verified by employing the derived airfoil characteristics as input to computations using the BEM technique and comparing the calculated axial and tangential forces to the measured data. The comparison also demonstrates that the used technique of determining the AOA is a reliable tool to extract airfoil data from experimental data. Copyright © 2010 John Wiley & Sons, Ltd.
High‐order numerical simulations of flow‐induced noise

In this paper, the flow/acoustics splitting method for predicting flow‐generated noise is further developed by introducing high‐order finite difference schemes. The splitting method consists of dividing the acoustic problem into a viscous incompressible flow part and an inviscid acoustic part. The incompressible flow equations are solved by a second‐order finite volume code EllipSys2D/3D. The acoustic field is obtained by solving a set of acoustic perturbation equations forced by flow quantities. The incompressible pressure and velocity form the input to the acoustic equations. The present work is an extension of our acoustics solver, with the introduction of high‐order schemes for spatial discretization and a Runge–Kutta scheme for time integration. To achieve low dissipation and dispersion errors, either Dispersion‐Relation‐Preserving (DRP) schemes or optimized compact finite difference schemes are used for the spatial discretizations. Applications and validations of the new acoustics solver are presented for benchmark aeroacoustic problems and for flow over an NACA 0012 airfoil. Copyright © 2010 John Wiley & Sons, Ltd.
Modeling of Airfoil Trailing Edge Flap with Immersed Boundary Method

The present work considers incompressible flow over a 2D airfoil with a deformable trailing edge. The aerodynamic characteristics of an airfoil with a trailing edge flap is numerically investigated using computational fluid dynamics. A novel hybrid immersed boundary (IB) technique is applied to simulate the moving part of the trailing edge. Over the main fixed part of the airfoil the Navier-Stokes (NS) equations are solved using a standard body-fitted finite volume technique whereas the moving trailing edge flap is simulated with the immersed boundary method on a curvilinear mesh. The obtained results show that the hybrid approach is an efficient and accurate method for solving turbulent flows past airfoils with a trailing edge flap and flow control using trailing edge flap is an efficient way to regulate the aerodynamic loading on airfoils.

Q3UIC – A new aerodynamic airfoil tool including rotational effects

Simulation of vortex sound using the viscous/acoustic splitting approach

A numerical viscous/acoustic splitting approach for the calculation of an acoustic field is applied to study the sound generation by a pair of spinning vortices and by the unsteady interaction between an inviscid vortex and a finite length flexible boundary. Based on the unsteady hydrodynamic information from the known incompressible flow field, the perturbed compressible acoustic terms are calculated and compared with analytical solutions. Results suggest that the present numerical approach produces results which are in good agreement with the analytical solutions. The present investigation verifies the applicability of the viscous/ acoustic approach to flow structure-acoustic interaction.
Study on wind turbine arrangement for offshore wind farms

In this paper, the separation distance between two neighboring offshore wind turbines has been carried out by using the Actuator Line/Navier-Stokes technique developed at the Technical University of Denmark (DTU). Under offshore atmospheric conditions, Large Eddy Simulation has been performed for two Tjæreborg 2 MW wind turbines in tandem with separation distances of 4D, 5D, 6D, 7D, 8D and 10D at the design wind speed of 10 m/s. The power performance of the wake turbine showed to be about 23% of the first turbine at a separation distance of 4D while its performance reached about 50% at 7D due to the turbulence mixing. This study hints that the optimal separation distance between neighboring turbines for offshore wind farms should be 7 rotor diameters.

The affect of wind turbine nacelle geometry on near wake structure

In this paper, the separation distance between two neighboring offshore wind turbines has been carried out by using the Actuator Line/Navier-Stokes technique developed at the Technical University of Denmark (DTU). Under offshore atmospheric conditions, Large Eddy Simulation has been performed for two Tjæreborg 2 MW wind turbines in tandem with separation distances of 4D, 5D, 6D, 7D, 8D and 10D at the design wind speed of 10 m/s. The power performance of the wake turbine showed to be about 23% of the first turbine at a separation distance of 4D while its performance reached about 50% at 7D due to the turbulence mixing. This study hints that the optimal separation distance between neighboring turbines for offshore wind farms should be 7 rotor diameters.
TOPFARM - next generation design tool for optimisation of wind farm topology and operation

The present report is the publishable final activity report for the EU project TOPFARM. The project has been running from 1st December 2007 to 30th November 2010, and has successfully addressed optimization of wind farm topology and control strategy based on aero-elastic modeling of loads as well as of power production as seen in an economical perspective. Crucial factors in this regard are the overall wind climate at the wind farm site, the position of the individual wind turbines, the wind turbine characteristics, the internal wind farm wind climate, the wind turbine control/operation strategy for wind turbines interacting through wakes, various cost models, the optimization strategy and a priori defined constraints imposed on the wind farm topology. In TOPFARM, the object function used in the optimization platform is formulated in economical terms, thus ensuring the optimal balance between capital costs, operation and maintenance costs, cost of fatigue lifetime consumption and power production output throughout the design lifetime of the wind farm. The report describes the project consortium and the project activities, which has been organized in 9 Work Packages. A summary description of the results is given, and reference is made to a large number of publications resulting from the project.

General information
Publication status: Published
Number of pages: 95
Publication date: 2011

Publication information
Publisher: Danmarks Tekniske Universitet, Risø Nationallaboratoriet for Bæredygtig Energi
Original language: English
(Denmark. Forskningscenter Risoe. Risoe-R; No. 1805(EN)).
Keywords: Risø-R-1805, Risø-R-1805(EN)
Electronic versions:
ris_r_1805.pdf
Research output: Book/Report › Report – Annual report year: 2012 › Research

Airfoils and method for designing airfoils
The present invention relates to airfoils and design and design optimization of airfoils, in particular airfoils of rotor blades for wind turbines. One aspect of the invention relates to an airfoil with an external shape provided by an airfoil profile defined by a limited number of parameters, such as a set of parameters. Another aspect of the invention relates to a method for designing an airfoil by means of an analytical airfoil profile, said method comprising the step of applying a conformal mapping to a near circle in a near circle plane, wherein the near circle is at least partly expressed by means of an analytical function, said conformal mapping transforming the near circle in the near circle plane to the airfoil profile in an airfoil plane. L

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering, Department of Physics
Publication date: 2010

Publication information
Country: China
Patent number: CN101898644 A
Filing date: 01/12/2010
Original language: English

Bibliographical note
Patent:
No.: WO2010136357
Date: 2010-12-02
Source: orbit
Source-ID: 274068
**Coupling analysis of wind turbine blades based on aeroelastics and aerodynamics**

The structural dynamic equations of blades were constructed for blades of wind turbines. The vibration velocity of blades and the relative flow velocity were calculated using the structural dynamics model. Based on the BEM (Blade Element Momentum) theory and traditional aerodynamics, the coupling model for wind turbines including the aeroelastic and aerodynamics was developed. Finally, the computation was completed for a 2MW wind turbine. The vibration deflection, speed, acceleration and load of the blades are computed at the rating wind speed. The aerodynamic model of wind turbines becomes more accurate because of the coupling analysis between the vibration speed of blade and the wind speed. The results have very significance to the design of wind turbines.

**General information**
- **Publication status:** Published
- **Organisations:** Fluid Mechanics, Department of Mechanical Engineering
- **Contributors:** Wang, X., Chen, J., Zhang, S., Shen, W. Z., Zhu, W.
- **Pages:** 96-100
- **Publication date:** 2010
- **Peer-reviewed:** Yes

**Publication Information**
- **Journal:** Taiyang Neng Xuebao
- **Volume:** 31
- **Issue number:** 1
- **ISSN (Print):** 0254-0096
- **Ratings:** Scopus rating (2010): SJR 0.185 SNIP 0.317
- **Original language:** English
- **Keywords:** Blades of wind turbines, Structural dynamics, Coupling analysis, Aeroelastics
- **Source:** orbit
- **Source-ID:** 270884

**Determination of the angle of attack on the mexico rotor using experimental data**

The reliability of airfoil data is an important factor to improve the prediction accuracy of aerodynamic loads and power using a Blade Element Momentum (BEM) code. The method of determination of angle of attack on rotor blades developed by Shen et al was successfully used to extract airfoil characteristics from experimental data on the MEXICO (Model Experiments in controlled Conditions) rotor. Detailed surface pressure and Particle Image Velocimetry (PIV) flow field at different rotor azimuth positions were examined for determining the sectional airfoil data. It is worthwhile noting that the present technique uses simultaneously both PIV data and blade pressure data that include the actual flow conditions (for example, tunnel effects), so it is advantageous over other techniques that use only the blade loading (pressure data). The extracted airfoil data is put into a BEM code and the calculated axial and tangential forces are compared to those from BEM computations with Glauert's and Shen's tip loss correction models and experimental data. The comparisons show that the present method of determination of AOA is correct, and the recalculated forces have good agreement with the experiment.

**General information**
- **Publication status:** Published
- **Organisations:** Department of Mechanical Engineering, Fluid Mechanics
- **Contributors:** Yang, H., Shen, W. Z., Sørensen, J. N., Zhu, W. J.
- **Number of pages:** 876
- **Pages:** 47-57
- **Publication date:** 2010

**Host publication information**
- **Title of host publication:** The Science of Making Torque from Wind 2010
- **Keywords:** rotor aerodynamics, airfoil data, Wind turbine
- **Source:** orbit
- **Source-ID:** 265728

**Optimization design of blade shapes for wind turbines**

For the optimization design of wind turbines, the new normal and tangential induced factors of wind turbines are given considering the tip loss of the normal and tangential forces based on the blade element momentum theory and traditional aerodynamic model. The cost model of the wind turbines and the optimization design model are developed. In the
optimization model, the objective is the minimum cost of energy and the design variables are the chord length, twist angle and the relative thickness. Finally, the optimization is carried out for a 2 MW blade by using this optimization design model. The performance of blades is validated through the comparison and analysis of the results. The reduced cost shows that the optimization model is good enough for the design of wind turbines. The results give a proof for the design and research on the blades of large scale wind turbines and also establish the foundation for further research and industrial application of wind turbines. © 2010 Journal of Mechanical Engineering.

Shape optimisation of wind turbine blades

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Shen, W. Z.
Pages: 90-92
Publication date: 2010
Peer-reviewed: No

Publication information
Journal: Modern Energy Review
Volume: 2
Issue number: 2
ISSN (Print): 2041-9570
Original language: English
Source: orbit
Source-ID: 270798
Research output: Contribution to journal

Study of airfoil trailing edge bluntness noise

This paper deals with airfoil trailing edge noise with special focus on airfoils with blunt trailing edges. Two methods are employed to calculate airfoil noise: The flow/acoustic splitting method and the semi-empirical method. The flow/acoustic splitting method is derived from compressible Navier-Stokes equations. It provides us possibilities to study details about noise generation mechanism. The formulation of the semi-empirical model is based on acoustic analogy and then curve-fitted with experimental data. Due to its high efficiency, such empirical relation is used for purpose of low noise airfoil design or optimization. Calculations from both methods are compared with exist experiments. The airfoil blunt noise is found as a function of trailing edge bluntness, Reynolds number, angle of attack, etc.

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Zhu, W. J., Shen, W. Z., Sørensen, J. N.
Number of pages: 876
Pages: 93-100
Validation of the actuator line/Navier Stokes technique using Mexico measurements

This paper concerns the contribution of DTU MEK in the international research collaboration project (MexNext) within the framework of IEA Annex 29 to validate aerodynamic models or CFD codes using the existing measurements made in the previous EU funded project MEXICO (Model Experiments in Controlled Conditions). The Actuator Line/Navier Stokes (AL/NS) technique developed at DTU is validated against the detailed MEXICO measurements. The AL/NS computations without the DNW wind tunnel with speeds of 10m/s, 15m/s and 24m/s. Comparisons of blade loading between computations and measurements show that AL/NS with original 2D airfoil data over-predicts the blade loads. To take into account the effects on a rotating blade, a set of modified airfoil data is used instead. Comparisons show that AL/NS with modified airfoil data agrees much better (for example, the maximum over prediction at the wind speed of 15m/s is decreased from 17% to 8% at 60%R). The discrepancy may be due to severe 3D effects caused essentially by the blade geometry where different types of airfoils are used. Comparisons of detailed near wake velocity show good agreements.

Aeroacoustic Computations for Turbulent Airfoil Flows

The How-acoustic splitting technique for aeroacoustic computations is extended to simulate the propagation of acoustic waves generated by three-dimensional turbulent flows. In the flow part, a subgrid-scale turbulence model (the mixed model) is employed for large-eddy simulations. The obtained instantaneous How solution is employed as input for the acoustic part. At low Mach numbers, the differences in scales and propagation speed between the flow and the acoustic field are quite large, and hence different meshes and time steps can be used for the two parts. The model is applied to compute flows past a NACA 0015 airfoil at a Mach number of 0.2 and a Reynolds number of 1.6 x 10(5) for different angles of attack. The flow solutions are validated by comparing lift and drag characteristics with experimental data. The comparisons show good agreements between the computed and measured airfoil lift characteristics for angles of attack up to stall. For the acoustic solutions, predicted noise spectra are validated quantitatively against experimental data. A parametrical study of the noise pattern for flows at angles of attack between 4 and 12 deg shows that the noise level is small for angles of attack below 8 deg, increases sharply from 8 to 10 deg, and reaches a maximum at 12 deg.
Analysis of numerical models for cavitation on 2D hydrofoil

General information
Publication status: Published
Organisations: Coastal, Maritime and Structural Engineering, Department of Mechanical Engineering, Fluid Mechanics
Contributors: Shin, K. W., Andersen, P., Shen, W. Z.
Publication date: 2009

Host publication information
Title of host publication: Proc. 12th Numerical Towing Tank Symposium
Editor: Bertram, V.
Keywords: cavitation
Source: orbit
Source-ID: 253361

Blade optimization for wind turbines

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering, Chongqing University
Number of pages: 10
Publication date: 2009

Host publication information
Title of host publication: Proceedings of EWEC 2009
Volume: PO 181
Source: orbit
Source-ID: 252318

Calculations of Flow around an Airfoil with a Trailing Edge Flap by Use of an Immersed Boundary Method

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering, Aeroelastic Design, Wind Energy Division, Risø National Laboratory for Sustainable Energy, Vestas Wind Systems AS
Contributors: Behrens, T., Zhu, W. J., Shen, W. Z., Sørensen, J. N., Sørensen, N. N., Wedel-Heinen, J. J.
Publication date: 2009

Host publication information
Title of host publication: EWEC 2009 Proceedings online
Publisher: EWEC
Keywords: Wind energy, Aeroelastic Design
Electronic versions:
2009_54.pdf
Computational Aerodynamics and Aeroacoustics for Wind Turbines

To analyse the aerodynamic performance of wind turbine rotors, the main tool in use today is the 1D-Blade Element Momentum (BEM) technique combined with 2D airfoil data. Because of its simplicity, the BEM technique is employed by industry when designing new wind turbine blades. However, in order to obtain more detailed information of the flow structures and to determine more accurately loads and power yield of wind turbines or cluster of wind turbines, it is required to resort to more sophisticated techniques, such as Computational Fluid Dynamics (CFD). As computer resources keep on improving year by year (about ten times every five years from statistics over the last twenty years), CFD has now become a popular tool for studying the aerodynamics of wind turbines. The present thesis consists of 19 selected papers dealing with the development and use of CFD methods for studying the aerodynamics and aero-acoustics of wind turbines. The papers are written in the period from 1997 to 2008 and numbered according to the list in page v. The work consists of two parts: an aerodynamic part based on Computational Fluid Dynamics and an aero-acoustic part based on Computational Aero Acoustics for wind turbines. The main objective of the research was to develop new computational tools and techniques for analysing flows about wind turbines. A few papers deal with applications of Blade Element Momentum (BEM) theory to wind turbines. In most cases the incompressible Navier-Stokes equations in primitive variables (velocity-pressure formulation) are employed as the basic governing equations. However, since fluid mechanical problems essentially are governed by vortex dynamics, it is sometimes advantageous to use the concept of vorticity (defined as the curl of velocity). In vorticity form the Navier-Stokes equations may be formulated in different ways, using a vorticity-stream function formulation, a vorticity-velocity formulation or a vorticity-potential-stream function formulation. In [1] - [3] two different vorticity formulations were developed for 2D and 3D wind turbine flows. In [4] and [5] numerical techniques for avoiding pressure oscillations were developed when solving the velocity-pressure coupling system in the in-house EllipSys2D/3D code, which originally was developed in a cooperation between DTU (Michelsen, 1992) and Risø (Sørensen, 1995). In [6] - [8] different actuator disc techniques combined with CFD are presented. This includes actuator disc, actuator line and actuator surface techniques, which were developed to simulate flows past one or more wind turbines. In [9] and [10] a tip loss correction method that improves the conventional models was developed for use in combination with BEM or actuator/Navier-Stokes computations. A simple and efficient technique for determining the angle of attack for flow past a wind turbine rotor was developed in [11], and in [12] tunnel wall corrections for wind tunnels with closed or open test sections were developed. The second part of the thesis deals with Computational Aero-Acoustics (CAA). With the spread of wind turbines near urban areas, there is an increasing need for accurate predictions of aerodynamically generated noise. Indeed, noise has become one of the most important issues for further development of wind power, and the ability of controlling and minimising noise emission may be advantageous when competing on the world energy market. To predict generation and propagation of aerodynamic noise, it is required to solve the compressible Navier-Stokes equations. As the scales of the flow and the acoustic waves are quite different (about 1/M, M=Mach number=U/c), it is difficult to resolve them together at the same time. Hardin and Pope proposed a non-linear two-step (viscous incompressible flow and inviscid acoustic perturbation) splitting procedure for computational aero-acoustics that is suitable for both generation and propagation. The advantage of the splitting approach, as compared to the acoustic analogies, is that the source strength is obtained directly and that it accounts for sound radiation as well as scattering. In [13] and [14] an inconsistency in the original formulation of Hardin and Pope 1994 was analysed and a consistent formulation was proposed and applied to laminar flows. An aero-acoustic formulation for turbulent flows was in [15] developed for Large Eddy Simulation (LES), Unsteady Reynolds Averaged Navier-Stokes Simulation (URANS) and Detached Eddy Simulation (DES). In [16] a collocated grid / finite volume method for aero-acoustic computations was developed and implemented in the Ellipsys2D/3D code. In [17] and [18] three dimensional flow-acoustic computations were carried out. Finally, the aero-acoustic formulation using high order Finite Difference schemes (Dispersion Relation Preserving (DRP) / Optimized Compact schemes) was developed in [19] and implemented in the Ellipsys2D/3D code.
Determination of the angle of attack on rotor blades

Two simple methods for determining the angle of attack (AOA) on a section of a rotor blade are proposed. Both techniques consist of employing the Biot-Savart integral to determine the influence of the bound vorticity on the velocity field. In the first technique, the force distribution along the blade and the velocity at a monitor point in the vicinity of the blade are assumed to be known from experiments or CFD computations. The AOA is determined by subtracting the velocity induced by the bound circulation, determined from the loading, from the velocity at the monitor point. In the second method, the full pressure distribution on the blade is assumed to be known and used to determine the local distribution of circulation along the surface contour of the blade. Using the local distribution of circulation to determine the influence of the bound vorticity enables the velocity monitor points to be located closer to the blade, and thus to determine the AOA with higher accuracy. Data from CFD computations for flows past the Tellus 95 kW wind turbine at different wind speeds are used to test both techniques. Comparisons show that the proposed methods are in good agreement with existing techniques. The advantage of the proposed techniques, as compared with existing techniques, is that they can be used to determine the AOA on rotor blades under general flow conditions (e.g. operations in yaw or with dynamic inflow).

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Shen, W. Z., Hansen, M. O. L., Sørensen, J. N.
Pages: 91-98
Publication date: 2009
Peer-reviewed: Yes

Publication information
Journal: Wind Energy
Volume: 12
Issue number: 1
ISSN (Print): 1095-4244
Ratings:
BFI (2009): BFI-level 2
Scopus rating (2009): SJR 0.885 SNIP 1.435
Web of Science (2009): Indexed yes
Original language: English
Keywords: rotor aerodynamics, wind turbine, airfoil characteristics
DOIs:
10.1002/we.277
Source: orbit
Source-ID: 233274
Research output: Contribution to journal › Journal article – Annual report year: 2009 › Research › peer-review

Integration study on airfoil profiles for wind turbines

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering, Chongqing University
Contributors: Wang, X. D., Chen, J., Shen, W. Z., Zhang, S.
Pages: 211-213
Publication date: 2009
Peer-reviewed: Yes

Publication information
Journal: Zhongguo Jixie Gongcheng
Volume: 20
Issue number: 2
ISSN (Print): 1004-132X
Ratings:
Scopus rating (2009): SJR 0.153
Original language: English
Source: orbit
Source-ID: 255199
Research output: Contribution to journal › Journal article – Annual report year: 2009 › Research › peer-review
**Numerical Study on Turbulent Airfoil Noise with High-Order Schemes**

High-order finite difference schemes are introduced in the flow/acoustics splitting technique for predicting flow generated noise. The flow equations are solved by a second-order finite volume method whereas the acoustic equations are solved by high-order finite difference schemes. At each time step, the incompressible pressure and velocity form input to the acoustic equations. In this paper, sound generation from a NACA 0012 airfoil in turbulent flow condition is studied. The noise source regions are found at the trailing edge and the strength of the sources is depended on the Reynolds number and the angle of attack.

**General information**
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Zhu, W. J., Shen, W. Z., Sørensen, J. N.
Publication date: 2009

**Host publication information**
Title of host publication: European Wind Energy Conference 2009
Source: orbit
Source-ID: 254128
Research output: Chapter in Book/Report/Conference proceeding – Annual report year: 2009 – peer-review

**Shape Optimization of Wind Turbine Blades**
This paper presents a design tool for optimizing wind turbine blades. The design model is based on an aerodynamic/aero-elastic code that includes the structural dynamics of the blades and the Blade Element Momentum (BEM) theory. To model the main aero-elastic behaviour of a real wind turbine, the code employs 11 basic degrees of freedom corresponding to I I elastic structural equations. In the BEM theory, a refined tip loss correction model is used. The objective of the optimization model is to minimize the cost of energy which is calculated from the annual energy production and the cost of the rotor. The design variables used in the current study are the blade shape parameters, including chord, twist and relative thickness. To validate the implementation of the aerodynamic/aero-elastic model, the computed aerodynamic results are compared to experimental data for the experimental rotor used in the European Commission-sponsored project Model Experiments in Controlled Conditions, (MEXICO) and the computed aero-elastic results are examined against the FLEX code for flow post the Tjereborg 2 MW rotor. To illustrate the optimization technique, three wind turbine rotors of different sizes (the MEXICO 25 kW experimental rotor, the Tjareborg 2 MW rotor and the NREL 5 MW virtual rotor) are applied. The results show that the optimization model can reduce the cost of energy of the original rotors, especially for the investigated 2 MW and 5 MW rotors. Copyright (C) 2009 John Wiley & Sons, Ltd.

**General information**
Publication status: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics, Chongqing University
Pages: 781-803
Publication date: 2009
Peer-reviewed: Yes

**Publication information**
Journal: Wind Energy
Volume: 12
Issue number: 8
ISSN (Print): 1095-4244
Ratings:
BFI (2009): BFI-level 2
Scopus rating (2009): SJR 0.885 SNIP 1.435
Web of Science (2009): Indexed yes
Original language: English
Keywords: aerodynamics, wind turbine design, aero-elasticity, blade optimization
DOI:
10.1002/we.335
Source: orbit
Source-ID: 253904
Simulation of Flow Past Wind Turbines Located on a Hill by a Hybrid Actuator/Navier-Stokes Method

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Shen, W. Z., Sørensen, J. N., Mikkelsen, R. F.
Number of pages: 10
Publication date: 2009

Host publication information
Title of host publication: Proceedings of EWEC 2009
Volume: PO 232
Place of publication: Marseille, France
Source: orbit
Source-ID: 252316

The Actuator Surface Model: A New Navier-Stokes Based Model for Rotor Computations
This paper presents a new numerical technique for simulating two-dimensional wind turbine flow. The method, denoted as the 2D actuator surface technique, consists of a two-dimensional Navier-Stokes solver in which the pressure distribution is represented by body forces that are distributed along the chord of the airfoils. The distribution of body force is determined from a set of predefined functions that depend on angle of attack and airfoil shape. The predefined functions are curve fitted using pressure distributions obtained either from viscous-inviscid interactive codes or from full Navier-Stokes simulations. The actuator surface technique is evaluated by computing the two-dimensional flow past a NACA 0015 airfoil at a Reynolds number of 10(6) and an angle of attack of 10 deg and by comparing the computed streamlines with the results from a traditional Reynolds-averaged Navier-Stokes computation. In the last part, the actuator surface technique is applied to compute the flow past a two-bladed vertical axis wind turbine equipped with NACA 0012 airfoils. Comparisons with experimental data show an encouraging performance of the method.

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Shen, W. Z., Zhang, J., Sørensen, J. N.
Pages: 011002
Publication date: 2009
Peer-reviewed: Yes

Publication information
Journal: Journal of Solar Energy Engineering
Volume: 131
Issue number: 1
ISSN (Print): 0199-6231
Ratings:
BFI (2009): BFI-level 1
Scopus rating (2009): SJR 0.649 SNIP 1.114
Web of Science (2009): Indexed yes
Original language: English
Keywords: electric actuators, Navier-Stokes equations, wind turbines, flow
DOIs:
10.1115/1.3027502

Bibliographical note
9 pp.
Source: orbit
Source-ID: 243276

Aero-Acoustic Modeling using Large Eddy Simulation
The flow-acoustic splitting technique for aero-acoustic computations is extended to simulate the propagation of acoustic waves generated by three-dimensional turbulent flows. In the flow part, a sub-grid-scale turbulence model (the mixed model) is employed for Large-Eddy Simulations. The obtained instantaneous flow solution is employed as input for the acoustic part. At low Mach numbers the differences in scales and propagation speed between the flow and the acoustic field are quite large, hence different meshes and time-steps can be utilized for the two parts. The model is applied to compute flows past a NACA 0015 airfoil at a Mach number of 0.2 and a Reynolds number of for different angles of attack.
The flow solutions are validated by comparing lift and drag characteristics to the experiments of Shedhal and Klimas. The comparisons show good agreement between computed and measured airfoil characteristics for angles of attack up to stall. For the acoustic solutions, predicted noise spectra are validated quantitatively against the experimental data of Brook et al. A parametrical study of the noise pattern for flows at angles of attack between 4 deg and 12 deg shows that the noise level is small for angles of attack below 8 deg, increases sharply from 8 deg to 10 deg and reaches a maximal at 12 deg.
Triplet of Helical Vortices

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering, Center for Fluid Dynamics
Contributors: Okulov, V., Naumov, I., Shen, W. Z., Sørensen, J. N.
Pages: 281-290
Publication date: 2008

Host publication information
Title of host publication: IUTAM Symposium on Hamiltonian Dynamics, Vortex Structures, Turbulence : IUTAM Bookseries
Volume: 6
Publisher: Springer
ISBN (Print): 978-1-4020-6743-3
Source: orbit
Source-ID: 208418

3D boundary layer study on a rotating wind turbine blade

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Hernández, G. G. M., Sørensen, J. N., Shen, W. Z.
Pages: 1-8
Publication date: 2007
Peer-reviewed: Yes

Publication information
Journal: Journal of Physics: Conference Series (Online)
Volume: 75
Issue number: 012032
ISSN (Print): 1742-6596
Ratings:
Scopus rating (2007): SJR 0.258 SNIP 0.407
Web of Science (2007): Indexed yes
Original language: English
Source: orbit
Source-ID: 209518
Research output: Contribution to journal – Conference article – Annual report year: 2007 – Research – peer-review

Actuator surface model for wind turbine flow computations

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering, GE Energy Engineering Division
Contributors: Shen, W. Z., Sørensen, J. N., Zhang, J.
Publication date: 2007

Host publication information
Title of host publication: Proceedings of European Wind Energy Conference 2007
Electronic versions:
proceeding_shen.pdf

Bibliographical note
Aero-acoustic modeling using large eddy simulation

The splitting technique for aero-acoustic computations is extended to simulate three-dimensional flow and acoustic waves from airfoils. The aero-acoustic model is coupled to a sub-grid-scale turbulence model for Large-Eddy Simulations. In the first test case, the model is applied to compute laminar flow past a NACA 0015 airfoil at a Reynolds number of 800, a Mach number of 0.2 and an angle of attack of 20 degrees. The model is then applied to compute turbulent flow past a NACA 0015 airfoil at a Reynolds number of 100 000, a Mach number of 0.2 and an angle of attack of 20 degrees. The predicted noise spectrum is compared to experimental data.

Airfoil noise computation using high-order schemes

High-order finite difference schemes with at least 4th-order spatial accuracy are used to simulate aerodynamically generated noise. The aeroacoustic solver with 4th-order up to 8th-order accuracy is implemented into the in-house flow solver, EllipSys2D/3D. Dispersion-Relation-Preserving (DRP) finite difference schemes and optimized high-order compact finite difference schemes are applied for acoustic computation. Acoustic equations are derived using so-called splitting technique by separating the compressible NS equations into viscous (flow equation) and inviscid (acoustic equation) parts. The viscous flow equations are solved using EllipSys2D/3D, the hydrodynamic variables are used as input to the inviscid acoustic equations at same time step.
Analysis of counter-rotating wind turbines
This paper presents a study on the performance of a wind turbine with two counter-rotating (CRWT) rotors. The characteristics of the two counter-rotating rotors are on a 3-bladed Nordtank 500 kW rotor. The analysis has been carried out by using an Actuator Line technique implemented in the Navier-Stokes code EllipSys3D. The analysis shows that the Annual Energy Production can be increased to about 43.5 %, as compared to a wind turbine with a single rotor. In order to determine the optimal settings of the CRWT turbine, parameters such as distance between two rotors and rotational speed have been studied.

General information
Publication status: Published
Contributors: Shen, W. Z., Zakkam, V. A. K., Sørensen, J. N., Appa, K.
Pages: 9 pages
Publication date: 2007
Peer-reviewed: Yes

Publication information
Journal: Journal of Physics: Conference Series (Online)
Volume: 75
ISSN (Print): 1742-6596
Ratings:
Scopus rating (2007): SJR 0.258 SNIP 0.407
Web of Science (2007): Indexed yes
Original language: English
Keywords: Counter Rotating Wind Turbine, Actuator Line Model
URLs:
http://www.iop.org/EJ/abstract/1742-6596/75/1/012003
Source: orbit
Source-ID: 207352
Research output: Contribution to journal › Conference article – Annual report year: 2007 › Research › peer-review

Computational Aero-Acoustic Using High-order Finite-Difference Schemes
In this paper, a high-order technique to accurately predict flow-generated noise is introduced. The technique consists of solving the viscous incompressible flow equations and inviscid acoustic equations using a incompressible/compressible splitting technique. The incompressible flow equations are solved using the in-house flow solver EllipSys2D/3D which is a second-order finite volume code. The acoustic solution is found by solving the acoustic equations using high-order finite difference schemes. The incompressible flow equations and the acoustic equations are solved at the same time levels where the pressure and the velocities obtained from the incompressible equations form the input to the acoustic equations. To achieve low dissipation and dispersion errors, either Dispersion-Relation-Preserving (DRP) schemes or optimized compact finite difference schemes are used for spatial discretizations of the acoustic equations. The classical fourth-order Runge-Kutta time scheme is applied to the acoustic equations for time discretization.

General information
Publication status: Published
Organisations: Department of Mechanical Engineering, Fluid Mechanics
Contributors: Zhu, W. J., Shen, W. Z., Sørensen, J. N.
Pages: 012084
Publication date: 2007
Peer-reviewed: Yes

Publication information
Journal: Journal of Physics: Conference Series (Online)
Volume: 75
ISSN (Print): 1742-6596
Ratings:
Scopus rating (2007): SJR 0.258 SNIP 0.407
Web of Science (2007): Indexed yes
Original language: English
Electronic versions:
1742-6596_75_1_012084.pdf
DOIs:
10.1088/1742-6596/75/1/012084
Oscillatory instability in a closed cylinder with rotating top and bottom
A numerical investigation of oscillatory instability is presented for axisymmetric swirling flow in a closed cylinder with rotating top and bottom. The critical Reynolds number and frequency of the oscillations are evaluated as function of the ratio of angular velocities of the bottom and the top. Earlier Linear Stability Analysis (LSA) using the Galerkin spectral method by Gelfgat et al. [Phys. Fluids, 8, 2614-2625 (1996)] revealed that the curve of the critical Reynolds number behaves like an “S” around in the co-rotation branch and around in the counter-rotation branch. Additional finite volume computations, however, did not show a clear “S” behaviour. In order to check the existence of the “S” shape, computations are performed using an axisymmetric finite volume Navier-Stokes code at aspect ratios 1.5 and 2.0. Comparisons with LSA at aspect ratio 1.5 show that the “S” shape does exist. At an aspect ratio, our results show that the critical Reynolds number curve has a “beak” shape in the counter-rotation region and a much wider “S” shape in the co-rotation region. This transformation of the “S” shape is caused by the change in aspect ratio from 1.5 to 2 and therefore the corresponding topological behaviour of the transition is different.

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Shen, W. Z., Sørensen, J. N.
Pages: 7
Publication date: 2007
Peer-reviewed: Yes

Publication information
Journal: Journal of Physics: Conference Series (Online)
Volume: 64
ISSN (Print): 1742-6596
Ratings:
Scopus rating (2007): SJR 0.258 SNIP 0.407
Web of Science (2007): Indexed yes
Original language: English
Keywords: swirl flow, rotating cylinder, bifurcation
URLs:
http://www.iop.org/EJ/article/1742-6596/64/1/012012/jpconf7_64_012012.pdf?request-id=eojV6-k3BGU-DvG2w7Kg
Source: orbit
Source-ID: 207462
Research output: Contribution to journal › Conference article – Annual report year: 2007 › Research › peer-review

Prediction and reduction of noise from a 2.3 MW wind turbine

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering, Siemens A/S
Pages: 1-9
Publication date: 2007
Peer-reviewed: Yes

Publication information
Journal: Journal of Physics: Conference Series (Online)
Volume: 75
Issue number: 012083
ISSN (Print): 1742-6596
Ratings:
Scopus rating (2007): SJR 0.258 SNIP 0.407
Web of Science (2007): Indexed yes
Original language: English
Source: orbit
Source-ID: 209050
**Rotational Effects on wind turbine blades for transition prediction**

**General information**
Publication status: Published  
Organisations: Fluid Mechanics, Department of Mechanical Engineering  
Contributors: Hernández, G. G. M., Sørensen, J. N., Shen, W. Z.  
Publication date: 2007

**Host publication information**
Title of host publication: 3nd PhD Seminar on Europe  
Source: orbit  
Source-ID: 209516  
Research output: Chapter in Book/Report/Conference proceeding  
Research: peer-review

**The influence of imperfections on the flow structure of steady vortex breakdown bubbles**

**General information**
Publication status: Published  
Organisations: Dynamical systems, Department of Mathematics, Fluid Mechanics, Department of Mechanical Engineering, Center for Fluid Dynamics  
Contributors: Brøns, M., Shen, W. Z., Sørensen, J. N., Zhu, W. J.  
Pages: 453-466  
Publication date: 2007  
Peer-reviewed: Yes

**Publication information**
Journal: Journal of Fluid Mechanics  
Volume: 578  
ISSN (Print): 0022-1120  
Ratings:  
Scopus rating (2007): SJR 2.432 SNIP 1.968  
Web of Science (2007): Indexed yes  
Original language: English  
Keywords: Fluid Mechanics  
DOIs:  
10.1017/S0022112007005101  
Source: orbit  
Source-ID: 206493  
Research output: Contribution to journal  
Research: peer-review

**Determination of Angle of Attack (AOA) for Rotating Blades**

**General information**
Publication status: Published  
Organisations: Fluid Mechanics, Department of Mechanical Engineering  
Contributors: Shen, W. Z., Hansen, M. O. L., Sørensen, J. N.  
Number of pages: 332  
Pages: 205-209  
Publication date: 2006

**Host publication information**
Title of host publication: Wind Energy : Proceedings of the Euromech Colloquium  
Place of publication: Berlin Heidelberg  
Publisher: Springer Verlag  
ISBN (Print): 978-3-540-33865-9  
Electronic versions: Euromech_shen2006.pdf  
Source: orbit  
Source-ID: 193618  
Research output: Chapter in Book/Report/Conference proceeding  
Research: peer-review
Effect of Mn-site vacancies on the magnetic entropy change and the Curie temperature of La0.67Ca0.33Mn1-xO3 perovskite

Single-phase polycrystalline samples of La0.67Ca0.33Mn1-xO3 (x = 0.00, 0.02, 0.04, 0.06) have been prepared using the sol-gel method. The structure, magnetocaloric properties and the Curie temperature of the samples with different Mn vacancy concentrations have been investigated. The experimental results show that vacancy doping at the Mn-sites has a significant influence on the magnetic properties of La0.67Ca0.33Mn1-xO3. The Curie temperature decreases monotonically with increasing the Mn-site vacancy concentration x. A remarkable enhancement of the magnetic entropy change has been obtained in the La0.67Ca0.33WO3 sample. The entropy change reaches |ΔS-M| = 3.10 J kg(-1) K-1 at its Curie temperature (264 K) under an applied magnetic field H = 10 kOe, which is almost the same value as that of pure Gd.

General information
Publication status: Published
Organisations: Department of Physics, Department of Electrical Engineering, Fluid Mechanics, Department of Mechanical Engineering
Pages: 165-168
Publication date: 2006
Peer-reviewed: Yes

Publication information
Journal: Solid State Communications
Volume: 138
Issue number: 4
ISSN (Print): 0038-1098
Ratings:
Scopus rating (2006): SJR 1.061 SNIP 0.931
Web of Science (2006): Indexed yes
Original language: English
Keywords: magnetically ordered materials, magnetocaloric effects, magnetic measurements, chemical synthesis
Source: orbit
Source-ID: 198056
Research output: Contribution to journal › Journal article – Annual report year: 2006 › Research › peer-review

Numerical study of swirling flow in a cylinder with rotating top and bottom

A numerical investigation of oscillatory instability is presented for axisymmetric swirling flow in a closed cylinder with rotating top and bottom. The critical Reynolds number and frequency of the oscillations are evaluated as function of the ratio of angular velocities of the bottom and the top (xi=Ω(bottom)/Ω(top)). Earlier linear stability analysis (LSA) using the Galerkin spectral method by Gelfgat [Phys. Fluids, 8, 2614 (1996)] revealed that the curve of the critical Reynolds number behaves like an "S" around xi=0.54 in the co-rotation branch and around xi=-0.63 in the counter-rotation branch. Additional finite volume computations, however, did not show a clear S behavior. In order to check the existence of the S shape, computations are performed using an axisymmetric finite volume Navier-Stokes code at aspect ratios (λ=H/R) 1.5 and 2.0. Comparisons with LSA at λ=1.5 show that the S shape does exist. The S shape of the stability diagram predicted by LSA is thus confirmed by a finite-volume based Navier-Stokes solver. The additional computations at aspect ratio λ=2 show that the curve of critical Reynolds number has a wider S shape in the co-rotating branch for xi about 0.7 whereas a sharp "beak" appears in the counter-rotating branch for xi approximately -0.5.

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Shen, W. Z., Sørensen, J. N., Michelsen, J.
Pages: 064102
Publication date: 2006
Peer-reviewed: Yes

Publication information
Journal: Physics of Fluids
Volume: 18
Issue number: 6
ISSN (Print): 1070-6631
Ratings:
Scopus rating (2006): SJR 1.943 SNIP 1.496
Web of Science (2006): Indexed yes
Wall correction model for wind tunnels with open test section
In the paper we present a correction model for wall interference on rotors of wind turbines or propellers in wind tunnels. The model, which is based on a one-dimensional momentum approach, is validated against results from CFD computations using a generalized actuator disc principle. In the model the exchange of axial momentum between the tunnel and the ambient room is represented by a simple formula, derived from actuator disc computations. The correction model is validated against Navier-Stokes computations of the flow about a wind turbine rotor. Generally, the corrections from the model are in very good agreement with the CFD computations, demonstrating that one-dimensional momentum theory is a reliable way of predicting corrections for wall interference in wind tunnels with closed as well as open cross sections.

An aerodynamic noise propagation model for wind turbines
A model based on 2-D sound ray theory for aerodynamic noise propagation from wind turbine rotating blades is introduced. The model includes attenuation factors from geometric spreading, sound directivity of source, air absorption, ground deflection and reflection, as well as effects from temperature and airflow. At a given receiver point, the sound pressure is corrected by taking into account these propagation effects. As an overall assumption, the noise field generated by the wind turbine is simplified as a point source placed at the hub height of the wind turbine. This assumption is reasonable, for the receiver is located in the far field, at distances from the wind turbine that are much longer than the diameter of the rotor.
Modeling of Aerodynamically Generated Noise From Wind Turbines

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Zhu, W. J., Heilskov, N., Shen, W. Z., Sørensen, J. N.
Pages: 517-528
Publication date: 2005
Peer-reviewed: Yes

Tip Loss Correction for Actuator/Navier-Stokes Computations

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Shen, W. Z., Sørensen, J. N., Mikkelsen, R. F.
Pages: 209-213
Publication date: 2005
Peer-reviewed: Yes
Tip loss corrections for wind research turbine computations

A colocated Grid Finite Volume Method for Aero-acoustic Computations of Low-speed Flows

Tip loss correction for actuator / Navier Stokes computations
Wall Correction Model for Wind Tunnels with Open Test Section

In the paper we present a correction model for wall interference on rotors of wind turbines or propellers in wind tunnels. The model, that is based on a onedimensional momentum approach, is validated against results from CFD computations using a generalized actuator disc principle. Generally, the corrections from the model are in very good agreement with the CFD computations, demonstrating that one-dimensional momentum theory is a reliable way of predicting corrections for wall interference in wind tunnels with closed as well as open cross sections. Keywords: Wind tunnel correction, momentum theory, CFD

An Improved SIMPLEC Method for Steady and Unsteady Flow Computations

A modified SIMPLEC scheme for flow computations on collocated grids has been developed. It is demonstrated that the standard SIMPLEC scheme (1) is inconsistent when applied on collocated grids. Hence, for steady computations the computed solution depends on the velocity underrelaxation parameter fu, whereas the solutions of unsteady computations for small time steps are polluted by unphysical wiggles. A revised scheme is proposed that extends the capability of the SIMPLEC method to cope with collocated grids in a general and consistent way. The efficiency of the new scheme is demonstrated by computing flows past a circular cylinder and an airfoil.
LDA-PIV Diagnostics and 3D Simulation of Oscillating Swirl Flow in a Closed Cylindrical Container

Results on unsteady vortex breakdown are obtained simultaneously using two diagnostics methods: a) determination of velocity fields by particle tracks (Particle Image Velocimeter - PIV), b) determination of velocity fields by Laser Doppler Anemometry (LDA), are presented. The experiments data are in some regimes supplemented by 3D computations of the incompressible Navier-Stokes equations in cylindrical coordinates. Perfect agreements between calculation and experiment reveal efficiency of both diagnostics method for pulsating vortex breakdown and numerical method for solution to 3D unsteady Navier-Stokes equations.

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Naumov, I., Okulov, V. L., Meyer, K. E., Sørensen, J. N., Shen, W. Z.
Pages: 143-148
Publication date: 2003
Peer-reviewed: Yes

Recent Development of Non-Linear Aeroacoustic Model for Wind Turbine Computations

A numerical algorithm for simulation of acoustic noise generation, based on collocated grids, has been developed. The approach, that was originally developed using a viscous/inviscid decomposition technique, involved two steps comprising a viscous incompressible flow part and an inviscid acoustic part. On collocated grids the inviscid solution is found to be mesh dependent due to unavoidable extrapolations of the acoustic pressure and density at walls, differing from the case on staggered grid where no extrapolation is needed. The situation is most pronounced when a sharp body is considered. A viscous acoustic algorithm is proposed to overcome the difficulty. The model is currently applied to the problem of an airfoil exposed to a gust and results are compared to the numerical results of Lockard and Morris [AIAA J. 36(6) (1998) 907].

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Shen, W. Z., Michelsen, J., Sørensen, J. N.
Publication date: 2003

Study of Tip-loss Using an Inverse 3D Navier-Stokes Method

The tip-correction for air-screws described by Prandtl (1919) and implemented into the Blade Element Momentum (BEM) theory by Glauer (1930), is founded on certain assumptions which the present analysis seeks to overcome. In the paper we propose a method to derive the tip-correction by solving the 3D Navier-Stokes equations combined with the actuator line technique where blade loading is applied using an inverse method. The numerical simulations show that the method captures the tip-correction when comparing with the theories of Prandtl and Goldstein, however, the accuracy of the obtained results reveal that further refinements still is needed. Keywords: Tip-loss; Actuator line; 3D Navier-Stokes methods.

General information
Validation of Tip Corrections for Wind Turbine Computations

Tip loss effect of rotors plays an important role in predictions of wind turbine performance. Classical tip corrections, based on the Prandtl tip reduction function, including Glauert’s, Wilson & Lissaman’s and De Vries’ corrections are considered in the paper. In the proximity of the tip, these classical models fail to predict the physical behaviour. A new tip correction model is proposed. Comparisons between numerical and experimental data for flows past the NREL combined experiment rotor and the Swedish WG 500 rotor show that only the new model can predict correctly the force in the tip region.

Vorticity-velocity formulation of the 3D Navier Stokes equations in cylindrical coordinates

Aero-acoustic Computations of Wind Turbines

A numerical algorithm for acoustic noise generation is extended to 3D flows. The approach involves two parts comprising a viscous incompressible flow part and an inviscid acoustic part. In order to simulate noise generated from a wind turbine, the incompressible and acoustic equations are written in polar coordinates. The developed algorithm is combined with a
so-called actuator-line technique in which the loading is distributed along lines representing the blade forces. Computations are carried out for the 500kW Nordtank wind turbine equipped with three LM19 blades.

©2001 The American Institute of Aeronautics and Astronautics, Inc. and ASME

Evaluation of the Prandtl Tip Correction for Wind Turbine Computations

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Shen, W. Z., Mikkelsen, R. F., Sørensen, J. N., Bak, C.
Publication date: 2002

Host publication information
Title of host publication: Global Windpower Conference and Exhibition
Source: orbit
Source-ID: 62533

Evaluation of Tip Correction Theories

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Shen, W. Z., Mikkelsen, R. F., Sørensen, J. N., Bak, C.
Publication date: 2002

Host publication information
Title of host publication: 15th IEA Symposium on the Aerodynamics of Wind Turbines
Place of publication: Sweden
Publisher: FOI Swedish Defence Research Agency
Source: orbit
Source-ID: 62594
Research output: Chapter in Book/Report/Conference proceeding – Article in proceedings – Annual report year: 2002 – Research

Numerical Modelling of Wind Turbine Wakes

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Sørensen, J. N., Shen, W. Z.
Aero-acoustic modelling of low Mach-number flows

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Shen, W. Z., Michelsen, J., Sørensen, J. N.
Publication date: 2001

Host publication information
Title of host publication: Proceedings of the 2001 European Wind Energy Conference and Exhibition
Place of publication: Munich
Publisher: WIP-Renewable Energies
Source: orbit
Source-ID: 64242

Aeroacoustic modelling of turbulent airfoil flows

General information
Publication status: Published
Organisations: Coastal, Maritime and Structural Engineering, Department of Mechanical Engineering, Fluid Mechanics
Contributors: Shen, W. Z., Sørensen, J. N.
Pages: 1057-1064
Publication date: 2001
Peer-reviewed: Yes

Publication information
Journal: A I A A Journal
Volume: 39
Issue number: 6
ISSN (Print): 0001-1452
Ratings:
Scopus rating (2001): SJR 1.322 SNIP 1.548
Web of Science (2001): Indexed yes
Original language: English
Source: orbit
Source-ID: 64068

An improved Rhie-Chow interpolation for unsteady flow computations

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Shen, W. Z., Michelsen, J., Sørensen, J. N.
Modelling and analysis of the flow field around a coned rotor

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Mikkelsen, R. F., Sørensen, J. N., Shen, W. Z.
Publication date: 2001
Peer-reviewed: Yes

Publication information
Journal: A I A A Journal
Volume: 39
Issue number: 12
ISSN (Print): 0001-1452
Scopus rating (2001): SJR 1.322 SNIP 1.548
Web of Science (2001): Indexed yes
Original language: English
Source: orbit
Source-ID: 64070

Research output: Contribution to journal › Journal article – Annual report year: 2001 › Research › peer-review

Modelling of aerodynamically generated noise

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Sørensen, J. N., Shen, W. Z.
Publication date: 2001

Host publication information
Title of host publication: Proceedings of 14th IEA Symposium on the Aerodynamics of Wind Turbines
Publisher: FFA
Source: orbit
Source-ID: 64210

Research output: Chapter in Book/Report/Conference proceeding › Article in proceedings – Annual report year: 2001 › Research › peer-review

Yaw analysis using a 3D actuator line model

General information
Publication status: Published
Organisations: Fluid Mechanics, Department of Mechanical Engineering
Contributors: Mikkelsen, R. F., Sørensen, J. N., Shen, W. Z.
Publication date: 2001

Host publication information
Title of host publication: Proceedings of the 2001 European Wind Energy Conference and Exhibition
Aeroacoustic Modelling of Airfoils Flows

General information
Publication status: Published
Organisations: Department of Energy Engineering
Contributors: Shen, W. Z., Sørensen, J. N.
Pages: 146-149
Publication date: 1999

Host publication information
Title of host publication: Proceedings of European Wind Energy Conference EWEC '99, Nice
Source: orbit
Source-ID: 174864
Research output: Chapter in Book/Report/Conference proceeding – Article in proceedings – Annual report year: 1999 – Research – peer-review

Aeroacoustic Modelling of Lowspeed Flows

General information
Publication status: Published
Organisations: Department of Energy Engineering
Contributors: Shen, W. Z., Sørensen, J. N.
Pages: 271-289
Publication date: 1999
Peer-reviewed: Yes

Publication information
Journal: Theoretical and Computational Fluid Dynamics
Volume: 13
ISSN (Print): 0935-4964
Ratings:
Scopus rating (1999): SJR 1.16 SNIP 1.071
Original language: English
Source: orbit
Source-ID: 174859
Research output: Contribution to journal – Journal article – Annual report year: 1999 – Research – peer-review

Analysis of a coned rotor using an actuator disc model

General information
Publication status: Published
Organisations: Department of Mechanical Engineering
Contributors: Mikkelsen, R., Sørensen, J. N., Shen, W. Z.
Pages: 119-132
Publication date: 1999

Host publication information
Title of host publication: Proceedings of the 13th IEA Symposium on the Aerodynamics of Wind Turbines
Publisher: Energy Technology Support Unit
Source: orbit
Source-ID: 176240
Research output: Chapter in Book/Report/Conference proceeding – Article in proceedings – Annual report year: 1999 – Research

Comment on the Aeroacoustic Formulation of Hardin and Pope
Computation of Wind Turbine Wakes using Combined Navier-Stokes/Actuator-line Methodology

Quasi-3D Navier-Stokes Model for Rotating Airfoil

Simulation of a coned rotor using an actuator disc model.
Analysis of wake states by a full-field Actuator disc model

General information
Publication status: Published
Organisations: Department of Energy Engineering
Contributors: Shen, W. Z., Munduate, X.
Pages: 73-88
Publication date: 1998
Peer-reviewed: Yes

Publication information
Journal: Wind Energy
Volume: 1
ISSN (Print): 1095-4244
Original language: English
Source: orbit
Source-ID: 171712
Research output: Contribution to journal › Journal article – Annual report year: 1998 › Research › peer-review

Quasi-3D model for rotating and oscillating airfoil

General information
Publication status: Published
Organisations: Department of Energy Engineering
Contributors: Shen, W. Z., Sørensen, J. N.
Pages: 440-447
Publication date: 1998

Host publication information
Title of host publication: Quasi-3D model for rotating and oscillating airfoil
Source: orbit
Source-ID: 174862
Research output: Chapter in Book/Report/Conference proceeding › Article in proceedings – Annual report year: 1998 › Research

A coupling finite difference/particle method for the resolution of 2D Navier-Stokes equations in velocity-vorticity form

General information
Publication status: Published
Organisations: Department of Energy Engineering
Contributors: Shen, W. Z., Loc, T.
Pages: 97-109
Publication date: 1997
Peer-reviewed: No

Publication information
Journal: Aerospace Science and Technology
Volume: 1
Issue number: (2)
Original language: English
Source: orbit
Source-ID: 176131
Research output: Contribution to journal › Journal article – Annual report year: 1997 › Research
A coupling finite difference/particle method for the resolution of 2D Navier-Stokes equations in velocity-vorticity form

**General information**
Publication status: Published
Organisations: Department of Energy Engineering
Contributors: Shen, W. Z.
Pages: 193-216
Publication date: 1997
Peer-reviewed: Yes

**Publication information**
Journal: Computer & Fluids
Volume: 26
Issue number: 2
Original language: English
Source: orbit
Source-ID: 168817
Research output: Contribution to journal › Journal article – Annual report year: 1997 › Research › peer-review

**Vortex ring state by full-field actuator disc model**

**General information**
Publication status: Published
Organisations: Department of Energy Engineering
Contributors: Sørensen, J. N., Shen, W. Z., Munduate, X.
Pages: 215-232
Publication date: 1997

**Host publication information**
Title of host publication: Proceedings of 10th Symposium on the Aerodynamics of Wind Turbines
Source: orbit
Source-ID: 169370
Research output: Chapter in Book/Report/Conference proceeding › Article in proceedings – Annual report year: 1997 › Research › peer-review

**Projects:**

**FarmOpt: Wind Farm Layout Optimization in Complex Terrain**
The overall objective of the project is to develop and provide new reliable tools for designing wind farms located in complex terrain through full scale measurements in wind farms. For wind farms located in flat terrain, the performance of the wind turbines is significantly influenced by the upstream wind turbines and slightly influenced by the ground. For wind farms located in complex terrain the ground effects are relatively more pronounced, as such effects may bend the wakes created by the upstream turbines significantly. The goal of the present Sino-Danish project is to further develop Danish wind farm technology by using measured wind farm data from complex terrain wind farms in China, which is convenient, as Denmark does not have complex terrain that can be used for developing/validating such technology. To improve the wind turbines' performance within wind farms in complex terrain, there are basically three important steps: (1) develop reliable CFD tools for predicting flow in complex terrain with and without wind turbines; (2) develop simplified flow models for predicting wind turbine performance in complex terrain; and (3) design high efficiency wind turbine parks in complex terrain.

Shen, W. Z., Project Coordinator, Department of Wind Energy, Fluid Mechanics
Zhu, W. J., Project Manager, Department of Wind Energy
Hansen, K. S., Project Manager, Department of Wind Energy, Fluid Mechanics
Bechmann, A., Project Manager, Department of Wind Energy, Resource Assessment Modelling
Larsen, G. C., Project Manager, Department of Wind Energy, Wind turbine loads & control
Feng, J., Project Manager, Department of Wind Energy, Fluid Mechanics

Project ID: EUDP-64013-0405
01/04/2014 → 31/12/2017
Keywords: Wind Farms
Collaborators: EMD International A/S
Project: Research

**OffWindChina: Research and Development of optimal Wind turbine rotors under offshore wind conditions in China**
The scientific objectives of the project are to develop new aerodynamic and structural design tools, and control techniques for optimizing wind turbine rotors for offshore wind energy applications in China. During the past five years, DTU has
established a strong research collaboration network with Chinese universities and research institutes in the area of wind energy. The present proposal will further strengthen the collaboration. To develop wind technology under offshore wind conditions in China, it demands the insights of the physics of wind turbine flows under local wind conditions and the development of novel computational techniques that are capable to design and predict the performance of wind turbines. The goal is to make offshore wind energy production more competitive through fundamental insights into the interaction between atmospheric turbulence and wind turbines. Further, wind turbines under offshore conditions in China can be operated optimally through the design of efficient control systems.

Shen, W. Z., Project Coordinator, Department of Wind Energy, Fluid Mechanics
Zhu, W. J., Project Manager, Department of Wind Energy
Aagaard Madsen, H., Project Manager, Department of Wind Energy, Aerodynamic design
Sørensen, J. N., Project Manager, Department of Wind Energy, Fluid Mechanics

Project ID: IFD-0603-005068
01/04/2012 → 30/06/2017

**Development of an advanced noise propagation model for noise optimization in wind farm**

Barlas, E., PhD Student, Department of Wind Energy
Shen, W. Z., Main Supervisor
Sørensen, J. N., Supervisor
Zhu, W. J., Supervisor
Aagaard Madsen, H., Examiner
Burdisso, R., Examiner
Sendergaard, B., Examiner
Samfinansieret - Andet
01/11/2014 → 06/03/2018

Award relations: Development of an advanced noise propagation model for noise optimization in wind farm
Project: PhD

**Experimental characterization of airfoil boundary layers for improvement of aeroacoustic and aerodynamic modelling**

Fischer, A., PhD Student
Aagaard Madsen, H., Main Supervisor
Bertagnolio, F., Supervisor
Shen, W. Z., Supervisor
Sørensen, J. N., Examiner
Keith, G., Examiner
Roger, M., Examiner
Riso (Løn)
01/10/2008 → 22/02/2012

Award relations: Experimental characterization of airfoil boundary layers for improvement of aeroacoustic and aerodynamic modelling
Project: PhD

**Development of Large Eddy Simulation Tools for Simulation of Atmospheric Boundary Layers in Wind Farms**

Dag, K. O., PhD Student, Department of Wind Energy
Sørensen, J. N., Main Supervisor
Shen, W. Z., Supervisor
Sørensen, N. N., Supervisor
Berg, J., Examiner
Churchfield, M. J., Examiner
Meyers, J., Examiner
Offentlig finansiering
01/11/2013 → 07/12/2017

Award relations: Development of Large Eddy Simulation Tools for Simulation of Atmospheric Boundary Layers in Wind Farms
Project: PhD

**Simulation of Moving Trailing Edge Flaps on a Wind Turbine Blade using Navier-Stokes based Immersed Boundary Method**

Behrens, T., PhD Student, Department of Mechanical Engineering
Shen, W. Z., Main Supervisor
Sørensen, J. N., Supervisor
Sørensen, N. N., Supervisor
Wedel-Heinen, J. J., Supervisor
Gaunaa, M., Examiner
Bijl, H., Examiner
Davidson, L., Examiner
Ansat eksternt
15/07/2008 → 21/12/2011
Award relations: Simulation of Moving Trailing Edge Flaps on a Wind Turbine Blade using Navier-Stokes based Immersed Boundary Method
Project: PhD

DCAMM
Zhu, W. J., PhD Student, Department of Mechanical Engineering
Shen, W. Z., Main Supervisor
Sørensen, J. N., Supervisor
Sørensen, N. N., Examiner
Ekaterianris, I. A., Examiner
Thomsen, P. G., Examiner
Forskningsrådsfinansiering
01/06/2004 → 07/03/2008
Award relations: DCAMM
Project: PhD

Design of Large wind turbines using fluid-structure coupling technique
Sessarego, M., PhD Student, Department of Wind Energy
Shen, W. Z., Main Supervisor
Ramos Garcia, N., Supervisor
Sørensen, J. N., Supervisor
Aagaard Madsen, H., Examiner
Madsen, J., Examiner
Schepers, G., Examiner
Samfinansieret - Andet
01/11/2013 → 16/02/2017
Award relations: Design of Large wind turbines using fluid-structure coupling technique
Project: PhD

Simulation and prediction of wakes and wake interaction in wind farms
Andersen, S. J., PhD Student, Department of Wind Energy
Sørensen, J. N., Main Supervisor
Mikkelsen, R. F., Supervisor
Shen, W. Z., Supervisor
Mann, J., Examiner
Meyers, J., Examiner
Ivanell, S. S. A., Examiner
1/3 FUU, 1/3 inst 1/3 Andet
01/06/2010 → 27/01/2014
Award relations: Simulation and prediction of wakes and wake interaction in wind farms
Project: PhD

Simulation of flows past a wind turbine with wind shear using Navier-Stokes based sliding mesh technique
Kolmogorov, D., PhD Student, Department of Wind Energy
Shen, W. Z., Main Supervisor
Sørensen, J. N., Supervisor
Zhu, W. J., Supervisor
Zahle, F., Examiner
Madsen, J. I., Examiner
Bijl, H., Examiner
Technical University of Denmark
01/02/2011 → 29/09/2014
Award relations: Simulation of flows past a wind turbine with wind shear using Navier-Stokes based sliding mesh technique
Project: PhD
**Simulation and Modelling of Wakes and Wake Interaction in Offshore Wind Farms**
Sariek Chivaee, H., PhD Student, Department of Wind Energy
Serensen, J. N., Main Supervisor
Mikkelsen, R. F., Supervisor
Shen, W. Z., Supervisor
Walthier, J. H., Examiner
Davidson, L., Examiner
Olesen, N. A., Examiner
Technical University of Denmark
01/01/2011 → 25/08/2014
Award relations: Simulation and Modelling of Wakes and Wake Interaction in Offshore Wind Farms
Project: PhD

**3D Modelling of Laminar-Turbulent Transition on Wind Turbine Blades**
Martinez Hernandez, G. G., PhD Student, Department of Mechanical Engineering
Serensen, J. N., Main Supervisor
Shen, W. Z., Supervisor
Serensen, N. N., Examiner
Matsubara, M., Examiner
Olesen, N. A., Examiner
Centerfinansiert
01/12/2005 → 09/11/2011
Award relations: 3D Modelling of Laminar-Turbulent Transition on Wind Turbine Blades
Project: PhD

**Quasi-3d aerodynamic code for analyzing dynamic flap response**
Ramos Garcia, N., PhD Student, Department of Mechanical Engineering
Serensen, J. N., Main Supervisor
Shen, W. Z., Supervisor
Andersen, P., Examiner
Sun, Y., Examiner
Voutsinas, S., Examiner
Technical University of Denmark
01/04/2008 → 28/09/2011
Award relations: Quasi-3d aerodynamic code for analyzing dynamic flap response
Project: PhD

**43033-4610: TURBOPT**
The project aims to develop the calculation of energy production and loads on wind turbines by develop and optimize integrated models, which is able to handle the multi-scale phenomena in complex terrain.
Aagaard Madsen, H., Project Manager, Department of Wind Energy, Aeroelastic Design
Henriksen, L. C., Project Participant
Fischer, A., Project Participant, Department of Wind Energy, Aeroelastic Design
Shen, W. Z., Project Participant, Department of Wind Energy, Fluid Mechanics
01/04/2011 → 30/06/2014
Collaborators: Chinese Academy of Sciences
Project: Research

**NextRotor: Design of next generation wind turbine rotors**
Shen, W. Z., Project Manager
Serensen, J. N., Project Manager
Zhu, W. J., Project Manager
Madsen, J., Project Manager, LM Wind Power
Project ID: 76206
Miljøstyrelsen: DKK16,716,501.00
01/07/2011 → 30/06/2014
Collaborators: Technical University of Denmark, LM Wind Power
Award relations: Design of next generation wind turbine rotors
Project: Research

**Aerodynamics and optimization of wind turbines in complex terrain**
This is a Sino-Danish collaboration project funded by DSF. The objective of the project is to develop numerical tools to simulate wind turbine flows in complex terrain, and optimize and control wind turbines in complex terrain.
Shen, W. Z., Project Manager
Zhu, W. J., Project Manager
Sørensen, J. N., Project Manager
Aagaard Madsen, H., Project Manager
Forskningsrådene - Andre
01/01/2011 → 31/12/2014
Collaborators: Technical University of Denmark
Award relations: Aerodynamics and optimization of wind turbines in complex terrain
Project: Research

**VISCWIND (EU) 3-D and dynamic stall modelling**
Sørensen, J. N., Project Manager, Department of Energy Engineering
Hansen, M. O. L., Project Participant, Department of Energy Engineering
Shen, W. Z., Project Participant, Department of Energy Engineering
01/01/1996 → ...
Project: Research

**Activities:**

**Wind farm design in complex terrain - the FarmOpt methodology**
Period: 18 Oct 2017
Ju Feng (Invited speaker)
Wen Zhong Shen (Other)
Department of Wind Energy
Fluid Mechanics

**Description**
Invited speaker at the conference on 18th October in the session "Wind Farm Micro Siting".
Degree of recognition: International

Documents:
Wind farm design in complex terrain - the FarmOpt methodology _Ju Feng _DTU (2017)

**Related event**

**China Wind Power 2017**
17/10/2017 → 19/10/2017
Beijing, China
Keywords: wind farm, design optimization, complex terrain, FarmOpt
Activity: Talks and presentations › Conference presentations