Technical University of Denmark

Wind turbine aerodynamics using CFD

Johansen, Jeppe; Zahle, Frederik; Bertagnolio, Franck; Sørensen, Niels N.

Publication date:
2007

Citation (APA):
Johansen, J., Zahle, F., Bertagnolio, F., & Sørensen, N. N. (2007). Wind turbine aerodynamics using CFD. Poster session presented at Internal DTU conference on future energy research at the Technical University of Denmark, Roskilde, Denmark.

Wind Turbine Aerodynamics using CFD

RISØ

Jeppe Johansen, Frederik Zahle and Franck Bertagnolio, Wind Energy Department, Risø-DTU
Niels N. Sørensen, Wind Energy Department, Risø-DTU and Ålborg University

Introduction

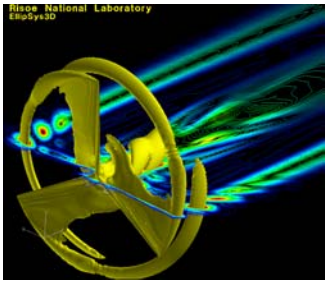
Computational fluid dynamics (CFD) gives a unique insight into the complex flow phenomena that govern the aerodynamics of wind turbines. Through a dedicated research and development effort CFD is steadily reaching a level of maturity, where it is becoming indispensable in the analysis of modern wind turbine designs.

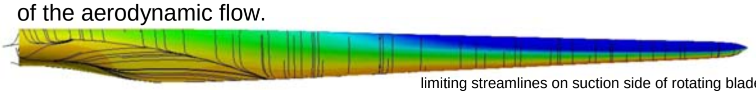
Over the last 15 years the incompressible general purpose flow solver, **EllipSys**, has been developed in a close cooperation between the Fluid Mechanics section, DTU and the Wind Energy Dept. Risø-DTU.

A variety of wind turbine aerodynamic applications are shown.

Rotor Aerodynamics

An important application of CFD is in the verification process of actual rotor designs. By simulating the flow around a given rotor design, power and thrust forces can be estimated to verify the design objectives prior to actual production. Detailed information on surface flow and tip vortices can be visualised to gain physical understanding of the aerodynamic flow.





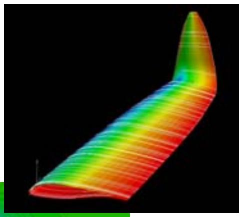
limiting streamlines on suction side of rotating blade

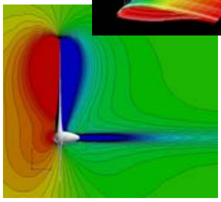
Model development

A continuous effort on further developing and optimising the code is taking place. Of the key issues can be mentioned turbulence modelling, modelling of laminar to turbulence transition, modelling of atmospheric inflow turbulence, fully coupled aeroelastic simulations, efficient high quality computational mesh generation, development of numerical algorithms etc..

Design changes

Apart from expensive full scale measurements only CFD can be used to analyse design changes such as e.g. adding winglets, increasing root chord or having an egg-shaped nacelle.

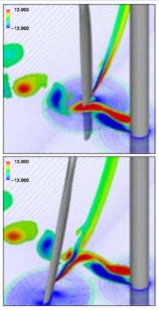


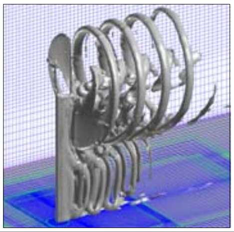


pressure contours in vertical plane

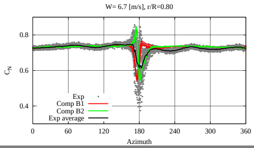
Rotor-Tower interaction

For investigating the interaction between the rotor and the tower, a new method called the overset grid method has been implemented, which allows for individual components in the grid to move relative to each other. This enables us to simulate the flow over the entire turbine including also the tower, nacelle and ground boundary.





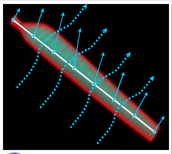
The figures show to the left an iso-vorticity surface around a two-bladed down-wind turbine. To the right interaction between a passing blade and vortex shedding behind the tower is shown.

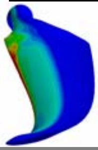


Aeroelasticity

By coupling the CFD solver to the aeroelastic code HAWC2, aeroelastic stability and damping of a given rotor blade can be determined.

The aerodynamic forces deform the surface of the blade according to the structural properties and modal properties can be determined.





Outlook

The strength of using CFD in wind turbine rotor design is the analysis of the very detailed flow physics. New physical insight can be gained to further develop faster engineering type models, which are used in the wind turbine industry for designing wind turbines.

In the future the CFD models will be coupled to rotor design codes, which currently are based on simpler engineering type models. By also applying optimisation methods the advanced CFD methods will eventually replace these simpler methods.

Another aim is to be able to compute flow around an entire flexible wind turbine including tower and nacelle placed on a surface with an inflow containing both atmospheric turbulence and shear.

Finally, continuous validation against high quality measurements is a very important issue and will still be an ongoing effort in the future.

Shear flows

Traditionally, rotor computations are performed assuming uniform inflow. In reality the mean inflow is increasing with height (red arrows) which causes unsteady variation of loads on the rotor during operation. To provide more physical input to engineering type models unsteady computations with logarithmic wind profiles are performed.