Faculty Engineering Technology Program
Mechanical Engineering

UNIVERSITY OF TWENTE.

COLLOQUIUM

Group: **Engineering Fluid Dynamics**

As part of his MSc thesis assignment

Sebastiaan ten Pas

will give a presentation, entitled:

CFD simulations of the MEXICO wind turbine

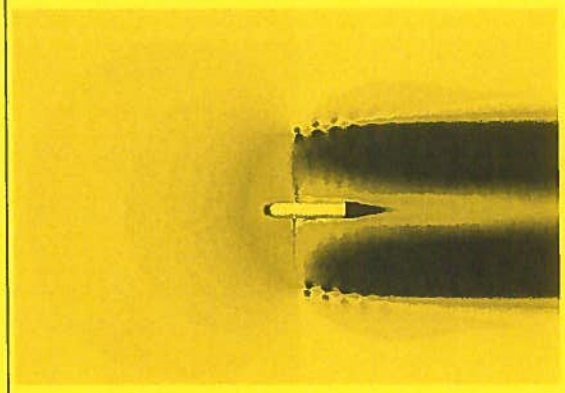
Validating ENSOLV for wind turbine flows

Date: Wednesday July 13, 2016
Time: 14:00
Room: Horst Building Room N-109

Summary:
To fulfil the increasing demand for sustainable energy, upscaling of wind turbines has been going on over the past decades. Large wind turbines are reported to lead to a number of specific aerodynamic and load problems. The current study aims at validating NLR's CFD solver ENSOLV for wind turbine flow. The experimental data is obtained from the New MEXICO project. A 3-bladed 4.5 meter diameter HAWT is tested in the LLF wind tunnel of DNW.

A structured multi-block grid is built consisting of approximately 9 million cells. Only one blade is simulated with the use of axisymmetric boundary conditions. Three simulation cases are selected which represent turbulent wake state, design conditions and separated flow conditions. The selected experimental data consists of pressure distribution data at five radial sections and PIV flow data at two axial traverses and two radial traverses.

The results of the simulated pressure distribution agree reasonably well with the measurements of the New MEXICO experiment. Differences are mainly found at the suction side of the airfoil. For the radial section of 35%, 82% and 92% these differences are reasonably small, but for 25% and 60% the differences are larger. For 25% radial section differences also occur at the pressure side of the airfoil. The differences at the suction and pressure side follow from the error in the experiments due to the uncertainty of the pressure sensors. The differences at 60% span are most probably due to the limitations of the turbulence models, but further research has to be done to confirm this hypothesis.



The velocities in the wake are measured with PIV sheets in two axial traverses (r = 1.37 m and r = 1.85 m) and two radial traverses (x = −0.15 m and x = 0.15 m). The results from the simulation agree reasonably well to the experimental data for both the axial and radial traverses. Differences are mainly found in the axial traverses after the flow passes the rotor, where the fluctuations in the velocity dissipate faster in the simulation.

Overall it can be concluded that NLR's CFD solver ENSOLV is capable of simulating wind turbine flow reasonably well.

Assessment committee:

Prof.dr.ir. C.H. Venner	(chairman)
Ir. J. van Muijden	(supervisor NLR)
Ir. A. van Garrel	(supervisor UT)
Dr. A. Pozarlik	(external member)

Chairman,

d.d. 15/06/2016