LES-SIMULATION OF A TURBULENT AND MEANDERING WAKE

Dipl.-Ing. Steffen Wussow, Fluid & Energy Engineering GmbH & Co. KG
Dr.-Ing. Thomas Hahm, Fluid & Energy Engineering GmbH & Co. KG

Fluid & Energy Engineering GmbH & Co. KG
Dr.-Ing. Thomas Hahm
Borsteler Chaussee 178
D-22453 Hamburg
Germany
Phone: +49-(0)40-55549081
Facsimile: +49-(0)40-55549084
e-mail: hahm@fluid2.de

SUMMARY

The airflow around two wind turbines of the type ENERCON E66, hub height 65m, was simulated. The simulations were undertaken with the Computational Fluid Dynamics (CFD) software ANSYS FLUENT 12 using the LES technique for turbulence modelling.

The incoming flow field was specified using a three component von-Karman turbulent wind model including wind shear.

A comparison with measurement data at a distance of 2.06 rotor diameter shows, that the current model captures both the velocity deficit and the turbulence intensity behind the wind turbine.

The wake inherently starts to meanders during the simulation without using any further model assumptions. Quick horizontal shifts of the wake from one side to another are detectable on a 25s time-scale but no general meandering pattern was visible during the simulated time of 300s. In the near wake region up to approximately 3 rotor diameter downstream of the WT high velocity gradients at the edges of the wake are present and contribute much to the local turbulence.

As the application of the current models for wind loads is uncertain in near wake situations there is a need to further clarify this especially in the context of onshore repowering projects where the spacing of wind turbines typically is very close.

CFD can provide enough data on a spatial and temporal resolution to predict loads in near wake situations more precisely and address questions like the influence of ambient turbulence intensity or wind speed on the wake characteristics.

Introduction

Wind turbines (WT) are subjected to environmental and electrical conditions that may affect their loading, durability and operation. The environmental conditions are further divided into wind conditions and other environmental conditions. Wind conditions are the primary external conditions affecting structural integrity.

The international IEC standard 61400-1 [1] defines the design requirements for wind turbines and classifies different wind turbine classes in terms of wind speed and turbulence parameters. The intention of these classes is to cover most applications. The main parameters are the reference wind speed average over 10 minutes (v-ref) and the expected value of the turbulence intensity at 15 m/s (I-ref). The latter one is defined as ratio of the wind speed standard deviation to the mean wind speed, determined from the same set of measured data samples of wind speed and in this context defined over a 10 minute period of time.
The values of wind speed and turbulence intensity are intended to represent many different sites and do not give a precise representation of any specific site or even a specific wind farm layout, where the individual WT is subject to the influence of nearby turbines

According to [1] the assessment of the suitability of the WT at a site in a wind farm shall therefore take into account the deterministic and turbulent flow characteristics associated with single or multiple wakes from upwind machines, including the effects of the spacing between the machines, for all ambient wind speeds and wind directions relevant to power production.

The increase in loading generally assumed to result from wake effects may be accounted for by the use of an effective turbulence intensity, which shall include adequate representation of the effect on loading of ambient turbulence and turbulent wake effects. For fatigue calculations the standard [1] describes a method to calculate the effective turbulence intensity as a superposition of wake and non-wake situation using the Wöhler (SN-curve) exponent for the considered material. Today these effective turbulence intensities together with the undisturbed wind speed probability distribution serve as input parameters for site specific load calculations. This means, all increase in loading from neighbouring WTs is projected onto an increased level of turbulence uniformly distributed across the rotor plane while a wind deficit is absent. This even holds true for single wake situations.
This approach may be limited in near wake situations where a distinct wind deficit with high velocity gradients is still present. At least the validity of the described method is uncertain for WT spacing less than 3 rotor diameters [1]. On the other hand, WT spacing of approximately 3 rotor diameters is common praxis in onshore repowering projects.

A better understanding of fatigue and ultimate loads in near wake situations would therefore be desirable but requires knowledge of the full three vector component turbulent wind velocity field. No model exists at the moment to describe a turbulent wind field within the full extent of the wake. Measured data are typically collected with a single mast to hold the anemometers. This means, no information is given to account for the spatial correlation structure of the velocity components.

It is the approach of this work to validate a Computational Fluid Dynamics (CFD) model of a single wake situation in order to provide more information about the near wake wind field. Calculations are focused on a WT of the type ENERCON E66, hub height 65m, to compare to data collected by ultrasonic anemometers during a research project of the “Deutsche Institut für Bautechnik” [2], which is the German authority that draws up the technical rules in the field of civil engineering.

Description of the model

The CFD-model contains the full geometry of two WT of the type ENERCON E66, hub height 65m. The blade geometry itself is based on a set of 2D-profiles and gives a reasonable representation of the aerodynamic relevant structure. However, it was not the intent to generate a full aerodynamic model of the blades, which would be able to reproduce lift, drag and overall power coefficients, as the computational effort needed would have been to expensive in the context of this work.

The rotor speed was fixed at a constant value, which corresponds to the mean wind speed at hub height of the incoming wind field. The blades of the WTs can be pitched individually during the simulation.

The computational domain extends from approximately 1.5 rotor diameter upwind of the first WT to three rotor diameter downwind of the second WT. The distance between the two WTs amounts to 4.25 rotor diameter.
The rotating part of the nacelles and the rotor are inserted in a dynamically meshed volume with sliding interfaces to the rest of the computational domain. The whole domain was meshed with 7.8 million hexaeder cells.

The simulations were undertaken with the Computational Fluid Dynamics (CFD) software ANSYS FLUENT 12 using the LES technique for turbulence modeling at a time step size of 0.1 seconds. The Smagorinsky-Lilly model was chosen as subgrid scale model. The incoming flow field was specified using a three component von-Karman turbulent wind model including wind shear, which is common praxis in BEM-based (blade element momentum theory) modeling of WT for load calculations. The ten minute averaged turbulence intensity was set at 0.1 in correspondence to measured mean values of ambient turbulence intensity on the site. Mean wind speed at hub height was 10m/s.

Approximately 50.000 measuring points at positions 2, 2.5, 3 and 3.5 rotor diameter behind the first WT were monitored at a rate of 10Hz during the simulation storing data of all three velocity components.

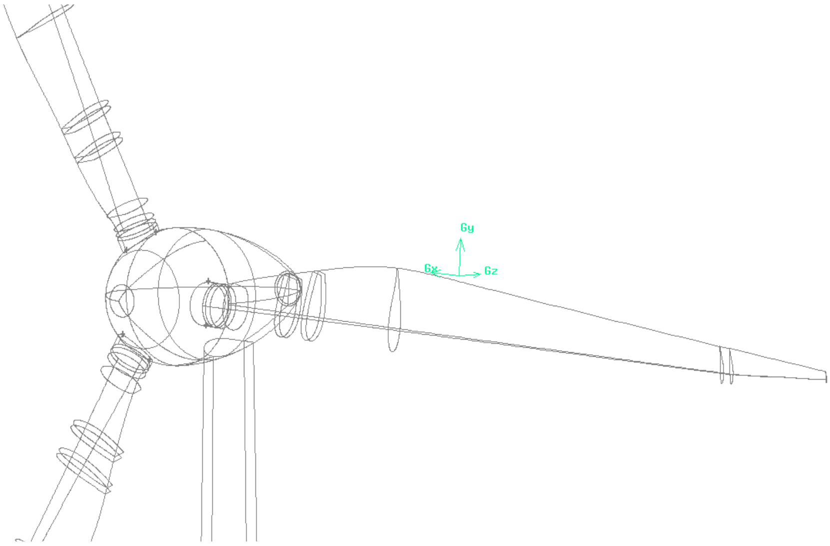

Figure 1: detail of the geometry model of the wind turbine ENERCON E66.

Comparison to measurements

Validation of the results was done using data collected at 2.06 rotor diameter behind a WT of the type ENERCON E66, hub height 65m, in [2]. Longitudinal, vertical and horizontal components of wind speed have been measured in [2] at hub height, hub height minus half rotor radius and hub height minus one rotor radius using ultrasonic anemometers. However only data at hub height in the range of 4 to 8 and 8 to 12m/s wind velocity was available for validation. The latter set of data was used for comparison with our simulation at 10m/s.

In figures 2 and 3 every point represents one ten minute mean value of measured data - either velocity magnitude or local turbulence intensity. The abscissa denotes wind direction, with the met mast and thus the centre of the wake located at 305°.

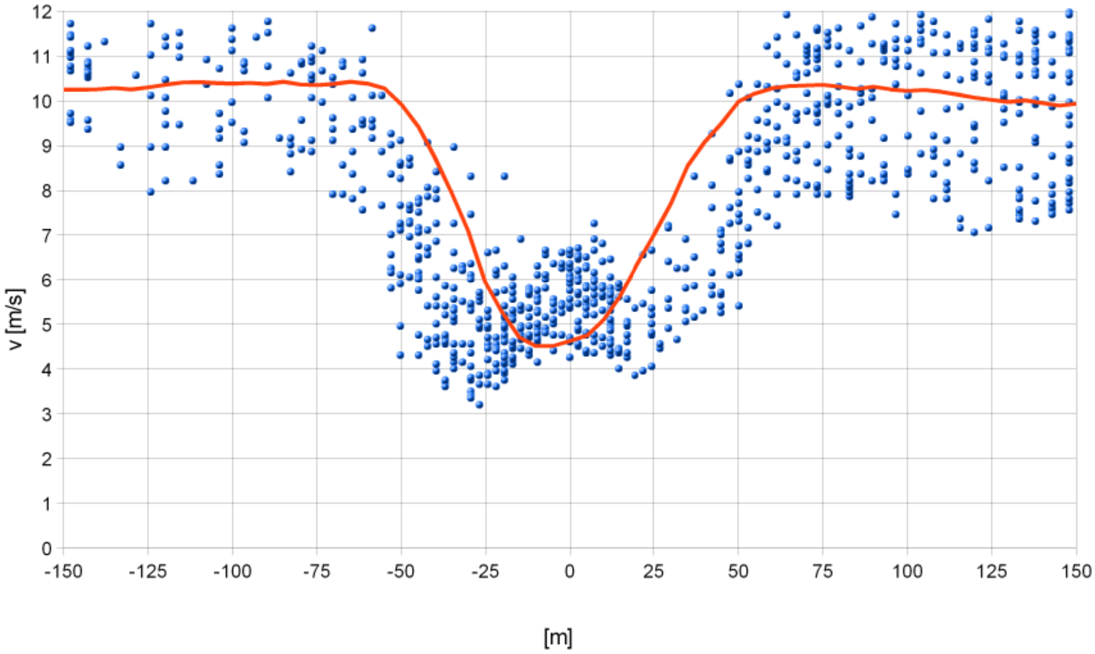

Figure 2: Measured data of velocity magnitude at hub height, incoming velocity 8-12m/s. Comparison of 10-minute mean values to 5-minute mean values of simulated data (red line) at an incoming velocity of 10m/s.

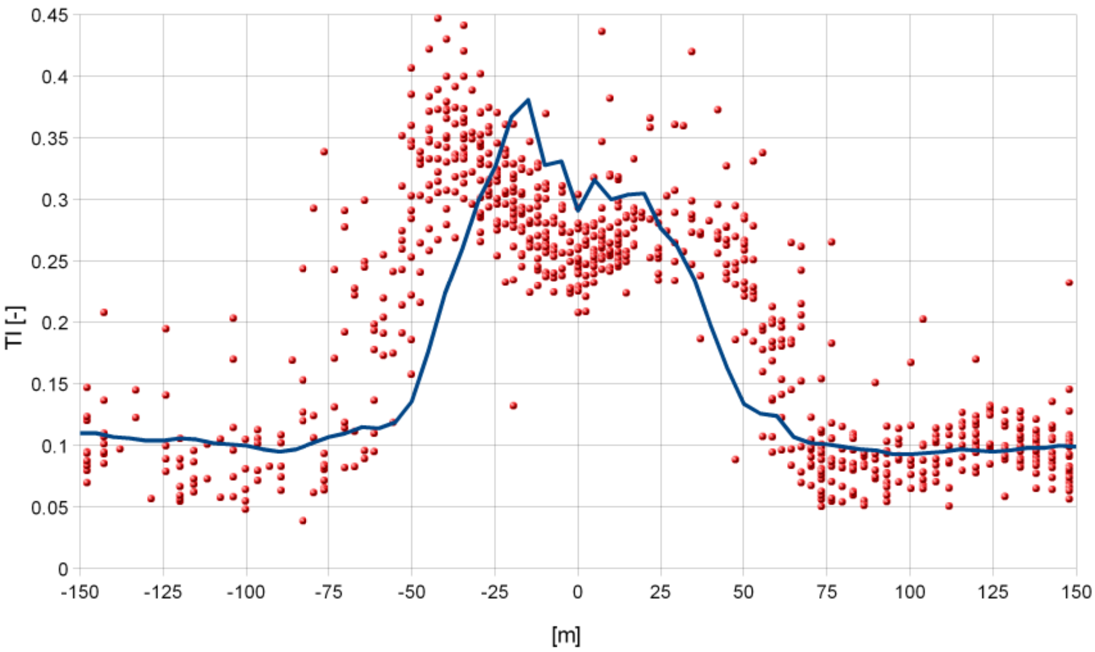

Figure 3: Measured data of local turbulence intensity at hub height, incoming velocity 8-12m/s. Comparison of 10-minute mean values to 5-minute mean values of simulated data (blue line) at an incoming velocity of 10m/s.

In contrast to our simulations in 2007 [3] the maximum deficit and the maximum turbulence intensity level is well captured. This is probably due to an improved computational grid. In [3] the thrust coefficient of the rotor taken from the simulation was constantly 20 – 30% lower than the expected value at 10m/s but is now in good agreement.

In both figures 2 and 3 the width of the wake is clearly smaller than the measured wake data.

The simulation data was averaged over a 5-minute-period only in contrast to the measured 10-minute averages. This might bring a bit more variation in the lateral direction and the measured data generally represents a much broader range of flow conditions concerning ambient turbulence intensity and average wind speed compared to the single simulation test case. It is not known to what extent the three component von-Karman model differs from the real wind field properties. But it is quite obvious, that an increase in wind direction changes will broaden the wake.

Figure 4 shows a comparison of measured and calculated time series of velocity magnitude in the centre of the wake two rotor diameter behind the first WT. A detailed analysis of this data concerning frequency spectrum etc. has not yet been performed.

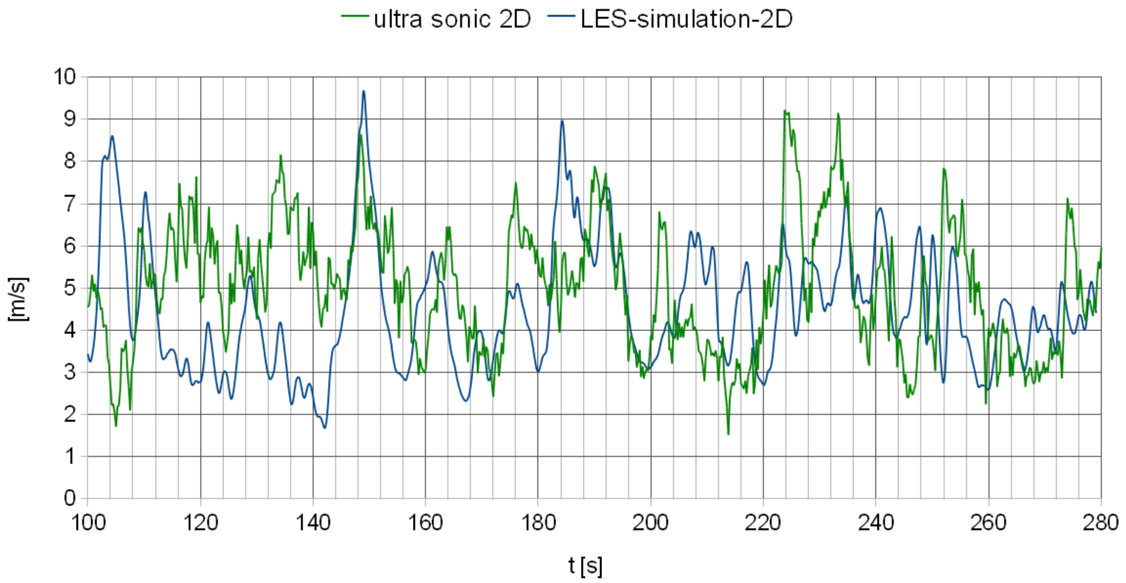

Figure 4: Example of measured (green) and calculated (blue) time series of velocity magnitude at hub height in the centre of the wake two rotor diameter behind the first wind turbine.

Wake meandering

The wake meanders during the simulation horizontally and vertically and is sufficiently smaller than known from measurement data averaged over ten minutes. I.e. the broadening of the wake as observed in this data is a consequence of the meandering and not present on smaller time scales.
Figure 5 gives a snapshot of the unsteady turbulent wake and shows contour plots of velocity magnitude ranging from 2 to 16m/s. White areas denote wind speeds outside this range, which in this case are areas of low wind speed in the vicinity of the walls and close downstream of the tower.

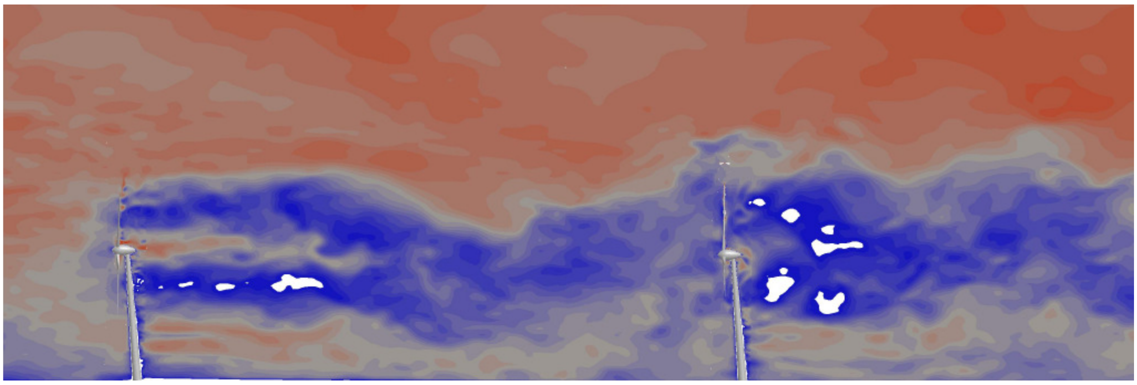

Figure 5: Contours of velocity magnitude in m/s at an average incoming velocity of 10m/s at hub height (snapshot in a vertical plane). See figure 8 for captions.

Calculated time series of four locations 3 rotor diameter downstream of the first WT were examined to have a closer look at the horizontal meandering. The four points are located at hub height 30m and 50m lateral of the centre wake line. Looking from the tip of the blade they approximately represent directions of 0° and 10° (see figure 6).

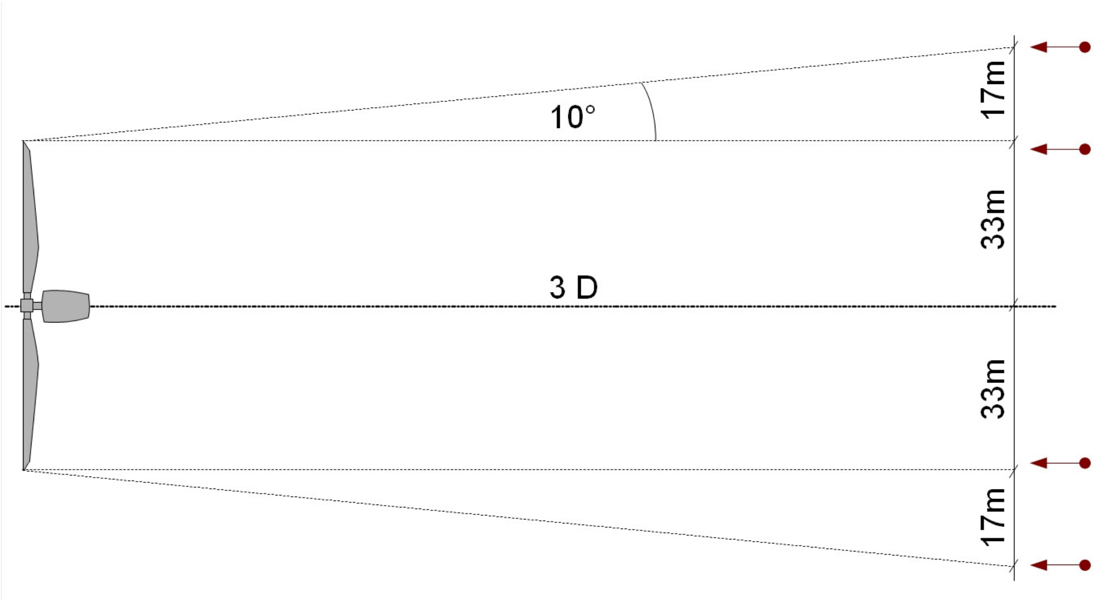

Figure 6: location of monitoring points used for evaluation of time series.

Figure 7 shows the calculated time series of velocity magnitude. The upper two curves at a distance of -30m and -50m from the wake centre line represent the left side of the wake (looking in flow direction) whereas the lower two curves represent the right side.

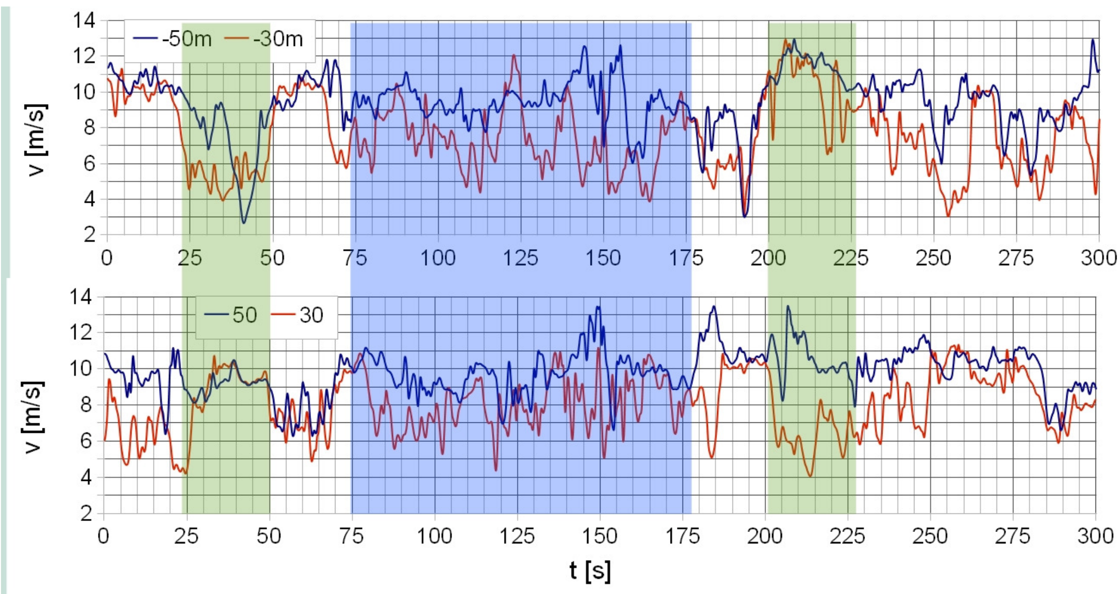

Figure 7: calculated time series of velocity magnitude at four monitoring points.

In figure 7 three time intervals are marked. The first one ranges from 25 to 50s. The calculated time series indicate that in this time interval the wake has completely shifted to the left and covers even the 10°-direction (top blue line) while the lower two lines both are around 10m/s wind velocity. This period is followed by an quite abrupt change of the wake to the right within a few seconds where the wake remains for another 25s. However there is no clear meandering pattern as can be seen in the second time interval which ranges from 75 to 175s. During this period little or no meandering of the wake is detectable as the 10°-directions (blue lines) do not show longer periods of velocity decrease. During the third time period from 200 to 225s the wake is only detectable at the 0°-direction of the right wake side (lower red line). This indicates that the wake is only slightly shifted and is comparable small in its horizontal extension. Snapshots of the velocity field in figure 8 illustrate these three different time intervals.

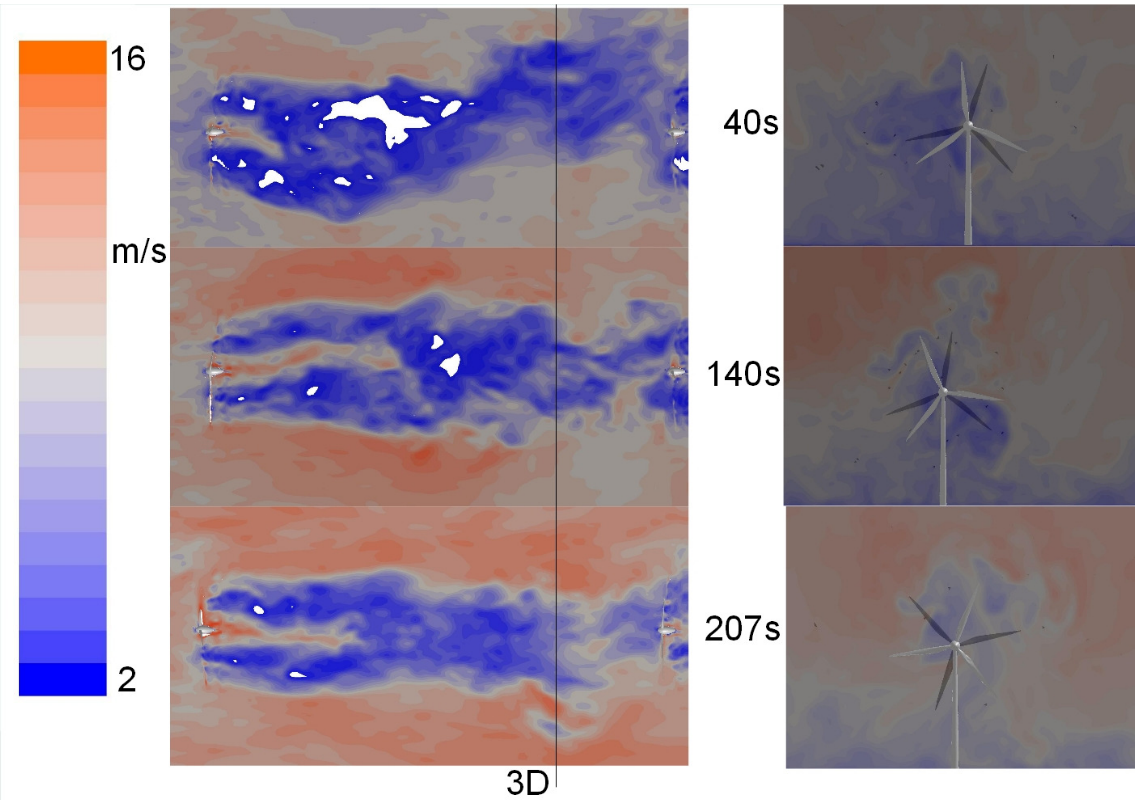

Figure 8: snapshots of the velocity field at 40s, 140s and 207s. Pictures on the left show contours of velocity magnitude on a horizontal plane at hub height and pictures on the right show contours of velocity magnitude on a plane 3 rotor diameter downstream of the first WT (here the second WT is visible in dark grey),

Concerning turbulence intensity the value of the upper blue curve in Figure 7 (10°-direction to the left side of the wake) for example amounts to 17.4% as a 300s-average but only to about 11.8% as a 100s-average taken for the time period from 75s to 175s which is quite close to the value of ambient turbulence intensity. This once again stresses that the high turbulent loads of the near wake mainly depend on the meandering of a quite sharply defined wake with high velocity gradients at the wake edges.

Conclusions

Computational Fluid Dynamics (CFD) using Large-Eddy-Simulation (LES) for turbulence modelling is able to predict the characteristics of a three-dimensional turbulent flow behind a WT. The wake meanders horizontally and vertically and is sufficiently smaller than known from measurement data averaged over ten minutes. Quick horizontal shifts of the wake from one side to another are detectable on a 25s time-scale but no general meandering pattern was visible during the simulated time of 300s.

In the near wake region up to approximately 3 rotor diameter downstream of the WT high velocity gradients at the edges of the wake are present and contribute much to the local turbulence. It is not clear if the present approach which covers the effects of wake loading by an increased level of turbulence uniformly distributed across the rotor plane while a wind deficit is absent is still applicable in the near wake region. Nor do full three-dimensional wind models exist which include the near wake characteristics to perform realistic load calculations with BEM-based (blade element momentum theory) models.

CFD can provide enough data on a spatial and temporal resolution to predict loads in near wake situations more precisely and address questions like the influence of ambient turbulence intensity or wind speed on the wake characteristics.

Especially for repowering of WTs in onshore projects spacing of WTs is often in the range of 3 rotor diameters and the topic of wake loads in these wind farm configurations becomes more and more urgent.

References

1. International Electrotechnical Commission; IEC 61400-1 Edition. 3.0, 2005-08: Wind turbines – Part 1: Design requirements.
2. Deutsches Institut für Bautechnik DIBt; Untersuchung des Nachlaufes von Windenergieanlagen und dessen Auswirkung auf die Standsicherheit der benachbarten WEA in Parkaufstellung; Forschungsvorhaben P 32-5-3.78-1007/02.
3. Steffen Wussow, Lars Sitzki, Thomas Hahm; 3D-simulation of the turbulent wake behind a wind turbine; The Science of Making Torque from Wind, Journal of Physics: Conference Series 75 (2007) 012033.