

Performance and wake development behind two in-line and offset model wind turbines –
"Blind test" experiments and calculations

This content has been downloaded from IOPscience. Please scroll down to see the full text.

2014 J. Phys.: Conf. Ser. 524 012171

(<http://iopscience.iop.org/1742-6596/524/1/012171>)

View [the table of contents for this issue](#), or go to the [journal homepage](#) for more

Download details:

IP Address: 158.39.2.66

This content was downloaded on 19/12/2014 at 09:41

Please note that [terms and conditions apply](#).

Performance and wake development behind two in-line and offset model wind turbines - "Blind test" experiments and calculations

Lars Sætran^a, Per-Åge Krogstad^a, Muyiwa Samuel Adaramola^b

^aNorwegian University of Science and Technology NTNU, Trondheim, Norway,

^bNorwegian University of Life Sciences, Ås, Norway

corresponding author: lars.satran@ntnu.no

Abstract. This is a report on data presented at the "Blind test 3" Workshop organized jointly by Nowitech and Norcowe in Bergen, December 2013. A number of research groups were invited to predict the performances and the wake development behind two model wind turbines that have been extensively tested at the Department of Energy and Process Engineering, NTNU. The turbines were arranged in-line, but slightly offset so that the wake of the upstream turbine only affected roughly half the area swept by the second rotor. This is a common event in most wind parks and produces flow fields that are both complicated and harmful for the downstream turbine. Contributions were received from five different groups using a range of methods, from fully resolved Reynolds averaged Computational Fluid Dynamics (CFD) models to Large Eddy Simulations (LES). The range of results was large but the overall trend is that the current methods predict the power generation as well as the thrust force reasonably well. But there is a large uncertainty in the prediction of the turbulence field in the wake.

Keywords: Wind turbines, prediction method comparison, blind test

1. The experiment

The models were mounted in-line, with a separation of only 3 diameters and tested in the wind tunnel that has a test section almost 12m long. The short distance between the turbines was necessary to allow a reasonable fetch for the wake to develop downstream of the second turbine. The test section height is 2m and the width 3m. For full details on the wind tunnel and instrumentation, see one of the references, [1] or [2]. The upwind turbine was placed two diameters from the entrance to the tunnel test section. We denote this turbine T1 (rotor diameter of $D_1 = 0.944\text{m}$). When seen from upstream, this turbine was located $\Delta y = 0.20\text{m}$ off the centre line towards the left. The downwind turbine, denoted turbine T2, ($D_2 = 0.894\text{m}$), was placed three diameters further downstream and shifted $\Delta y = 0.20\text{m}$ off the centre line towards the right from the centre line. This gives a total offset between the turbines of $\Delta y = 0.40\text{m}$, and so the projected area of the upwind rotor covers exactly 50 % of the downwind rotor area. The empty wind tunnel has a low turbulence level of 0.23 %. In order to make the conditions more similar to the atmospheric conditions, background turbulence was generated using a large scale bi-planar mesh at the inlet of the test section (Mesh size 0.24m, Solidity 35 %). This gave a turbulence level at the location of T1 of 10 %, while the level had dropped to about 5 % at the location of the second turbine.



Thrust forces were measured by a 6 component balance on which the model was mounted. The power coefficient was measured using a torque transducer mounted directly onto the rotor shaft, which also provided pulses to measure the speed of rotation. Velocities were measured using x-wire hot-wire anemometers. For verification, some of the measurements were repeated using pitot-static tubes and laser Doppler anemometry. The uncertainty in the measurements of the mean velocity were estimated to be less than $\pm 4.6\%$ of U_{ref} and $\pm 5.9\%$ of U^2 for k^2 (turbulent kinetic energy).

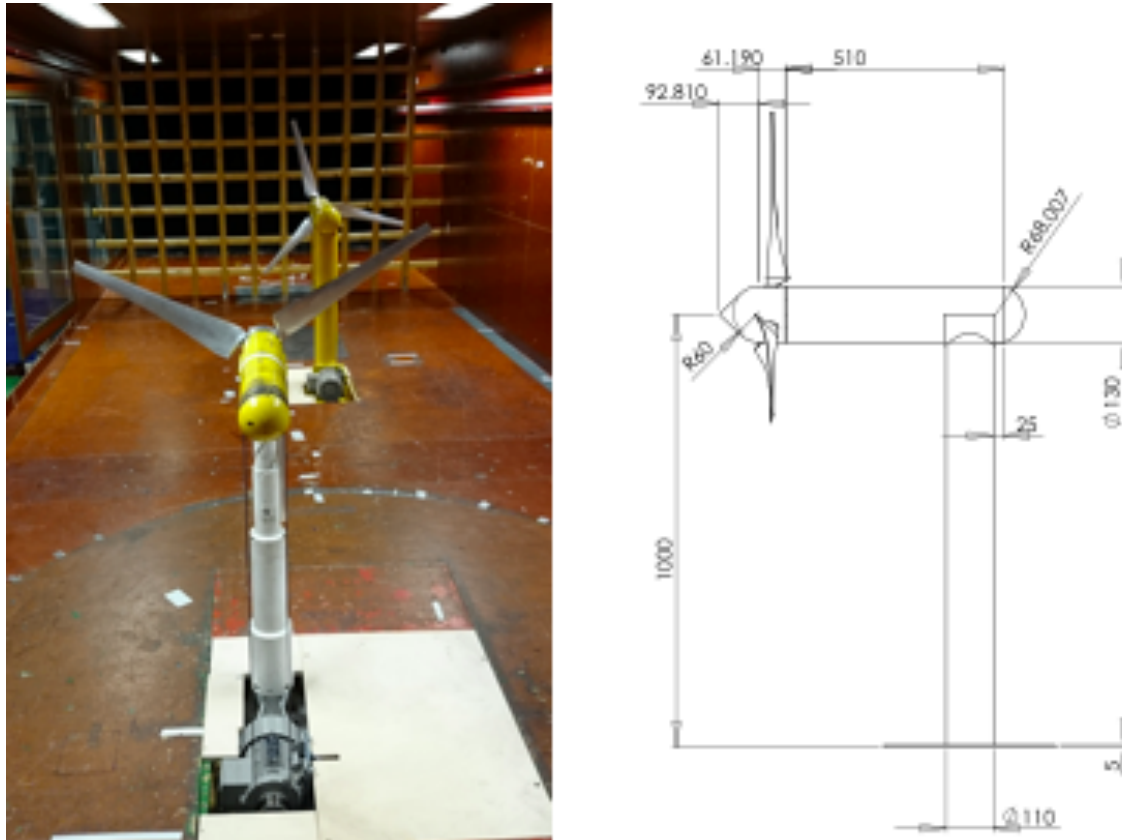


Fig 1: 2 turbines and turbulence grid seen from downstream. Details of turbine T1

2. Participants and methods

Calculations were submitted by five groups. In order to start the wake calculations, the power and the thrust coefficients of the turbines must first be estimated. This was therefore compulsory output. Two participants resolved the flow down to the boundary layer on the turbine blades, while the rest relied to some extent on a Blade Element Momentum method calculation. For the other methods that use some imbedded force method, estimates of the airfoil performance were needed, e.g. using software such as XFOIL [5]. However, suggested values of airfoil lift and drag coefficients from unpublished experimental data were also provided. Below is a short list of the participants and the essences of their methods.

2.1. Alcona Flow Technology

E. Manger of Alcona Flow Technology, Skien, Norway, modeled the entire experimental setup, including the towers and nacelles with the turbines located inside the test section.

The rotors were included in a short cylindrical sliding mesh within a fixed grid. The boundary layers on the blades were resolved down to $y^+ \approx 5$. The flow was solved using the Ansys Fluent v.14-5 software and the $k - \omega$ SST turbulence model was used to model the turbulent field. The final mesh used approximately 12 million cells. The computations were rather time consuming and therefore only Case A was completed in time for the meeting.

2.2. DTU Mechanical Engineering

The group of Professor Sørensen at DTU, Lyngby, Denmark, delivered data for a combined actuator line / Large Eddy Simulation (LES) using a program called EllipSys3D. This uses a block structured finite volume approach. The time increment was sufficiently small so that the tip of the blade advanced less than half a cell per step. The blades are represented by forces along rotating lines and the loads were estimated from their own unpublished measurements for the NREL S826 airfoil. The computational domain is a regular Cartesian grid divided into 750 blocks using a total of 24.5 million mesh points. The tunnel walls are included in the calculations, but the towers and nacelles are missing. The tunnel turbulence was simulated by inserting synthetic turbulence 1.5 diameters upstream of the first turbine and the level was adjusted to match the experimental conditions at the T₁ rotor plane. Four grid resolutions were tested to find the required density for grid independency.

2.3. GexCon

M. Khalil of GexCon, Bergen, Norway, performed calculations using the software package FLACS – Wind which is developed by GexCon. This is a transient CFD solver which in this case used the standard $k - \epsilon$ turbulence model. The computational domain was similar to the wind tunnel dimensions, but the increase in tunnel height to compensate for the growth of side wall boundary layers was not included. The rotor was represented as an actuator disk and the disk data was obtained using a BEM method. The effects of the towers and nacelles were not included in the simulations.

2.4. CD-adapco

S. Evans from CD-adapco, London, UK, provided predictions using their own software Star-CCM+. Around the rotors, cylindrical domains were created so that the arbitrary sliding interface functionality of STAR-CCM+ could be used to model the rotor motion. A polyhedral mesh was created containing 14 million mesh elements with the boundary layer on the blades being resolved down to a Y^+ value of less than 2, dependent on the operating conditions. Predictions were provided using the $k - \omega$ SST DES model. The mesh was created in such a way that the DES model operated in LES mode in the wake regions of the flow. The simulation ran in a two-step approach. Firstly, the case was run using a steady approach with multiple rotating frames. After the simulation was considered to be sufficiently initialised, it was switched to transient simulation.

2.5. CMR

A. Hallanger and I.Ø. Sand of CMR Instrumentation, Bergen, Norway, used a CFD code called Music developed in-house. The rotors, including the hubs, were modeled in the wind tunnel confinement, but the rest of the nacelle, as well as the towers, were omitted. The forces on the blades were estimated using a generalized Blade Element Momentum model with

rotation and included as source terms in the axial and rotational momentum conservation equations. A total of 30 elements were distributed along the radial direction of the blade and a total of 0.5 million grid nodes were used to represent the wind tunnel test section with the turbine rotors. The turbulence was described using the standard $k - t$ model with a sub-grid model. The turbulence intensity and length scales for the two test cases were applied as specified in the case description.

3. Results

3.1. Turbine performance

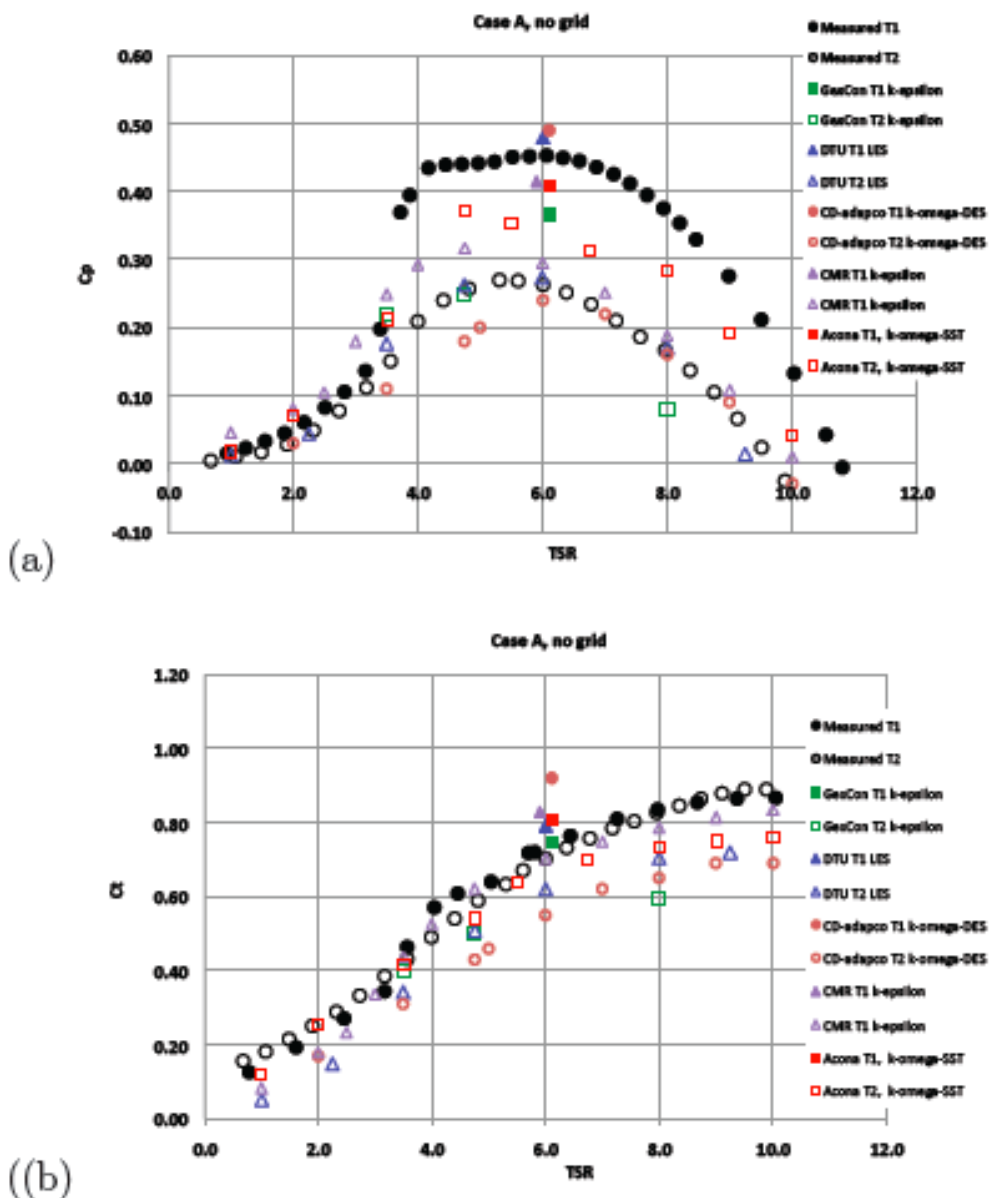


Fig. 2. C_p and C_t for low turbulence case A.

We start by presenting the results for the turbine performances for Case A, i.e. with the low background turbulence level. The power coefficients are shown in Figure 2(a) and the thrust

coefficients in Figure 2(b). The symbols used will always have the same colour and shapes in all figures, with filled symbols for the upstream turbine and open symbols for T2. The measurements will always be presented as black circles.

Even though the two turbines have the same blades, the shape of the CP curves are seen to be different for the two turbines. The upstream turbine shows a rapid drop when TSR falls below 4. This is due to the sudden onset of stall. For T2, this development appear to be less dramatic. One may therefore speculate that this is due to a much higher turbulence level felt in the incoming flow to T2 which has generated a more gradual separation development.

The data for the second turbine predictions vary by more than 50 % in some regions, but this is understandable, since we here have a complicated test case. However, some of the predictions for T2 are in fact very good, e.g. the curves obtained by DTU, CMR Instrumentation and CD-adapco for $TSR > 6$.

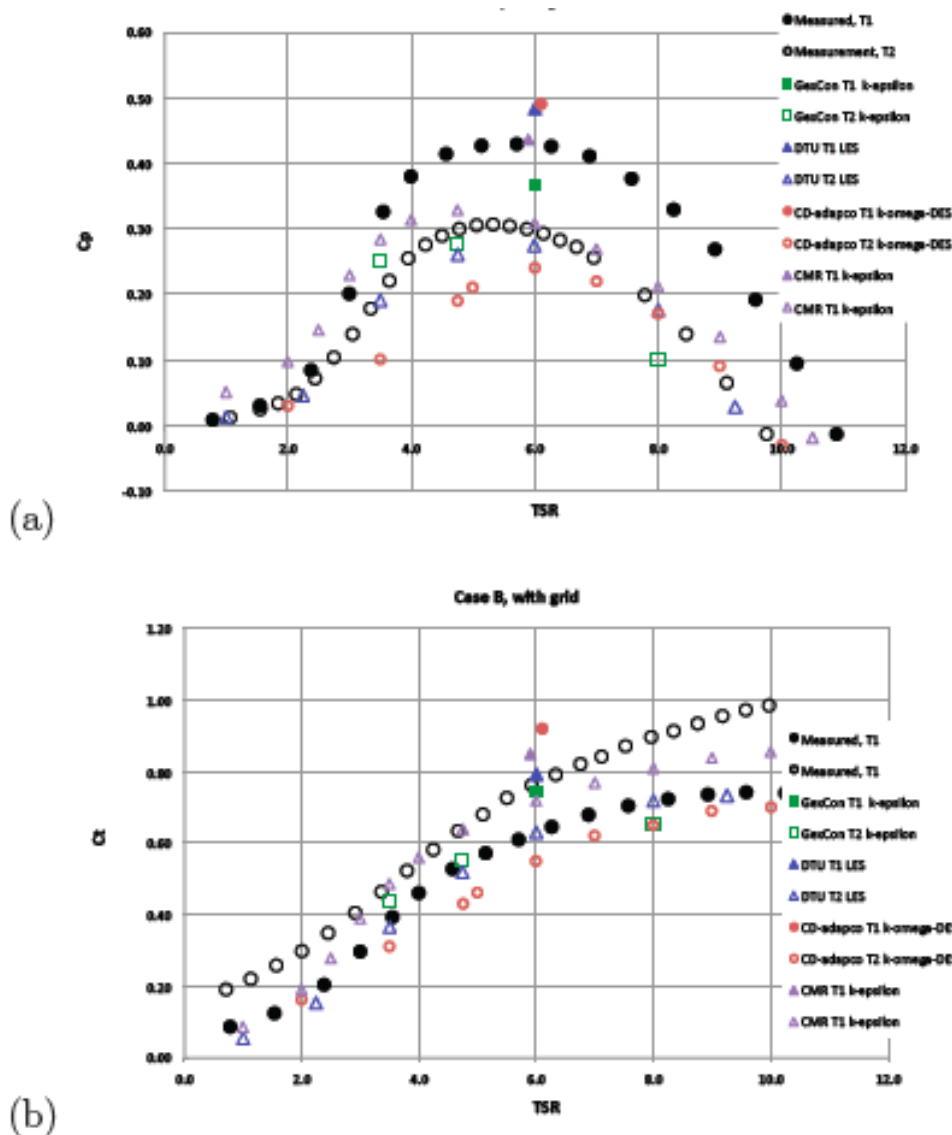


Fig. 3. C_p and C_t for high turbulence case B

For the very low TSRs, where the flow over the blades are severely stalled, the deviation from the measurements is mostly small, while there is significant spread in the predictions for high tip speed ratios. It was expected that one of the principal problems would be to predict the onset of stall. The first signs of stall were found to occur around $TSR=4$ when the T2 turbine operates alone and this is where the largest differences are found. But for the deep stall region for $TSR < 3$, all methods behave well. The fully resolved predictions of Alcona and CD-adapco do not agree. Alcona predicts consistently the highest CP while the predictions by CD-adapco are generally low. This points to sensitivities to the turbulence models used or significant differences in the numerical grids constructed.

Looking at the thrust coefficient (Figure 2(b)), the measurements of the upstream turbine again show that something happens to the flow as TSR is reduced below 4. Except for a short region here, the CT data for both turbines are very similar. This is puzzling, since the velocity fields seen by the two rotors are very different. Despite this, the coefficients are the same when scaled with the same parameters, showing that the physical forces are in fact almost identical even though the momentum available at T2 ought to be less than for T1. The general trend is that CT is mainly over-predicted for the T1 turbine, but under-predicted for T2. GexCon has matched the force on turbine T1 very well, but underpredicts the forces on T2 dramatically at $TSR=8$.

Next we present the performance data for the case with high turbulence level, Case B. The measurements indicate that the peak performance of the upstream turbine has been slightly reduced (Figure 3(a)) and the shape is seen to be smoother, again suggesting that the free stream turbulence significantly affects the onset of stall on the blades. None of the performances predicted for T1 appear to be sensitive to the freestream level. The same applies to the T2 predictions. However, the measured performance of T2 is slightly increased by the turbulence. This is consistent with the reduction in energy extraction by T1. The turbulence effect shows even more clearly on the measurements for CT (Figure 3(b)), which indicate a dramatic sensitivity to the freestream turbulence for both turbines. Compared to Case A, CT for T1 is reduced by between 10 and 15 %, and thus T2 has increased CT by about the same amount.

3.2. Wake data

3.2.1. Operation at peak performance, $TSR_1=6$, $TSR_2=4.75$. The participants were asked to predict the wake development behind turbine T2 when T1 was operating at its design TSR and T2 was operating at $TSR = 3.5, 4.75$, and 8.0 . $TSR = 4.75$ is close to the peak performance for T2 and will be presented first. This should be the operating condition that generates the most homogeneous wake and therefore be the simplest case to predict. However, only part of the wake from T1 hits the rotor of T2, so T2 will see a very inhomogeneous inflow and therefore the wake becomes quite complicated also in this case.

Output of mean velocities and kinetic energy were requested for both case A and B at $X/D = 1$ and 3 downstream of T2. We present only the streamwise normal component $\langle u_x^2 \rangle$ here. The mean velocity profiles at $X/D=1$ along a horizontal diagonal are shown in Figure 4(a), for Case A and Figure 4(b) for Case B, respectively. There are a few obvious observations that may be made immediately. Compared to the wake behind a single turbine operating at its best performance, this is a much more complicated wake. The outer edges of

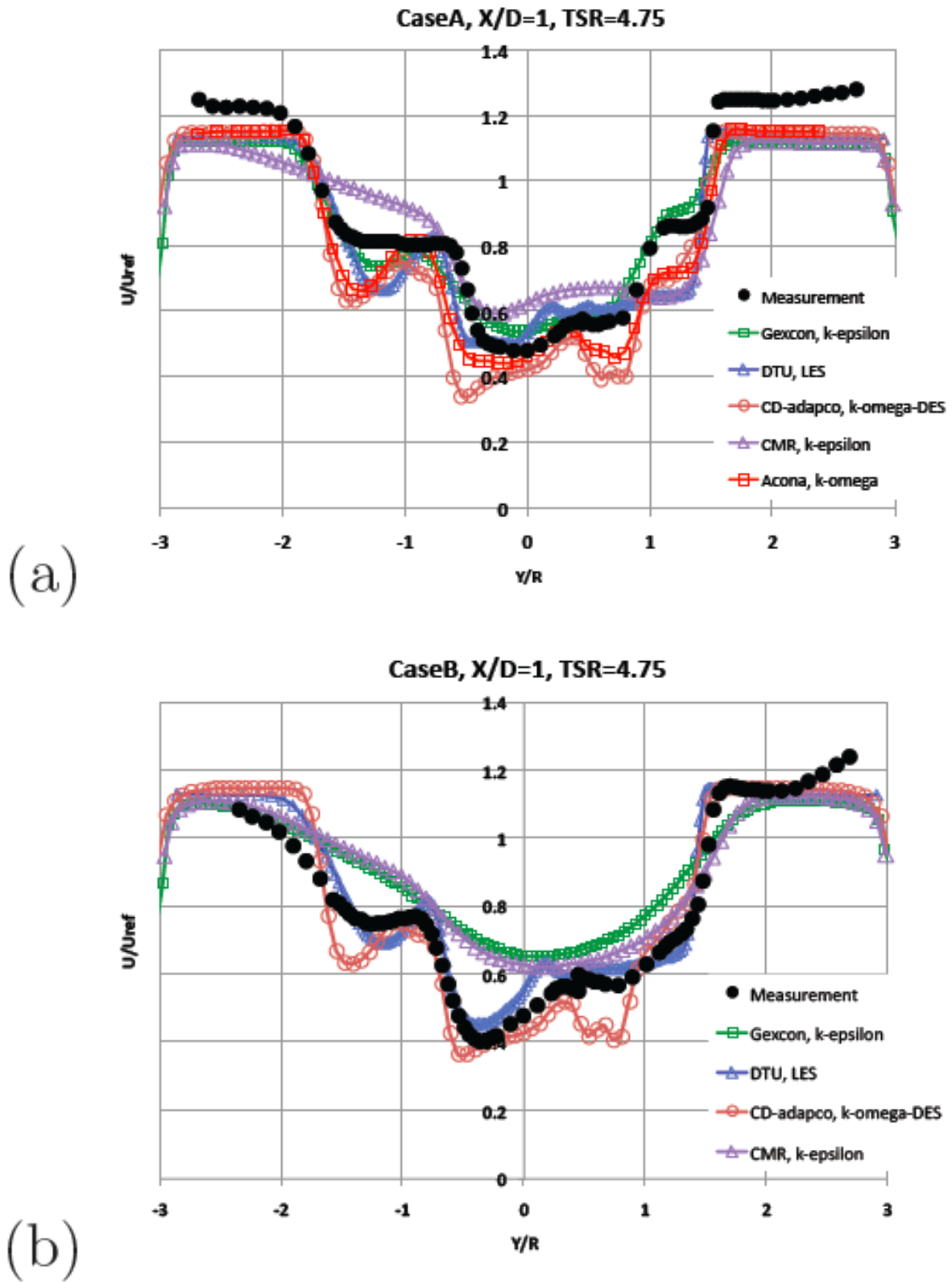
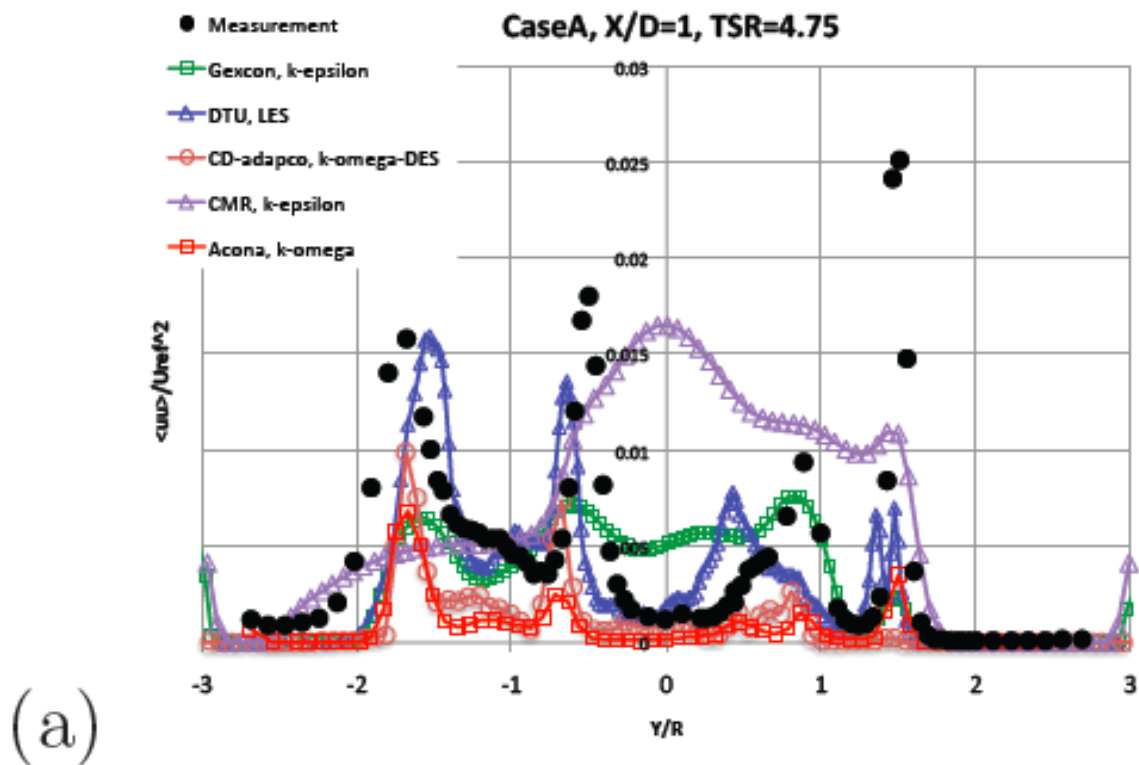


Fig. 4. Mean velocity profiles for $TSR_1=6$, $TSR_2=4.75$ at $X/D=1$. (a) case A, (b) case B

the two individual wakes are quite evident, but the central part is a mix of influences from the two wakes. As expected, the wake for Case B is much smoother than for Case A. This smoothing effect comes out much clearer in the predictions than in the measurements if we compare e.g. the predictions by GexCon and CMR with the measurements (Figure 4(b)). Acona did not provide predictions for Case B. The LES predictions by DTU appear to have captured all the details of the wakes very well for both cases and are followed closely by the CD-adapco DES predictions.

The differences in the mean velocity profiles are reflected in the turbulent stresses as well. These are shown in Figure 5(a), for Case A and Figure 5(b) for Case B, respectively. While the methods of Acona, CD-adapco and DTU appear to have captured most of the details of the stress distributions for both cases, the level of detail in the GexCon and CMR predictions are not good. The energy level predicted by DTU is mostly close to the measurements, but computations by the Acona and CD-adapco methods are predominantly low.



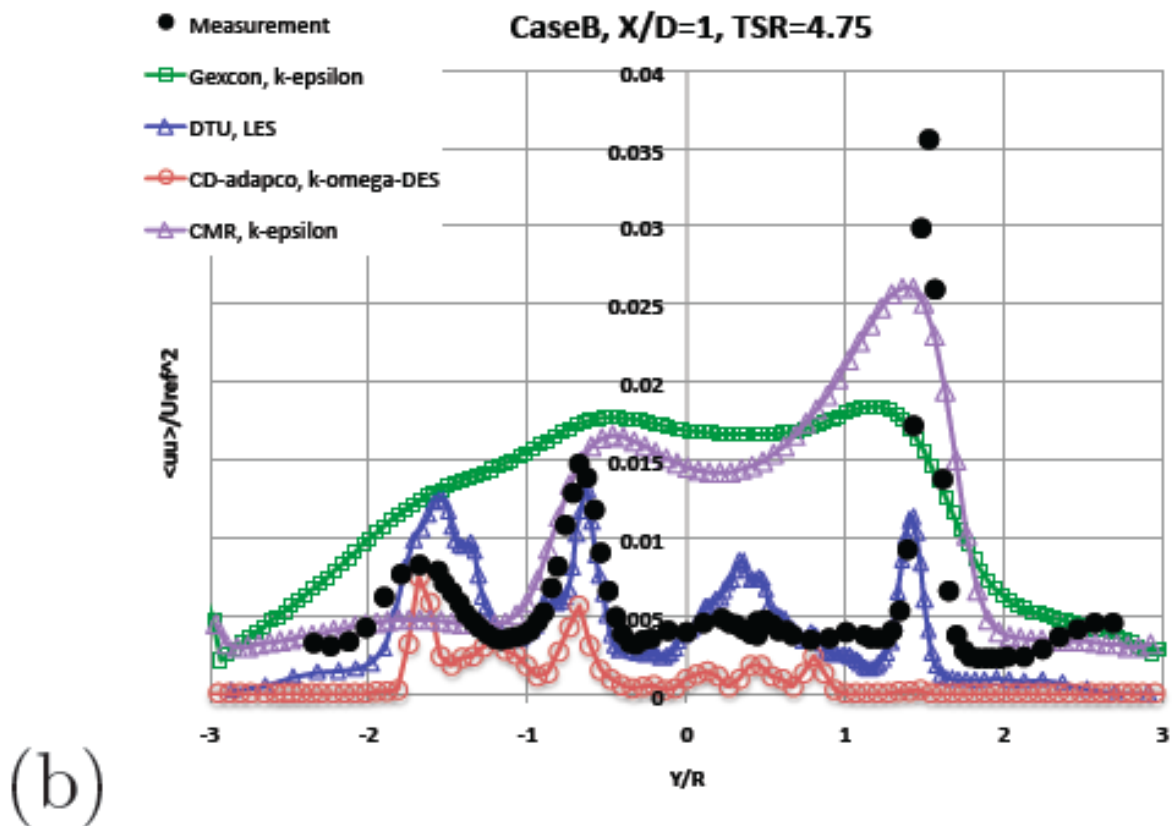


Fig. 5. Turbulent stress $\langle u_x^2 \rangle$ for $TSR_1=6$, $TSR_2=4.75$ at $X/D=1$. (a) case A, (b) case B

Next we move downstream to $X/D=3$. The mean velocity profiles along a horizontal diagonal are shown in Figure 6(a), for Case A and Figure 6(b) for Case B, respectively. Diffusion has now significantly modified the flows and so the measured mean velocity profiles show very little detail in both cases. The effect of diffusion appear to be overestimated by CMR for both cases, while Acona seems to have the opposite problem with a profile that are very similar to those at $X/D = 1$. While the method by GexCon performs quite well in Case A, diffusion is much too strong in Case B. However, the DES of CD-adapco produces profiles that are very close to the LES by DTU for both cases.

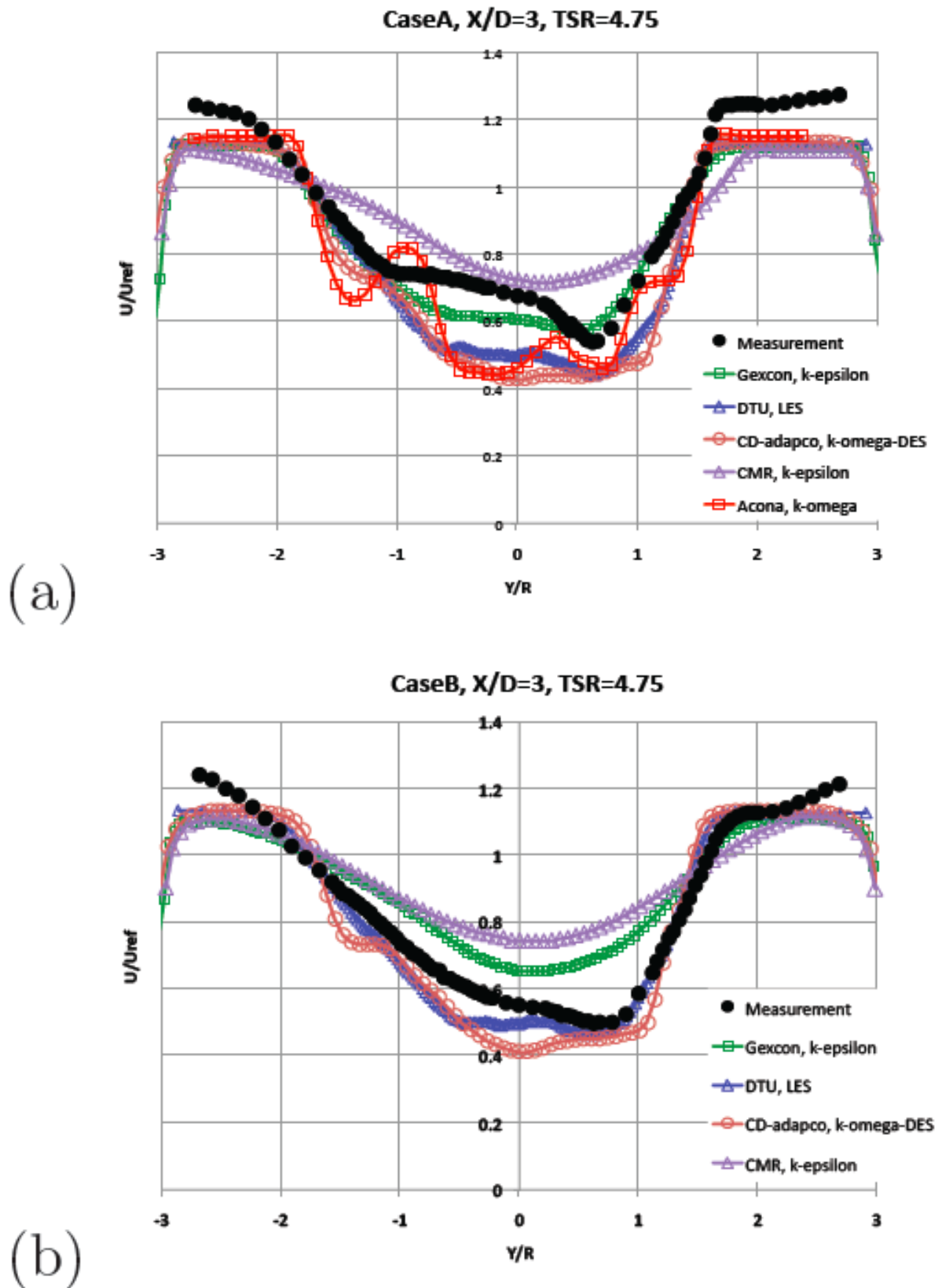
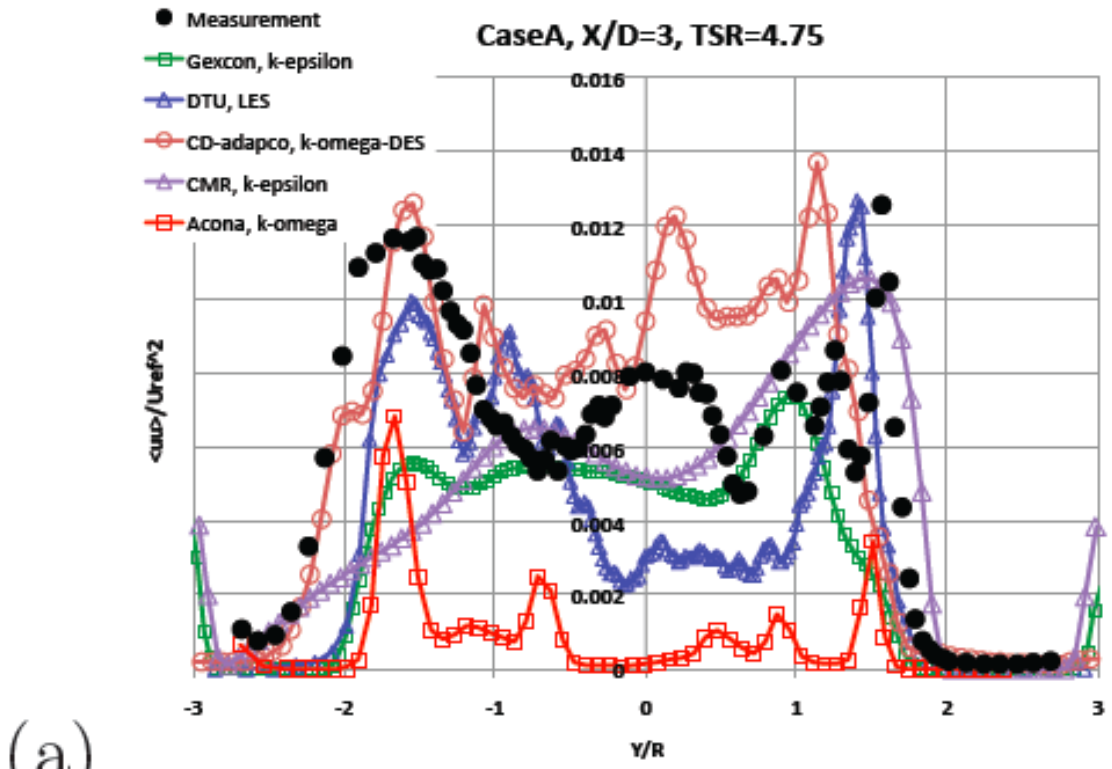
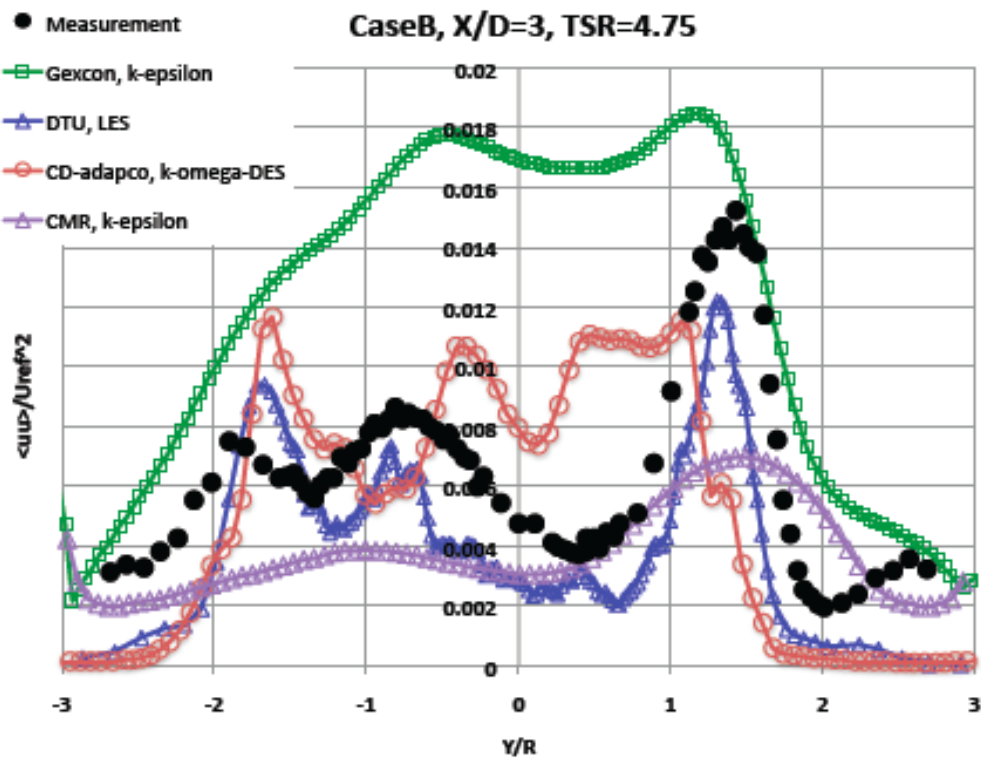


Fig. 6. Mean velocity profiles for TSR1= 6, TSR2= 4.75 at X/D=3. (a) case A, (b) case



(a)



(b)

Fig. 7. Turbulent stress $\langle u^2 \rangle_x$ for $TSR_1=6$, $TSR_2=4.75$ at $X/D=3$. (a) case A, (b) case B

Figure 7(a) shows the turbulent stresses for Case A and Figure 7(b) for Case B, respectively. The figures show that the LES method of DTU and the DES of CD-adapco perform best, although the fine details near the centre in Case A are missing in both predictions. Again the results for Case A and B are very different in the predictions by the GexCon method. But the predictions of CMR are now at the correct level for both cases although the finer details of the tip vortices are missing. Acona produced the correct stress distribution for Case A, but again the level is much too low.

4. Some concluding remarks

The comparison between the model tests and predictions of the turbine performances and the wake development behind the second turbine have been presented in this report. Five research groups delivered simulation results. The methods ranged in complexity from standard CFD methods to Large Eddy Simulations. It is surprising that even the performance data of the upstream turbine in a uniform, low turbulence flow appear to be a challenge with results differing by about $\pm 10\%$ near the design condition. Obviously, if the performance of the upstream turbine is not correctly predicted, the results for the performance of the second turbine must be even more uncertain. And so the predictions of CP for the downstream turbine showed a spread of about 50 % of the measured values near the peak performance. Similar uncertainties were found for the predictions of the thrust coefficients. For all cases investigated the upstream turbine was always operated at the same conditions, which was its best performance point (TSR= 6). This is the condition where the wake produced is the least complicated. But even when the second turbine is also operated at its best performance point the wake produced is very non-uniform with complicated interactions between the two wakes.

Most methods captured this surprisingly well in the low turbulence case. However, it was a trend that the effects of turbulent diffusion were over-predicted by the methods using the k - ϵ turbulence model in the high turbulence cases, giving a much smoother profile than in the measurements. Only the LES method did faithfully reproduce the data for both free stream conditions. Two groups used commercial CFD packages, while the other groups used software developed in-house. Two of these incorporated a k - ϵ turbulence model but produced very different results. This indicates that the choice of turbulence model is less critical than the implementation of grids etc. which relies heavily on the judgement of the model operator.

The overall conclusion for this blind test was that the LES method produces the most reliable predictions and when the cases are set up properly, the results are very consistent when the boundary conditions are changed

References

- [1] P.-A. Krogstad, M. Adaramola, Performance and near wake measurements of a model horizontal axis wind turbine, *Wind Energy* 15 (2012) 743–756.
- [2] M. Adaramola, P.-A. Krogstad, Experimental investigation of wake effects on wind turbine performance, *Renewable Energy* 36 (2011) 2078– 2086.
- [3] P.-A. Krogstad, P. Eriksen, Blind test calculations of the performance and wake development for a model wind turbine, *Renewable Energy* 50 (2013) 325–333.
- [4] F. Pierella, P.-A. Krogstad, L. Sætran, Blind Test 2 calculations for two in-line model wind turbines where the downstream turbine operates at three different rotational speeds, Accepted for publication in *Renewable Energy* (2014)
- [5] M. Drela, Xfoil 6.99, <http://web.mit.edu/drela/Public/web/xfoil/>.