Numerical Simulation of the Interaction between Tandem Wind Turbines

Kidambi Sreenivas*, Anshul Mittal†, Levi Hereth‡, Lafayette K. Taylor§, and C. Bruce Hilbert**

University of Tennessee at Chattanooga, Chattanooga, TN 37403

Two model wind turbines operating in tandem are simulated using *Tenasi*, a node-centered, finite volume unstructured flow solver. The turbine blades are designed using the NREL S826 airfoils. The entire test section of the wind tunnel is simulated since the blockage (based on swept area of the rotor and tower area) was 12%. The simulations included the tunnel walls, wind turbine blades, hubs, nacelles and towers. Detailed experimental data is available for a variety of flow conditions (varying tip-speed-ratios). The results presented here are for tip-speed-ratios of 2.5, 4, and 7 for the rear turbine while the front turbine was always operated at the design tip speed ratio of 6. The tunnel wind speed was 10 m/s and the wind turbine RPM was varied to achieve the desired tip-speed-ratios. A DES version of the Menter's SST turbulence model is utilized for the turbulence closure. Turbine performance as well as wake data at various locations is compared to experiment (Blind Test 2 study carried out at NTNU, Norway). Very good agreement is observed for the turbine performance. Good agreement was obtained for velocity fluctuations in the wake region with trends captured very well. Mean velocity predictions agree reasonably well.

* Research Professor

† Research Engineer, Convergent Science

‡ Structural Engineer

§ Professor (Retired)

** Algorithms | Optimization, Branch Technology

I. Introduction

With the emergence of wind energy as a viable renewable energy source, large wind farms are being constructed around the world. These wind farms frequently employ hundreds of multi megawatt wind turbines. In a typical wind farm, the turbines are usually operating in the wakes of upstream turbines. This reduces their power output while increasing the dynamic loading of the blades. Therefore, it is of considerable interest to the wind farm operator to design and operate a wind farm that minimizes these wake effects.

To alleviate the adverse wake effects, a good understanding of the wake of a single wind turbine is a must. A thorough understanding of the wake dynamics can be obtained using either experimental or computational approaches. In either case, experimental data is required as the computations need to be validated before they can be used to make predictions. Ideally, one would want to utilize data from a full scale wind turbine for validation purposes. However, there are issues in doing so such as proprietary nature of the data and the blade geometry, and uncertainties in the initial and boundary conditions due to atmospheric conditions. Thus, model scale experiments are preferred. Examples of such studies include the NREL Phase VI rotor [1], the MEXICO rotor [2] and the BT1 rotor [3]. Both the NREL Phase VI and the MEXICO rotor were isolated rotors only, i.e., there were no plans to include a second turbine in the wake of the first and experimentally measure the impact of the wake. The BT1 rotor, on the other hand, was designed from the beginning to be part of a two rotor setup with one rotor operating in the wake of the other. BT1 results computed using *Tenasi* were presented in Sreenivas et al. [4] and Mittal et al. [5] with three different turbulence models and the effect of temporal convergence were investigated. Good agreement was obtained between predicted power and thrust coefficients and experimental data for all the turbulence models. A DES version of the Menter's SST model performed significantly better for the mean velocity profiles and the velocity fluctuations in the wake of the turbine.

The follow-on study for the BT1 test case was the BT2 (Blind Test 2) test case conducted at NTNU in Norway. This involved two rotors operating one behind the other with a separation of three rotor diameters. As part of the BT2 workshop, results from various academic and industrial entities were compared to experimental data and a summary of the comparison is available in Pierella et al. [6] Most of the results included in that comparison utilized either the Actuator Disc (AD) or the Actuator Line (AL) models for modeling the rotor with only one participant including the entire geometry of the test setup. The velocity fluctuations predicted by this participant were almost

one order of magnitude smaller than the experimental values. [6] The solution methods used ranged from standard Reynolds-Averaged Navier-Stokes (RANS) modeling to Large Eddy Simulation (LES), with one participant using an unsteady free-wake vortex panel method. There was significant variation between the predicted thrust and power coefficients using various methods. Similar differences were also observed in the velocity deficit and turbulence levels in the wake. Furthermore, the results obtained by two different groups that used the same airfoil data as input to their AD and AL models did not match with each other or with the experimental thrust and power values.

The focus of this paper is on the computational simulation of the BT2 test case. Given the uncertainty associated with using the AL and AD methods (arising from the distribution of the forces in the vicinity of the blade locations) and the inability to include the tower and nacelle (in general, though there are exceptions [7]), it was decided to model the entire experimental setup, i.e., the computational domain considered in this study included the entire test section with tunnel walls and both the turbines (with fully resolved blades, towers, hubs, and nacelles). It is known that fully-resolved rotor CFD simulations perform better over a wider operating range than BEM based simulations due to the capability of the method to capture the flow separation over the blade at the off-design conditions. [8] The BT1 [4,5] study showed that out of the three turbulence models tested (Spalart-Allmaras, Menter's SST, and a DES version of the Menter's SST), the DES version of the Menter's SST model performed better than the other two. Therefore, only the SST-DES model is utilized in this study and its results are presented. The remainder of the paper is organized as follows. First, a brief description of *Tenasi*, the flow solver used in these simulations is given. Then, the BT2 test case is described followed by the details of the numerical simulations. Results are presented for three conditions (varying tip speed ratios). Finally, conclusions are drawn regarding the ability of *Tenasi* to predict the flow fields around multiple wind turbines accurately.

II. Flow Solver

The baseline flow solver in *Tenasi* employs a finite volume, implicit scheme with high resolution fluxes and a dual time stepping, Newton sub-iteration procedure for time accuracy. The linear system at each time step is solved using a Symmetric Gauss-Seidel algorithm. Some of the features of the unstructured solver are highlighted below. The details of the numerical algorithm are available in Hyams et al.[9]

A. Flow Regimes

An important distinguishing feature of *Tenasi* is that the same executable can be used to solve different sets of governing equations, i.e., iso-energetic incompressible [10], incompressible with heat transfer [11], compressible, preconditioned compressible [12], surface capturing, electromagnetics, etc. by choosing a flow regime at runtime. The implementation is such that a new flow regime can be added in a fairly straightforward manner by implementing the required flux functions and the boundary conditions. The rest of the infrastructure such as processing of the grid, the sliding interface capabilities, the solution of the linear systems etc. are available to all flow regimes. For the present study, incompressible formulation is utilized. The incompressible form of the Navier-Stokes equations is solved using the artificial compressibility [13] method. The method yields a divergence-free velocity field as the residual is reduced and the solution converges. For further details, please refer to Mittal et al. [5]

B. Residual Computation

The unstructured solver is based on a node-centered formulation. Higher order spatial accuracy is achieved using a linear or quadratic reconstruction of the dependent variables at the control volume faces and using these reconstructed values to evaluate the fluxes. Viscous terms are evaluated using a directional derivative based approach. The gradients required for the variable reconstruction are computed using an un-weighted least squares approach while the gradients for the viscous terms are computed using a weighted least squares approach. The numerical fluxes are evaluated using Roe's flux difference split approach.

C. Time Evolution

The flow solver employs a discrete Newton relaxation approach coupled with dual time-stepping to solve the unsteady equations. The dual time stepping approach is used to accelerate the convergence of the unsteady residual. Newton's method is used to drive the unsteady residual to zero, thereby ensuring time accuracy. For further details of the time advancement algorithm, the reader is referred to the study by Mittal et al. [5] The flux Jacobians arising from this linearization can be evaluated using numerical derivatives or the complex Taylor series method. They can also be replaced by approximations which result in substantial savings in computational time. The resulting linear system is solved using a Symmetric Gauss Seidel algorithm (point relaxation). For deforming grids, the Geometric Conservation Law (GCL) has to be satisfied in order to prevent the occurrence of spurious sources in the solutions. This leads to an additional contribution to the residual.

D. Turbulence Modeling

The turbulence models are implemented in a loosely coupled manner. The flow solver has the Spalart-Allmaras model, the Menter SAS model, the $k\text{-}\epsilon/k\text{-}\omega$ hybrid model (with and without SST), the standard $k\text{-}\epsilon$ model, the Wilcox $k\text{-}\omega$ model, and the Wilcox Reynolds Stress model. In addition, DES models are available for the Spalart-Allmaras, the Menter SAS and the $k\text{-}\epsilon/k\text{-}\omega$ hybrid models. A DES model is a blend between a RANS turbulence model and a LES model. RANS model is utilized near the viscous walls since a LES model would require a very fine mesh. Away from the walls, turbulence is modeled using the LES approach. Furthermore, a wall adapting LES model (WALE) is also available.

E. Parallel Implementation

The parallel solution procedure consists of a scalable solution algorithm implemented to run efficiently on subdomains distributed across multiple processes and communicating through MPI. The algorithm has multiple nested kernels viz. time step, Newton iteration, LU/SGS iteration etc., and the subdomain coupling is at the innermost level, i.e., in the solution of the linear system. A block-Jacobi type updating of the subdomain boundaries ensures efficient parallelization with a small incremental cost incurred in terms of sub-iterations required to recover the convergence rate of the sequential algorithm. Details about the parallel algorithm can be found in Hyams [14].

F. Relative Motion

The unstructured sliding interface technique utilized in *Tenasi* was developed and validated at the SimCenter. The underlying concept is to extrude the pairs of sliding interfaces into adjoining domains and to interpolate for the flow variables and their gradients at the newly created nodes. Note that the sliding interfaces are not merged in any way and this has the potential to lead to multi-valued solutions at a given point. However, this has not been observed in practice and points that are very close to each other have approximately the same solution values. Further details on the parallel implementation of the sliding interface algorithm are available in Hyams et al. [5,15,16]

III. Description of the BT2 test case

The BT2 test case consists of two similar wind turbines operating in tandem. The experimental setup is shown in Figure 1. Both of the turbines are three bladed, with the blades made up of 14% thick NREL S826 airfoils. The turbines sit on top of towers that have two different shapes. The front turbine (T1) has a cylindrical tower (110 mm

diameter), while the aft turbine (T2) sits on top of a stepped tower consisting of 4 cylinders of different diameters. The height of the two turbines is 0.817 m. Note that the aft turbine is identical to the turbine used in the BT1 [3] test case. The diameters of the turbines are 944 mm for the front turbine (D_{T1}) and 894 mm for the aft turbine (D_{T2}). The turbines are separated by $3D$, with the front turbine $2D$ from the test section inlet, with D being defined as the diameter of the aft turbine (894 mm). The turbines were placed in the middle of the test section with a distance of 1350 mm from the center of the turbine to the side walls. The nacelles of the two turbines are circular cylinders with diameters of 130 mm (T1) and 90 mm (T2) respectively. Both nacelles are capped with hemispherical hubs at the back, while at the front T1 has a pointed hub and T2 had a hemispherical one.

Figure 1: Experimental setup for the BT2 test case [17]

The wind tunnel had a test section that was 11.14 m (length) x 2.7 m (width) and employed a slightly diverging roof (1.801 m at inlet to 1.851 m at outlet) in order to achieve a zero pressure gradient at a tunnel speed of approximately 14 m/s. The inflow velocity was maintained at 10 m/s while the turbine RPMs were varied in order to achieve the desired Tip Speed Ratios (TSRs; defined as the ratio of tip speed to incoming speed). The front turbine was always operated at its design point ($TSR = 6$), while the aft turbine was operated at $TSR = 2.5$, 4, and 7 corresponding to stall, peak production and runaway conditions respectively. Experimental data available includes thrust and power coefficients in addition to mean velocities and RMS of streamwise velocity fluctuation. The velocities and fluctuations were obtained on horizontal and vertical sweeps at $x/D = 1$, 2.5, and 4, measured downstream of T2 (Figure 2).

IV. Grid Generation

The geometry that was meshed included the rotors, towers, nacelles and the walls of the wind tunnel. All solid surfaces were meshed with viscous off-wall spacing except for the side walls of the tunnel. Viscous spacing was not employed for the side walls of the tunnel in an effort to reduce the size of the overall mesh so that the simulations could be carried out using available resources. Since the simulation included relative motion between the rotors and the rest of the solution domain, sliding meshes were employed. The mesh was generated in three pieces, one for each of the rotors and the corresponding sliding interfaces and the other outside of the sliding interfaces and included the tunnel walls and the towers. The three grids were then combined to obtain the final mesh. These meshes were generated using Pointwise® [18] and the Viscous Layer Extrusion (VLE) method of Karman. Special attention was paid to ensure reasonable wake resolution both between the turbines and downstream of the aft turbine by including uniformly resolved grids (Figure 2).

The mesh generation process started with generation of surface meshes in Pointwise on the viscous surfaces. From those surface meshes, VLE was used to construct the necessary viscous layer meshes. VLE creates these meshes by extruding the triangles and quadrilaterals that comprise the surface into stacks of prisms and hexahedra, respectively. This extrusion is done on an element-by-element basis using normals and stack heights that respect neighboring elements. Note that the individual stack height generated off each element is not dependent upon its neighboring elements. However, each stack is usually restricted, for quality purposes, from advancing more than two or three layers past its neighbors' stacks. The result of this process is a complete boundary layer mesh with a viscous front surface. That surface along with any non-viscous boundary surfaces are used to create the remainder of the mesh.

Figure 3 shows the mesh at 50 % span location on the blade of the rear turbine. Figure 4 is the zoomed-in view of the same cross-section near the leading edge. The blade is resolved using three different regions, each with its own spacing characteristics (Figure 4) The leading edge of the blade is resolved using 3250 nodes in the spanwise direction (spacing of 0.119 mm) and 20 nodes along the chordwise direction (spacing of 0.04 mm). The resulting leading edge mesh is diagonalized so that only triangles are present (Figure 4, region a). An unstructured transition patch is utilized to transition from the leading edge mesh to the remaining mesh. The unstructured patch has 550 nodes along the span (spacing of 0.7 mm) and 9 nodes along the chord of the blade (Figure 4, region b). The

remaining surface mesh of the blade is a diagonalized structured mesh with 550 nodes in the span and 75 nodes along the chord. The last patch extends up to the trailing edge of the blade (Figure 4, region c). The total number of nodes for the surface mesh is 251,681 nodes for a single blade.

The sliding mesh interface is an axisymmetric surface that encloses the rotating components (hub and blades). Figure 5 shows the T2 turbine and its associated sliding interface, with a similar interface being used for T1. The sliding interface extends 20 mm beyond the tip of the blades and 20 mm from the hub with its axis coinciding with the axis of rotation. The sliding interface mesh is an unstructured Delaunay mesh with 150 nodes in the radial direction and 400 nodes on the circumference. The final mesh had 33.11 M nodes, 48.98 M tetrahedra, 1.98 M pyramids, 42.28 M prisms, and 2.30 M hex elements. The grid employed an off-wall spacing of 0.006 mm which resulted in a y^+ of less than 2.0 over most of the viscous surfaces. The uniform refinement seen in Figure 2 had a spacing of 0.026 m.

Figure 2: Cross section of the grid showing wake resolution and comparison locations

Figure 3: Cross-section at 50 % span showing the mesh around the blade

Figure 4: Zoomed-in view at the leading edge of the airfoil showing the regions of different patches of meshes

Figure 5: Rotating interface shown for the rear turbine

V. Simulation Details

One aspect of the experiment that was significant was the boundary layer thickness that was seen by the wind turbines. The thickness of the boundary layer was reported to be 340 mm [19] for the BT1 case, though it was not reported for the BT2 test case. The effect of the boundary layer thickness was evaluated for the BT1 study by comparing the performance of the turbine with and without the incoming boundary layer. It was found the incoming boundary layer had considerable effect on the turbine's performance. [5] The boundary layer thickness was matched in the BT1 test case by imposing an incoming velocity profile. Since the tunnel setup was identical between the BT1 and BT2 test cases (except for the one versus two turbines and the location of the front turbine), the same incoming boundary layer profile was imposed in the simulations performed as part of this study. It should be noted that if LES turbulence modeling approach is used, some technique to generate turbulence at the inflow boundary, such as in [20], might be utilized to reduce the computational cost.

Simulations were carried out using the following procedure: A steady state simulation (local time-stepping) was carried out starting from uniform flow conditions with the CFL being ramped from 1 to 40 over 360 time steps and then held constant. This ensured that the gross flow features were established before a time accurate (constant time step) simulation was started. The solution was restarted from the "steady" state simulation in unsteady mode and

continued until the forces on T2 settled into a periodic pattern and the cycle-to-cycle variation in the wake velocity profiles were minimal. The solutions were then time averaged over six rotor revolutions. The time step used was equivalent to 180 time steps per revolution of the front rotor. 20 Newton sub-iterations coupled with a dual time-stepping approach was used to ensure temporal convergence of the resulting simulation. These guidelines for temporal convergence have been established in a previous (BT1) study. [5] It should be noted that the BT1 study was for a single wind turbine and no study investigating temporal convergence for multiple turbines is available to authors' knowledge. Solutions were obtained using the two-equation Menter SST (with DES modifications) turbulence model.

All cases presented here were simulated using 60 cores of an in-house cluster. The runtime per time step was approximately 130 seconds. The steady solution utilized 1800 time steps whereas the unsteady simulations were carried out for 9000 time steps after restarting from a steady solution. Total run times for each simulation averaged about two weeks.

VI. Results

The results presented here cover a wide range of operating conditions. The operating points considered here have T1 running at a TSR of 6.0, while T2 was operated at three different TSR's (2.5, 4.0, and 7.0). Since T1 and T2 are essentially identical, a TSR of 6.0 indicates the design point for both. A TSR of 4.0 for T2 corresponds to peak power production, while a TSR of 2.5 results in T2 operating in a propeller mode and a TSR of 7.0 corresponds to a runaway condition for the turbine.

The primary objective of a wind turbine is to generate torque which can be used to turn the generator and in turn generate power. The power generation depends on inflow characteristics like wind speed, wind shear, etc. These inflow characteristics are significantly modified, relative to the freestream, when turbines operate in each other's wakes and can affect their operation and power generation. [21] Thus, having a good idea of the flowfield seen by a downstream turbine will enable one to predict its performance with reasonable accuracy. The BT1 experiment, which was also carried out at NOWITECH, was focused on obtaining this data. Good agreements for the mean flow as well as the fluctuations were obtained by the authors in earlier research. The purpose of the BT2 study was to assess the impact of wake on the performance of a wind turbine. In the following sections, impact of the wake of T1

on T2 is assessed by looking at the power and thrust coefficients of these turbines as well as the wake data of the T2 turbine.

A. Time-averaged thrust and torque predictions

As part of the tests carried out on the BT2 turbines, thrust and power were measured for the cases described earlier. [6] Computed results (time averaged over six rotor revolutions) using the SST-DES turbulence model are compared to the experimental data in Figure 6 for T2. Note that the power and thrust coefficients were calculated using the respective turbine diameters.[6]

For the T1 turbine, the computed values of power and thrust coefficients are 0.4518 and 0.826 (averaged for the three cases with maximum deviation of 4×10^{-4} and 3.66×10^{-4} . The simulation under-predict the power and thrust coefficients with an error of 2.1 % and 4.9 % respectively. Figure 6 shows the comparison of the thrust and power coefficients for T2. The overall power production for T2 is substantially lower than that of T1 and shows the impact of operating in another turbines wake. Excellent agreement is obtained between the computational results and the experimental data for the power and thrust coefficients with a slight under-prediction at the TSR of 7. The power and thrust coefficients are slightly under-predicted at high TSRs (this was noticed for the BT1 case also [5]) and could be due to compressibility effects no longer being negligible. The error at TSR of 4 is 4.4 % and 5.3 % for power and thrust coefficients respectively.

Figure 6: Comparison of power and thrust for T2

B. Time-averaged Wake Data (Mean Flow)

Wake data was collected downstream of the T2 turbine at three different locations in the BT2 experiment. These locations corresponded to $x/D = 1, 2.5$, and 4, corresponding to near-wake, intermediate-wake and far-wake

locations. The experimental wake surveys included horizontal and vertical sweeps providing a three-dimensional view of the wake behind the aft turbine.

Time-averaged (over six rotor revolutions) computed results using the SST-DES turbulence model are compared to the experimental data for several stations downstream of T2. In order to facilitate easier comparison between the various TSRs, the same layout is used for all the figures. In each figure, the horizontal and vertical wake surveys, grouped by the axial station, are shown next to each other. The locations of the axial stations are shown in Figure 2.

1. Peak Power ($TSR_{T2} = 4.0$):

Error! Reference source not found. shows the comparison of the mean velocity profiles for the peak power production case ($TSR_{T1} = 6.0$, $TSR_{T2} = 4.0$). Overall, the peaks in the velocity profiles near the tip of the blades are not distinct and the velocity profile smoothens quickly with no distinctive features visible from the second station onwards. This is, in part, due to the enhanced mixing due to wake of the front turbine impacting the aft turbine and mixing with its wake.

The effect of the presence of the nacelle is clearly seen in the wake at the $x/D = 1.0$ location (**Error! Reference source not found.**(a)). At the $x/D = 2.5$ location, the effects of the nacelle are smoothened out in the experimental data, while the computational results still exhibit some nacelle effects. The slight asymmetry at $z/R = -0.3$ (**Error! Reference source not found.**(a)) is due to the turbine and tower interaction and was reported for the BT1 case [4,5] also. There is a large discrepancy in all the vertical profiles with respect to the location where the velocity starts to recover. As an example, in **Error! Reference source not found.**(b) we see that the experimental data shows that the velocity starts to recover at $y/R = 0.7$ whereas the computations suggest a location of $y/R = 1.0$. This was observed in the BT1 test case [4,5] and was also reported in the results published by the organizers of the BT2 workshop[6].

Another discrepancy is seen in the velocity outside of the core region of the wake (around z/R of ± 2.0 in the horizontal sweeps in **Error! Reference source not found.**) where the computations under-predict the acceleration. It is expected that the flow will accelerate around the turbines due to the blockage effects of the turbines. None of the participants in the BT2 workshop [6] were able to predict this flow acceleration accurately and all under-predicted the flow velocity similar to the results presented here.

Further examination of the horizontal sweep velocity profiles reveals an anomaly (small peaks) around $z/R = \pm 2.85$. The source of this anomaly was investigated using a simple flat plate case, wherein a turbulent flow past a

272 flat plate was simulated using various off-wall spacings. It was found that such a peak could be reproduced by
 273 changing the grid resolution in the wall normal direction. The peak could be caused by inadequate grid resolution in
 274 the wall normal direction possibly leading to a severe momentum deficit in the first few control volumes next to the
 275 wall. This in turn leads to over-compensation immediately after the boundary layer (local acceleration outside the
 276 boundary layer). Another possible cause could be the turbulence model which depends on the accurate velocity
 277 gradients for estimating the eddy viscosity field. In this particular case, viscous spacing was not employed on the
 278 side walls of the tunnel leading to the presence of this anomaly.

Figure 7: Comparison of axial velocity profiles for peak power case at various locations

2. Runaway ($TSR_{T2} = 7.0$):

Velocity plots for the second test case of runaway condition ($TSR_{T1} = 6.0$, $TSR_{T2} = 7.0$) are shown in Figure 8. The trends for these cases are similar to those for the peak power case ($TSR_{T2} = 4.0$) and comments made earlier in regard to good overall agreement, acceleration outside the core region of the wake, mismatch of the location where the velocity starts to recover in vertical profiles and the peak in velocity observed near the side walls apply here too. The velocity is under-predicted near the edge of the turbine ($z/R = \pm 1$) as seen in Figure 8(a) and (b). The nacelle wake is distinctly visible at the first station and slightly observed at the second station. The computational results are not as smooth as the experimental ones at the last station $x/D = 4.0$ (both horizontal and vertical sweeps; Figure 8(e), (f)). This might indicate the necessity of carrying out the simulation for a longer period of time or averaging the solutions over a larger time period than the 6 revolutions used here.

Figure 8: Comparison of axial velocity profiles for the runaway case at various locations

3. Stalled ($TSR_{T2} = 2.5$):

The comparisons shown in Figure 9 follow more closely the trend for the peak power case. The observations and comments made earlier in regard to mismatch of the location in the vertical profiles, velocity peak near the side walls and the acceleration outside the core region of the wake apply here too. The nacelle wake is distinctive, stronger and persists for longer as observed in the computational results. There is lot of variability in the vertical velocity profile at the first station (Figure 9(b)) and this can be attributed to the computations capturing some of the subtleties of the wake behind the stepped tower that was used for T2.

Horizontal Sweep

Vertical Sweep

Figure 9: Comparison of axial velocity profiles for the stalled case at various locations

299

300 C. Time-averaged Velocity Fluctuations

301 The experimental data also included measurements of the fluctuations in the axial velocity. These measurements
 302 were taken at the same locations as the mean velocity profiles. In this section, computational results are compared
 303 with the experimental data for the three cases. The layout of the figures is the same as was for the velocity profiles

with horizontal and vertical sweeps grouped together by axial stations. These results are also time-averaged over six rotor revolutions.

1. Peak Power ($TSR_{T2} = 4.0$):

Figure 10 shows the comparison of the velocity fluctuations for the peak power production case ($TSR_{T1} = 6.0$, $TSR_{T2} = 4.0$). Overall, the trends are captured very well but the computational results are not as smooth as the experimental data. At the first station (Figure 10(a) and (b)), peaks in the results near $z/R = -1$ and $y/R = -1$ are well captured but they are missing on the other end ($z/R = 1$ and $y/R = 1$). From second station onwards, computations pick-up the peaks at both ends in horizontal and vertical profiles. The mismatch of the location in vertical velocity profiles observed in the previous section is evident here also in vertical profiles, especially at the last two axial stations.

Figure 10: Comparison of axial velocity fluctuations for the peak power case at various locations

2. Runaway ($TSR_{T2} = 7.0$):

Velocity fluctuation plots for the second test case of runaway condition ($TSR_{T1} = 6.0$, $TSR_{T2} = 7.0$) are shown in Figure 11. The comments made earlier for the peak power production case apply here too in regard to mismatch in location in the vertical profiles and results being not as smooth as data. However, a difference is noted at the first station ($x/D = 1.0$) where the peaks are captured well at both the ends in horizontal and vertical profiles. There is some over-prediction of the velocity fluctuations at the peaks.

Figure 11: Comparison of axial velocity fluctuations for the runaway case at various locations

3. Stalled ($TSR_{T2} = 2.5$):

The comparisons shown in Figure 12 follow more closely the trend for the peak power case. The overall trends are well captured and the locations of the peaks as well as their magnitudes are reasonably predicted. The observation made earlier about the missing peaks on one end at the first station apply here also, although to a much lesser extent in the vertical profile. The mismatch of the location in the vertical profiles is observed at the first two stations and less evident at the last station.

Figure 12: Comparison of axial velocity fluctuations for the stalled case at various locations

D. Comparison of wake structure

Figure 13 shows the wake interaction obtained using the SST-DES model. The isosurface of the Q -criterion (value = 0.05) colored by axial velocity (range -10 m/s to 15 m/s) for the three test cases is shown. Recall that for all these cases, T1 is always operating at a TSR of 6.0. Consequently, the wake structure immediately behind T1 is very similar among the three cases. The wake behind T2, however, is different. The TSRs for T2 are 2.5, 4, and 7 for (a), (b), and (c) respectively. One would expect that the wake structure would be different since the TSR of T2 is varying from 2.5 to 7. However, this is not the case as evident from the figures. There are no discernible differences in the wake structure and this can be attributed to wake of the T1 turbine being fed as the inflow to the T2 turbine. However, we do notice that as the rotational speed of T2 increases, the vortex breakdown occurs sooner leading to turbulent structures occurring earlier. In contrast, results for the BT1 case (isolated turbine) showed well defined helical patterns even up to a TSR of 10.0 [4,5].

VII. Summary and Conclusions

The flow through two model wind turbines operating in tandem was simulated using a DES version of the SST turbulence model. The DES version allowed the velocity fluctuations to be captured well across the range of TSR. It was found that the impact of the wake of the front turbine on the performance of the rear turbine was severe with the peak power coefficient being dropped from 0.5 to 0.12.

Very good agreement between the computational and experimental data is observed for the performance of the turbine. The agreement was good for the velocity fluctuations with the trends being captured well. Reasonable agreement is observed for the velocity profiles in the wake region. This test case provides validation of the capabilities of *Tenasi* in predicting flow fields associated with multiple wind turbines. The jaggedness and high variability in some of the data might indicate that the simulations should be run for longer, however, with the available computing resources, it was not feasible to do so.

Acknowledgments

The research presented here was supported by the National Science Foundation (NSF) through a grant (CBET # 1236124) with Dr. Ram Gupta and Dr. Gregory Rorrer as technical monitors. This support is gratefully acknowledged. Experimental data for the BT2 test case was provided by Prof. Per-Åge Krogstad of NOWITECH/NTNU, Norway. Images were created and data extractions carried out using FieldView as provided by Intelligent Light through its University Partners Program.

References

- [1] D.A. Simms, S. Schreck, M. Hand, L.J. Fingersh, NREL unsteady aerodynamics experiment in the NASA-Ames wind tunnel: a comparison of predictions to measurements, National Renewable Energy Laboratory Colorado, USA, 2001.
- [2] J.G. Schepers, H. Snel, Model experiments in controlled conditions, 2007. <http://www.ecn.nl/docs/library/report/2007/e07042.pdf>.
- [3] P.-Å. Krogstad, P.E. Eriksen, “Blind test” calculations of the performance and wake development for a model wind turbine, *Renew. Energy*. 50 (2013) 325–333. doi:<http://dx.doi.org/10.1016/j.renene.2012.06.044>.
- [4] K. Sreenivas, C.B. Hilbert, A. Mittal, L. Hereth, L.K. Taylor, High-Fidelity Computational Simulation of the Wake Characteristics of a Model Wind Turbine, in: 31st AIAA Appl. Aerodyn. Conf., American Institute of Aeronautics and Astronautics, 2013. doi:[doi:10.2514/6.2013-2416](https://doi.org/10.2514/6.2013-2416).
- [5] A. Mittal, K. Sreenivas, L.K. Taylor, L. Hereth, C.B. Hilbert, Blade-Resolved Simulations of a Model Wind Turbine: Effect of Temporal Convergence, *Wind Energy*. (2015). doi:[10.1002/we.1949](https://doi.org/10.1002/we.1949).
- [6] F. Pierella, P.-Å. Krogstad, L. Sætran, Blind Test 2 calculations for two in-line model wind turbines where the downstream turbine operates at various rotational speeds, *Renew. Energy*. 70 (2014) 62–77. doi:<http://dx.doi.org/10.1016/j.renene.2014.03.034>.
- [7] M.J. Churchfield, S. Lee, S. Schmitz, Z. Wang, Modeling Wind Turbine Tower and Nacelle Effects within an Actuator Line Model, in: 33rd Wind Energy Symp., American Institute of Aeronautics and Astronautics, Orlando, 2015. doi:[doi:10.2514/6.2015-0214](https://doi.org/10.2514/6.2015-0214).
- [8] B. Plaza, R. Bardera, S. Visiedo, Comparison of BEM and CFD results for MEXICO rotor aerodynamics, *J. Wind Eng. Ind. Aerodyn*. 145 (2015) 115–122. doi:<http://dx.doi.org/10.1016/j.jweia.2015.05.005>.
- [9] D.G. Hyams, K. Sreenivas, R. Pankajakshan, D.S. Nichols, W.R. Briley, D.L. Whitfield, Computational simulation of model and full scale Class 8 trucks with drag reduction devices, *Comput. Fluids*. 41 (2011) 27–40. doi:[10.1016/j.compfluid.2010.09.015](https://doi.org/10.1016/j.compfluid.2010.09.015).
- [10] L.K. Taylor, Unsteady three-dimensional incompressible algorithm based on artificial compressibility, Mississippi State University, 1991.

- [11] L.K. Taylor, K. Sreenivas, R.S. Webster, J. Kress, An Artificial Compressibility Algorithm for Convective Heat Transfer, in: 44th AIAA Thermophys. Conf., American Institute of Aeronautics and Astronautics, 2013. doi:doi:10.2514/6.2013-2894.
- [12] W.R. Briley, L.K. Taylor, D.L. Whitfield, High-resolution viscous flow simulations at arbitrary Mach number, *J. Comput. Phys.* 184 (2003) 79–105. doi:http://dx.doi.org/10.1016/S0021-9991(02)00018-9.
- [13] A.J. Chorin, A Numerical Method for Solving Incompressible Viscous Flow Problems, *J. Comput. Phys.* 135 (1997) 118–125. doi:http://dx.doi.org/10.1006/jcph.1997.5716.
- [14] D.G. Hyams, An investigation of parallel implicit solution algorithms for incompressible flows on unstructured topologies, Mississippi State University, 2000.
- [15] D. Hyams, K. Sreenivas, R. Webster, N. Currier, A Generalized, Unstructured Interpolative Interface method for Rotor-Stator Interactions, in: 20th AIAA Comput. Fluid Dyn. Conf., American Institute of Aeronautics and Astronautics, 2011. doi:10.2514/6.2011-3700.
- [16] D.G. Hyams, R.W. Webster, K. Sreenivas, A Generalized Interpolative Interface for Parallel Unstructured Flow Solvers, in: 40th Fluid Dyn. Conf. Exhib., Chicago, IL, 2010.
- [17] F. Pierella, P.E. Eriksen, L. Sætran, P.-Å. Krogstad, Invitation to the 2012 “Blind Test 2” Workshop, (2012). [http://www.norcowe.no/doc/NORCOWE-NOWITECH joint workshop 2012 - invite\[2\].pdf](http://www.norcowe.no/doc/NORCOWE-NOWITECH%20joint%20workshop%202012%20-%20invite%5B2%5D.pdf) (accessed July 07, 2015).
- [18] Mesh Generation Software for CFD - Pointwise, (n.d.). www.pointwise.com.
- [19] P.-Å. Krogstad, M.S. Adaramola, Performance and near wake measurements of a model horizontal axis wind turbine, *Wind Energy*. 15 (2012) 743–756. doi:10.1002/we.502.
- [20] H. Aboshosha, A. Elshaer, G.T. Bitsuamlak, A. El Damatty, Consistent inflow turbulence generator for LES evaluation of wind-induced responses for tall buildings, *J. Wind Eng. Ind. Aerodyn.* 142 (2015) 198–216. doi:http://dx.doi.org/10.1016/j.jweia.2015.04.004.
- [21] A.M. Abdelsalam, K. Boopathi, S. Gomathinayagam, S.S. Hari Krishnan Kumar, V. Ramalingam, Experimental and numerical studies on the wake behavior of a horizontal axis wind turbine, *J. Wind Eng. Ind. Aerodyn.* 128 (2014) 54–65. doi:http://dx.doi.org/10.1016/j.jweia.2014.03.002.