High Fidelity Fluid-Structure Interaction Analysis of a Wind Turbine Blade

Mark E. Braaten¹ and Charles Seeley²
GE Global Research Center, Niskayuna, NY 12309 USA

Michael Tooley³
ANSYS Canada Ltd., Waterloo, Ontario N2J 4G8 Canada

A two-way coupled fluid structure interaction (FSI) analysis for a realistic wind turbine blade has been successfully demonstrated using ANSYS Mechanical and ANSYS CFX software. This analysis couples a sophisticated composite shell finite element model (FEM) model of the blade with an efficient hybrid Reynolds-Averaged Navier-Stokes / Vortex Line Method Computational Fluid Dynamics (RANS/VLM CFD) aerodynamic analysis. The FSI procedure was applied to the analysis of an aerodynamic twist (AT) blade. A simplified isotropic model of the blade structure was used initially to develop the overall FSI procedure, but it was found that a composite shell model was required to get realistic results. The results of the FSI analysis yield bending and induced twist of the blade, and stresses and strains within the blade structure. Steady analyses were performed to compute the deformed shape of the blade under different steady wind loads, and unsteady analyses were performed to study the effects of yaw and unsteady wind gusts. The computed tip deflections and induced twists of the steady FSI analyses compare favorably to ADAMS computations for the same case. The use of the hybrid CFD procedure allows the CFD mesh to be small enough to allow the transient FSI computation to be completed in a reasonable amount of time, typically less than a day on a modest number of processors.

Nomenclature

\[ \text{c}^{\text{stiff}} = \text{exponent} \]
\[ d_{\text{wall}} = \text{distance to wall} \]
\[ \mathbf{\Delta} = \text{mesh displacement relative to previous mesh} \]
\[ \mathbf{\beta}^{\text{stiff}} = \text{mesh diffusivity} \]

I. Introduction

All wind turbine blades bend and twist as they experience wind loads. For most existing wind turbine blade designs, the blades are rigid enough that the deflections and induced twist are small relative to the blade, allowing the computation of the aerodynamic properties in an uncoupled manner using an assumed shape of the blade. Aero-elastic Twist (AT) blades are designed to untwist as the wind speeds increase, to passively reduce the wind loads. Such AT blades may bend as much as several meters in the wind direction, and the induced twist can become quite large. For the blade to perform at the optimal operating condition (angle of attack) and avoid tower clearance issues, the blade must be designed with a proper amount of prebend and pretwist. An accurate knowledge of the bending and induced twist caused by the wind loads requires a coupled Fluid-Structure Interaction (FSI) computation to properly design the blade with the correct prebend and pretwist.

¹ Senior Engineer, Fluid Mechanics Laboratory, GE Global Research, K1-2C36A, Niskayuna, NY 12309.
² Senior Engineer, Vibrations Laboratory, GE Global Research, KW-D212 Niskayuna, NY 12309, Member, AIAA.
³ Senior Technical Services Specialist, Support and Services, ANSYS Canada Ltd., 283 Northfield Dr E., Unit 21, Waterloo, Ontario, N2J 4G8, Canada.
A common approach for FSI analysis of wind turbine blades utilizes a simplified beam model for the structure of the blade, and a simple aerodynamics model for computing the aero loads. Examples include FAST \(^1\) or ADAMS \(^2\) coupled with AeroDyn \(^3\) for computing the aero loads. A detailed review of such models for the aeroelastic analysis of wind turbines is given in Ref. 4. The Blade Element Momentum (BEM) aero model in AeroDyn, that is not capable of capturing the effect of sweep or dihedral, can be replaced by a Vortex Line Method (VLM) \(^5\) for applications to modern blades with 3D aero features such as sweep and winglets, as described in Ref. 6.

Such beam-based FSI analyses, while useful for getting basic loads information for a large number of wind load cases, are unable to provide detailed distributions of the 3D aero loads or the resulting local stresses and strains within the blade structure. The purpose of this work was to explore a higher fidelity 3D computational procedure using a more accurate 3D structural and aerodynamic model. Some earlier studies using a combination of Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA) structural models for FSI analyses of aircraft and wind turbine blades are given in Ref. 7-8. Following is a description of the development of the structural FEA model, the aerodynamics CFD model, and results of the fully coupled FSI model. A realistic GE Wind aerodynamic twist (AT) blade is used to demonstrate the procedure.

II. Methodology

The primary goal of this effort was to demonstrate FSI analysis of a wind turbine blade using a realistic blade with high fidelity, fully coupled FEA and CFD models. This necessitated development of an FEA model with structural properties that represented a realistic wind turbine blade. A hybrid CFD model was also developed that combined a Reynolds Averaged Navier-Stokes (RANS) CFD analysis in the near-blade region with a vortex line method (VLM) analysis in the far field. This approach offered high fidelity 3D accuracy of the flow with significantly reduced run time compared to the full domain approach. The FEA and CFD models, developed independently, were then integrated for the coupled FSI solution. The development of the FEA, CFD, and full FSI models is described in the following sections.

A. Structural Model

An existing GE AT wind turbine blade design was selected to demonstrate the ANSYS / CFX FSI procedure. A structural FEA model of the original blade was available from the original blade design effort, but there were some issues with the geometry that made it unsuitable for use with the present demonstration. Therefore, the structural model was re-created in a form that was useful for the present study. Then, static and modal analyses was performed to validate that the new structural model accurately reproduced the response of the original blade model.

The original structural model of the blade, termed the “source” model, was an ANSYS finite element (FEA) model. Shell elements (SHELL99), with both membrane and bending capabilities, were used to discretize the domain of the structural components including the composite blade skin, spar cap and shear web. The source model was only a single blade that was fixed at the root. The composite layer properties were defined on an element by element basis. This meant that there was an ANSYS “real” property set for each and every element. The challenge was to map these properties onto the new model to be used with the FSI procedure.

The first attempt to re-create the source blade FEA model in a form that was suitable for the FSI demonstration was a simplified shell element model. The material properties were simplified to a single representative isotropic material, and the blade and spar cap were each defined by a single thickness. An attempt was made to adjust the simplified isotropic material properties and thicknesses to match the first few natural frequencies of the source blade. Parameters were found that could match either the first flap frequency, or the first torsion frequency, but no parameter set could match both. Reasonable matching of both of these natural frequencies was considered to be important due to the aero-elastic twist (AT) design, so it was suspected that the isotropic simplification would not adequately represent the AT blade selected for demonstration, as later proved to be the case.

For the improved structural model, the new ANSYS FEA model, termed the “target” model, was to be used with the FSI procedure based on the physical properties and orthotropic materials of the source model. The CAD geometry of the blade that was available had thin slivers of surfaces at the trailing edge necessary to define the aerodynamic surface. Unfortunately, these surfaces were problematic for meshing the structure because they led to elements with vastly different sizes, leading to an unnecessarily large number of elements. Instead, a blocking structure was created in ANSYS / ICEM with patch independent meshing that easily led to a high quality quad shell
mesh and avoided other potential issues from high surface edge length ratios and gaps. The mesh was then copied two times and rotated, creating the three bladed rotor required for the FSI procedure. The target mesh for a single blade is shown in Figure 1. Newer, four node SHELL181 elements were assigned to the target model.

Next, the orthotropic composite layer properties needed to be mapped from the elements in the source model to the elements in the target model. The lack of alignment between the two meshes, except for the original CAD surfaces from which they were created, made this a challenge. A script was created that looped through all of the target elements and found the closest source element based on the element centroids. A table of target elements and corresponding source element “real” property set identifiers was created from this script. Another script was created to reformat the source “real” properties into the target “section” properties that are normally used with the SHELL181 elements. A similar script was developed to reformat the twenty or so source orthotropic material properties into input for the target model. The mapped orthotropic composite material properties needed to be aligned with the appropriate coordinate system for each blade. For example, the fiber direction needed to run along the blade span-wise axis to ensure that the spar cap had the correct stiffness. Three independent global coordinate systems were created, one for each blade. The element coordinate systems in each blade were then aligned to the appropriate global blade coordinate system. These coordinate systems insured that the orthotropic material properties were correctly aligned with each blade. The composite element property mapping process is shown in Figure 2.

Once the new structural model was built, it was compared with known modeling results from the AT blade to ensure that it had similar physical behavior. Results from an ADAMS model, with a simplified beam representation of the blades, and simplified aerodynamics, were available for comparison. The new FEA model predicted a static tip deflection with a ratio of 0.93 compared to that predicted by ADAMS for a selected steady wind case (similar to a static load). This was judged to be acceptable given some uncertainties in the details of the blade construction between the two models.

Next, natural frequencies and mode shapes were compared between the source ANSYS model and the new structural model. It was critical that the first flap natural frequency match as closely as possible since it dominated the dynamic response of the blade in most load cases. The first edge and first torsion natural frequencies were also of interest, although more leniency could be given since these modes did not contribute as much to the blade response as the first bending mode. The results are shown in Figure 3. Here, mode shapes for the first flap, edge and torsion modes are shown for the new target structural model. In all cases, the new mode shapes were indistinguishable from the mode shapes calculated by the source ANSYS model. Ratios of the target to source natural frequencies are also given for these modes. The frequency ratio for the first flap mode was 1.00, indicating a very close match. The ratios for the edge and flap modes were 1.15 and 0.96, respectively. Although not as good of a match as the first flap model, they were judged to be close enough for a realistic demonstration of the FSI
procedure. Therefore, the new target model, with mapped orthotropic material properties, was able to match the first flap, edge, and torsion natural frequencies simultaneously.

![Figure 3. Structural model validation. The first flap, edge and torsion mode shapes are indistinguishable between the source and the target structural models. Natural frequency ratios indicated that the first flap frequencies were very close, and the other modes were close enough to proceed with the model.](image)

Based on the comparison of the static deflection, the mode shapes, and the natural frequencies of the first flap, edge and twist modes to previous model results, the new structural model was judged sufficient to represent the physical behavior of the GE AT wind blade. It was ready to integrate into the ANSYS CFX FSI model for demonstration of the procedure.

**B. Aerodynamic Model**

Conventional full-domain CFD for a wind turbine blade can become very expensive in terms of both mesh size and CPU time. Consider a typical wind turbine CFD domain, as shown in Figure 4. For a three-bladed rotor, periodicity requires a 120° sector. The inlet, outlet, and far field boundary conditions must be imposed many blade radii away from the blade (often as much as ten) to get a solution that is independent of their exact position. It is extremely difficult to accurately capture the effect of the wake downstream of the blade due to the numerical dissipation inherent in any conventional CFD scheme. Even with a mesh in excess of 10-20 million points, wall functions must usually be resorted to in order to capture the boundary layer effects on the blade.

Hybrid CFD approaches (see Ref. 9-11) combine a Reynolds Averaged Navier-Stokes (RANS) CFD analysis in the near-blade region with a vortex line method (VLM) analysis in the far field. This reduces the size of the RANS domain dramatically, as shown in Figure 5, reducing mesh size and run time by an order of magnitude or more over the full domain approach. With this reduction in required computational resources, hybrid CFD enables modeling of turbulent transition effects on wind turbine blades by enabling much finer resolution of the boundary layer region True unsteady 3D computations become manageable, allowing the effects of yaw, tower interaction, and wind gusts to be captured. The ability to do true unsteady computations in a reasonable time also enables detailed FSI analyses such as described in this paper.
Unlike the structural model, the hybrid CFD mesh fully resolved the thin trailing edge surface to ensure accurate aerodynamic results. This resulted in a non-matching geometry and corresponding gaps between the fluid and structural meshes at the trailing edge FSI interface region. When interpolating forces from the fluid to the structural mesh, a small number of unmapped nodes (compared to the number of nodes on the FSI interface) were expected and seen in the trailing edge region. The forces acting on the thin trailing edge surface were not considered important to the overall structural response. Mesh displacements received by CFX for un-mapped nodes are set to the displacement of the closest mapped node.

The hybrid CFD model used here was implemented in CFX User Fortran, and is described in Ref. 11. The VLM method is used to compute induced velocities at the boundary points of the RANS domain, which are imposed as boundary conditions. In the VLM, the blade is represented as a combination of bound vortices on the lifting line and trailing vortices in the wake. Since the blade deformations are expected to be large, these vortex elements ideally need to move as the blade deforms. The polylines used to compute the circulation from the RANS domain are tied to the blade cross sections, and also should move as the blade deforms. To handle this, the motion of the vortices and the polyline points can be taken from the mesh displacement of the closest mapped points in the mesh. For the steady FSI results shown later, the VLM wake was updated periodically during the solution accounting for this motion, so that when the final steady state was reached, the VLM solution was consistent with the deformed CFD mesh.

Several major differences between transient and steady analyses arise. For a transient case, all three blades need to be modeled simultaneously, whereas for a steady analysis only one blade need be modeled. The CFD analysis has to be run in transient mode, increasing the required run time. Here the existing CFD and structural meshes were replicated. For the CFD mesh, this was easily accomplished in the CFX pre-processor CFX-Pre by rotation. For the structural shell model, not only must the mesh be instanced, but the composite element properties (which depend on direction) must also be properly interpolated as well.

Hybrid CFD creates three isolated regions, that may or may not overlap, as shown in Figure 6. The coupling between these regions comes through the application of the induced boundary velocities computed by the VLM. The presence of isolated domains does create an issue, however, as ANSYS requires that the pressure level in each domain be consistent to get consistent loads on the blades. Initially, an attempt was made at modeling the transient case with three isolated regions in a single domain (as required by our existing User Fortran for hybrid CFD), and tried setting the pressure level in each region by creating a small patch with prescribed pressure in each domain at the same corresponding point of the “outlet” boundary. While CFX ultimately would hold the desired pressure, it responded violently to new displacements from ANSYS in coupled FSI computations. This caused very large pressure spikes (and correspondingly large force spikes). Although they would die out as solution returned to the proper pressure level, allowing steady state solutions to be achieved, the pressure spikes inevitably caused failure of transient restarts. It was determined that the small pressure boundary patches were not a suitable way to enforce a pressure level in the domains in unconverted solutions.
An alternative approach, that eliminated this issue, was to specify each blade as a separate domain, and directly specify the same pressure level at the same corresponding point in each domain. This approach strictly enforced the pressure level at all times. The ability to set the pressure level in multiple disconnected domains in CFX required the use of the CFX Command Language (CCL). This approach precluded the use of the existing User Fortran, that was written for a single domain, and it was beyond the scope of this project to extend it. Recall that the purpose of the hybrid CFD is to provide the distribution of induced velocity on the boundary of the RANS domain. As a simplification, for the transient cases, the hybrid CFD was replaced with the specification of a constant induced velocity at the rotor plane. For the case of interest here, it was found that setting the axial induction to 0.3 on the RANS boundary yielded similar loads and deflections for the steady case as the hybrid CFD treatment.

C. ANSYS CFX/FSI Coupling

ANSYS is a general purpose finite element based code that is widely used for non-linear structural simulations. Its extensive element library covers a wide range of physics (including structural, thermal, acoustic, electric, piezoelectric and various coupled-field elements) and element topologies (including solid, shell and beam elements). Linear (low order) and non-linear (high order) elements are available.

CFX is a general purpose finite volume based CFD code. It uses unstructured 3D meshes made from hexahedral, tetrahedral, prism and/or pyramid elements. CFX is a cell vertex code that forms the control volumes from the mesh-dual of the elements.

The CFX solution algorithm allows for moving and deforming meshes, which may include a non-zero mesh velocity and changing control volume size/shape. The approach used is described in Demirdzic and Peric. Full conservation is maintained on the moving/deforming mesh. The transient term is discretized using a second order backwards Euler scheme (by default), which is conservative and implicit in time. The advection term is discretized using a high-resolution scheme (by default) similar to Barth and Jesperson. Other transient and advection discretization schemes are available but were not used here.

The ANSYS structural and CFX CFD solvers are coupled using an iteratively implicit coupling approach based around the ANSYS Multifield (MFX) solver. In a transient simulation a series of coupling iterations are executed within each time step until convergence metrics for the forces and displacements at the fluid-solid interface are achieved. This produces an implicit (in time) solution at convergence of each time step. Within each coupling iteration the individual ANSYS structural and CFX solvers perform their own iterative solutions to converge their respective solution fields. Coupling iterations within each time step continue, up to the maximum specified, or until the individual field solvers and the fluid-solid interface force and displacement transfers are converged to the user specified levels. Coupled steady-state simulations proceed in a similar way and can be compared to a single time step of a transient simulation.

The force values transferred from the CFX solver to the structure are based on the face momentum flows at the FSI interface. These flows drop out of the coupled mass-momentum equation solution in the CFX solver and naturally include pressure and viscous contributions. In interpolating the momentum flows onto the structural mesh, strict conservation is maintained. This is achieved by dividing each CFX element sector flow between the
overlapping structural element sectors. An intermediate control surface is used to intersect the dissimilar meshes and divide the flow contributions. The control surface flows are assigned to the structural element sectors and then gathered at the structural nodes to produce a nodal force. This algorithm is the same as the CFX solver uses to connect dissimilar meshes. See Galpin et al. \(^{14}\) for a more detailed description. There is no requirement for node matching between the two codes at the FSI interface.

The transfer of displacements from the ANSYS to the CFX solver is more straightforward, since displacement is not a conserved quantity. A distance weighted algorithm is used, that is described further in the ANSYS documentation \(^{15}\) where it is referred to as the Profile Preserving algorithm. The mesh displacements received by the CFX solver result in boundary motion at the FSI interface. The displacement of the non-boundary nodes is computed by solving a displacement-diffusion equation:

\[
\nabla \cdot \left( \Gamma_{\text{stiff}} \nabla \delta \right) = 0
\]

Here \(\Gamma_{\text{stiff}}\) is the mesh diffusivity and \(\delta\) is the mesh displacement relative to the previous mesh. A mesh motion boundary condition of “Unspecified” may also be set, in which case those boundary nodes are treated in the same way as internal nodes, with their location determined by the movement of other boundaries and the solution to the displacement diffusion equation.

An under-relaxation factor can be applied to the force and displacement transfers between the solvers. Under-relaxation is applied using the forces or displacements from the previous coupling iteration.

Communication between the ANSYS and CFX solvers occurs at run-time using standard internet socket communication (TCP/IP) without any file-based I/O. The required libraries exist in a standard ANSYS/CFX installation and do not require the installation of any additional inter-solver communication tools.

### III. Results

The focus of this work was to simultaneously compute the aerodynamic loads on the wind turbine blade, and the structural bending and induced twist of the blades under load, under both steady and unsteady wind conditions.

To demonstrate the ANSYS /CFX FSI procedure, a "realistic" wind turbine blade was desired for the analyses. At the same time it was felt important to protect the proprietary details of existing GE production blades, so the blade chosen was an early design of an aerodynamic twist (AT) blade, that is not in production. This blade was designed to untwist as the wind loads increased, to provide for a passive means of reducing the extreme blade loads. When viewing the computed results, it is important to note that this particular blade has no prebend, which is a bit atypical.

A series of steady and unsteady computations were performed, and the resulting aerodynamic loads, deflections, induced twists, and internal blade stresses and strains were computed. The software used was ANSYS/CFX 13.0. The shell mesh for the blade and the CFD mesh were both computed using ICEM CFD. The shell mesh for the blade was comprised of 6800 quadrilateral shell elements (7500 nodes), while the CFD mesh had 407,000 hexahedral cells. The flow was treated as incompressible.

The steady solutions obtained used a relatively small under-relaxation factor of 0.5 applied to the force/displacement transfers between the CFX and Mechanical solvers. This allowed the solvers to progress gradually towards a converged solution. The transient solutions were stable without the use of under-relaxation (i.e. under-relaxation factor of 1).

Run times for the computations were relatively modest. The steady runs reached a converged solution in about 500 iterations, taking 4 hours on 12 processors. Unsteady runs took about 18 hours on 12 processors for 250 time steps.

#### A. Steady FSI Runs
The initial runs were made for a steady wind speed of 8 m/s, and a blade rotation rate of 13 rpm. Initially the simplified isotropic shell model described earlier was used for the structural analysis. Problems were initially encountered with mesh folding due to the large bending deformations of the blade. Typically, once the bending deflection reached about one meter, the CFX runs failed due to mesh folding, resulting in either negative mesh volumes causing immediate termination of the solver, or excessively distorted meshes that led to negative sector volumes, causing the solver to diverge. The mesh folding problem was overcome by switching from a prescribed zero mesh deformation at the outer boundary to an unspecified prescription that allows the outer boundary to move with the blade, and through the following modified expression for the mesh diffusivity \( \Gamma_{stiff} \):

\[
\Gamma_{stiff} = \left( \frac{1}{d_{wall} + \varepsilon} \right)^2 CWT_{stiff} + \Gamma_{stiff, min}
\]

Here \( C_{stiff} = 2 \), and \( \Gamma_{stiff, min} = 1.0 \text{ m}^2/\text{s} \). This specification ensured that the mesh remained stiff near the blade, where the boundary layer mesh was finest, and enforced a lower limit on the mesh stiffness with causes the mesh motion to propagate out to the outer boundary, so that mesh folding was avoided.

Comparison of the initial results from the isotropic shell model indicated that the simplified model was not yielding realistic levels of either tip deflection or induced twist. To get a more realistic result, the shell thickness for both the skin and the stiffener were reduced by a factor of two. The resulting tip deflection of the blade, as a function of the number of iterations, is shown in Figure 7. The tip deflection is normalized by the value obtained from the ADAMS model. The initial and deformed mesh at the tip is shown in Figure 8, and the initial and deformed blade geometry is shown in Figure 9. The results from the isotropic shell model indicated that the simplified model was still not yielding realistic levels of induced twist (likely as a result of not matching the first torsional mode), so further effort was focused on developing and using a more realistic composite shell model of the blade that became the new target structural model with orthotropic material properties.

The same steady FSI run was repeated for the composite shell model, with the same unspecified mesh motion outer boundary condition and the same mesh diffusivity specification. This time, no jiggering of the shell model parameters was required, and the computed tip deflection (see Figures 10 and 11) and the induced twist (see Figure 12) compare more closely with that predicted by ADAMS.

![Figure 7. Tip displacement vs. iterations - isotropic shell model](image)

![Figure 8. Initial (red) and deformed (blue) mesh near tip - isotropic shell model (coarse grid shown for clarity)](image)
B. Unsteady FSI Runs

A number of unsteady runs were made to demonstrate the capability of the FSI procedure for such cases. The unsteady cases included yaw, and various unsteady wind gusts with both velocity and direction change. The unsteady gusts used were generated using the IECWind software package. This software is widely used to create wind files for loads analyses for the AeroDyn-based analysis programs such as FAST and ADAMS/AeroDyn. Certification of a wind turbine requires demonstration that maximum permissible loads are not exceeded over a large series of different wind conditions.
The unsteady FSI runs were restarted from a very well-converged steady-state solution. The results presented here are all normalized using the appropriate steady-state values. The first unsteady runs showed unbounded oscillations at the tip, as seen in Figure 13(a), leading eventually to failure of the FSI procedure due to excessive mesh deformation. A simple damping model was developed and used to specify sufficient Rayleigh structural damping to cause these oscillations to damp out, as seen in Figure 13(b). The logarithmic decrement method was used to compute the effective damping from the variation in tip displacement observed in the computations.

Yaw occurs when the wind direction is not in line with the axis of rotation of the wind turbine. Each blade then sees a sinusoidal variation in wind speed and direction as it revolves around the rotor axis in the θ direction. For the yaw case considered here, the wind direction changes from axial to 20° yaw over a 1 second time interval. To specify this in CFX 13, the boundary velocities are specified in the relative frame, and the θ offset is explicitly accounted for using CFX expression language.

As expected, the three blades have similar solutions that are out of phase by 120° to each other, once the periodic steady state is reached. The initial transient going from 0° yaw to 20° yaw causes a slight overshoot in tip displacement and forces, and the blades see a different response, depending on where they were oriented when the yaw started, as seen in Figure 14.

The first unsteady gust case considered here is an extreme operating gust that involves just a change in the wind velocity. As a result, all three blades experience the same wind conditions, and their solutions are the same, as expected, as seen in Figures 15 and 16 below. The peak wind velocity during the gust reaches a speed slightly above the rated speed for this machine. Since the rotor power varies as $V_{wind}^3$, it might be expected that the power output would exceed the rated power, but the peak power only reaches 88% of the rated power, due to the short duration of the gust. The tip deflection and induced twist also vary with the gust speed, in the manner expected.

The time histories of the span-wise and chord-wise strains at a point on the pressure side of the blade at 45% span, extracted from the transient ANSYS solutions, are shown in Figure 16. Note here that tension gives a positive strain, while compression gives a negative strain. The steady solution is clearly dominating the response, the blades are clearly bent in downstream direction, and the unsteady effects appear as perturbations about the steady response. The strains for this case in the fiber direction are well below the allowable strains for the material used in the blade construction.
The next unsteady case considered here involves an extreme coherent gust with direction change. The initial case attempted involved a wind speed that gusted to 23 m/s, and a yaw angle that went all the way to -90°. This initial case could not be solved due to extreme distortion of ANSYS mesh that resulted from the extreme aero loads, that ultimately led to ANSYS solver failure. An FSI solution was able to be obtained for a less extreme case where both the gust strength and direction change were reduced by 50%, leading to a peak wind speed of 15 m/s and a yaw angle of -45°.

As seen in Figure 17, for this case, the power drops off dramatically due to the high yaw angle, even as the wind speed increases. The peak blade forces occur during the initial transient, while the yaw angle is still not too large. The tip deflections follow the variations in force, with very large blade deformations and induced twist occurring in this case.

Figure 18 shows that the span-wise and chord-wise strains. These strains follow the forces and tip deflections closely, as expected, adding confidence to the FSI solution.
I. Concluding Remarks

This paper describes the demonstration of a coupled fluid structure interaction procedure using ANSYS and CFX to compute the aero and structural response of an AT wind turbine blade under both steady and unsteady wind conditions. Various challenges were encountered and some of the steps taken to overcome them are described. The results demonstrate that the ANSYS/CFX FSI capability is capable of analyzing realistic aero-elastic wind blade behavior.

Future work should include generalizing the hybrid CFD procedure to the three-bladed unsteady case, to allow for more accurate computation of the aerodynamic loads in such cases.

Acknowledgments

The authors would like to thank Peng Yuan, Jeff Koplic, and Mike Chudiak of ANSYS for their assistance, and Slawomir Kolasa of GE Polska for his help with generation of the CFD mesh.

References