Aeroelastic Investigation of the Sandia 100m Blade Using Computational Fluid Dynamics

David Corson
Altair Engineering, Inc.

Todd Griffith
Sandia National Laboratories

Tom Ashwill (Retired)
Sandia National Laboratories

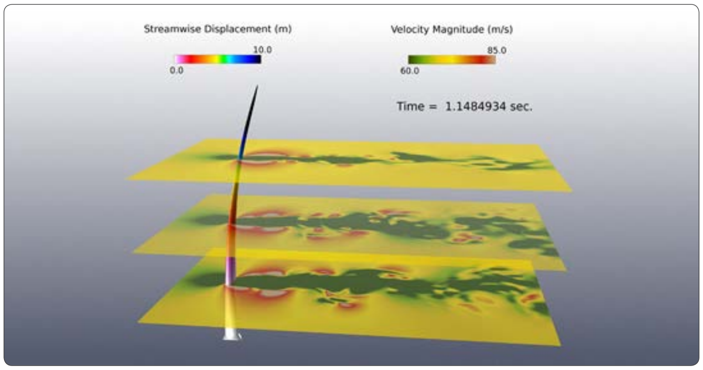
Farzin Shakib
Altair Engineering, Inc.

Abstract

Recent trends in wind power technology are focusing on increasing power output through an increase in rotor diameter. As the rotor diameter increases, aeroelastic effects become increasingly important in the design of an efficient blade. A detailed understanding of the fluid elastic coupling can lead to improved designs; yielding more power, reduced maintenance, and ultimately leading to an overall reduction in the cost of electricity. In this work, a high fidelity Computational Fluid Dynamics (CFD) methodology is presented for performing fully coupled Fluid-Structure Interaction (FSI) simulations of wind turbine blades and rotors using a commercially available flow solver, AcuSolve. We demonstrate the technique using a 13.2 MW blade design that has been developed by Sandia National Laboratories[1]. The results obtained using AcuSolve are compared against results obtained using other more commonly used simulation techniques, namely FAST[2] and WT-Perf[3].

Background

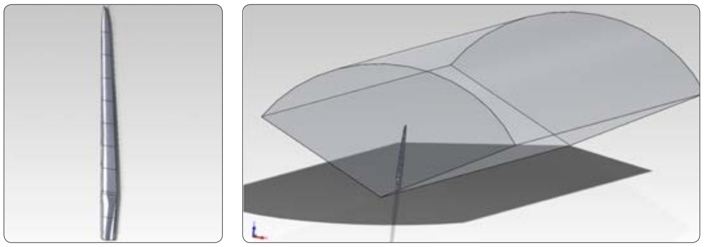
Current wind turbine design practices look towards desktop engineering tools such as FAST and ADAMS[4] to provide information about the aero-elastic behavior of the turbines. Additionally, frequency based approaches, such as those of Lobitz are also used[5]. Each of these techniques has advantages and disadvantages. An evolving approach for generating performance data on wind turbine rotors is through the use of Computational Fluid Dynamics (CFD). In addition to performing detailed flow field predictions around the rotor, modern CFD codes are increasingly capable of performing multiphysics simulations. These capabilities include the ability to run aeroelastic simulations in which the full fluid-structure interaction computation is performed. In this work, the AcuSolve CFD package is used to do precisely that. The fully coupled fluid/structure interaction problem is simulated using a modal superposition approach. This technique, known as Practical Fluid Structure Interaction (or P-FSI) requires the eigenvalues and eigenvectors of the structure as input to the CFD model. Once this information is provided, AcuSolve is able to independently compute the structural deformation in response to the fluid forces on the wetted surfaces.



AcuSolve fluid-structure interaction results showing the response of the Sandia 100m blade to a wind gust.

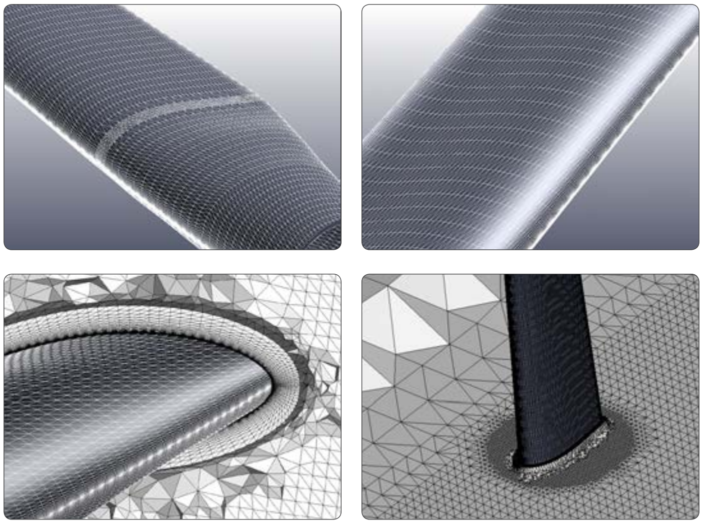
Modeling Methodology

The starting point for this analysis is a CAD model of the 13.2 MW blade design. The geometric model is created based on the specified airfoil sections that make up the blade geometry. For the CFD side of the analysis, the bounding fluid volume around the rotor is created using a simple cylindrical solid region. A boolean subtraction operation is used to subtract the blade volume from the surrounding volume, leaving only the air volume represented as a geometric solid. To keep the computational expense to a minimum, we exploit the rotational periodicity of the 3-bladed rotor. By modeling a 120 degree sector of the rotor, only a single blade is modeled, but the aerodynamics of the full rotor are taken into account through the periodic constraints. Since this analysis requires both a fluid and structural solution, a common blade model was used as the basis for both the AcuSolve CFD model as well as the OptiStruct structural model. A total of 100 eigenvalues were computed in OptiStruct, then mapped to the AcuSolve simulation to model the structural deformation.



Geometry of the 100 meter blade used for the aero-elastic simulations.

The unstructured meshing software, AcuMeshSim was used to discretize the fluid side geometry. Due to the high geometric aspect ratio of wind turbine blades,we utilize structured surface meshing along the span of the blade. The nodal distribution along the curves that represent the blade cross section are clustered near the leading and trailing edges of the blade to capture the high levels of curvature in these regions. The nodal spacing is increased with a gradation rate of 1.2 as the center of the blade is approached. This point distribution is then extruded in the span wise direction of the blade to create structured, anisotropic surface elements. Upon completion of the surface meshing process, the elements on the blade are extruded in the surface normal direction to create elements that are clustered near the surface to resolve the steep near wall gradients in the boundary layer. The extrusion process is continued until the last layer of extruded elements has a similar height as what is specified for the surrounding volume elements. The remaining volume is then meshed and united with the existing surface and boundary layer mesh to produce the final conformal mesh. Local volume refinement zones are used in the vicinity of the blade and in the wake region immediately downstream of the blade to maintain a high level of resolution in those areas.

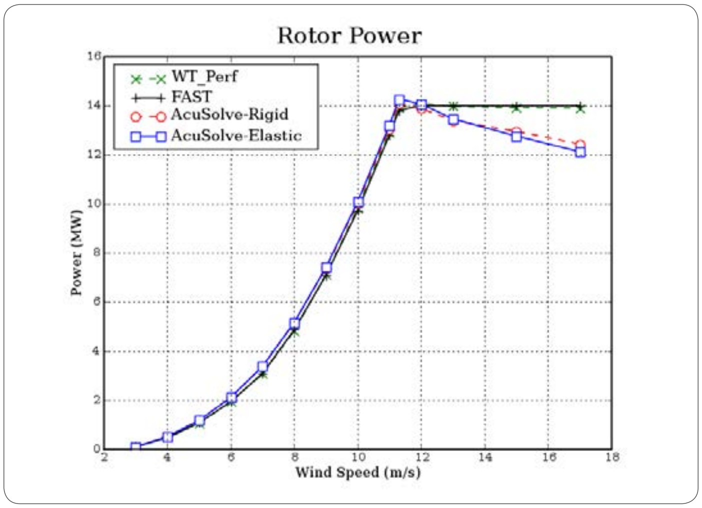


Images illustrating the CFD mesh used to perform the fluid-structure interaction simulations.

Results

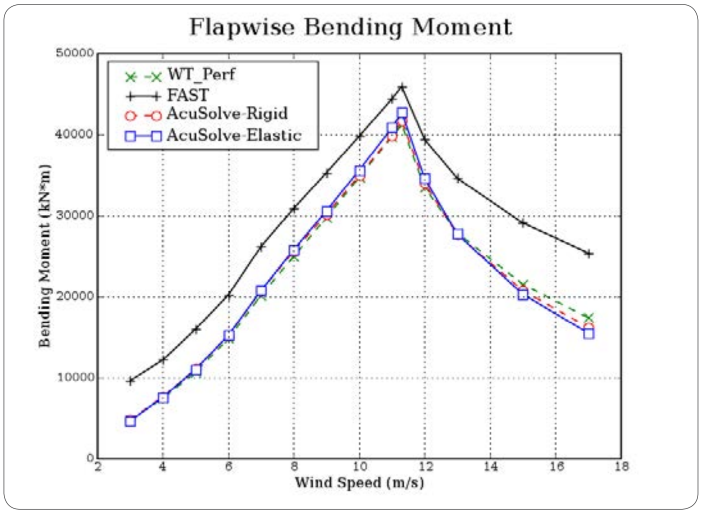
A power sweep of the 100m blade was performed using AcuSolve, FAST, and WT_Perf. For the AcuSolve simulations, a steady state solution approach was employed, using the Spalart-Allmaras RANS model. Two different types of simulations were performed. The first approach assumed a completely rigid structure. When performing this type of simulation, AcuSolve converges directly to the time averaged flow field by iterating the solution to a steady state. The second set of simulations that was performed employed a steady state solution procedure that incorporated flexible blades. Using this technique, AcuSolve is continuously updating the position of the blade in response to the flow forces. The solution is iterated to convergence , and the resulting deformation represents the mean deformed shape of the blade. These two simulation techniques allow direct comparison of the rotor thrust, flapwise bending moment, power, and tip displacement against FAST results. WT_Perf does not contain a structural representation of the blade, and therefore, only the aerodynamic quantities are compared against the other simulation techniques.

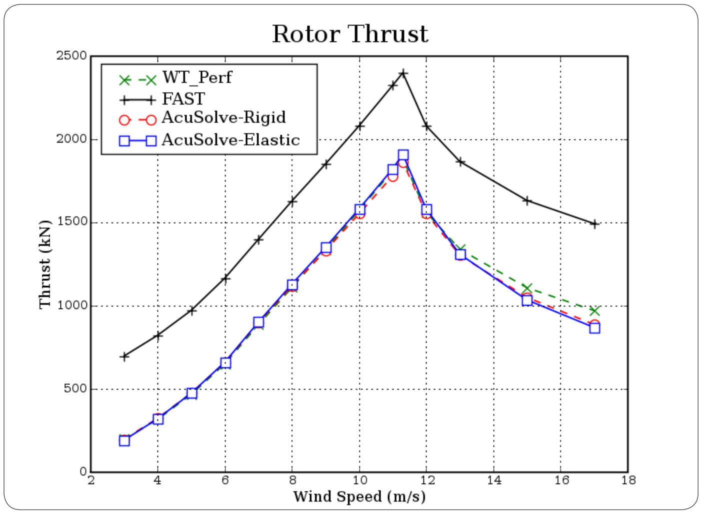
Excellent agreement is obtained for the power predictions amongst all three codes up to the 11.3 m/s wind speed. As the wind speed increases beyond this operating point, the blade pitch is changed to control the power output of the machine. Once the pitch control is active, the AcuSolve results show some discrepancies when compared to FAST and WT_Perf. The aerodynamics become increasingly complex in these operating conditions and differences are expected between the full solution of the RANS equations and the simplified techniques employed by FAST and WT_Perf.



Rotor power predictions resulting from AcuSolve, FAST, and WT_Perf.

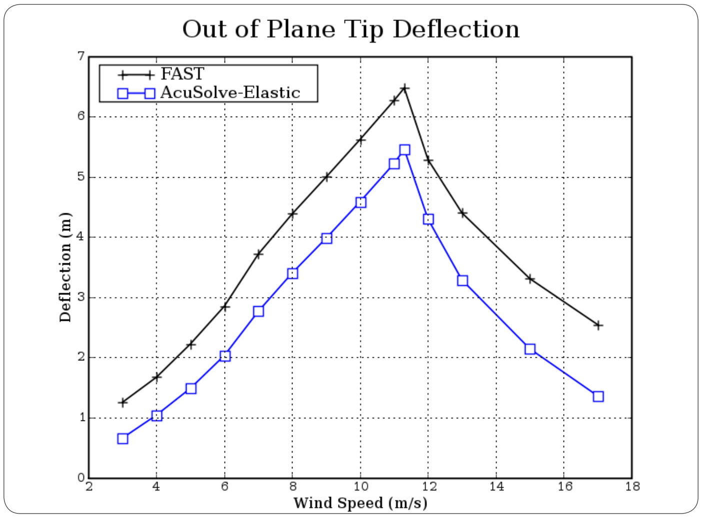
A further investigation of the aerodynamic predictions for the rotor is made by investigating the rotor thrust and flapwise bending moment for the blade. The thrust provides a comparison of the total forces acting on the blade in the streamwise direction, while the bending moment gives some indication of how the forces are distributed radially along the blade. Despite the good agreement that was seen for power, we see significant differences between the three codes when comparing the thrust. The AcuSolve results compare very well to WT_Perf, whereas the FAST results show an over prediction in thrust at all wind speeds when compared against the others. Accompanying this increased level of thrust is an increase in the flapwise bending moment. We see again that the WT_Perf and AcuSolve results agree very well when comparing the flapwise bending moment, while FAST tends to show a higher value.

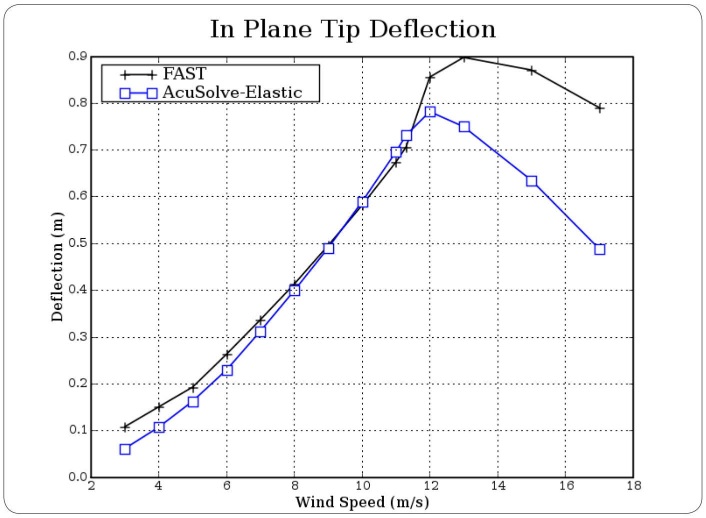




Flapwise bending moment and rotor thrust predictions from AcuSolve, FAST, and WT_Perf simulations.

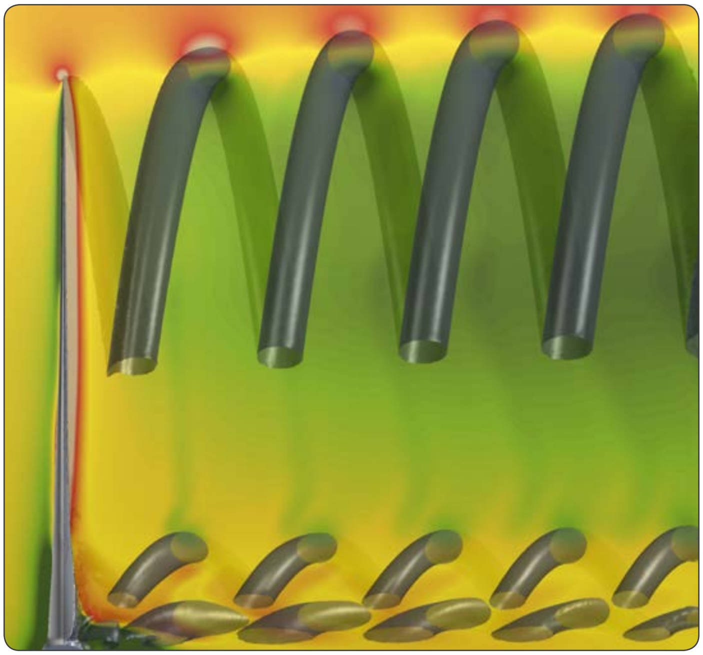
The higher loading predicted by FAST also leads to an increase in blade tip displacement, as shown below.

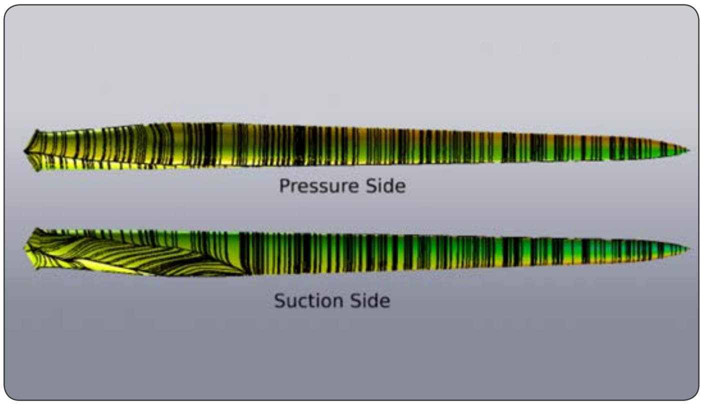




Prediction of blade tip deflection from AcuSolve and FAST.

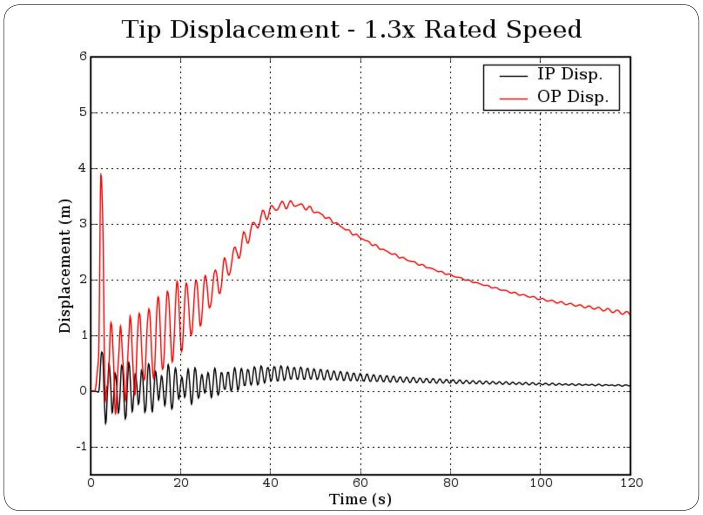
Representative results showing the flow features predicted by the CFD model are shown in the images.

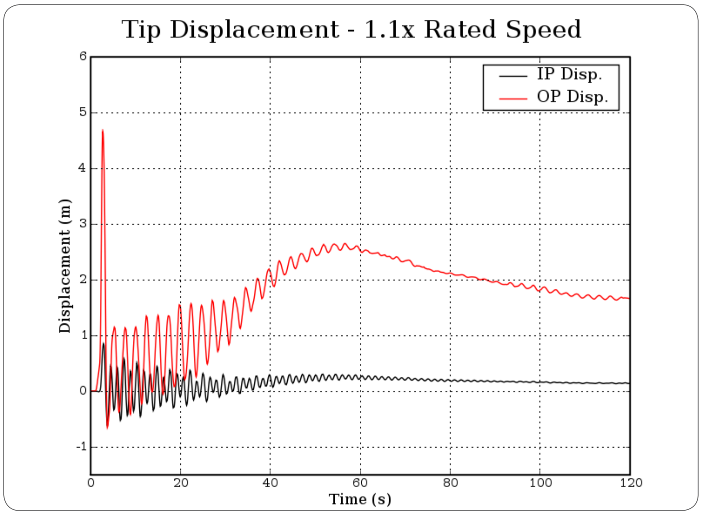


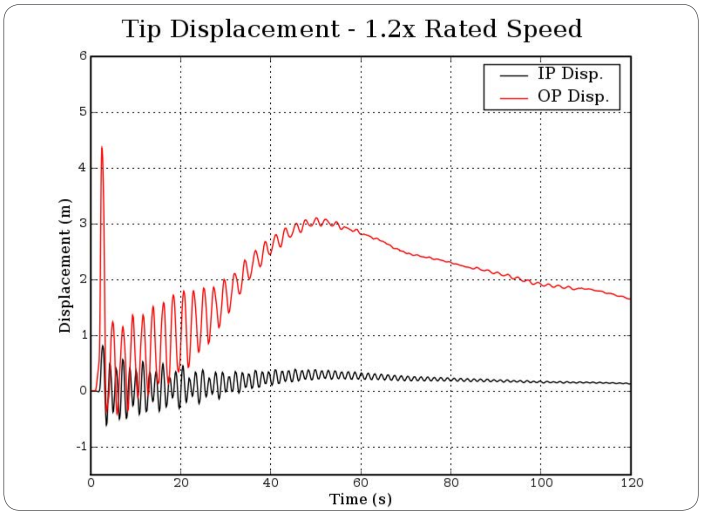


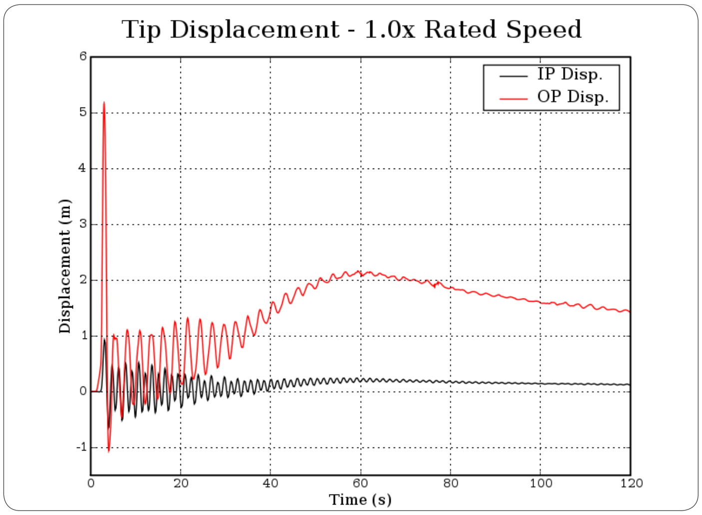
Once the accuracy of the CFD model was established, the aeroelastic stability of the blade was investigated. For these simulations, a fixed rotor speed was selected, and the blade was perturbed by an impulse force. Once the force was removed, the blade was allowed to oscillate freely. A total of 4 simulations were performed using this approach. Rotor speeds of 1.0, 1.1, 1.2, and 1.3 times the maximum rated rotor speed were investigated.

All rotor speeds investigated were characterized by an initial perturbation caused by the high amplitude displacement in the in plane and out of plane directions. After the initial displacement has passed, a smaller amplitude oscillation appears in both the in plane and out of plane directions. For the in plane direction, the oscillation appears to sustain itself for the duration of the simulation, but with a very low amplitude. For the out of plane motion, the behavior is characterized by a high frequency oscillation superimposed on top of a drifting displacement towards a mean displaced state. In all cases, the time record that was simulated indicates a decay of this signal, and no evidence of any type of fluid-elastic coupling that may lead to an undamped oscillation of the blade.


Tip Displacement - 1.3x Rated Speed


Tip Displacement - 1.1x Rated Speed


Tip Displacement - 1.2x Rated Speed


Tip Displacement - 1.0x Rated Speed

Plots showing the response of the blade to a structural perturbation. The results indicate no unbounded growth or self sustaining excitation of the blade in either the in plane our out of plane directions.

Conclusions/Summary

A Computational Fluid Dynamics based modeling approach has been developed to enable fully coupled FSI simulations of wind turbine rotors. The approach was first demonstrated on both a rigid and flexible blade using steady state modeling techniques. Comparison of these results with WT_Perf simulations of the 100m blade indicated favorable agreement. Comparisons to FAST showed deviations in thrust, bending moment, and tip displacement. However, the deviations reflect an over prediction of these quantities by FAST that has been documented by other researchers[6] and is believed to be attributed to the aerodynamics model employed by the code. Overall, the aerodynamic predictions of the CFD models were judged to be in excellent agreement with accepted techniques, and the CFD models were extended to investigate the aeroelastic stability of the blade. The response of the blade resulting from an impulsive load was shown to decay in time, and no indications of flutter were evident. It should be noted that the simulations performed in this work represent a starting point for the development of the CFD model. Future work will focus on extending the technique further to simulate more realistic operating conditions including variable velocity wind profiles, gusts, and off-axis loading cases.

References

1. Griffith, D.T. and Ashwill, T.D, “The Sandia 100-meter All-glass Baseline Wind Turbine Blade: SNL100-00,” Sandia National Laboratories Technical Report, June 2011, SAND2011-3779.
2. NWTC Design Codes (FAST by Jason Jonkman). http://wind.nrel.gov/designcodes/simulators/fast/. Last modified 12-August-2005; accessed 12-August-2005.
3. NWTC Design Codes (WT_Perf by Marshall Buhl). http://wind.nrel.gov/designcodes/simulators/wtperf/. Last modified 17-February-2011; accessed 17-February-2011.
4. MSC.ADAMS Software, http://www.mscsoftware.com. Last accessed 10-August-2010.
5. Lobitz, D.W., “Aeroelastic Stability Predictions for a MW-Sized Blade”, Wind Engineering, 2004, 7:211-224.
6. Resor, B.,Wilson, D., Berg, D., Berg, J., Barlas, T., Wingerden, J., Kuik, G. , “Impact of Higher Fidelity Models on Simulation of Active Aerodynamic Load Control for Fatigue Damage Reduction”, 48th AIAA Aerospace Sciences Meeting and Exhibit, 4-7 January, 2010, Orlando, Florida, AIAA Paper 2010-253.

Sandia National Laboratories is a multi-program laboratory managed and operated by Sandia Corporation, a wholly owned subsidiary of Lockheed Martin Corporation, for the U.S. Department of Energy’s National Nuclear Security Administration under contract DE-AC04-94AL85000.