Fluid Structure Interaction simulation of wind turbine blade using 3DEXPERIENCE platform

Suryakant Nagdewe[1], Jean-Daniel Lecuyer[2], Rajesh Nagose[1], Niranjansing Pardeshi[1]

[1] 3DPLM Software Solutions Limited, Pune-411057, India

[2] Dassault Systèmes, Vélizy-Villacoublay, France

Abstract: Numerical prediction of wind turbine pose number of challenges such as the flow is fully turbulent, presence of fluid boundary layer and long and slender blade with complex distribution of composite material properties. Fluid Structure Interaction (FSI) simulations are essential for accurate prediction of structural forces. 3 dimensional single blade composite models is designed using CATIA Composite Design workbench. Simulation is carried out using fully coupled FSI procedure from SIMULIA Fluid Scenario Creation workbench to predict safety of composite blade for two load cases. This study demonstrates the effectiveness of 3DEXPERIENCE platform to carry out integrated flow and composite structural simulations for wind turbine blade application.

Keywords: Wind Turbine, Fluid Structure Interaction, CFD Coupling, Damage, Composite

1. Introduction

The airflow around a wind turbine blade induces a number of complex aerodynamic phenomena. As a result, accurate estimates of the blade service loads are not easily predictable. Numerical prediction of wind turbine pose number of challenges such as the flow is fully turbulent, presence of fluid boundary layer and long and slender blade with complex distribution of material properties. The deformation of wind-turbine blade depends on the wind speed and air flow, and air flow pattern depend on deformation of blade. Thus governing equation of air flow and blade deformations need to solve simultaneously. Hence, Fluid Structure Interaction (FSI) simulations are essential for accurate modeling. The analysis presented here is based on a robust, seamless, easy-to-use fluid-structure interaction (FSI) solution using 3DEXPERINCE R2014x platform. The co-simulation is executed simultaneously by the Abaqus Simulation Engine, exchanging data at certain time intervals between the flow simulation and structural simulation.

In this study, validation study is carried out for the aerodynamic forces on the NREL S809 airfoil which is widely used in wind turbine blades. FSI simulation are further built using SIMULIA Fluid Scenario Creation workbench for idling condition with extreme wind load and in-operation condition with normal wind to ensure safe design.

2. Geometry and Model

The complete blade design right from creation of the geometry to FSI simulation is performed using 3DEXPERIENCE Platform. The structural model of the blade shown in Figure 1 is created in Generative Shape Design workbench.

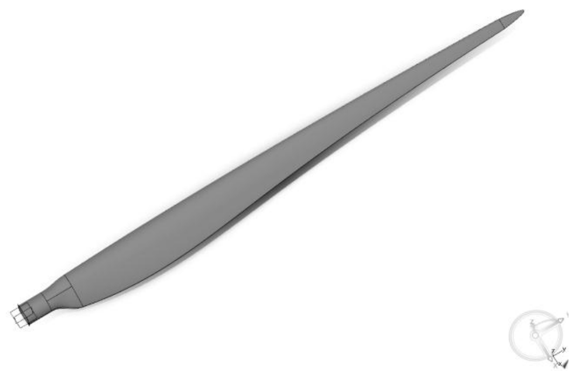

Figure 1. Wind Turbine Blade Geometry

2.1 Wind Turbine Composite Blade Design

The Composite properties are modeled using ‘by zones’ approach using the CATIA Composite Design workbench shown in Figure 2. The complete blade is divided into 70 different zones with 43 laminate definitions. Zone section is a closed contour having fixed fiber orientations however thickness and laminate material varies. The composite blade consists of Carbon fiber, Glass fiber and Balsa material laminates. Producibility is defined to take into account the actual fiber orientations when actual component is manufactured. This is taken into account by extracting the plies from zones.

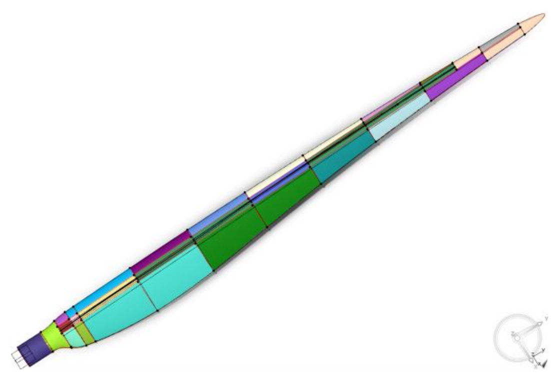

Figure 2. Wind Turbine Blade by zones definition

2.2 Integrating Composite Blade Design in Physics Modeling

The structural mesh is generated using SIMULIA Mesher workbench with quad dominated shell elements shown in Figure 3. The plies\zones created in CATIA Composite Design workbench can be selected directly in Composite shell section property. The composite parameters like fiber orientations, ply thickness and composite material definitions are automatically fetched from and are linked with composite design. Any change in the composite design parameters such as change in fiber orientations, material or ply thickness is associative and taken into account by updating the model definition.

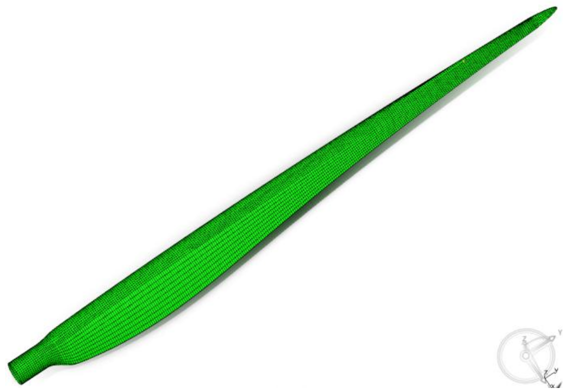

Figure 3. Wind Turbine Blade by zones definition

2.3 CFD Mesh

The CFD mesh is created with automatic Hex Dominant Mesher in SIMULIA Mesher workbench. The tool allows specifying boundary layer thickness and number of boundary layers. Additionally, user can specify the fine mesh in localized area. The CFD model consists of approximately 2.2 Million elements with 78% Hex elements and remaining Wedge, Pyramid and Tetrahedron elements. The Figure 4 shows CFD fluid domain mesh around the Structural mesh of wind turbine blade. Figure 5 shows the boundary layer mesh around the blade external surface.

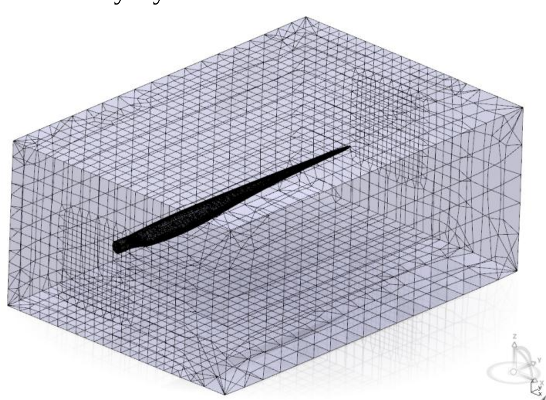

Figure 4. CFD Fluid Domain Mesh

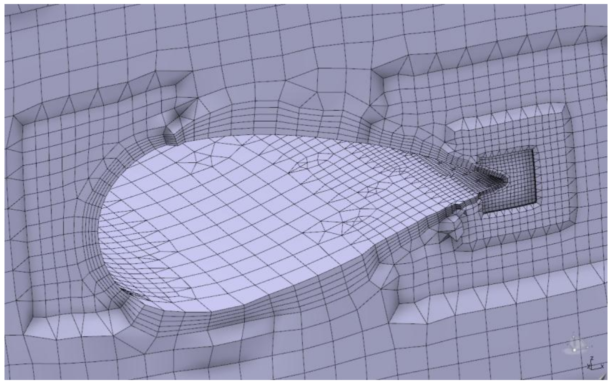

Figure 5. CFD Fluid Mesh showing the boundary layer mesh around blade profile

3. Analysis Approach

It is important to understand the dynamic characteristics of the structural model and then tune the FSI analysis attributes to ensure overall system stability. 3DXPERIENCE provides Structural Dynamics Scenario to perform modal analysis.

During wind turbine operations, the structural deformation of the blade necessitates the use of unsteady aerodynamic loads. In this study, unsteady blade aerodynamics loads are obtained from fluid solver. 3DEXPERIENCE FSI app provides full coupling between the fluid (CFD) and structure (CSM) solver.

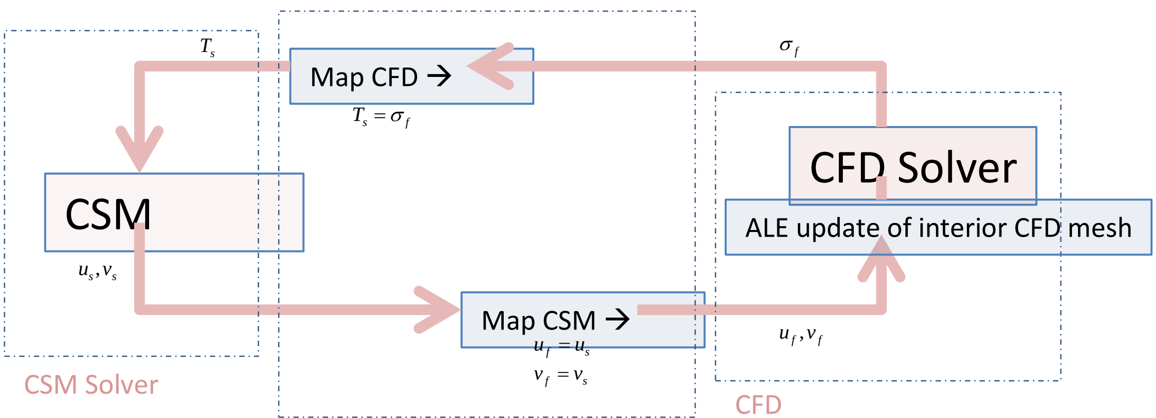

Figure 6. FSI coupling between structure and fluid

Coupling between the fluid and structure occurs across a wetted surface or interface region on which conjugate fields are transferred. In present study, the interface region is defined by the outer wall of the blade. Structure solver computes displacement in solid domain and exports displacement and velocity at the interface. The CFD solver simultaneously imports displacement and velocity from the interface and evaluates distribution of pressure and velocity on the entire fluid domain. It then transfers lumped mass values and traction vector back to the structure solver as shown in Figure 6. The run-time data exchange and synchronization between the CFD and structure solvers is done by the SIMULIA Co-Simulation Engine, which is in built in 3DEXPERIENCE platform.

The incompressible flow solver is used for CFD simulation. It uses integral form of Navier Stokes conservation equations. A node-centered finite-element discretization for the pressure and a cell-centered finite volume discretization of all the other transported variables are adopted. This hybrid approach guarantees accurate solutions and eliminates the possibility of spurious pressure modes while retaining the local conservation properties associated with traditional finite volume methods. An advanced second-order projection method is used for time-dependent problems. In present analysis, flow is considered as three dimensional, transient and turbulent. Turbulence equations are approximated with the widely used k- ε Realizable turbulent model. Hybrid wall function approach is used. Backward Euler time integration scheme is selected. By default, the Abaqus Simulation Engine uses an automatic time incrementation algorithm that continually adjusts the time increment size to satisfy the Courant-Friedrichs-Lewy (CFL) stability condition for advection [4].

An Implicit Dynamics steps is used with Moderate Dissipation for FSI structural simulation. An implicit dynamic step performs a stress/displacement analysis of transient dynamic or quasi-static problems using implicit time integration. The response is generally nonlinear. Moderate dissipation applications encompass a more general range of dynamic events in which a moderate amount of energy is dissipated by plasticity, viscous damping, or other effects. Geometric nonlinearity is accounted during the step [4]. FSI simulation can also be carried out with explicit dynamic step.

Current FSI app version has following limitations,

- Present turbulence model is not able to predict turbulent transition flows.
- Moving mesh is not supported.
- Periodic boundary condition is not supported.
- 2D flow simulation is not possible.
- Symmetric boundary condition is not available for flow analysis.

4. Numerical Simulation

4.1 FSI Simulation

Fully coupled FSI simulation is carried out with wind turbine blade at idling condition with extreme wind load and in operation condition with normal wind load. Detailed analysis of these load cases are explained in following sub-sections.

4.1.1 Idling Condition with Extreme Wind Load Case

During this load case the maximum reaction force gets generated at the base of the blade under extreme wind condition.

CFD LBC: No-slip boundary condition is applied on the blade surfaces. Velocity inflow and pressure outflow boundary condition is applied on the inlet and outlet surfaces respectively. No motion boundary condition is specified on inlet, outlet, top, bottom and side surfaces. Inflow velocity of 50 m/s with angle of attack 0 is considered as shown in Figure 7.

Structural LBC: The blades are clamped at the base using kinematic coupling to represent the rigid connection at the rotor. Tsai-Wu criterion is used for the composite blade failure and reaction forces, moments are requested at the base of the blade

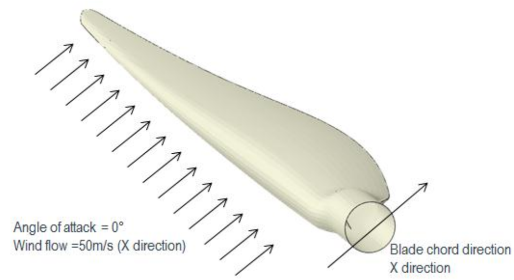

Figure 7. Idling Condition with Extreme Wind

4.1.2 In Operation Condition with Normal Wind Load Case

This load case is simulated to make sure that the blade design is safe under operating condition with normal wind speed.

CFD LBC: No-slip boundary condition is applied on the blade surfaces. Velocity inflow and pressure outflow boundary condition is applied on the inlet and outlet surfaces respectively. No motion boundary condition is specified on inlet, outlet, top, bottom and side surfaces. For In Operation Condition with Normal Wind Load Case, The wind flow velocity is 10m/s with an angle of attack 45^0.

Structural LBC: The blades are clamped at the base using kinematic coupling to represent the rigid connection at the rotor. In addition to this angular speed of 15 rpm is provided to the Blade. This is shown in Figure 8. Tsai-Wu criterion is used for the composite blade failure and reaction forces, moments are requested at the base of the blade

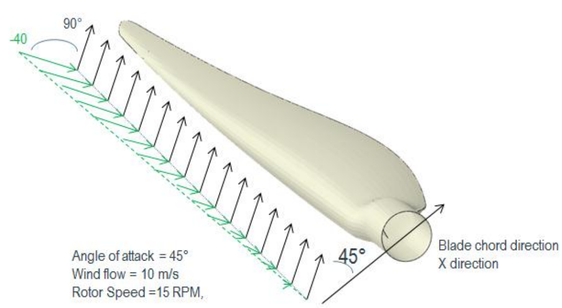

Figure 8. In Operation Condition with Normal Wind Load Case

5. Results and Discussion

The results are displayed using the SIMULIA Physics Results Explorer workbench. The dynamic characteristics of the blades are studied and a first natural bending frequency of 4.086 Hz was obtained for the structural model alone as shown in Figure 9. Additionally first pre-stressed frequency in bending model is found to be 1.8 Hz. Based on this result and a blade rotation period of 4 seconds (15 RPM), the flow and blade speed assumptions yield a stable model [1].

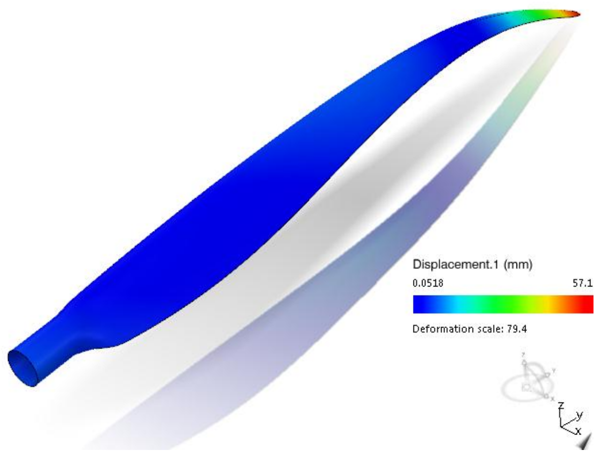

Figure 9. First natural bending mode for wind turbine blade

5.1 Results: Idling Condition with Extreme Wind

The max values of Tsai-Wu criterion is less than 1 as shown in Figure 10. Thus the plies considered in the present design are safe under the extreme wind condition load case. Maximum blade displacement at extreme wind velocity is 341 mm as shown in Figure 11. Maximum Principle stress acting on wind turbine blade is as shown in Figure 12.

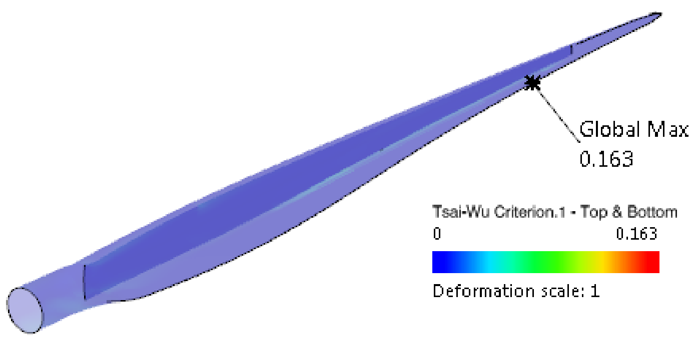

Figure 10. Tsai-Wu Max Envelope plot

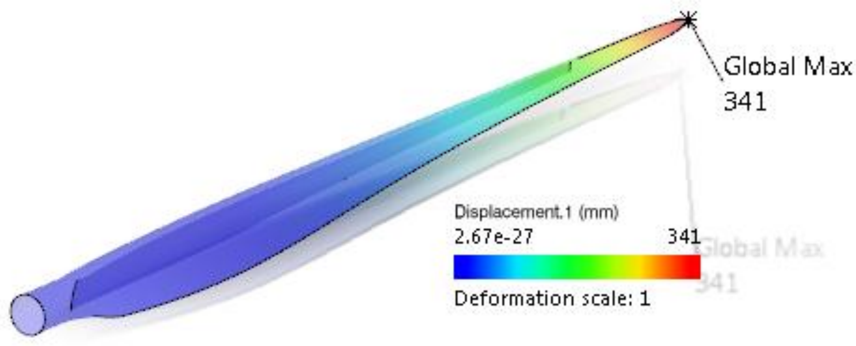

Figure 11. Maximum Displacement plot

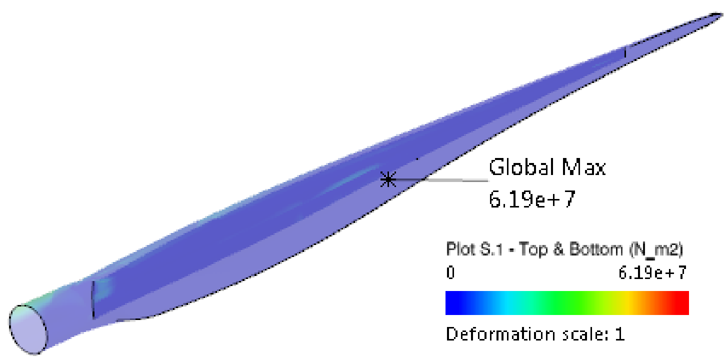

Figure12. Principal stress plot (Maximum)

It is important to resolve the boundary layer flow for accurate prediction by refining mesh near wall boundary. Y+ value is a non-dimensional distance from the wall to the first element center. In the present case, distribution of Y+ values over blade surface wall is shown in Figure 13. Hybrid wall function takes care of the mesh resolution from very fine to coarse mesh near wall surface. Pressure distribution over whole turbine blade surface is seen in Figure 14. In addition, we can observe the behavior of blade tip due to wind velocity fluctuations in history plot shown in Figure 15.

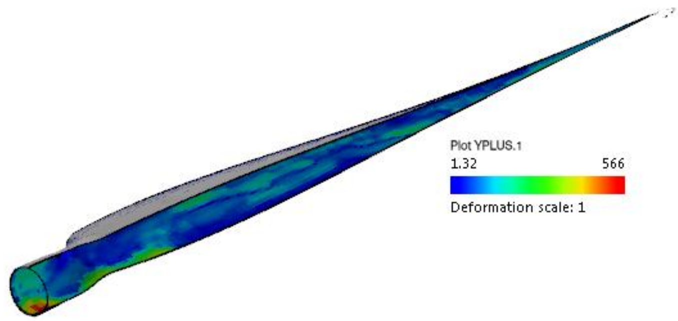

Figure13. Distribution of Y+ on wind turbine blade surface.

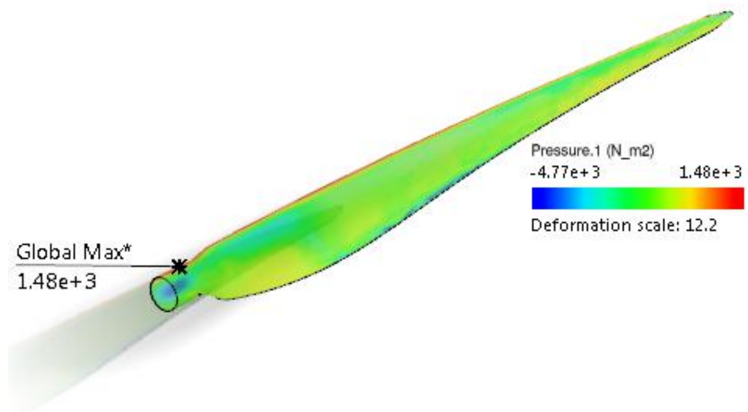

Figure14. Distribution of Pressure wind turbine blade surface.

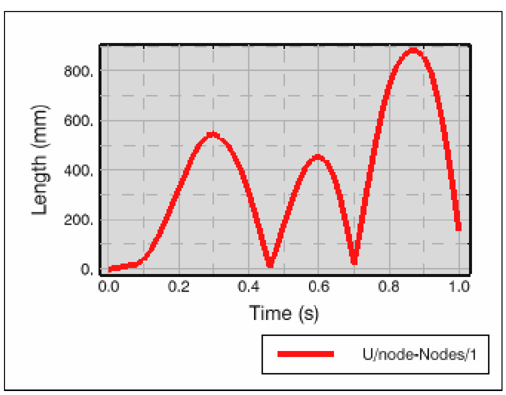

Figure 15. History plot for displacement at tip

5.2 Results: In Operation Condition with Normal Wind

In Operational condition, we will able to see streamline and blade displacement plot for complete rotation cycle as shown in Figure 16 and 17 respectively. Behavior of blade tip due to high wind velocity is seen in history plot as shown in Figure 18. Likewise, representative force distribution on leading edge is shown in Figure 19. Maximum and minimum principle stresses produced on wind blade surface shown in Figure 20 and Figure 21 respectively.

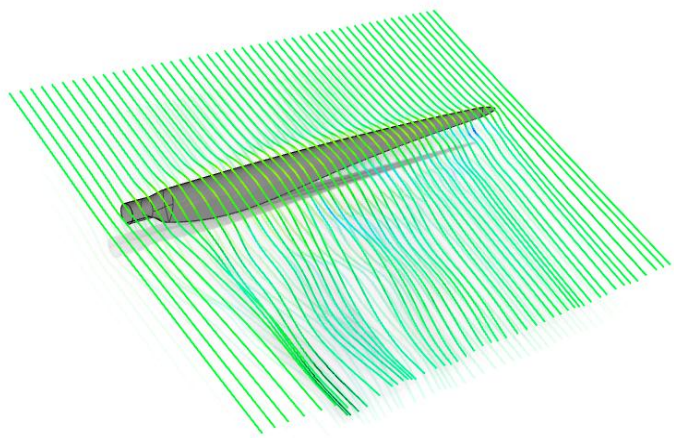

Figure 16.CFD Stream lines plot

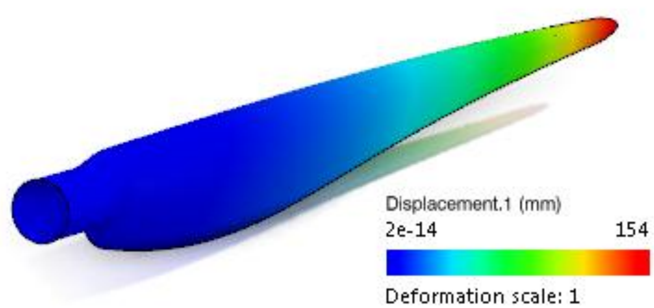

Figure 17. Displacement plot for the complete blade rotation cycle

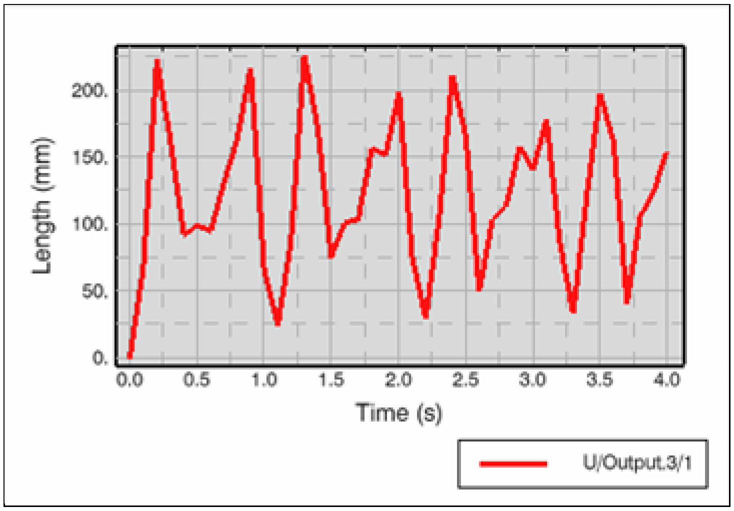

Figure 18. History plot for displacement at tip

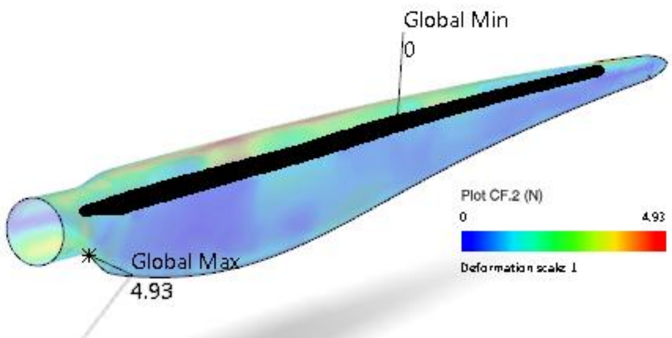

Figure 19. Representative force distribution on the blade leading edge.

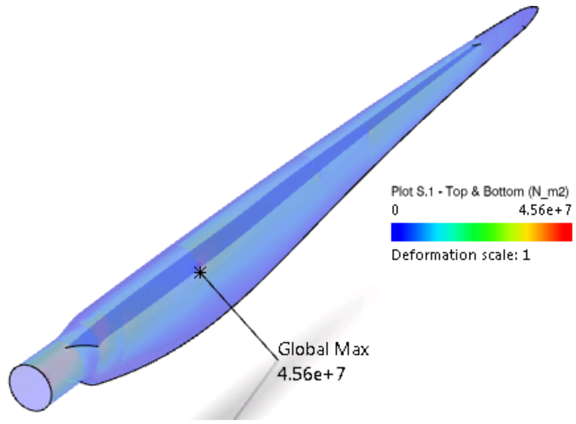

Figure 20. Maximum principle stress.

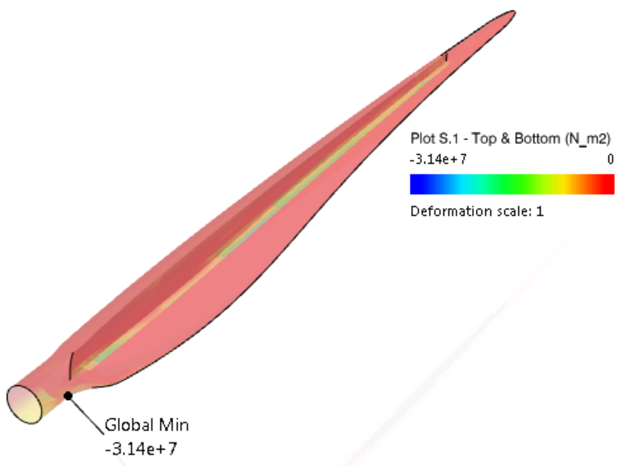

Figure 21. Minimum principle stress

6. Conclusion:

In present work, numerical simulations are performed on a composite wind turbine blade using 3DEXPERIENCE fluid structure interaction capability to predict blade tip displacement and stresses due to wind velocity in idling and operational conditions. As per Tsai-Wu criterion design seems to be safe under idling and operational conditions. This study clearly demonstrates the effectiveness of 3DEXPERIENCE platform to carry out integrated flow and composite structural simulations.

7. References

1. "Fully Coupled Fluid-Structure Interaction Analysis of Wind Turbine Rotor Blades", Abaqus Technology Brief, TB-12-WT-1, Revised: April 2012
2. "National Renewable Energy Laboratory (NREL) S809 airfoil", Wind Turbine Airfoil Catalogue, F. Bertagnolio, N. Sørensen, J. Johansen and P. Fuglsang, RISØ-R-1280(EN), August 2001.
3. SIMULIA Abaqus 6.10.3 User Manual
4. SIMULIA 3DEXPERIENCE R2014x Documentation