STRUCTURAL ANALYSIS OF COMPOSITE WIND TURBINE BLADE USING FINITE ELEMENT MODEL

[1]D.Muniraj, [2]K.Prabhakaran, [3]C.Karthi, [4]V.BalaMurugan [5]R.Pradeep

[1]Professor, Department of Aeronautical Engineering, Park College of Engineering and Technology, Tamil Nadu, India.
[2]Student, Department of Aeronautical Engineering, Park College of Engineering and Technology, Tamil Nadu, India.

Abstract-This paper is concerned with the structural optimization of horizontal axis composite wind turbine blade. Structural Analysis has been performed to achieve minimum weight and maximum strength in order to fulfil the requirement of efficient power production. The finite element modelling of theblade is performed through ANSYS software with various meshescreated on each structural part considering shell type for all surfacegeometries. The rigidity of the blade was evaluated in terms of three distinct components: Flapwise, Edgewise,and Torsional rigidity based on the optimized layup schedule .The Extreme Wind loading condition is imposed on the pressure side of the blade and is analyzed as a cantilever setup.The dynamic analysisperformed in order to obtain the natural frequencies andcorresponding mode shapes focusing the first five in and out-of-planebending and the torsional modes of the blade. The results obtained from static and dynamic analysis of the composite wind turbine blade indicates that the blade will not fail for the extreme wind loading condition.

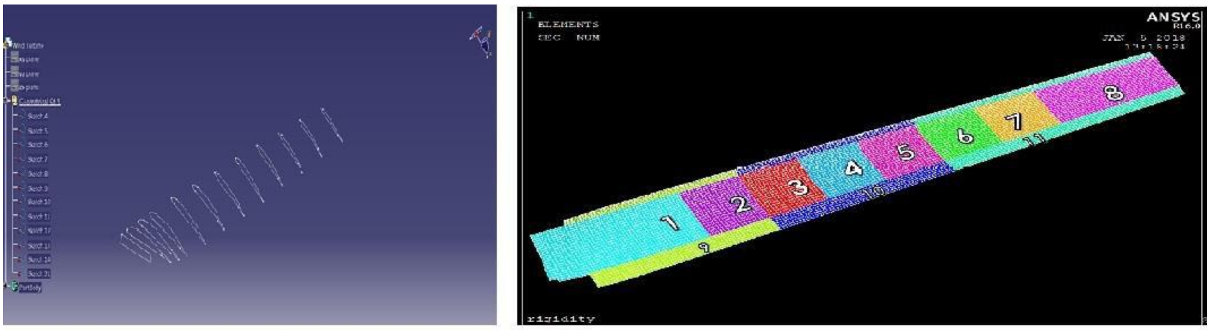
Keywords- composite, rigidity, static and dynamic analysis, mode shapes.

1 INTRODUCTION

A wind turbine is a device that converts the wind’s kinetic energy into electrical energy. Wind turbine operates on a simple principle, the energy in the wind turns the propeller blades around the rotor. The rotor is connected to the main shaft, which spins a generator to create electricity. This project deals with the design of structurally light weight and reliable 1kw horizontal axis wind turbine blade using composite (glass fibre) material. Static and Dynamic behaviour of the blade is examined to ensure the safe operating conditions by using finite element analysis. The flexural and torsional rigidity of blade was calculated to find the blade stiffness. Extreme Wind Loading condition is imposed on the blade and checked for the Tip deflection, maximum stress and strain.

2 BLADE GEOMETRY

The length of the blade from rotor axis to tip is 1m. The root of the blade starts at the hub connection, at a radius of 50 mm from the center of the hub. It has taper a ratio of 0.64, wherein the chord at the root and tip is 150mm and 96mm respectively. The maximum thickness at the root and tip is 21mm and 7.8mm respectively and the corresponding angle of twist is 32.06° and 5.40° respectively.



(a) Blade catia model

(b) Blade section ID

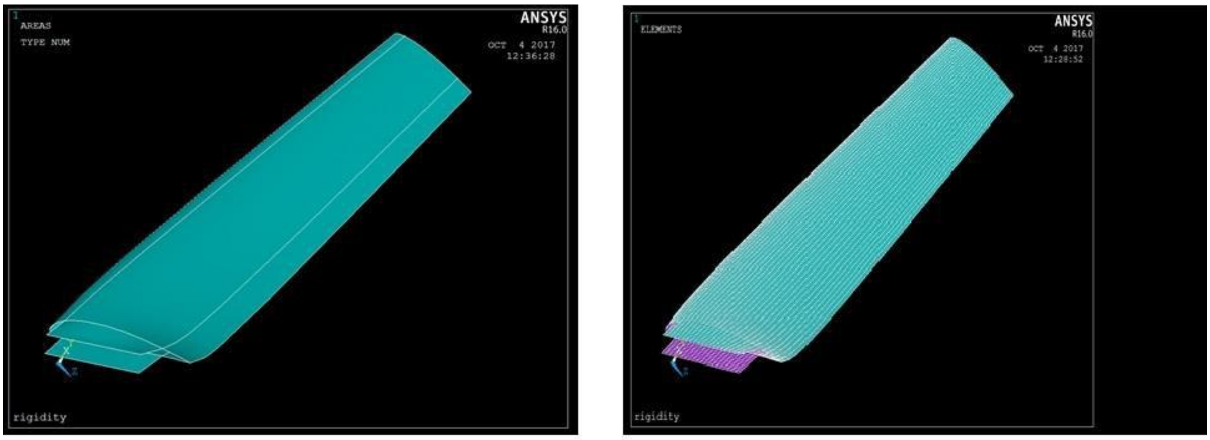
Figure 1: Blade Geometry and Layup

Using spline, the blade surface region was designed. Also the surface was divided into spar region and shell region, as eight sections and three sections respectively as shown in figure. 1(b). The partition of the blade was done along span-wise.

3 FINITE ELEMENT MODEL

For the finite element model (FEM), developed using ANSYS, the structure of the blade is modeled with shell elements (ANSYS element types SHELL181) capable of representing layer characteristics throughout the shell thickness. For the overall design and analysis of the blade, the ply layup in the root region was considered “stiff” relative to the outboard sections of the blade as shown in figure. 2

The orientation of the material axes (fiber directions) varies from one layer to the next. The D155 and DB 120 lamina use E-glass fibers that are embedded in polymer matrix and are chosen for fabrication. The D155 lamina was considered for the 0 degree ply layups and the DB120 lamina was used for the 45 degree ply layups.



(a) Blade surface model

(b) Blade mesh model

Figure 2: Finite Element Model

At the root end of the blade, the connection to the hub is assumed to be rigid, relative to the blade. As a consequence, all six degrees of freedom for the nodes in the root plane of the blade are fixed and tip of the blade is free which assumed to be cantilever beam.

4 STATIC BEHAVIOUR:

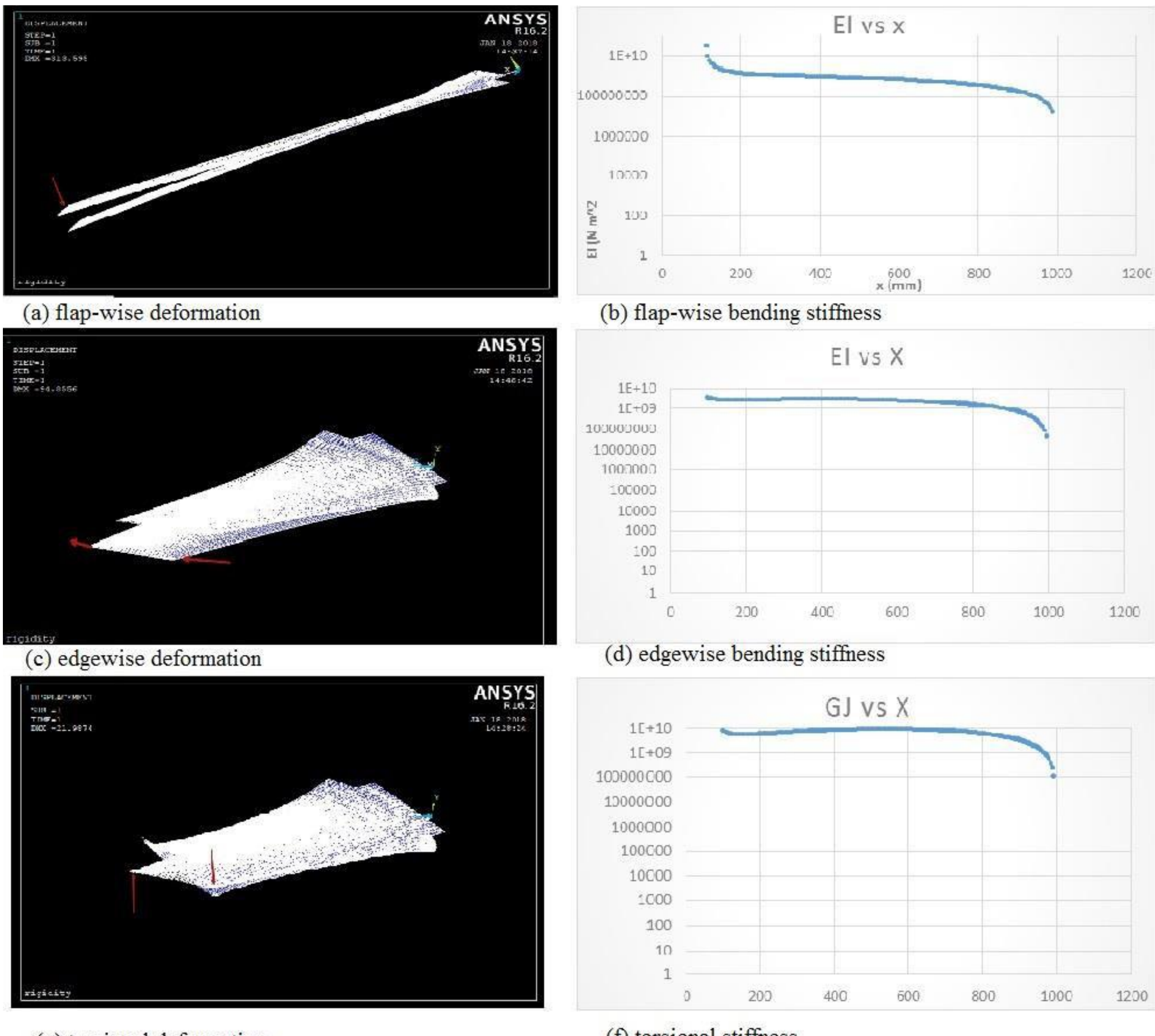
The Composite wind turbine blade is designed to be stiff in character, i.e., relative to the tower. The material of the blade is modeled with higher stiffness at the root and relatively lower stiffness at the tip. The rigidity of the blade was evaluated in terms of three distinct components: flap wise, edgewise, and torsional rigidity.

4.1 FLEXURAL RIGIDITY:

The flexural rigidity is simply termed as resistance offered by the structure while undergoing bending.

$$\frac{M}{d\Theta/dz} = EI$$

Where*E* represents the effective elastic modulus and I represents the effective moment of inertia.



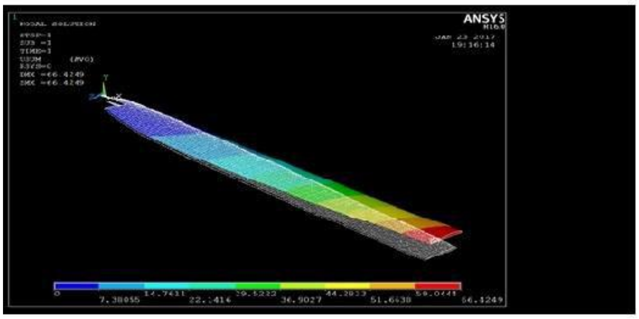
(a) flap-wise deformation
(b) flap-wise bending stiffness
(c) edgewise deformation
(d) edgewise bending stiffness
(e) torsional deformation
(f) torsional stiffness

Figure 3: Three distinct Rigidities of Blade

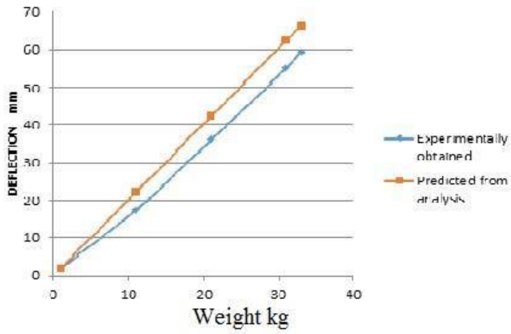
5 STATIC ANALYSIS

The maximum velocity in the wind history of 50 years isapplied as dynamic pressure on the pressure side of the blade. Pressure was applied onto the blade to simulate the wind that flows into the structure.

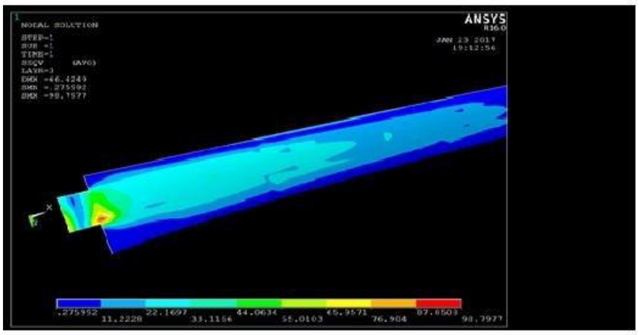
For the tip deflection test, the root of the blade is rigidly fixed with the vice. Operating load on the blade is 13.23N distributed over the lift generating portion of the blade. The test is done with 325.26N located at the 75% of the span from the root, to represent a worst case scenario.



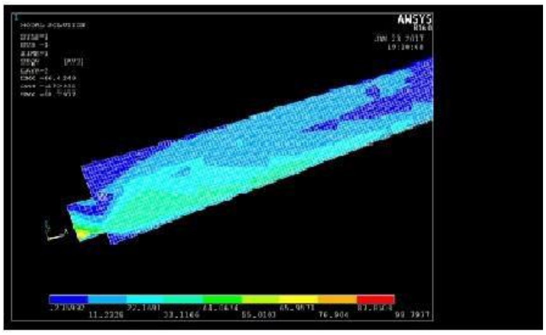
(a) deformation plot



(b) deflection curve



(c) stress plot on suction surface



(d) stress plot on pressure surface

Figure 4: Static Analysis

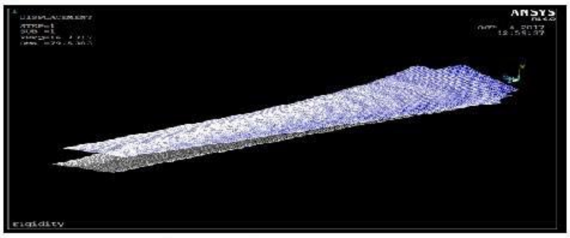
The deformation plot shows the maximum displacement of 66.42 mm at the tip of the blade. The stress plot clearly shows the variation of stress along the span of the blade, particularly in the ply drop regions. The maximum stress occurs at the root of the blade as was expected and is found to be below the yield stress. So the design is said to be safe from the static structural point of view.

6 DYNAMIC BEHAVIOUR

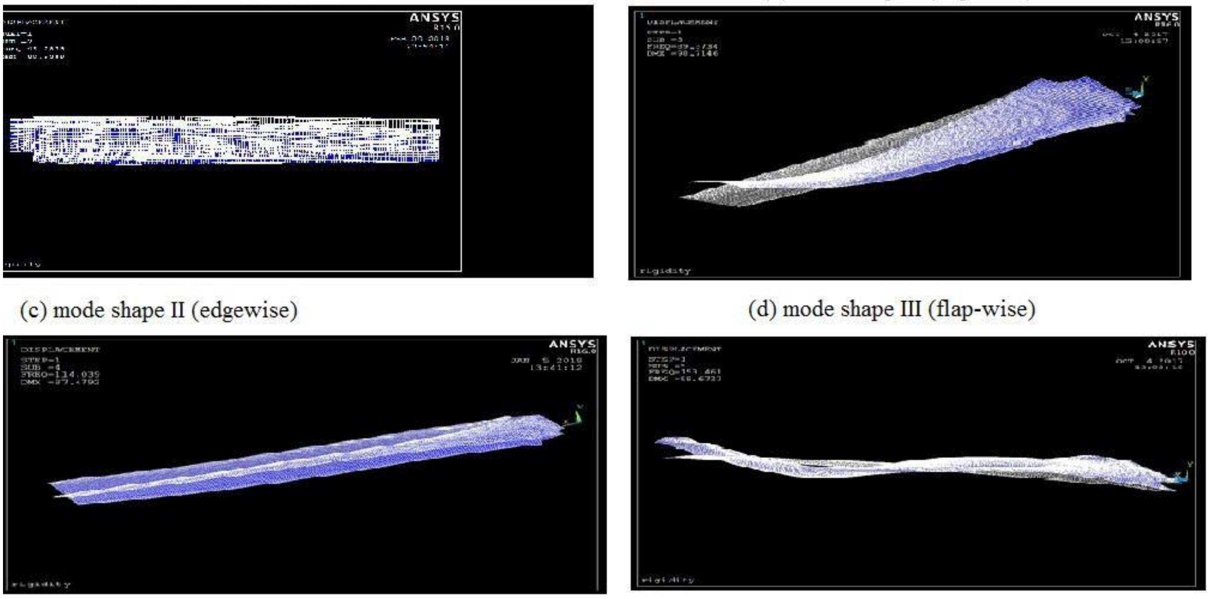
One feature of interest in the analysis of a dynamically loaded structure is the harmonic frequencies of free vibration for the structure, as excitations at or near these frequencies can generate large structural displacements and, as a consequence, large stresses and strains. These natural frequencies are dependent on the fundamental characteristics of the structure, such as geometry,density, and stiffness.

Set	Frequency	Load step	Substep	Cumulative
1	16.731	1	1	1
2	42.059	1	2	2
3	89.373	1	3	3
4	114.04	1	4	4
5	153.46	1	5	5

(a) blade natural frequencies



(b) mode shape I (flap-wise)



(c) mode shape II (edgewise)

(d) mode shape III (flap-wise)

(e) mode shape IV (torsional)

(f) mode shape V (mixed)

Figure 5: Mode Shapes and Natural Frequencies

To validate the results with simple hand calculations, because the blades were firmly attachedat the root and were otherwise free to deform under load, the blade vibrations could be modeled, in an approximate manner, using a cantilevered beam model. For a prismatic beam, the first natural frequency can be modeled by

$$f_1 = \frac{1.875^2}{2\pi l^2}\sqrt{\frac{EI}{m}}$$

Where, *l* is the length of the beam and *m* is the mass per unit length.

7 CONCLUSION

The deflection for the severe loading condition is in the acceptable range from the assembly point of view also, since the distance between the tower and the rotational plane is far above the displacement value.The maximum stress occurs at the root section of the blade, which is well below the ultimate stress of the glass fiber . The stress variation is smooth and it decreases along the length of the blade.The deflection test was repeated multiple times to check for fiber failure. If fiber failure had occurred, tip deflection would change after each failure. The tip deflection did not change after each successive application of the load. Tip deflection for each load case was identical for all the trails. The finite element analysis result is correlated with the deflection test and the dynamic behavior is compared with the simple prismatic beam hand calculation.

REFERENCES

1. McKittrick, Ladean R., et al. "Analysis of a composite blade design for the AOC 15/50 wind turbine using a finite element model." *SAND2001-1441. Sandia National Laboratories Contractor Report* (2001).

2. Amer, C., and M. Sahin. "Structural Analysis of a Composite Wind Turbine Blade." *World Academy of Science, Engineering and Technology, International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering* 8.7 (2014): 1264-1270.

3. Petrova, Radostina, and Hirpa G. Lemu. "Design study for dynamic behavior of wind turbine blade." *Proc. of Int. Workshop of Advanced Manufacturing and Automation (IWAMA 2012), Tapir Academic Press*. Vol. 131. No. 8. 2012.

4. Deepak J N, Chandan R, Doddanna K, "Design & Structural Analysis of Wind Turbine Blade for Operation at low Wind Speed." IRJET, November 2017.

5. James M. Gere, "Mechanics of Materials", CENGAGE learning custom publisher, Jan 2017.