ISSN NO. 2320-5407

Journal homepage: http://www.journalijar.com

INTERNATIONAL JOURNAL
OF ADVANCED RESEARCH

RESEARCH ARTICLE

OPTIMUM STACKING SEQUENCE IN COMPOSITE WIND TURBINE BLADE TO MINIMISE THE INTERLAMINAR STRESS

Thirunavukarasu M, Balaji Kasaram and C.Balaji Ayyanar
Mechanical Engineering, VeltechMultitech Engineering college,Avadi/Chennai,TN,INDIA.

Manuscript Info

Manuscript History:

Received: 12 November 2013
Final Accepted: 28 November 2013
Published Online: December 2013

Key words:
Wind turbine blade, Interlaminar shear stress, CLT, NACA, Testing.

Abstract

The nation is in need of renewable sources of energy to meet its energy requirement. Among the renewable energy sources, role of wind energy is an important one. Since the energy extracted from wind is a seasonal one, fault free operation during its operating regime is essential. Upon field survey it was found that more than 88% of wind turbine blade failure is due to matrix cracking which is mainly because of induced interlaminar stress. This failure can be eradicated by appropriately tailoring the stacking sequence of the laminate. Classical lamination theory (CLT) is employed to determine the induced interlaminar shear stress for the currently employed laminate of the wind turbine blade. The possible alternative stacking sequences for the currently employed laminate is found out as 12. Using CLT procedure, induced interlaminar stresses were found out for these alternative cases. The results are validated using Finite Element analysis using Shell 99 and Plane 46 elements of ANSYS package. Here solid46 element is an 8-noded, 3-D solid having 3-DOF per node is used to model the metallic part of the blade, whereas Shell 99 element is used to model the composite section of the blade. The real constants of the blade were decided based on rule of mixture equation. The section of the blade is identified using NACA coding. Three point bending test, four point bending test and modified short beam shear test were performed for the fabricated laminate to validate the results obtained in the above two procedures.

Introduction

During the past 20 years, large wind turbine blades have been fabricated from steel, aluminum, and composite materials such as wood, fiberglass, and carbon fibers. For a given blade strength and stiffness, the blade should be as light as possible to minimize inertia and gyroscopic loads, which contribute to blade fatigue. Blades made from steel and aluminum suffers from excessive weight and low fatigue life relative to modern composites. Because of these limitations, during the past 10 years almost all blades have been fabricated from composite materials, usually fiberglass.

1.1Analysis of Lamina;

It involves the following stages

- ✓ Determination of Lamina properties along the fiber direction
- ✓ Determination of Lamina properties along the loading direction
- ✓ Determination of [S]ij and [Q]ij matrices
- ✓ Determination of midplane strain and curvature
- ✓ Determination of induced strain in each lamina
- ✓ Determination of induces stresses in each lamina

The material properties of the laminate of the composite wind turbine blade are shown in table-1.

Table 1: Material Properties

Variable	Material Property
Young's modulus	E_{xx} = E_{zz} = 51.196 *GPa*; E_{yy} = 10.18 *GPa*
Poisson's ratio	$\gamma_{xy} = \gamma_{zx} = 0.278$; $\gamma_{yz} = 0.05$
Shear modulus	G_{xy} = G_{zx} = 2.433 *GPa;* G_{yz}=1.698*GPa*
Density	$\rho_m = 2\,X\,10^{-6}\ kg/mm^3$

2. Modeling of Laminate:

To create the laminates in the ANSYS, the element type is the important parameter. Shell 99 element is suitable for creating the layer by layer laminas; there are 37 layers which are defined. Each layers having different material property so we must give the material properties for all layers and define the layer thickness(0.16mm), ply orientation angle[0/±45]5,[0/90]2,[±45/0]6. The values of Exx, Eyy and Gxy are calculated from the equations of rule of mixture. After modeling the laminate shown as the stack has seen to be only twenty lamina hence the total composite model having 37 layers is viewed as 20 layers in one set and remaining 17 layers in another set.

Table 2: Material Specification

Material Name	Laminate specification
M1	Lamina 1 – [0]
M2	Lamina 1 – [45]
M3	Lamina 1 – [-45]
M4	Lamina 1 – [0]
M5	Lamina 1 – [45]
M6	Lamina 1 – [-45]
M7	Lamina 1 – [0]
M8	Lamina 1 – [45]
M9	Lamina 1 – [-45]
M10	Lamina 1 – [0]
M11	Lamina 1 –[45]
M12	Lamina 1 – [-45]
M13	Lamina 1 –[0]
M14	Lamina 1 – [45]
M15	Lamina 1 – [-45]
M16	Lamina 1 – [0]
M17	Lamina 1 –[90]
M18	Lamina 1 –[0]
M19	Lamina 1 – [90]
M20	Lamina 1 – [-45]
M21	Lamina 1 – [45]
M22	Lamina 1 – [0]
M23	Lamina 1 – [-45]
M24	Lamina 1 –[45]
M25	Lamina 1 –[0]
M26	Lamina 1 – [-45]
M27	Lamina 1 – [45]
M28	Lamina 1 –[0]
M29	Lamina 1 – [-45]
M30	Lamina 1 –[45]
M31	Lamina 1 – [0]

M32	Lamina 1 –[-45]
M33	Lamina 1 –[45]
M34	Lamina 1 – [0]
M35	Lamina 1 – [-45]
M36	Lamina 1 – [45]
M37	Lamina 1 – [0]

2.1. Defining Elements, Real Constants, and Materials:

There are several steps that must be taken in order to properly define the composition of a composite structure with in Ansys. First is the selection of the element type. First is the selection of the element type. ANSYS offers five element types such as Shell 91,shell 99,Shell 181,Solid46,Solid 191, which can be used to define layered composites. Of these shell elements are used for two dimensional analysis and solid elements are used for 3D analysis. Here solid46 element is used which has 8-node, 3-D solid, 3-DOF per node, up to 250 layers can be specified. Element is shown in figure 3. These elements are designed to model thin to moderately thick plate and shell structures, with a side-to-thickness ratio of 10 or greater. It is this fact that allowed for the generation of a moderately fine mesh, as opposed to a very fine mesh that takes longer to solve. Once the element type has been selected the material properties, layer orientation, and layer thickness must be defined within each element. In Ansys these properties are set using real constants. Real constants are user-defined element Characteristics, which represent the configuration of the element. In real constant as each element is divided into the no of layers specified the no of element in the thickness direction is specified to be one. In the real constant dialog window, the preliminary option is to define the number of layers in the model. Once the number of layers is specified, the composition of each individual layer is defined. Each layer can be represented by any one of the material models that have been defined. The layer orientation is defined as the direction of the layer coordinate system relative the global coordinate system, and the orientation is defined by entering the angle between the x-axes of each coordinate system. Finally the thickness of each individual layer can be defined to meet the specifications of the composite. There may be as many real constants as are necessary to accurately represent the structure being modeled. The Possible materials are accessed through the material models section of Ansys. In this dialog window, the orthotropic mechanical properties can be set for any number of materials.

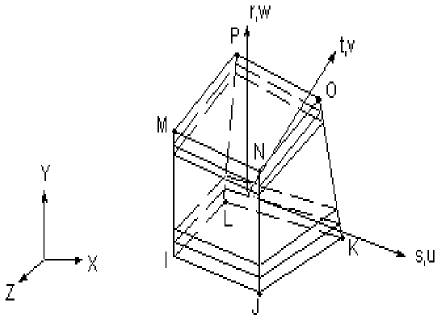

Figure 1: Solid 46 Elements

Material properties for a typical E-Glass-Polyester composite material for various fiber orientation angle as follows
For 0^0

E_{xx} = 51196 Mpa, E_{yy} = 10180 Mpa

$\gamma_{xy} = \gamma_{xz}$ = 0.278, γ_{yz} = 0.05

$G_{xy} = G_{xz}$ = 38430 MPa , G_{yz} = 25906 Mpa

For 45^0

E_{xx} = 34508 Mpa, E_{yy} = 26868 Mpa

γ_{xy} = γ_{xz} = 0.278, γ_{yz} = 0.05

G_{xy} = G_{xz} = 25906 MPa , G_{yz} = 10062 Mpa

For - 45^0

E_{xx} = 4560 Mpa, E_{yy} = 75740 MPa

γ_{xy} = γ_{xz} = 0.278, γ_{yz} = 0.05

G_{xy} = G_{xz} = 3420 MPa, G_{yz} = 126230 Mpa

For 90^0

E_{xx} = 17700MPa, E_{yy} = 53480 MPa

γ_{xy} = γ_{xz} = 0.278, γ_{yz} = 0.05

G_{xy} = G_{xz} = 13280 MPa , G_{yz} = 89130 Mpa

The following figure 2.shows formation of layers in ANSYS tool. In a single frame, it is possible to view 20 layers only. The remaining 17 lamina are shown in figure 3. It is important to note that these 37 layers are having different material properties, thus making the composite laminate as anisotropic.

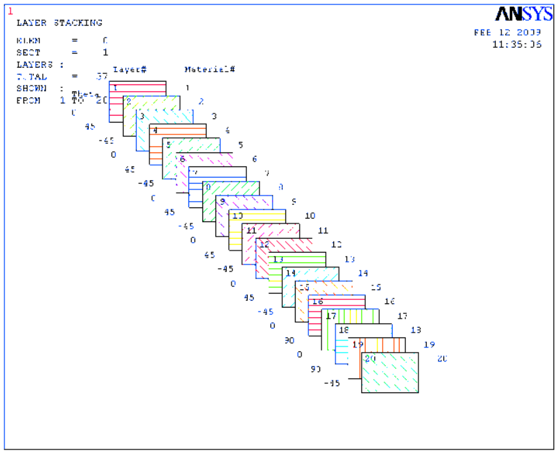

Figure 2: Stacking of first 20 lamina

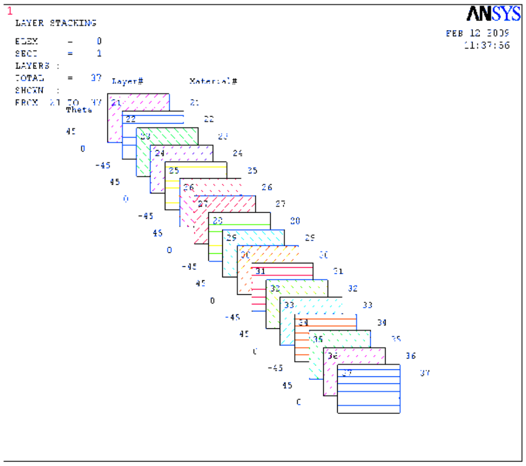

Figure3: Stacking of remaining 17 lamina

2.2 Modeling, Meshing, Processing and Post Processing:

After defining the material properties, the aerofoil section of the blade is modeled. This can be achieved by identifying the keypoints for the specified profile in NACA package. NACA 4412 profile is used for the current application. Considering the advantage in the symmetry in the section, half of the blade alone is modeled and is extruded for its complete length using the specified stacking sequence. Then the blade is fixed at its hub side and the load is applied in the form of pressure distribution. The following figures 4,5 and 6 shows the stress plot in the directions respectively.

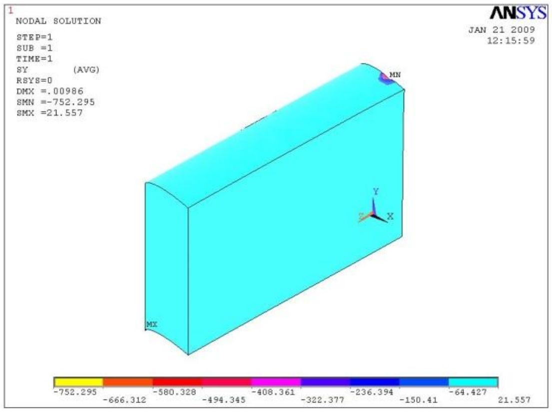

Figure 4: Induced stress in current laminate

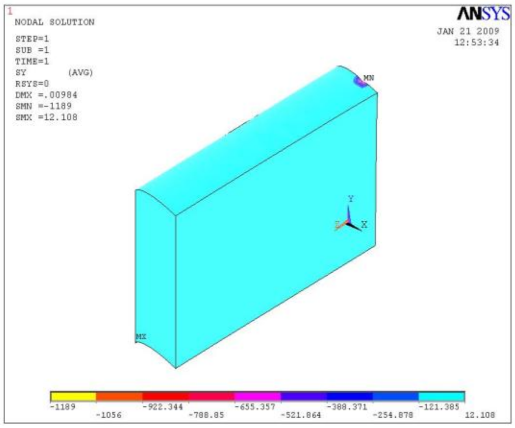

Figure 5: Minimum Induced stress

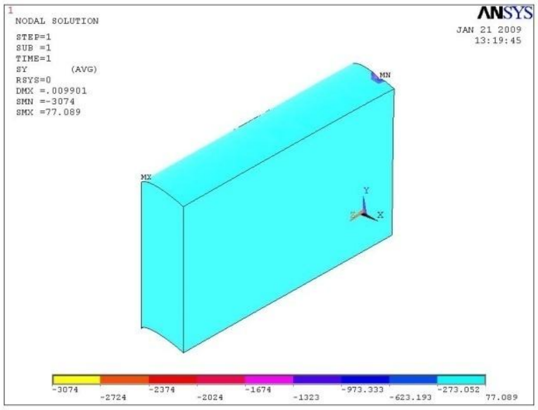

Figure 6: Maximum Induced stress

Table 3: Results of Stresses (MPa) for different Stacking sequences

S.No	Laminate Stacking sequences	Minimum stress	Maximum stress
1.	$(0/45/-45)_{11}(0/90)_2$	-752.295	21.557
2.	$(0/-45/45)_{11}(0/90)_2$	-759.777	21.124
3.	$(45/-45/0)_{11}(0/90)_2$	-1189	12.108
4.	$(-45/45/0)_{11}(0/90)_2$	-3059	76.756
5.	$(45/0/-45)_{11}(0/90)_2$	-1182	12.848
6.	$(-45/0/45)_{11}(0/90)_2$	-3065	77.177
7	$(0/45/-45)_{11}(90/0)_2$	-754.002	21.666
8	$(0/-45/45)_{11}(90/0)_2$	-761.512	21.238
9	$(45/-45/0)_{11}(90/0)_2$	-1192	12.217
10	$(-45/45/0)_{11}(90/0)_2$	-3068	76.671
11	$(45/0/-45)_{11}(90/0)_2$	-1184	12.956
12	$(-45/0/45)_{11}(90/0)_2$	-3074	77.089

3. Conclusion

Blade is the most important component in a wind turbine, which is made of composite material with aerodynamic profile, twisted tip and varying thickness along its length. The complex geometry of the composite blade is modeled utilizing ANSYS software. The most predominant failure occurred in the blade is due to interlaminar stresses. Hence efforts have been taken to estimate the magnitude of interlaminar stresses in all possible alternative stacking sequences. From the results obtained from the software, it can be concluded that the minimum interlaminar stress occurred in third case.

References

1.Pipes, R. B. and Pagano, N. J., Inter-laminar stresses in composite laminates under uniform axial extension, J. composites Mater., 4,538, 1970

2.Pagano, N.J. and pipes, R. B., The influence of stacking sequence on laminate strengths, Int. J. Mech. Sci., 15, 679, 1973.

3.Wang, S.S, Edge delamination in angle-ply composite laminate, AIAA J., 21, 256,1984.

4.Williams, J.G., O, Brien, T. K., and Chapman, A. J., III, Comparison of toughened composite laminates Using NASA standard Damage Tolerance Tests, NASA CP 2321, ACEE Composite structure Technology conference, Seattle, WA, August 1984.

5.Chen, S. F. and Jeng, B. Z., Fracture behavior of interleaved fiber-resin composites, composites Sci. Technol., 41,77, 1991.

6.Kaw, A. K. and Goree, J. G., Effect of interleaves on fracture of laminated composites. II, ASME J. Applied Mech., 57, 175, 1990.

7.Andrew Makeev, A Simple Elasticity Solution for Predicting Inter-laminar Stresses in Laminated Composites, to appear in Journal of the American Helicopter Society, April 1999

8.Ramesh.S, Government College of Engineering Tirunelveli June, 2006

9.Valeria La Saponara, Hanifah Muliana, Rami Haj-Ali,George A. Kardomateas, "Experimental And Numerical Analysis Of Delamination Growth In Double Cantilever Laminated Beams", 6 March 2000,pp 687–699

10.W.Wagner, F. Gruttmann, W. Sprenger "A Finite Element Formulation for the simulation of Propagating Delamination in layered Composite Structures", pp1337-1359

11.Ronald Krueger, " A Shell/3d Modeling Technique For Delamination In Composite Laminates" pp.843-852, 1999.