THE ANNALS OF "DUNAREA DE JOS" UNIVERSITY OF GALATI
FASCICLE XIV MECHANICHAL ENGINEERING, ISSN 1224-5615
2013

RESISTANCE STUDY OF THE SHELL OF A WIND TURBINE BLADE

PhD.Lect.Eng. Mariana Carmelia DRAGOMIR[1]
Eng. Master student Catalin Mihai POPESCU[2]
PhD. Lect. Eng. Viorel CARTAS[2]
[1]"Dunarea de Jos" University of Galati - Romania,Cross- border Faculty of Humanities, Economic and Engineering Sciences,
[2]"Dunarea de Jos" University of Galati - Romania, Faculty of Engineering of Braila,

ABSTRACT

The paper aims at presening a resistance study on the shell of a wind turbine blade and highlights the main forces that act on its surface. Femap program with finite element analysis was used to simulate a linear static analysis of the forces on the shell surfaces during operation of a wind turbine blade.

KEYWORDS: FEM, Femap, finite elements

1. GENERAL CHARACTERISTICS OF THE MAIN METHODS USED TO STUDY MATERIAL STRENGTH

In the recent years, with the development of software engineering design and 3D modeling, it was possible to develop and optimize products using virtual prototype. All subsequent steps will be used to simulate the behavior of the design and manufacture of the same virtual prototype, improved and optimized at each stage of the life cycle. The phase that occurs after conception and design and pre-manufacture is called simulation and behavior analysis mainly aims at subjecting the virtual prototypes to different requests. The behavior study is done using CAE (Computer Aided Engineering) software applications. The simulation software for analysis meets the most stringent international standards and the results obtained from the analysis, where applications have been correctly used, are digital certificates and validate the virtual prototype.

Most companies whose main activity is the design, development and manufacture of new products, regardless of industry, have specialized departments that validate the digital prototype. Depending on the complexity of manufactured products, the existence of such a department is required in a company.

Analysis and simulation have influence in the following areas:

- Stress analysis (static, dynamic) components or products using the finite element method (FEM / FEM / FEA);
- Thermal analysis and fluid flow dynamics, known as CFD (Computational Fluid Dynamics);
- Simulation of mechanisms, Cinematic;
- Pressure die casting simulation (injection into molds), pouring molten metal into molds or cold deformation (punching / stamping).

The main simulation and analysis capabilities common to most CAE software using the finite element method are described as follows:

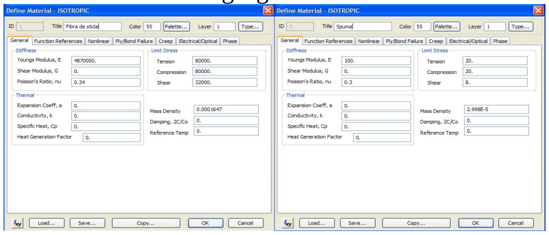
- Linear static analysis refers to the ability to determine stresses and strains for a problem in which requests are constant over time, assuming that the material is linearly elastic
- Modal dynamic analysis consists in calculating natural modes of vibration;
- Buckling analysis (Buckling) or general loss of structural model;
- Static nonlinear when deformations are very large and exceed the elastic range, the analysis takes into account the permanent deformations of the system.

- Dynamic analysis (dynamics) where requests vary over time, stress and strain can be obtained for any time the user considers it necessary;
- Thermal analysis calculates the temperature in a structure under various conditions of heat stress;
- Advanced implicit non-linear analysis is the analysis of large deformations using nonlinear materials and time-dependent loading;
- Advanced explicit non-linear analysis is used for high-speed impact;
- Analysis of fluid flow (CFD) in an advanced analysis mode that allows determining the distribution of speeds and accelerations in a fluid flowing along a structure on the boundary of the solid flow modeling;
- Analysis engines, predicting the dynamic response of the rotating systems (shafts, turbines);
- Analysis of fatigue (fatigue analysis), response prediction systems subject to variables such as time and frequency applications.

2. RESISTANCE STUDY

Femap program with finite element analysis was used to simulate a linear static analysis of the forces on the shell surfaces during operation of a wind turbine blade. For this we consider a blade of 50 meters length made of composite, fiberglass and foam. The model used was a simple one.

The values considered for the two materials are shown in the following figure.



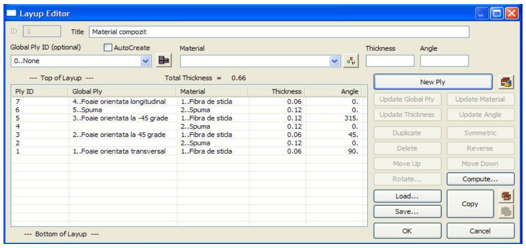
ID 1
Title Fibra de sticla
Color 55
Layer 1
Youngs Modulus, E 4870000.
Shear Modulus, G 0.
Poisson's Ratio, nu 0.34
Tension 80000.
Compression 80000.
Shear 32000.
Expansion Coeff, a 0.
Conductivity, k 0.
Specific Heat, Cp 0.
Heat Generation Factor 0.
Mass Density 0.0001647
Damping, 2C/Co 0.
Reference Temp 0.

ID 2
Title Spuma
Color 55
Layer 1
Youngs Modulus, E 100.
Shear Modulus, G 0.
Poisson's Ratio, nu 0.3
Tension 20.
Compression 20.
Shear 8.
Expansion Coeff, a 0.
Conductivity, k 0.
Specific Heat, Cp 0.
Heat Generation Factor 0.
Mass Density 2.998E-5
Damping, 2C/Co 0.
Reference Temp 0.

Fig. 1 Characteristics of the shell materials

The composite material was defined as consisting of seven different layers as follows:

- Glass fiber layer with longitudinally aligned fibers
- Glass fiber layer with transversally aligned fibers
- Glass fiber layer aligned at 45 ° with the length
- Glass fiber layer aligned at -45 ° with the length
- Between each layer of fiber there is a layer of foam.



ID 1
Title Material compozit
Global Ply ID (optional) 0..None
Total Thickness = 0.66

Ply ID	Global Ply	Material	Thickness	Angle
7	4..Foaie orientata longitudinal	1..Fibra de sticla	0.06	0.
6	5..Spuma	2..Spuma	0.12	0.
5	3..Foaie orientata la -45 grade	1..Fibra de sticla	0.12	315.
4		2..Spuma	0.12	0.
3	2..Foaie orientata la 45 grade	1..Fibra de sticla	0.06	45.
2		2..Spuma	0.12	0.
1	1..Foaie orientata transversal	1..Fibra de sticla	0.06	90.

Fig. 2 Composite material structure

The finite element method is a general method to approximate the partial differential equations describing physical phenomena. In principle, the finite element method is to decompose analyzed virtual prototype portions of simple geometric shapes, to analyze them and rebuild the virtual prototype respecting certain mathematical requirements. In solving partial differential equations, one challenge is to create an equation that approximates the equation to be studied, but is numerically stable, meaning that errors of input and intermediate calculations do not accumulate and the result of the analysis is meaningful. FEM replaces the mechanical system with a infinite number of degrees of freedom, by an equivalent system with a finite number of degrees of freedom, namely the system of differential equations of equilibrium (strong mathematical form: 6 Cauchy, 6 Hook, 3 internal equilibrium conditions) is replaced by an equivalent manner to the full effect (low mathematical form, the formulation of variation of Rayleigh), which leads to a system of algebraic equations. Essentially FEM represents the numerical implementation of the material deformations method, respectively unknowns, the nodal degrees of freedom are at a structural analysis the displacements and rotations.

An important issue in the analysis of systems subjected to voltage stress is to determine the yield strength of the experimental measurements and finite element analysis results. For a simple one-dimensional problem, we can easily compare the measured tension, the material yield strength, as measured from a uniaxial tension strength test (or as of reference material). The reference yield stress σ_y is generally noted.

It is much more difficult when you have a two-dimensional a three-dimensional problem. We have tensions in two or three directions (x, y, and z), and also the shear stresses are involved. What tension should be used for comparison with σ_y? The elasticity does not always appear because only one of the directional tensions exceeds the yield stress, even so it would be very conservative, elasticity will also occur in certain times before all the directional tensions exceed the yield point. Therefore an intelligent system is needed to forecast the appearance of elasticity based on measured directional tensions, such a device is known as criteria on elasticity. There is a number of criteria used in the technical analysis of overall elasticity, all based on the main stresses.

Rankine elasticity criterion is also known as the criterion of "normal tension peak", it assumes that the elasticity will occur when the normal tension reaches the maximum at any point of the uniaxial σy flow for the material. In dimensional analysis, this criterion implies that the elasticity in tension will occur when $\sigma_1 = \sigma_y$ and elasticity in compression will occur when $\sigma_3 = \sigma_y$. Stresses σ_1, σ_2, σ_3 are the principal stresses.

Rankine is quite a conservative criterion and, in general, is suitable for brittle materials (such as concrete, cast iron), and is not suitable for ductile materials, it does not take into consideration the shear stresses which greatly influence the flow of plastic ductile material.

Tresca elasticity criteria estimate that elasticity will occur when the shear stress reaches at any point the maximum permissible shear stress of the material. For this reason, the Tresca criterion is often referred to as the maximum shear stress criterion.

On a one dimensional plane maximum shear stress is given by:

$$\tau_{max} = \sigma_y / 2 \qquad (1)$$

On a three dimensional plane, maximum shear stress is given by:

$$\tau_{max} = (\sigma_1 - \sigma_2)/2 \qquad (2)$$

Taking the above equations together, in a three dimensional plane Tresca elasticity criterion says that elasticity will occur when $\sigma_1 - \sigma_3 = \sigma_y$.

The criterion of maximum normal deformation estimates that elasticity will occur when the largest of the three main deformations is equal to the limit of elasticity, when $\varepsilon_1 = \varepsilon_y$. The maximum normal deformation theory is not useful for the ductile material, but may be appropriate in the case of metals such as perspex type material (plexiglass).

The Von-Mises method is based on the criterion that the gap is produced when the strain energy is equal to the energy of the spring / rupture in uniaxial tension. It is less conservative than the other criteria of flexibility and, therefore, can help eliminate the addition of the ductile material where it is applied. When using Von-Mises criterion for stress analysis, the results, generated by using FEM analysis in general postprocessor finite element software, are used to generate the Von-Mises tension and if this tension exceeds σ_y, it can be assumed that the elasticity occurred at these points.

The blade surface stress analysis will use the Von-Mises criteria.

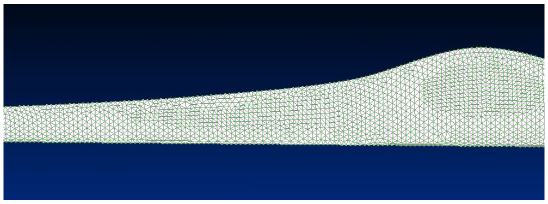


Fig. 3 Decomposition in finite elements

The surfaces analyzed were divided into finite elements of size 0.2.

In the first case of the study it was hypothetically considered that the lifting force was the only forces acting on surface. The blade was embedded in the hub mounting surface. It was considered the amount of force that acts perpendicularly to the surface bearing 400 N.

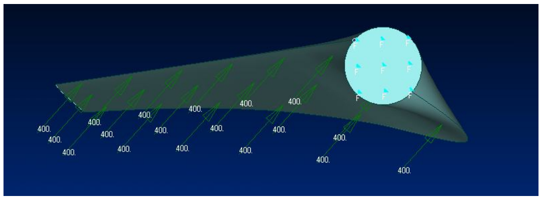


Fig. 4 Lifting force

3. RESULTS

After the analysis the following aspect was noted:

- the maximum displacement recorded was 4.83 m;

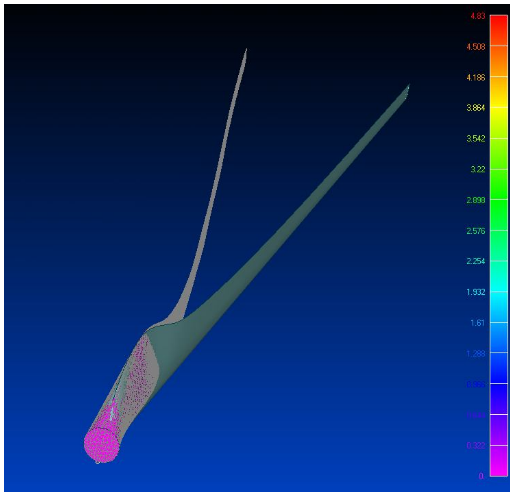


Fig. 5 Displacement recorded under the lifting force

- recorded maximum surface tension was 86 kPa.

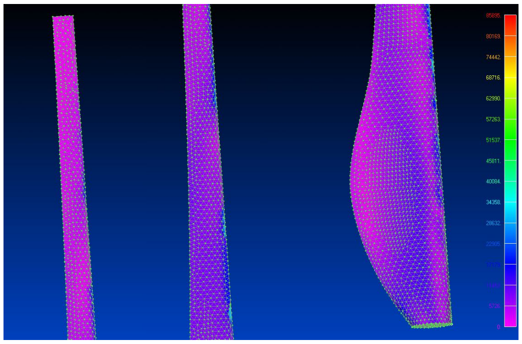


Fig. 6 Surface tensions for the first case

For the second case study it was considered hypothetical that on the blade is acting only a driving force of 30 N.

The results were:

- the maximum displacement of the shell was 0.21 m

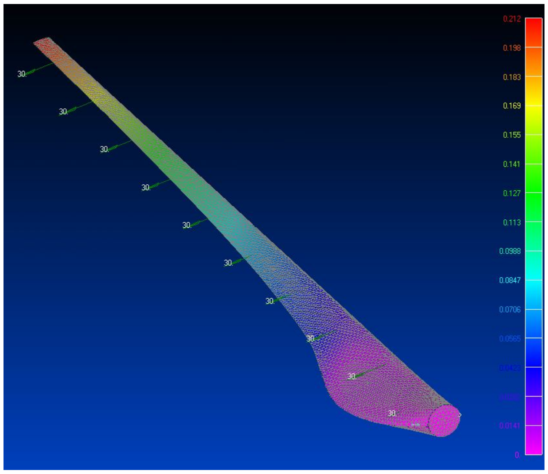


Fig. 7 Displacement recorded under the driving force

- the recorded maximum surface tension was 2.4 kPa

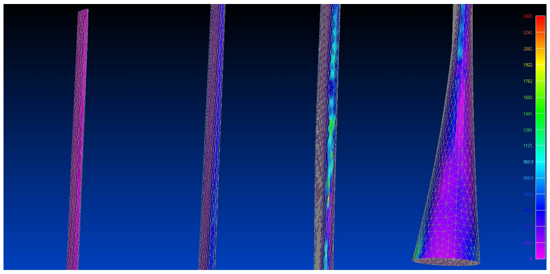


Fig. 8 Tensions on the edge in the second case

In case 3, analysis was done using the two previous forces, namely the driving force of 30 N and the lifting force of 400 N.

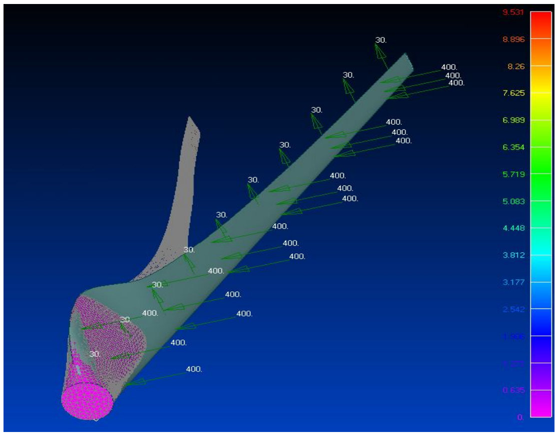


Fig. 9 Displacement under the action of the two forces

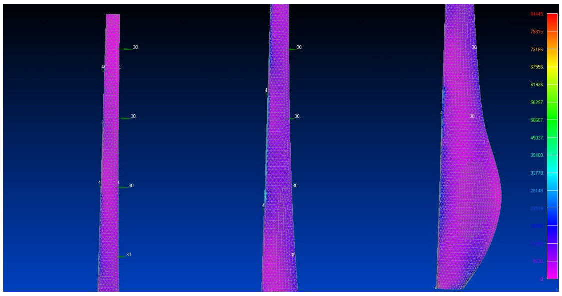


Fig. 10 Surface tensions in the third case

The results in the last case were:

- maximum displacement of 9.5 m
- maximum surface tension of 85 kPa.

4. CONCLUSIONS

The displacements on the blade may seem high, but this is a good result considering that the model used had no reinforcing structure and the blades during operation undergo an elastic deformation. Usually the shape of the blades is designed curved for all acting forces during operation to bend them to the optimal aerodynamic shape.

The deformations of up to 2 m are ranging with length and the materials used in construction.

Tensions arising on the surface of the blade are very small compared to the maximum supported by glass fiber, approximately 2205 psi or 15.2 MPa. The main role of the fiber glass is to withstand as many bending cycles as possible. The reliability of this component depends on the number of such supported cycles.

REFERENCES

[1] **Popescu C.** *“Dissertation paper” 2013.*

[2] **Rao S.S.** *“The Finite Element Method in Engineering”* Elsevier, 2005.

[3**] Mac Donald, B.** “*Practical Stress Analysis with Finite Elements*” Glasnevin Publishing, 2007.

[4] **Adams, V., Askenazi, A.** “*Building better products with Finite Elements Analysis*”, OnWord Press 1999.

[5] **Pilkey, W.D.** “*Formulas for Stress, Strain and Structural Matrices*” Wiley 1994.

[6] **Armenakas, A.** “*Advanced Mechanics of Materials and Applied Elasticity*” CRC Taylor & Francis, 2006.

[7] **Shanley, F.R.** “*Strenght of Materials*” McGraw Hill, 1957.