Stress concentration in the structural elements of wind generators

A V Maistrenko
Don State Technical University, 1, Gagarin sq., Rostov-on-Don, 344010, Russia

E-mail: anatol-maystrenko@yandex.ru

Abstract. A three-dimensional finite element calculation of the stress-strain state for a complex-shaped metal structure has been considered. Modeling the loading process accuracy is determined. The stress fields of wind generators blade elements at the stage of their manufacture were calculated, which made it possible to identify stress concentration areas and negative reasons leading to them in the technological process of manufacturing these structures.

Problem statement in general
During the next five years, global wind energy total power increase will be about 300 GW. The grounds for such expectations are mainly related to Asia, Africa and Latin America markets [1]. Developed wind energy industry will also be created in the Russian Federation till 2024 according to the Russian Federation Ministry of Energetic plans [2]. It is emphasized that the main task of "green" energy development in the country is creation of its own technological and production base. It is important that Russian wind farms will become an export product and will be in demand in the world as the best samples [3]. To this end, the work to improve design and technological processes of wind generators basic elements production is underway. First of all, this refers to the wind turbine main element - the blades where the main intellectual component of the installation is concentrated [4]. Producing a blade from a metallic material is a complex process. The resulting stress concentration in the elements of power plants is one of the main dangerous causes that leads to violation of their strength and reduces survivability. In this regard, the exact calculation of the stress-strain state and the identification of possible stress concentration areas in the blade body during its manufacturing are of great practical importance.

Recent achievements and publications analysis
Determining stress concentration zones has been a fundamental problem for the strength theory for many years. The areas of increased stresses are sources for the cracks occurrence and propagation, therefore, stress concentration is considered as one of the main factors of structural failure and accidents of wind turbines. Many works focus on the calculation in the design of products with cutouts. Thus, the three-dimensional distribution of stresses and strains in front of V-shaped cutouts in random thickness plates of elastic setting was studied in [5]. In [6-10] - stress concentration in beams with various forms of perforation (hexagonal, sinusoidal, etc.) was empirically and numerically investigated. The beams carrying capacity analysis and measurement of stresses in the cutouts area made it possible to approach the creation of an engineering method for calculating the stress state in beams with a complex perforation geometry.
Stresses and deformations calculation of a complex-shaped structure leads to the necessity for applying numerical methods and, first of all, the finite element method (FEM). In [11] - a calculated finite element scheme was proposed for analyzing and optimizing the composite properties. In [12] - the residual thermal stress fields were calculated using the FEM. Using the finite element model, the effective elastic characteristics of a multilayer structure with dissimilar materials layers were calculated [13]. The work [14] is devoted to the breakdown of the model into finite elements with different sizes and properties in the assumed stress concentration area. In order to reduce computing time for calculation of stresses and strains in a finite element model with a dense local grid it proposes to use submodeling, problem area fragmentation on a subregion with further imposition of its own adapted grids. Most of the studies based on the finite element analysis were performed using the universal software system ANSYS, which allows, among other things, to solve linear and nonlinear spatial strength problems. However, when determining stresses in ANSYS computational problems appear in the model even with a correctly constructed grid and an elastic homogeneous material. Stress concentration occurs in areas with sharp geometric non-linearity or physical inhomogeneity, which are difficult to cover with an adequate finite-element grid. It is possible to eliminate the erroneous stress concentration calculation by adjusting the model: grinding the grid, removing sharp corners, replacing concentrated forces with distributed ones, etc. [15].

Despite the large number of works devoted to numerical strength calculations, the topical issue of predicting stress concentration in the construction of blade installations still remains. Increasing requirements for the quality of wind turbine elements production leads to the need for further refinement of the finite element models based on a more complete account of various external and internal factors.

**Formulating goals and setting work objectives**

The goal of the work is to study the stress-strain state of a complex shape volume element for a wind generator design in the manufacturing process, taking into account changes in the geometric characteristics of its cross section.

While manufacturing a metal spar of a wind generator a small partial displacement of the profile of the outer surface may occur in its most loaded part for a number of technological reasons. This part of the spar has the greatest thickness in the product, and such deviations might not significantly affect the strength. However, in these places there are structural cutouts, and the displacement of the profile can significantly reduce the wall thickness of the spar. Here a stress concentration is formed, which, in its turn, can lead to the destruction of the spar under operational loads. Therefore, the search for places of stress concentration and the calculation of their possible value depending on the degree of profile change is important.

The setting work objectives is as follows. It is required to create a finite element model of a blade element, calculate its volumetric stress-strain state at the manufacturing stage under normal loads, determine the accuracy of modeling the structure loading processes, and establish the influence degree when changing the section profile of the blank part on the stress concentration at the places of change. This will clarify and justify the requirements for improving the technological process of making similar samples.

**Basic mathematical dependencies**

For linearly deformable materials, the relationship between stresses and strains is known as

$$\{\sigma\} = [D]\{\varepsilon^{el}\}. \quad (1)$$

Where: $$\{\sigma\} = \begin{bmatrix} \sigma_x & \sigma_y & \sigma_z & \sigma_{xy} & \sigma_{xz} & \sigma_{yz} \end{bmatrix}^T$$ - stress vector,

$$[D]$$ - the stress-strain state matrix,

$$\{\varepsilon^{el}\} = \{\varepsilon\} - \{\varepsilon^{th}\}$$ - elastic deformations vector,
\[
\{ \varepsilon \} = \begin{bmatrix} \varepsilon_x & \varepsilon_y & \varepsilon_z & \varepsilon_{xy} & \varepsilon_{yz} & \varepsilon_{xz} \end{bmatrix}^T - \text{total strain vector},
\]
\[
\{ \varepsilon^\text{th} \} - \text{temperature deformations vector}.
\]

From Eq. 1 you can get an expression for the deformations:
\[
\{ \varepsilon \} = \{ \varepsilon^\text{th} \} + \{ D \}^{-1} \{ \sigma \}.
\] (2)

Then the expressions for the deformations take the form:
\[
\varepsilon_x = \alpha_x \Delta T + \frac{\sigma_x}{E_x} - \frac{v_{xy} \sigma_y}{E_x} - \frac{v_{xz} \sigma_z}{E_x}
\]
\[
\varepsilon_y = \alpha_y \Delta T - \frac{\sigma_y}{E_y} + \frac{v_{xy} \sigma_x}{E_y} + \frac{v_{yz} \sigma_z}{E_y}
\]
\[
\varepsilon_z = \alpha_z \Delta T - \frac{\sigma_z}{E_z} - \frac{v_{xz} \sigma_x}{E_z} + \frac{v_{yz} \sigma_y}{E_z}
\]
\[
\varepsilon_{xy} = \frac{\sigma_{xy}}{G_{xy}}
\]
\[
\varepsilon_{yz} = \frac{\sigma_{yz}}{G_{yz}}
\]
\[
\varepsilon_{xz} = \frac{\sigma_{xz}}{G_{xz}}
\]

where: \( \varepsilon_x \) - deformations in the \( x \) direction, \( \sigma_x \) - stresses in the \( x \) direction, \( \varepsilon_{xy} \) - shear deformation in the \( xy \) plane, \( \sigma_{xy} \) - shear stress in the \( xy \) plane, \( \alpha_x \) - coefficient of thermal expansion in the \( x \) direction.

The equations for explicit finding the stresses are as follows:
\[
\sigma_x = \frac{E_x}{h} \left( 1 - (v_{yz})^2 \right) \varepsilon_x
\]
\[
\sigma_y = \frac{E_y}{h} \left( v_{xy} + v_{yz} \frac{E_z}{E_y} \right) \varepsilon_x
\]
\[
\sigma_z = \frac{E_z}{h} \left( v_{xz} + v_{yz} \frac{E_y}{E_z} \right) \varepsilon_x
\]
\[
\sigma_{xy} = \frac{E_y}{h} \left( v_{xy} + v_{yz} \frac{E_z}{E_y} \right) \varepsilon_x
\]
\[
\sigma_{xz} = \frac{E_z}{h} \left( v_{xz} + v_{yz} \frac{E_y}{E_z} \right) \varepsilon_x
\]
\[
\sigma_{yz} = \frac{E_y}{h} \left( \frac{E_z}{E_y} \right) \varepsilon_x
\]

where:
\[
h = 1 - (v_{xy})^2 \frac{E_y}{E_x} - (v_{yz})^2 \frac{E_z}{E_x} - (v_{xz})^2 \frac{E_x}{E_y} - 2v_{xy}v_{yz}v_{xz} \frac{E_z}{E_x}.
\]

**Finite element model**

The task was solved using the finite element method in the ANSYS multifunctional software package with the implementation of the calculation method in the parametric programming language APDL. A mathematical model of the wind power spar section has been created taking cutouts into account. The calculated volume area is divided into tetragonal finite elements. Calculated longitudinal and transverse loads and fastenings are applied to the sample: a movable hinge on one side and a seal on the other. Figure 1 shows a general view of a three-dimensional computational model, a loading scheme, volume finite elements, and calculation examples.

The APDL program determines the stress-strain state of the spar at each design node during loading. Particular attention is paid to areas with cutouts where stress concentration appears. The calculation algorithm consists of the following steps: setting the source data (geometry, strength properties of the
material, etc.); division of the area into finite elements; the power loads and fixtures application; determination of displacements, stresses and deformations at each point; printing results.

**Figure 1.** Design model: a) loading scheme; b) the division of the analyzed area into finite elements; c) stress field; d) stress field, rear view

To verify the correctness of the program was tested. The spar was physically loaded by predetermined efforts, surface stresses were measured by strain gauges at certain points. The same loading was simulated by the created program. The calculated and practically measured values of bending stresses were compared (Table 1). The comparison results analysis allows us to conclude that the APDL voltage field modeling program has an acceptable accuracy (within 9%).

**Table 1.** Comparison of the settlement and measured longitudinal stresses in control points

| Checkpoint numbers | Calculated values of longitudinal stress, [MPa] | Measured values of longitudinal stress, [MPa] | Absolute difference of stress values, [MPa] | Relative error of stress calculation by the developed program, [%] |
|--------------------|-------------------------------------------------|---------------------------------------------|-------------------------------------------|---------------------------------------------------------------|
| 1                  | 17                                              | 22                                          | 5                                         | 22.73                                                         |
| 2                  | 18.3                                            | 21.6                                        | 3.3                                       | 15.28                                                        |
| 3                  | 20.6                                            | 19.5                                        | 1.1                                       | 5.64                                                         |
| 4                  | 20.1                                            | 19.6                                        | 0.5                                       | 2.55                                                         |
| 5                  | 20.6                                            | 20.8                                        | 0.2                                       | 0.96                                                         |
| 6                  | 20.1                                            | 20.7                                        | 0.6                                       | 2.90                                                         |
|                   | Average value                                   | 1.78                                        | 8.34                                      |                                                               |

**Main results**

Simulation of testing product samples at the stage of the technological process was carried out by loading them with tensile and bending forces in the plane of rotation. Conditions for the appearance of stress concentration were created by changing the cross section of the structure. Therefore, the calculation was carried out for two sections: the original and the modified (weakened). The characteristic view of the resulting picture of the stress-strain state is shown in Figures 2 and 3. So, figure 2 shows the distribution of spherical stresses on the surface for the two cases studied. It can be seen that the cross section weakening leads to a sharp stress concentration.

Integrated indicators, for example, stress intensity, give the most objective picture of the stress-strain state. Figure 3 shows that dangerous phenomenon is observed for a weakened sample in the cutouts area – that is a significant stress concentration. Compared to the background stress (63 MPa) in certain limited areas, the stress increases to 125 MPa, i.e. increase by 2 times.
Figure 2. Spherical stresses in Pa on the spar surface in the cutouts area: a) initial section geometry (red contour line); b) modified section geometry

Figure 3. The stress intensity in Pa on the spar surface in the cutouts area: a) the original section geometry; b) modified section geometry

With a decrease in wall thickness, not only areas with increased stresses (stress concentration) appear in the cutout areas, but also average stresses increase. So, with a decrease in the wall thickness of the spar by a few mm, the average stresses increase by 20-25% (Figure 4).

Figure 4. Stress comparison (1 - modified section geometry; 2 - initial section geometry) in the cut-out area for: a) maximum stress intensity; b) medium stress

The developed sample model makes it possible to determine the stress-strain state in its entire volume, as well as to predict the behavior of a weakened structure for various loading modes.

Summary
The strength analysis of wind power plants structural elements at the stage of the production process reveals the characteristic features of their stress-strain state and quantifies the influence degree of changes in cross section geometry on the size and location of areas with stress concentration, which are potential causes of operational damage.

References
[1] Information on https://www.windpower.by/news/1001.html
[2] Information on https://minenergo.gov.ru/node/10307
[3] Information on https://minenergo.gov.ru/node/11554
[4] Maistrenko A V 2018 Building Structures Thermal Calculation (Materials Science Forum) 931 164-169.
[5] Berto F, Lazzarin P, Wang C H 2004 *Not only in the shape of plates* (Int. J. Fracture).

[6] Pritykin A 2016 *Stress concentration in castellated I-beams under transverse bending* (Mechanika) 19 466–473.

[7] Jamadar F M, Kumbhar P D 2015 *Parametric study of circular beams with circular and diamond shaping openings* (International Research Journal of Engineering and Technology) 2 (2) 715–722.

[8] Wang P, Wang X, Ma N 2014 *Vertical shear, engineering structures* (Engineering) 75 315–326.

[9] Wang P, Ma Q., Wang X., Investigation on Vierendeel mechanism failure of castellated steel beams with fillet corner web openings, Engineering Structures. 74 (2014) 44–51.

[10] Durif S, Bouchair A, Vassart O 2014 *Experimental and numerical follow-up of a network of engineers* (Engineering Structures) 59 587–598.

[11] Boguszewski T, Ciupinski L, Kurzydlowski K 2012 *Designing for optimizing for thermal management* (APNFM 2008).

[12] Maligno A R 2008 *Finite element investigations on fiber-reinforced composites* (Express Polymer Letters) 2 (9) 665–676.

[13] X Yu J 2011 *Microphase and Multilayer Composites* (Materials Science and Technology) 7 (27) 1213–1221.

[14] Fricke W 2010 *Guideline for Fatigue Assessment by Stress Analysis for Welded Structures* [Text] (International Institute of Welding. - IIW-Doc. XIII-2240-2-08 / XV-1289r2-08. - Hamburg University of Technology Shipbuilding Design and Analysis) pp. 144.

[15] Information on https://cae-club.ru/forum/metody-borby-s-oshibochnoy-koncentraciey-napryazheniy -v-ke (In Russian)