Calculation of the bending of electromechanical aircraft element made of the carbon fiber

Galina Danilova-Volkovskaya[1], Anton Chepurnenko[2], Aleksandr Begak[1] and Andrey Savchenko[2]

[1]North-Caucasian Federal University, Ave 40 Let Oktyabrya, 56, Pyatigorsk, 357500, Russia
[2] Don State Technical University, Sotcialisticheskaya st., 162, Rostov-on-Don, 344022, Russia

E-mail: anton_chepurnenk@mail.ru

Abstract. We consider a method of calculation of an orthotropic plate with variable thickness. The solution is performed numerically by the finite element method. The calculation is made for the springs of a hang glider made of carbon fiber. The comparison of the results with Sofistik software complex is given.

1. Introduction
At present, synthetic materials and reinforced plastics are widely used in various branches of engineering. Since most reinforced plastics are characterized by a pronounced anisotropy of mechanical properties, the possibility of using methods developed for isotropic materials to calculate products from them is excluded. The physical and mechanical properties of anisotropic materials are described in extensive scientific and engineering literature, including [1-8].

Many works have been devoted to the methods of calculation of products from anisotropic materials, including [9-15]. However, some questions remain poorly studied, in particular the problem of calculating anisotropic plates of variable thickness. The finite element method opens up wide possibilities for solving this problem.

The investigated structural element of the aircraft and its design scheme are shown in figure 1. At the cut-off zones, concentrated forces act on the element, the peak value of which is 25 kN. In the calculation, we assume that these forces are uniformly distributed along the contour of the hole. Since this element is rigidly attached to the frame, only variable-width sections that work as cantilever beams are involved. The thickness of these areas is also variable: at the base - 10 mm, and at the end of the console - 8 mm. The load acting on the element is not normal to the plane of the console, but forms an angle of 10.5 degrees with the normal, so it will be decomposed into a normal and tangential component. The normal component causes bending, and under the action of the tangential component, the element is in the plane problem of the theory of elasticity (plane stress state). The action of the normal and tangential components can be considered separately, and then add up the stresses and displacements using the superposition principle.

2. Materials and Methods
We consider the effect on the element of the normal component of the load. The plate will be modeled by triangular finite elements. When calculating for bending, the considered finite element (figure 2) has

3 degrees of freedom: deflection w_i and 2 angles of rotation φ_{ix} and φ_{iy}. The displacement field of the finite element is represented as:

$$\{U\} = \begin{Bmatrix} \{\rho_1\} \\ \{\rho_2\} \\ \{\rho_3\} \end{Bmatrix}, \tag{1}$$

where $\{\rho_i\} = \{w_i \quad \varphi_i^x \quad \varphi_i^y\}^T = \left\{ w_i \quad -\dfrac{\partial w}{\partial y}\Big|_i \quad -\dfrac{\partial w}{\partial x}\Big|_i \right\}^T$.
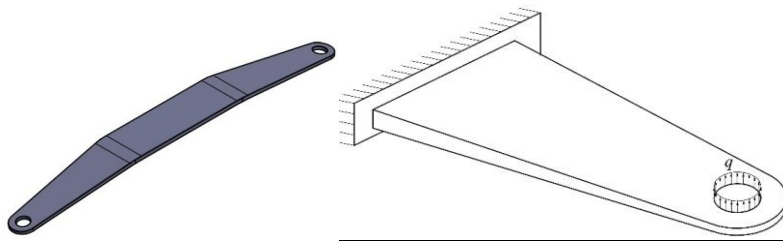

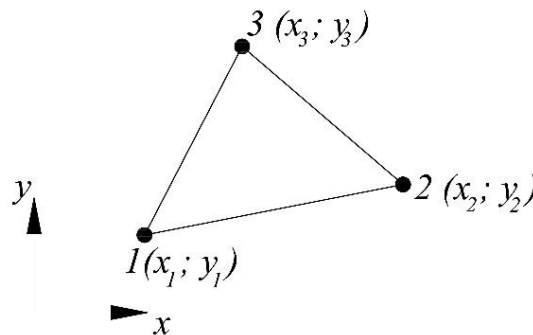**Figure 1.** The investigated structural element and its design scheme

Figure 2. Triangular finite element

For the deflection function we take the following approximation:

$$w = \beta_1 L_1 + \beta_2 L_2 + \beta_3 L_3 + \beta_4 (L_2^2 L_1 + \frac{1}{2} L_1 L_2 L_3) + ... + \beta_9 (L_1^2 L_3 + \frac{1}{2} L_1 L_2 L_3), \tag{2}$$

where $\beta_{1...9}$ – undetermined coefficients, L_1, L_2, L_3 – natural coordinates.

$$L_i = \frac{1}{2A}(a_i + b_i x + c_i y), i = 1...3, \tag{3}$$

where $A = \dfrac{1}{2} \begin{vmatrix} 1 & x_1 & y_1 \\ 1 & x_2 & y_2 \\ 1 & x_3 & y_3 \end{vmatrix}$ – the area of the triangular finite element, $a_1 = x_2 y_3 - x_3 y_2, b_1 = y_2 - y_3,$

$c_1 = x_3 - x_2$.

Coefficients a_i, b_i, c_i at $i = 2...3$ are determined by cyclic replacement of indices $1 \rightarrow 2 \rightarrow 3 \rightarrow 1$. Coefficients $\beta_{1...9}$ are found by substituting in (2) the nodal values of the deflections and angles of rotation. To calculate the rotation angles, it is required to differentiate function (2) in x and y. The calculation of the partial derivatives with respect to x and y is performed as follows:

$$\frac{\partial}{\partial x} = \frac{\partial L_1}{\partial x}\frac{\partial}{\partial L_1} + \frac{\partial L_2}{\partial x}\frac{\partial}{\partial L_2} + \frac{\partial L_3}{\partial x}\frac{\partial}{\partial L_3} = \frac{1}{2A}\left(b_1\frac{\partial}{\partial L_1} + b_2\frac{\partial}{\partial L_2} + b_3\frac{\partial}{\partial L_3} \right);$$

$$\frac{\partial}{\partial y} = \frac{\partial L_1}{\partial y}\frac{\partial}{\partial L_1} + \frac{\partial L_2}{\partial y}\frac{\partial}{\partial L_2} + \frac{\partial L_3}{\partial y}\frac{\partial}{\partial L_3} = \frac{1}{2A}\left(c_1\frac{\partial}{\partial L_1} + c_2\frac{\partial}{\partial L_2} + c_3\frac{\partial}{\partial L_3} \right). \tag{4}$$

Finally, the expression for deflection through nodal displacements takes the form:

$$w = \{\{N_1\} \quad \{N_2\} \quad \{N_3\}\}\{U\}, \tag{5}$$

where $\{N_1\}$, $\{N_2\}$, $\{N_3\}$ – form functions.

$$\{N_1\}^T = \left\{ \begin{array}{c} L_1 + L_1^2 L_2 + L_1^2 L_3 - L_1 L_2^2 - L_1 L_3^2 \\ b_3(L_1^2 L_2 + \frac{1}{2}L_1 L_2 L_3) - b_2(L_3 L_1^2 + \frac{1}{2}L_1 L_2 L_3) \\ c_3(L_1^2 L_2 + \frac{1}{2}L_1 L_2 L_3) - c_2(L_3 L_1^2 + \frac{1}{2}L_1 L_2 L_3) \end{array} \right\}. \tag{6}$$

To obtain expressions for $\{N_2\}$ and $\{N_3\}$, it suffices to perform a cyclic replacement of the indices in (6).

To obtain the resolving equations, we apply the variational principle of Lagrange. The potential strain energy is written as:

$$W = \frac{1}{2}\int_V \{\sigma\}^T \{\varepsilon\} dV, \tag{7}$$

where $\{\sigma\}^T = \{\sigma_x \quad \sigma_y \quad \tau_{xy}\}$ – stress vector, $\{\varepsilon\} = \{\varepsilon_x \quad \varepsilon_y \quad \gamma_{xy}\}^T$ – strain vector.

For orthotropic material, the relationship between stresses and strains has the form:

$$\varepsilon_x = \frac{\sigma_x}{E_1} - \nu_2\frac{\sigma_y}{E_2};$$

$$\varepsilon_y = \frac{\sigma_y}{E_2} - \nu_1\frac{\sigma_x}{E_1}; \tag{8}$$

$$\gamma_{xy} = \frac{\tau_{xy}}{G}.$$

Elastic constants of unidirectional carbon plastic are given in [2]: $E_1 = 259.3\,GPa$; $E_2 = 4.49\,GPa$; $G = 3.1\,GPa$; $\nu_1 = 0.404$; $\nu_2 = 0.007$.

We express from (8) the stresses through deformations:

$$\sigma_x = \frac{E_1}{1 - \nu_1\nu_2}(\varepsilon_x + \nu_2\varepsilon_y);$$

$$\sigma_y = \frac{E_2}{1 - \nu_1\nu_2}(\nu_1\varepsilon_x + \varepsilon_y); \tag{9}$$

$$\tau_{xy} = G\gamma_{xy}.$$

Or in the matrix form:

$$\{\sigma\} = [D]\{\varepsilon\}, \tag{10}$$

where $[D] = \dfrac{1}{1-v_1v_2} \begin{bmatrix} E_1 & E_1v_2 & 0 \\ E_2v_1 & E_2 & 0 \\ 0 & 0 & G(1-v_1v_2) \end{bmatrix}$.

The relationship between deformations and displacements has the form:

$$\{\varepsilon\} = = -z \begin{Bmatrix} \dfrac{\partial^2 w}{\partial x^2} \\ \dfrac{\partial^2 w}{\partial y^2} \\ 2\dfrac{\partial^2 w}{\partial x \partial y} \end{Bmatrix} = -z \begin{bmatrix} \dfrac{\partial^2 \{N\}}{\partial x^2} \\ \dfrac{\partial^2 \{N\}}{\partial y^2} \\ 2\dfrac{\partial^2 \{N\}}{\partial x \partial y} \end{bmatrix} \{U\} = -z[B]\{U\}. \tag{11}$$

The elements of the matrix $[B]$ depend on x and y. This matrix can be obtained using the symbolic math toolbox of the Matlab package.

Taking into account (11), the stress and strain vectors can be written in the form:

$$\{\varepsilon\} = -z[B]\{U\}; \quad \{\sigma\} = -z[D][B]\{U\}. \tag{12}$$

Substituting (12) into (7), we obtain:

$$W = \frac{1}{2}\int_V z^2 \{U\}^T [B]^T [D][B]\{U\}dV = \frac{1}{2}\{U\}^T \frac{h^3}{12}\int_A [B]^T [D][B]dA\{U\}. \tag{13}$$

The work of external nodal forces represents the sum of their products to the corresponding nodal displacements:

$$A = \{U\}^T \{F\}. \tag{13}$$

Minimizing the Lagrange functional $\Lambda = W - A$ with respect to the nodal displacements $\{U\}$, we obtain a system of linear algebraic equations:

$$[K]\{U\} = \{F\}, \tag{14}$$

where $\{F\}$ – vector of external node loads, $[K] = \dfrac{h^3}{12}\int_A [B]^T [D][B]dA$ – stiffness matrix.

Since the elements of the matrix $[B]$ depend on x and y, numerical integration is used to calculate the stiffness matrix. The variable thickness of the plate is taken into account quite simply, since it can be set different for each finite element. In the calculation, we assume that the thickness varies linearly with x:

$$h(x) = h_1 + \frac{h_2 - h_1}{l}x, \tag{15}$$

where h_1 – plate thickness in the support zone, h_2 – thickness at free end, l – distance from free end to support area.

When calculating the thickness of each finite element, as x we consider the coordinate of its center of gravity, measured from the support zone.

3. Results
The calculation of a plate with variable thickness was carried out for the vertical component of the load at $h_1 = 10\,mm$ and $h_2 = 8\,mm$. We also performed calculation of plate with constant thickness $h_1 = h_2 = 8\,mm$. For the second case, the calculation was also made in the program complex Sofistik.

Figure 3 shows the displacement w curves as a function of x and y, obtained by us in the Matlab package. The net surface corresponds to the result for a plate of constant thickness, painted over surface - for a plate of variable thickness. The finite element model in the Sofistik software package and the isospin of vertical displacements for a plate of constant thickness are shown in figure 4 and figure 5.
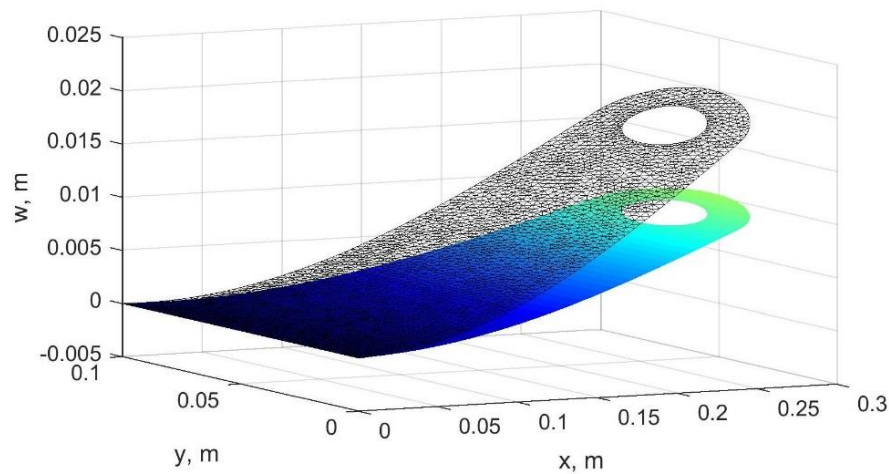
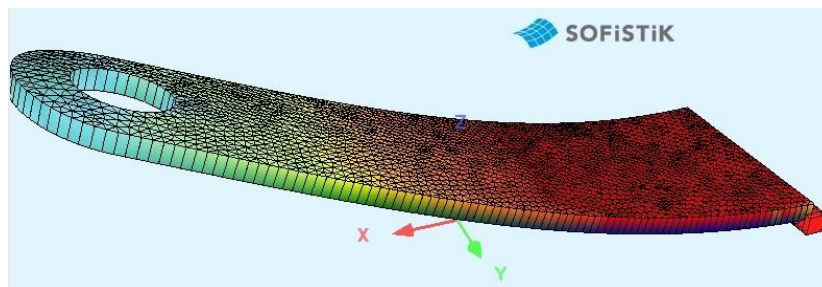**Figure 3.** Change of displacements as a function of x and y for a plate of constant and variable thickness

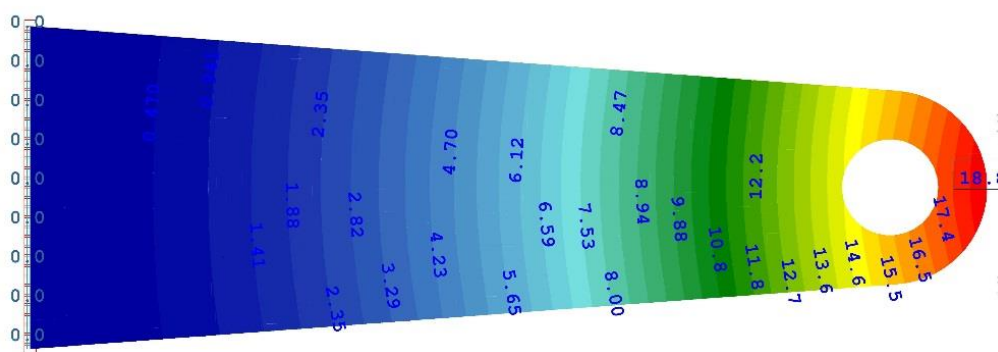**Figure 4.** Finite element model in the program complex Sofistik

Figure 5. The displacement isole in mm, obtained in Sofistik

4. Discussion

When calculating in the Matlab package, the maximum deflection for a plate of constant thickness was 20.1 mm, for a plate of variable thickness - 10.5 mm. In the program complex Sofistik in the case of constant thickness $w_{\max} = 18.8$ mm. The discrepancy between the results obtained it two software complexes is 6.5%.

This discrepancy can be explained by the fact that the material has a strongly pronounced orthotropy: the module of elasticity in two mutually perpendicular directions differ in 58 times.

5. Conclusions

The developed methodology and software package in Matlab environment allow calculating orthotropic plates of variable thickness of arbitrary shape. Comparison of the results with the Sofistik software was performed, and a satisfactory match of the results was obtained.

References

[1]   Oliveira B and Creus G 2004 *Composite Structures* **65** pp 443-457
[2]   Maranci B and Plesovskih A 2014 Factory laboratory *Diagnostics of materials* **79** pp 12-16
[3]   Mallick P 1997 *Composites engineering handbook* (New York: CRC Press) 1251 pp
[4]   Gorokhov A and Trufanov N 2011 Herald of PNIPU *Mechanics* **1** pp 170–182
[5]   Turusov R and Memarianfard H 2016 *Mechanics of Composite Materials* **52** pp 1-10
[6]   Sergeev A, Baurova N and Turusov R 2016 *Mechanics of Composite Materials* **52** pp 89-98
[7]   Tsybin N, Turusov R and Andreev V 2016 *Procedia Engineering* **153** pp 51-58
[8]   Chepurnenko A, Andreev V, Beskopylny A and Jazyev B 2016 *MATEC* **67** 06059
[9]   Andreev V, Yazyev B and Chepurnenko A 2014 *Advanced Materials Research* **900** pp 707-710
[10]   Litvinov S, Klimenko E, Kulinich I and Yazyeva S 2015 *International Polymer Science and Technology* **42** pp 23-25
[11]   Litvinov S, Trush L and Yazyev S 2016 *Procedia Engineering* **150** pp 1686-1693
[12]   Andreev V 2013 *Applied Mechanics and Materials* **353-354** pp 1267-1270
[13]   Turusov R, Kuperman A and Andreev V 2015 *Advanced Materials Research* **1102** p 155
[14]   Andreev V and Turusov R 2014 *Applied Mechanics and Materials* **670-671** pp 153-157
[15]   Andreev V, Turusov R and Kuperman A 2015 *Advanced Materials Research* **1102** pp 155-159