Finite-element modeling of loading of spring from an orthotropic material

Andrey Veremeenko[1], **Anton Chepurnenko**[1], **Pavel Shvetsov**[1], **Lidiia Zorchenko, and Serdar Yazyev**[1]

[1]Don State Technical University, Sotcialisticheskaya St., 162, Rostov-on-Don, 344022, Russia

E-mail: anton_chepurnenk@mail.ru

Abstract. In the paper, the stress-strain state of the springs from carbon fiber reinforced plastic was analyzed at its loading by a vertical load. The orthotropy of the material was taken into account. The solution was implemented in the software complexes Lira, ANSYS, and also using the program specially developed by the authors in the Matlab package. In the first case, the part was modeled by three-dimensional finite elements in the form of tetrahedral. In Lira and Matlab we used the plane finite elements of the plates. Comparison of the results obtained in three software complexes is given.

1. Introduction

Composite materials offer great prospects for improving the strength and performance properties of structures in the industry. They are characterized by high strength and rigidity at low weight, and in their characteristics significantly exceed traditional metals and alloys [1, 2].

The use of composite plates and shells as load-bearing elements in the structures of a responsible design makes it necessary to take into account additional factors, in particular, a pronounced anisotropy of the deformative properties of polyarmored materials. This led to the creation of nonclassical versions of the theories of plates and shells [3-6]. Calculation of anisotropic plates and shells is associated with the solution of complex systems of partial differential equations [7-10]. The finite element method opens up a wide range of possibilities in the study of anisotropic materials [11-19]. In this paper, we consider some aspects of finite element modeling of anisotropic plates using as the example the spring of flying device made of carbon fiber reinforced plastic.

2. Materials and Methods

To assess the stress-strain state of the spring (figure 1), we use finite element modeling of its loading (FEM) in the software complexes ANSYS and Lira. In addition, the program developed by the authors in the package Matlab is used. In the software complex ANSYS the problem is solved in a three-dimensional setting. The graphical model is built in the Compass 3D environment and exported to ANSYS in the ACIS format (figure 2a). Since the problem is symmetric with respect to the coordinate planes of the Cartesian coordinate system, ½ of the console part was considered to simplify the model and to shorten the counting time. In the first approximation, the sheet was considered a constant thickness, which was 8 mm. The distance from the origin to the center of the hole was 300 mm. Ten Nodular Quadratic Elements SOLID92 (II order) in the form of a tetrahedron were used to model the spring body (figure 2b). The number of elements ranged from 2 000 to 80 000, the number of nodes ranged from 4 000 to 120 000 in order to achieve sufficient accuracy of the solution.

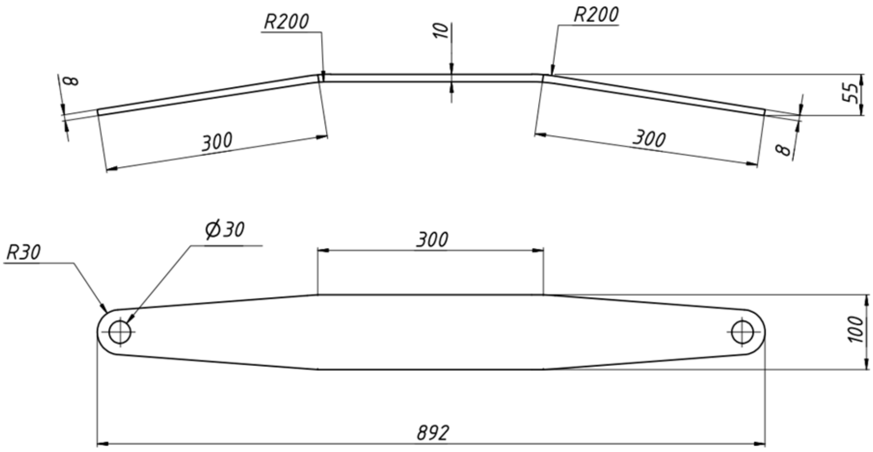

Figure 1. Spring drawing

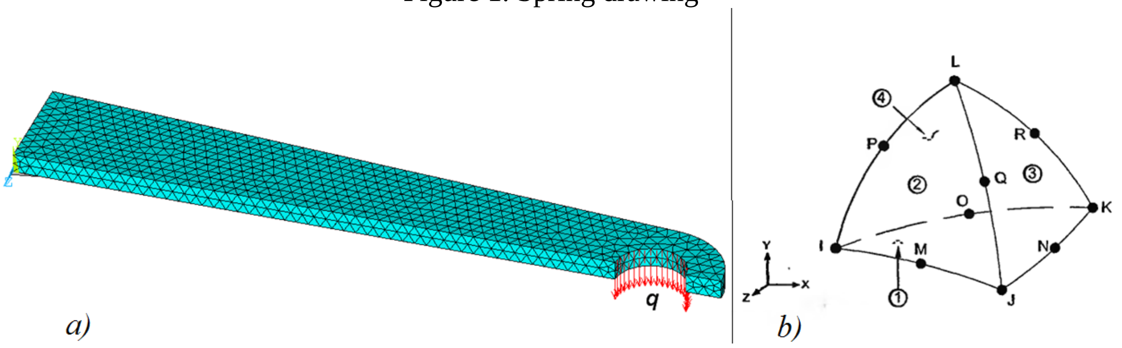

Figure 2. (a) – finite element model; (b) – geometry of the applied element SOLID92

The mechanical characteristics of the element were determined by the elastic constants of the orthotropic material in accordance with the directions of the axes (figure 2):
- modulus of elasticity in the direction of the axis *Ox* (E_x): $2.59 \cdot 10^{11}$ Pa;
- shear modulus in planes *xOy*, *xOz*, *yOz* (*G*): $3.1 \cdot 10^9$ Pa;
- Poisson's ratio in the direction of the *Ox* axis (ν_x): 0.404;
- Poisson's ratio in the direction of the axes *Oy* (ν_y) and *Oz* (ν_z): 0.007.

The modulus of elasticity along the directions of the axes *Oy* and *Oz* can be determined by the formula:

$$E_y = E_z = E_x \frac{\nu_y}{\nu_x}. \quad (1)$$

The values of the elastic parameters of the material are taken from the works [20].

The boundary conditions (figure 3):
- all nodes with the coordinate $x = 0$ are fixed in the direction of the *Ox* axis ($U_x = 0$);
- all nodes with the coordinate $z = 0$ are fixed in the direction of the *Oz* axis ($U_z = 0$);
- all nodes with coordinates $x = 0$ and $y = 0$ are fixed in the direction of the *Oy* axis ($U_y = 0$);
- the force $F = 2.5$ *kN* is applied to the circumference line (figure 2a) in the form of a distributed load *q* vertically downwards.

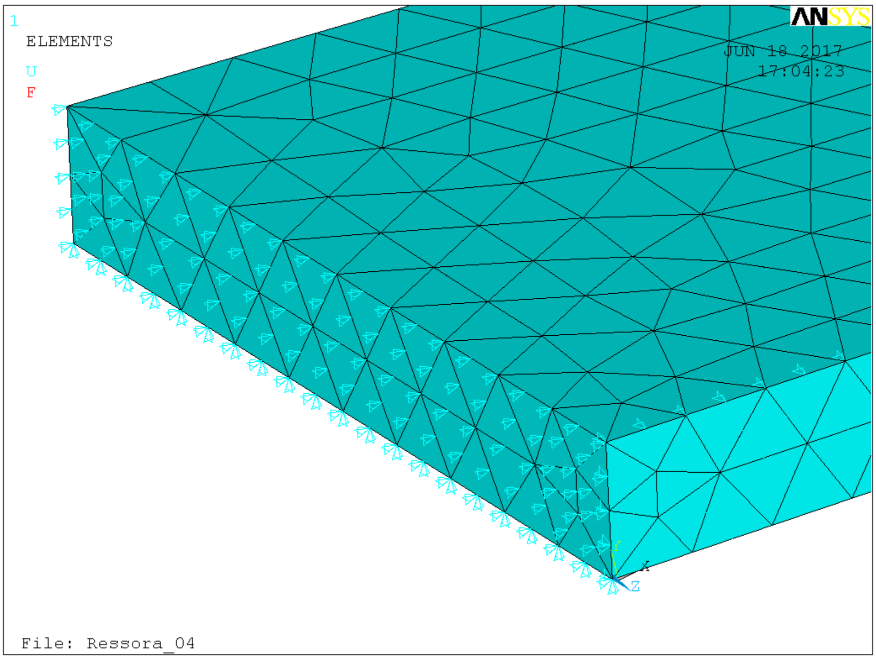

Figure 3. The boundary conditions.

In the Lira and Matlab software complexes, the spring is modeled by the flat finite elements of the plate (figure 4). In Lira, quadrangular and triangular elements are used, and in the Matlab we use only triangular finite elements with three degrees of freedom. Splitting of the grid is regular in Lira and irregular in Matlab with concentration near the hole.

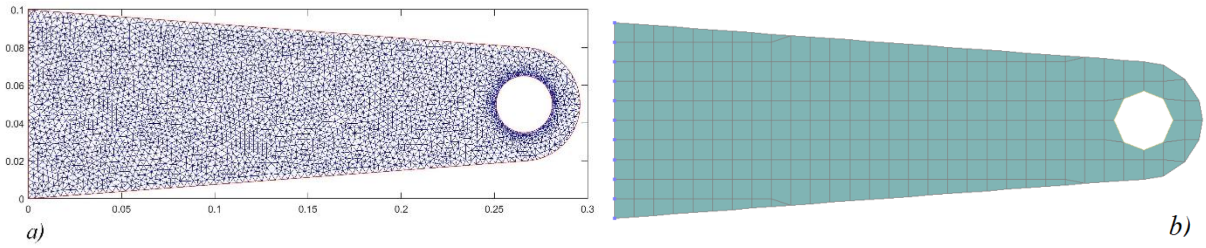

Figure 4. Finite element mesh: (a) – in Matlab, (b) – in Lira.

3. Results and Discussion

The results of solving the problem in ANSYS software in the form of stress and displacement fields are shown in figure. 5. Analysis of the results shows that the maximum deflection *Uy* under the action of the load F = 2.5 kN was 20.7 mm. The maximum tensile stresses in the direction of the *Ox* axis are 713 MPa, compressive stresses are 748 MPa.

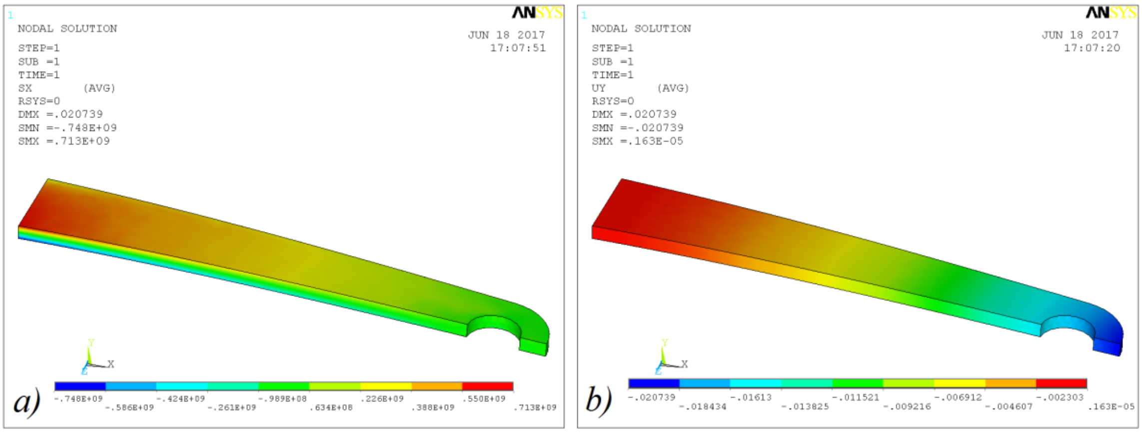

Figure 5. (a) – the fields of normal stresses σ_x (Pa) at a constant sheet thickness; (b) – fields of displacements for a constant sheet thickness in the direction of the axis *Oy* (U_y), *m*

In the program complexes Lira and Matlab, a similar character of the isopoles of stresses and displacements was obtained. The maximum deflection for a plate of constant thickness was 20.1 mm in Matlab and 18.8 mm in Lira.

The results of solving the problem for variable thickness of the sheet in accordance with the geometry of figure 1 in the form of displacement and stress fields are shown in figure 6. Analysis of the results shows that the maximum deflection U_y under the action of the load F = 2.5 kN was 12.8 mm; The maximum tensile stresses in the direction of the *Ox* axis are 473 MPa, compressive stresses are 467 MPa.

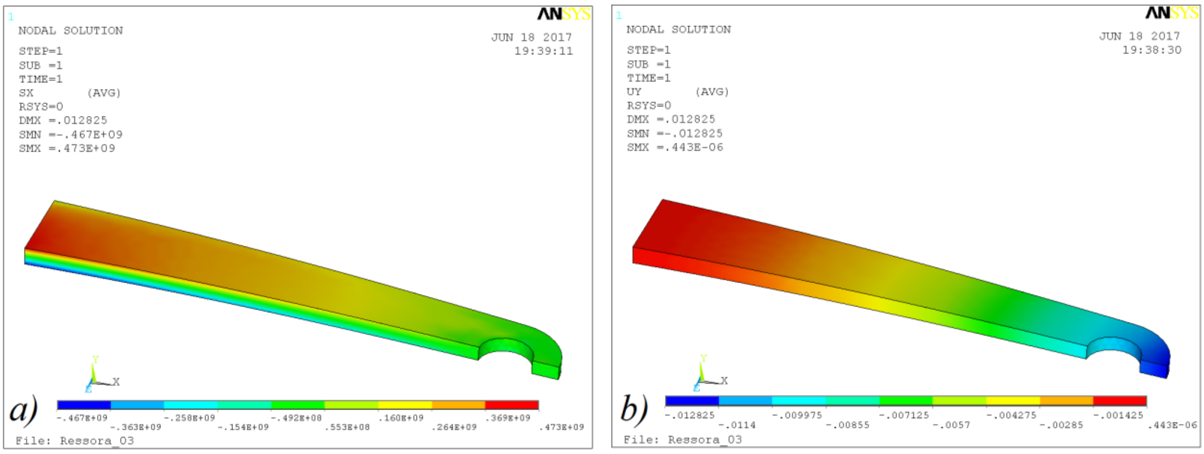

Figure 6. (a) – the fields of normal stresses σ_x (Pa) at a variable sheet thickness; (b) – fields of displacements for a variable sheet thickness in the direction of the axis *Oy* (U_y), m

Comparison of the solution results obtained for a spring of equal thickness and variable thickness showed a reduction in the maximum deflection by 38%, in tensile and compressive stresses by 34%.

4. Conclusions

The stress-strain state of a spring made of carbon fiber reinforced plastic was analyzed under the action of a vertical load on it, which causes bending. The solution was implemented in software packages ANSYS, Lira, Matlab. The greatest discrepancy in the results of displacements was 9.6%. Such a spread can be explained by the fact that the material under study is characterized by a strongly pronounced anisotropy of the properties. Modulus of elasticity in the x and y direction differ by 57.7 times.

References

[1] Oliveira B and Creus G 2004 *Composite Structures* **65** pp 443-457
[2] Mallick P 1997 *Composites engineering handbook* (New York: CRC Press) 1251 pp
[3] Treschev A 2007 *Anisotropic plates and shells of materials of different resistance* (Tula: RAASN) 341 p
[4] Golushko S et al 2014 *Computational technologies* **19.5** pp 24-36
[5] Dong S 1962 *Journal of the Aerospace Sciences* **29** pp 969-975
[6] Aghalovyan L 2015 *Asymptotic Theory of Anisotropic Plates and Shells* (Singapore: World Scientific Publishing) 359 p
[7] Andreev V et al 2014 *Advanced Materials Research* **900** pp 707-710
[8] Mailyan L et al 2017 *MATEC* **106** 04010
[9] Chepurnenko A et al 2017 *MATEC* **106** 04011
[10] Chepurnenko A et al 2016 *Procedia Engineering* **150** pp 1694–1700
[11] Gorokhov A and Trufanov N 2011 *Herald of PNIPU Mechanics* **1** pp 170–182
[12] Turusov R and Memarianfard H 2016 *Mechanics of Composite Materials* **52** pp 1-10
[13] Sergeev A et al 2016 *Mechanics of Composite Materials* **52** pp 89-98
[14] Tsybin N et al 2016 *Procedia Engineering* **153** pp 51-58
[15] Litvinov S et al 2016 *Procedia Engineering* **150** pp 1686-1693
[16] Andreev V 2013 *Applied Mechanics and Materials* **353-354** pp 1267-1270
[17] Turusov R et al 2015 *Advanced Materials Research* **1102** p 155
[18] Andreev V and Turusov R 2014 *Applied Mechanics and Materials* **670-671** pp 153-157
[19] Andreev V et al 2015 *Advanced Materials Research* **1102** pp 155-159
[20] Maranci B and Plesovskih A 2014 *Factory laboratory. Diagnostics of materials* **79** pp 12-16