PARAMETRIC CALCULATION OF THE STRESS STATE OF TEETH IN SCREW-NUT TRANSMISSION

Kravchuk, Alexander S., *Belarusian State University (BSU), Minsk, Belarus.*
Kravchuk, Anzhelika I., *Belarusian State University (BSU), Minsk, Belarus.*
Tarasyuk, Ivan A., *Belarusian State University (BSU), Minsk, Belarus.*

ABSTRACT

Screw-nut transmissions are used in control mechanisms to convert rotary motion to forward motion or vice versa. The authors propose a technique for determining the strength and wear resistance of screw-nut sliding transmission based on the solution of the contact problem for its radial section, taking into account the difference in the materials of the transmission parts. The object of the study is a screw that is made of steel, and a tin bronze nut. The article presents the solution of the problem for the trapezoidal thread profile, the basic step-by-step actions in ANSYS 10 ED, as well as the parametrized code APDL, allowing to choose the optimal geometric parameters of the transmission and the mechanical parameters of its details in terms of durability.

Keywords: *mechanics, screw-nut transmission, stress state, finite element method, ANSYS, parametric model, motion transformation.*

Background. *Such transmissions as a screw-nut are used in control mechanisms of air and water vessels, in drives of machine tools, manipulators and measuring machines, as well as in various construction tools [1].*

The paper presents a technique for calculating flexural rigidity of teeth of an arbitrary screw-nut sliding transmission profile using the ANSYS 10 ED finite element analysis program using the example of solving the contact problem for a trapezoidal thread profile. The technical sense of the calculation is to ensure that, by choosing the geometry of the profile of the teeth, it is possible to ensure a complete fit in the transmission of sliding of their contacting working surfaces and to eliminate the redistribution of contact stresses due to bending deformations. In addition, reduction of flexural deformations of teeth in the screw-nut transmission during design will reduce their tendency to break [2–5]. In this case, the proposed methodology takes into account the possibility of using different materials in the transmission links.

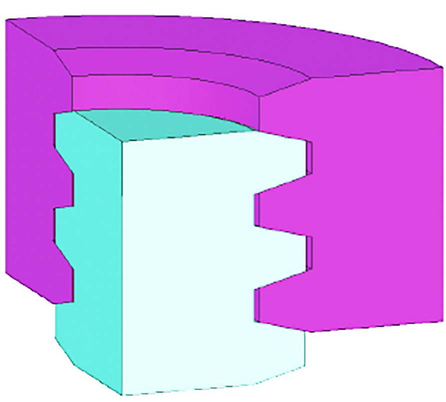
Objective. *The objective of the authors is to consider parametric calculation of the stress state of teeth in screw-nut transmission.*

Methods. *The authors use general scientific and mechanical engineering methods, comparative analysis, evaluation approach, mathematical method, computer-aided simulation methods.*

Results.

Method of construction

The axial cross section of the transmission is considered, the screw of which is made of steel, and the nut is made of antifriction tin bronze. In the evaluation decision, taking into account the limitations imposed on the number of elements in ANSYS 10 ED, the inclination of the thread with respect to the screw and nut axes is ignored, and the task is reduced to the interaction of axisymmetric gears, which allows us to move from the spatial problem for screw threads to the plane problem for the radial section of screw and nut (Pic. 1).



Pic. 1. Interaction of teeth in the screw-nut transmission model.

This unimportant simplification from the point of view of calculation accuracy results in a large saving of system resources and allows using the demonstration version of ANSYS 10 ED [9–12]. It is assumed that to demonstrate the technique it is sufficient to leave three teeth on a screw and a nut.

Considering the fact that the trapezoidal profile of the gear thread has a lower reduced friction coefficient, assuming that grease is applied to the working surfaces of the teeth, we will neglect friction in the contact area when solving the problem [1].

Construction of the model of the radial section of the screw-nut transmission. *The model will be created «from the bottom up», taking into account that the axis of symmetry of the construction is OY, and the whole model should be constructed in the first quarter of the XOY coordinate plane [9, 10]. And all the geometric dimensions will be given in a parametric fashion. To solve the problem, the dimensions of the Tr16 4 profile are used (Pic. 2), which correspond to GOST 9484-81 [13] and GOST 24737-81 [14].*

To simplify the construction of the model, it is necessary to change the angle units using the item Utility Menu>Parameters>Angular Units. In the appeared window it is required to select Degrees in the list Units for angular-parametric functions. Next, it is necessary to specify the parameters that will be used when creating the screw-nut transmission model using Utility Menu> Parameters> Scalar Parameters. In the Selection field of the window that appears, it is necessary to enter the required parameters (Table 1).

In the first step, the key points of the screw-nut transmission model are constructed. In this case, the key points are the angular points of the section of the screw tooth model, as well as the points lying on the axis of symmetry of the section. They are created using the menu item Main Menu > Preprocessor > Modeling > Create > Keypoints > In Active CS. In the window Create Keypoints in Active Coordinate System it is necessary to enter the numbers and coordinates of the points in the fields Keypoint number and X, Y, Z Location in active CS (Table 2).

Table 1

Physical and geometric parameters of the model

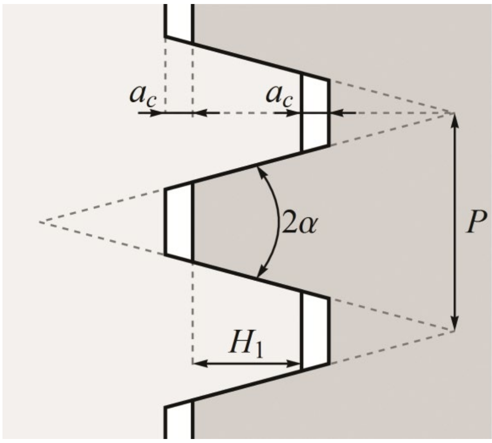
Parameter designation and value	Parameter description
E1 = 2.10E11	Young's modulus of steel, E_{st} (Pa)
NU1 = 0.29	Poisson's ratio of steel, υ_{st}
E2 = 1.22E11	Young's modulus of tin bronze, E_{br} (Pa)
NU2 = 0.33	Poisson's ratio of tin bronze, υ_{br}
P = 4.00E-3	Thread pitch, P (m)
H1 = 2.00E-3	Working height of the profile, H_1 (m)
AC = 0.25E-3	Clearance at the top of the thread tooth, a_c (m)
RV = 5.75E-3	Internal radius of the screw, R_{sc} (m)
TG = TAN(15)	Half of the profile angle, α (°)
DY = 5.00E-7	Displacement of the nut, Δy (m)
X0 = H1+AC	Height of the thread tooth, x_0 (m)
Y0 = X0 • TG	Half-difference of the base and vertex of the tooth, y_0 (m)
H0 = P/2–H1 • TG	Vertex of the thread tooth, h_0 (m)
RG = RV+X0+AC	Internal radius of the nut model, R_n (m)

In the second step, the surface of the radial section of the screw tooth model is constructed. Since the boundaries of the section are straight lines, ANSYS allows to create a surface directly at the key points, without producing the lines limiting it. Lines will appear automatically when creating a surface using the menu item Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > Through KPs. Specification of the points of the tooth model should be made only in one direction of the bypass of the area (counterclockwise, for example, 1, 2, 3, 4).

The third step is to copy the surface that defines the cross section of the tooth model: two times with a shift in the thread pitch P along the OY axis. To do this, we use the menu item Main Menu > Preprocessor > Modeling > Copy > Areas. In the window that appears, it is necessary to specify the number of copies (including the original) 3 in the field Number of copies – including original and the copy step P (m) in the field Y-offset in active CS.

The next step is to create a surface of the radial section of the body model of the screw. After using the menu item Main Menu > Preprocessor > Modeling > Create > Areas > Arbitrary > Through KPs, it is necessary to point in succession points that lie on the internal radius on the symmetry axis of the section of the screw model. It is necessary to select points only in one direction of the bypass of the region (clockwise/counterclockwise, for example 1, 4, 5, 8, 9, 12, 13, 14) (Pic. 3).

In the fourth step, the radial section of the nut model is constructed, the thickness of which is assumed to be equal to the inner radius of the screw body R_s. To do this, the surfaces corresponding to the screw model must be reflected relative to the Y-Z plane using the menu item Main Menu > Preprocessor



Pic. 2. Dimensions of the trapezoidal screw-nut transmission profile.

> Modeling > Reflect > Areas, and in the list labeled IMOVE, it is necessary to indicate that the surfaces are copied (Pic. 4).

New surfaces must be moved along the OX axis by the sum of the internal radii of the screw R_s and the nut R_n, along the OY axis by half the pitch of P/2 thread using the menu item Main Menu > Preprocessor > Modeling > Move / Modify > Areas > Areas. To do this, in the window that appears, it is necessary to name the number of copies (including the original) 1 in the field Number of copies – including original and the copying steps RV+RG (m) and P/2 (m) in the fields X-offset in active CS and Y-offset in active CS. It is obvious that by rotating around the OY axis, it is possible to obtain a three-dimensional model of transmission within the limits of the accepted geometric simplifications.

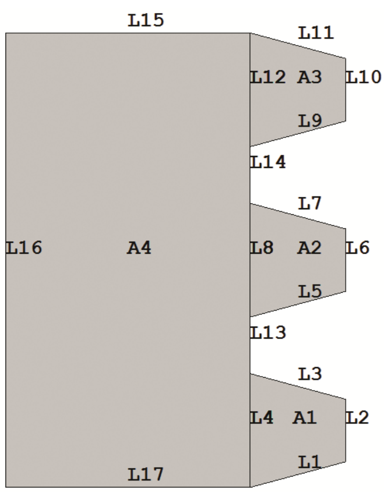
Choice of the task type and material properties of the model. *In accordance with the logic of organization of the main menu, the choice of the type of the problem being solved is offered at the first step before preparation of the geometric model. However, the practice of working in ANSYS 10 ED with such additional modules as LS-DYNA or FLOTRAN, shows that it is better to make the choice after preparing the entire geometry of the model. To do this, use the menu item Main Menu > Preferences... and in the window Preferences for GUI Filtering specify Structural.*

Linear elastic materials will be used to solve the problem: steel for the screw model and antifriction tin

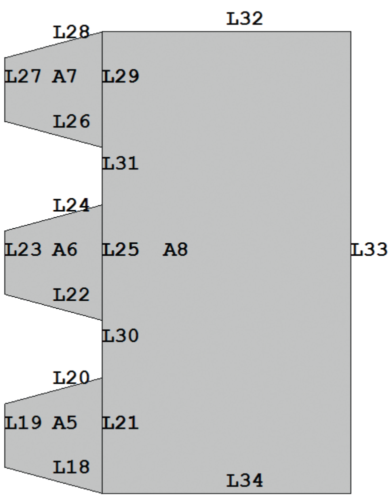
Table 2

Key points of the radial section of the screw tooth model

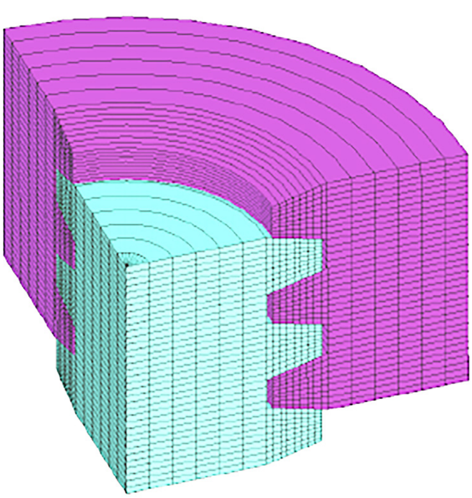
Key point number KP	X–Loc (m)	Y-Loc (m)
1	RV	0
2	RV+X0	Y0
3	RV+X0	Y0+H0
4	RV	2 • Y0+H0
13	0	2 • (P+Y0)+H0
14	0	0



Pic. 3. Radial section of the screw model.



Pic. 4. Radial section of the nut model.



Pic. 5. Ordered partitioning of the screw-nut transmission model.

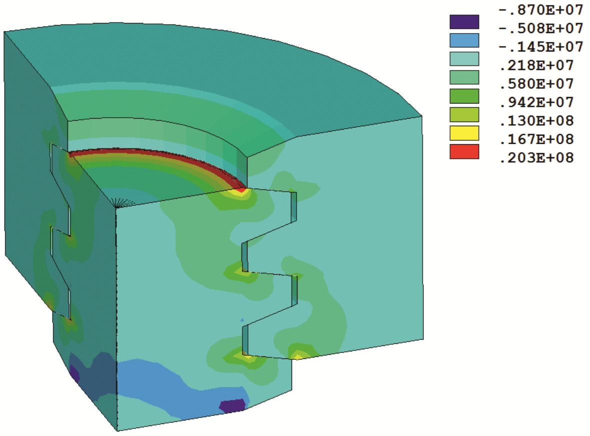
bronze for the nut model (Table 3) [1]. To create two material models, it is necessary to call the Define Material Model Behavior window using the Main Menu>Preprocessor>Material Props> Material Models main menu item. Using the items of the upper drop-down menu and the sections of the material library Material Models Available (on the right in the general window of the Define Material Model Behavior), you need to specify the physical characteristics of two different materials, enumeration and values of which are recorded in the left window Material Models Defined in the general window Define Material Model Behavior.

In the Define Material Model Behavior window, one material Material Number 1 is available by default (displayed in the left window and highlighted by the cursor). In the right window, using the Define Material Model Behavior> Material Models Available > Structural > Linear > Elastic > Isotropic item, the material is linearly elastic, and in the appeared Linear Isotropic Properties for Material Number 1window, you enter the desired values for the elastic constants of the steel 1): the Young's modulus E1 (Pa) in the field labeled EX and the Poisson's ratio NU1 in the field labeled NUXY, and then click OK.

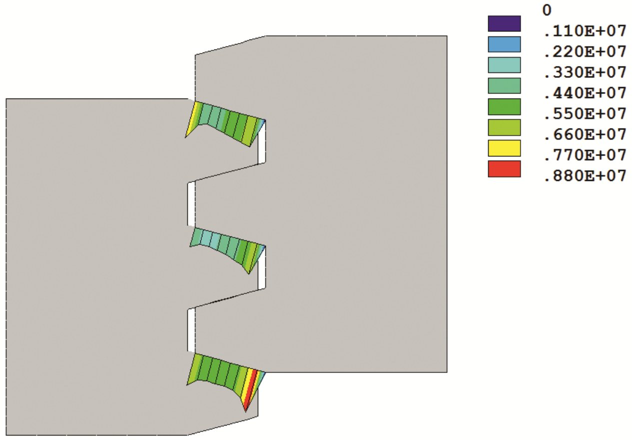
Then, returning to the general window Define Material Model Behavior, you need to use the menu item Define Material Model Behavior > Material > New Model to add the second material. After confirming that the number of the material to be added should be left «default» (i.e. 2), you can select another model for behavior of the material with different characteristics. In this case, the material will also be linearly elastic (this is indicated via the menu item Define Material Model Behavior > Material Models Available > Structural > Linear > Elastic > Isotropic) with elastic constants that correspond to the tin bronze (Table 1): Young's modulus E2 (Pa) and the Poisson's ratio NU2. They should be entered in the appeared Linear Isotropic Properties for Material Number 2 window in the EX and NUXY fields respectively, and then click OK.

Choice of an element type for the model partition. *Since the task in question corresponds to a structural analysis, you must select the type of Structural Solid elements and the elements Quad 4node 182. The selection is made using the main menu item Main Menu > Preprocessor > Element Type > Add/Edit/Delete after clicking the Add button in the Element Types window. Then, in the Element Types window, click the Options button and enable the Axisymmetric option in the Element Behavior list, then click OK and close the window.*

Assigning attributes to model surfaces. *In ANSYS, the attributes of the geometric components of the model are material number Material number (different for screw and nut models), element type Element type number (same for all model components), Real constant set number and Element section sections (not used). The enumerated set of attributes is assigned to each component of the geometric model separately using the item Main Menu > Preprocessor > Meshing > Mesh Attributes > Picked Areas. In the menu that appears, select the surfaces corresponding to the section of the screw model (A1, A2, A3, A4), and then click OK (Pic. 3). In the Area Attributes window, all values should be left «by default» and then click Apply. Then you need to select the surfaces corresponding to the section of the nut model (A5, A6, A7, A8), click OK and change the material number to 2 in the Area Attributes window in the Material number list (Pic. 4).*



Pic. 6. Distribution of the first principal stresses in the screw-nut transmission model.



Pic. 7. Distribution of contact pressures in the section of the screw-nut transmission model.

Construction of an ordered partition of the model. *To do this, you need to make a concatenation of several lines (logical join) using the menu item Main Menu > Preprocessor > Meshing > Concatenate > Lines. Using the menu that appears, select the lines lying on the inner radius of the section of the screw model (L4, L8, L12, L13, L14), and then click Apply (Pic. 3). Next, select the lines lying on the inner radius of the section of the nut model (L21, L25, L29, L30, L31), and click OK (Pic. 4).*

To set the size of the partitioning of surfaces, you need to use the menu item Main Menu>Preprocessor>Meshing>Size Cntrls>Manual Size> Areas>All areas and in the window that appears specify the length of the side of the element (P-2•Y0–H0)/4 (m), that corresponds to dividing the lines simulating the thread cavities into four parts. Since a coarser grid can be used away from the contact area when solving a contact problem due to the limitation on the number of elements in ANSYS 10 ED, the linear dimension of the element in the radial direction should be reduced for surfaces simulating the cross sections of the screw and nut bodies. To do this, using the item Main Menu > Preprocessor > Meshing > Size Cntrls > Manual Size > Lines > Picked Lines, you need to select the section lines for the screw and nut models that lie perpendicular to the axis of symmetry of the nut model (for example, L15 and L32), and then in the Element Size on Picked Lines window, specify the number of divisions 6 in the No of element divisions field (Pic. 3, 4).

After performing the above operations, the cross section of the transmission model is ready to construct an ordered partition using the item Main Menu > Preprocessor > Meshing > Mesh > Areas > Mapped > 3 or 4 sided. When you use it, the Mesh Areas window appears, in which the Pick All button should show that the entire model will be split (see Pic. 5).

After building the partition, you must delete the concatenation of lines using the menu item Main Menu > Preprocessor > Meshing > Concatenate > Del Concats > Lines. When the selection menu appears, select Pick All.

Assignment of boundary conditions. *The boundary conditions for the screw-nut transmission model begin with the simplest restrictions. First, it is necessary to fix the cross section of the rotor model along the axis of symmetry (in this case, to prohibit its movement along the OY axis) and, secondly, to assign a non-zero displacement of the external boundary of the nut model in the axial direction (set the movement in the negative direction of the OY axis).*

The menu item Main Menu > Preprocessor > Loads > Define Loads > Apply > Displacement > On Lines is turned on. To specify the first boundary condition, it is required to select the section lines of the nut model that lie perpendicular to its symmetry axis (L15 and L17), click OK, select the UY label in the appeared window, set the displacement value 0 (m) in the VALUE field and click the Apply button (Pic. 3). Then, to specify the second boundary condition, select the section lines of the screw model that lie perpendicular to the axis of symmetry of the section of the nut model (L32 and L34), click OK, select the UY label in the appeared window and set the displacement value in the VALUE field to DY (m).

One of the most complex types of boundary conditions in structural mechanics is creation of contact pairs between interacting surfaces of unconnected

Table 3

The program code ANSYS APDL for solving the task

No.	Command	No.	Command
1	*AFUN, DEG	46	LSEL, A, LOC, X, RG+RV/2
2	E1 = 2.10E11	47	LESIZE, ALL,,,6
3	NU1 = 0.29	48	MSHKEY,1
4	E2 = 1.22E11	49	AMESH, ALL
5	NU2 = 0.33	50	LSEL, S, LCCA
6	P = 4.00E-3	51	LDELE, ALL
7	H1 = 2.00E-3	52	LSEL, S, LOC, X, RV/2
8	AC = 0.25E-3	53	DL, ALL,, UY,0
9	RV = 5.75E-3	54	LSEL, S, LOC, X, RG+RV/2
10	TG = TAN(15)	55	DL, ALL,, UY,-DY
11	DY = 5.00E-7	56	ALLSEL, ALL
12	X0 = H1+AC	57	MP, MU,1,0
13	Y0 = X0 • TG	58	MAT,1
14	H0 = P/2-H1 • TG	59	ET,2, TARGE169
15	L0 = 2 • (P+Y0)+H0	60	ET,3, CONTA172
16	RG = RV+X0+AC	61	KEYOPT,3,5,1
17	/PREP7	62	KEYOPT,3,9,0
18	K,1, RV,0	63	KEYOPT,3,10,2
19	K,2, RV+X0, Y0	64	LSEL, S, LOC, Y,3 • Y0/2+H0
20	K,3, RV+X0, Y0+H0	65	LSEL, A, LOC, Y,3 • Y0/2+H0+P
21	K,4, RV,2 • Y0+H0	66	LSEL, A, LOC, Y,3 • Y0/2+H0+2 • P
22	K,13,0,2 • (P+Y0)+H0	67	CM,_TARGET, LINE
23	K,14,0,0	68	TYPE,2
24	A,1,2,3,4	69	NSLL, S,1
25	AGEN,3, ALL,,,0, P,0,,1,0	70	ESLN, S,0
26	A,1,4,5,8,9,12,13,14	71	ESURF
27	ARSYM, X, ALL,,,,1,0	72	LSEL, S, LOC, Y, Y0/2+P/2
28	AGEN,1,5,8,, RV+RG, P/2,0,,1,1	73	LSEL, A, LOC, Y, Y0/2+3 • P/2
29	MP, EX,1, E1	74	LSEL, A, LOC, Y, Y0/2+5 • P/2
30	MP, NUXY,1, NU1	75	CM,_CONTACT, LINE
31	MP, EX,2, E2	76	TYPE,3
32	MP, NUXY,2, NU2	77	NSLL, S,1
33	ET,1, PLANE182	78	ESLN, S,0
34	KEYOPT,1,3,1	79	ESURF
35	ASEL, S, AREA,,1,4	80	CMDEL,_TARGET
36	AATT,1,,1	81	CMDEL,_CONTACT
37	ASEL, S, AREA,,5,8	82	ALLSEL, ALL
38	AATT,2,,1	83	FINISH
39	LSEL, S, LOC, X, RV	84	/SOL
40	LCCAT, ALL	85	TIME,1
41	LSEL, S, LOC, X, RG	86	NSUBST,100
42	LCCAT, ALL	87	AUTOTS, ON
43	ALLSEL, ALL	88	NLGEOM, ON
44	AESIZE, ALL,(P-2 • Y0-H0)/4	89	SOLVE
45	LSEL, S, LOC, X, RV/2	90	–

solids. In the two-dimensional case, it is necessary to define the injection lines and the contact lines. The first are the upper lines of the section for the nut thread patterns (L3, L7, L11), the second are the lower lines of the section for the screw thread patterns (L18, L22, L26) (Pic. 3, 4).

To get the contact pair, you need to run Contact Manager by using the menu item Main Menu > Preprocessor > Modeling > Create > Contact Pair, clicking the Contact Wizard button and using it to start the pair creation wizard. In the first window that appears, leave all options «by default» (the implant surface acts as a compliant line), click the Pick Target button, select the embedding lines (L3, L7, L11), confirm the selection with the OK button, and then go to the next wizard window by clicking Next (Pic. 3). In the new wizard window, all the options are «default» (the surface and the contact area are lines), click the Pick Contact button, select the contact lines (L18, L22, L26), confirm the selection with OK and move to the next window by clicking Next (Pic. 4). In the last window again, leave all the settings «by default» (do not create a symmetric contact due to the limitation on the number of elements in ANSYS 10 ED, take into account the initial implementation, neglect friction, and other options in the Optional settings section) and create a contact pair by clicking the Create button. After that, the created contact elements will be displayed in the ANSYS Graphics window.

Generally speaking, both side surfaces of the cross-section of the thread teeth models should be indicated as the contacting surfaces, however this leads to exceeding the limit on the number of elements in the training version of ANSYS 10 ED.

Features of displaying results

Before starting the solution, in view of the non-linearity of the task, you should use the menu item Main Menu > Solution > Analysis Type > Sol'n Controls and turn on the Large Displacement Static option in the Analysis Options drop-down list. In addition, you must specify the number of iterations of solution 100 in the Number of substeps item (the number of iterations is specified for nonlinear or non-stationary analysis), and also enable the option to automatically select the time step for the solution by selecting On from the Automatic time stepping drop-down list.

If you do not install the steps for integrating the task, after the solution is launched, an information message (warning) appears about the mandatory assignment of the size of the iteration or the number of it. To solve the task, you need to use the item Main Menu > Solution > Solve > Current LS.

To display the results of solution in an axisymmetrically extended screw-nut transmission model, use the menu item Utility Menu > PlotCtrls > Style > Symmetry Expansion > 2D Axi-Symmetric… and select the size of extension in the window that appears. Return to the original view is provided by clicking No Expansion. The window for selecting the displayed physical parameter calculated in the nodes of the finite element model is called up via the main menu item Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu. Next, in the Contour Nodal Solution Data window that appears, in the Item to be contoured list, select the result you want to display.

The display of the first main stresses – one of the main parameters for calculating the strength of parts – the Nodal Solution > Stress > 1st Principal Stress function (Pic. 6). To display the contact pressures, the Nodal Solution > Contact > Contact Pressure item is activated (Pic. 7).

The code APDL (Table 3) helps to get the solution of the problem for different values of the physical and geometric parameters of the model (Table 1).

Conclusions. *A method is proposed for determining the strength and wear resistance of screw-nut sliding transmission based on the solution of the contact problem for its radial section, taking into account the difference in the materials of the transmission parts. As an example, the variant for the trapezoidal thread profile is taken, all the main features of the solution of the problem are analyzed in ANSYS 10 ED, and a parameterized code APDL is presented, which allows choosing the optimal geometric parameters of the transmission and the mechanical parameters of its parts in terms of durability. The methodology can find wide application in the design organizations engaged in the development of this type of mechanisms.*

REFERENCES

1. Roshchin, G. I., Samoilov, E. A., Alekseeva, N. A. [*et al*]. Details of machines and the basis of construction: Textbook [*Detali mashin i osnovy konstruirovaniya: Uchebnik*]. Moscow, Drofa publ., 2006, 415 p.

2. Lodygina, N. D. Dependence of stresses on geometrical parameters of the screw mechanism twist [*Zavisimost napryazhenii ot geometricheskih parametrov vitka vintovogo mehanizma*]. *Sovremennie naukoemkie tehnologii*, 2013, Iss. 2, pp. 26–28.

3. Kiri, B., Lacetera, N., Zirulia, L. Above a swamp: A theory of high-quality scientific production. *Research Policy*, 2018, vol. 47(5), pp. 827–839.

4. Timofeev, G. A. Theory of mechanisms and machines [*Teoriya mehanizmov i mashin*]. Moscow, Yurayt publ., 2016, 429 p.

5. Dilnot, C. Thinking design: A personal perspective on the development of the Design Research Society. *Design Studies*, 2018, vol. 54, pp. 142–145.

6. Nestler, M., Nitschke, I., Praetorius, S., Voigt, A. Orientational Order on Surfaces: The Coupling of Topology, Geometry, and Dynamics. *Journal of Nonlinear Science*, 2018, vol. 28(1), pp. 147–191.

7. Li, Z., Pehlken, A., Qian, H., Hong, Z. An automated functional decomposition method based on morphological changes of material flows. *Journal of Engineering Design,* 2016, vol. 28(1), pp. 47–75.

8. Lucero, B., Linsey, J., Turner, C. J. Frameworks for organising design performance metrics. *Journal of Engineering Design*, 2016, vol. 27, iss. 4–6 [Electronic resource]: https://www.tandfonline.com/doi/full/10.1080/09544828.2015.1135235. Last accessed 23.07.2018.

9. Kravchuk, A. S., Smalyuk, A. F., Kravchuk, A. I. Electronic library of mechanics and physics. Lectures on ANSYS with examples of solving problems: in 5 parts. Part 1. Graphical interface and command line. Tools for creating a geometric model [*Elektronnaya bibilioteka mehaniki i fiziki. Lektsii po ANSYS s primerami resheniya zadach: v 5 ch. Chast 1. Graficheskiy interfeis i komandnaya stroke. Sredstva sozdaniya geometricheskoi modeli*]. Minsk, BSU publ., 2013, 130 p. [Electronic reource]: http://elib.bsu.by/handle/123456789/43440. Last accessed 23.07.2018.

10. Kravchuk, A. S., Smalyuk, A. F., Kravchuk, A. I. Electronic library of mechanics and physics. Lectures on ANSYS with examples of solving problems: in 5 parts. Part 2. Tools for displaying and editing geometric components of a solid model. Examples of creating solid models by means of ANSYS for solving physical problems [*Elektronnaya biboletka mehaniki i fiziki. Lektsii po ANSYS s primerami resheniya zadach: v 5 ch. Chast 2. Sredstva otobrazheniya i redaktirovaniya geometricheskih komponentov tverdotelnoi modeli. Primery sozdaniya tverdotelnyh modelei sredstvami ANSYS dlya resheniya fizicheskih zadach*]. Minsk, BSU publ., 2013, 145 p. [Electronic resource]: http://elib.bsu.by/handle/123456789/43443. Last accessed 23.07.2018.

11. Kravchuk, A. S., Smalyuk, A. F., Kravchuk, A. I. Electronic library of mechanics and physics. Lectures on ANSYS with examples of solving problems: in 5 parts. Part 3. Definition of physical constants of materials and finite element decomposition of a solid model. Examples of doing these actions with previously built models [*Elektronnaya biblioteka mehaniki i fiziki. Lektsii po ANSYS s primerami resheniya zadach: v 5 ch. Chast 3. Opredelenie fizicheskih constant materialov i konechnoelementnoe razbienie tverdotelnoi modeli. Primery vypolneniya etih deistvii s postroennymi ranee modelyami*]. Minsk, BSU publ., 2013, 193 p. [Electronic resource]: http://elib.bsu.by/handle/123456789/43444. Last accessed 23.07.2018.

12. Kravchuk, A. S., Smalyuk, A. F., Kravchuk, A. I. Electronic library of mechanics and physics. Lectures on ANSYS with examples of problem solving: in 5 parts. Part 4. Limitations and loads. Sections Solution and General Postproc of the main menu. Examples of statement of boundary value problems, their solutions and viewing of results [*Elektronnaya biblioteka mehaniki i fiziki. Lektsii po ANSYS s primerami resheniya zadach: v 5 ch. Chast 4. Ogranicheniya i nagruzki. Razdely Solution i General Postproc glavnogo menu. Primery nostanovki kraevyh zadach, ih resheniya i prosmotra rezultatov*]. Minsk, BSU publ., 2013, 118 p. [Electronic resource]: http://elib.bsu.by/handle/123456789/43445. Last accessed 23.07.2018.

13. GOST 9484-81. The basic norms of interchangeability. Thread is trapezoidal. Profiles. Instead of GOST 9484-73. Introduced from 01.01.82 [*GOST 9484-81.Osnovnie normy vzaimozamenyaemosti. Rezba trapetseidalnaya. Profili. Vzamen GOST 9484-73; Vveden s 01.01.82*]. Moscow, Izd-vo standartov, 1981, 4 p.

14. GOST 24737-81. The basic norms of interchangeability. Thread trapezoidal single-thread. Basic dimensions. Instead of GOST 24737-73. Introduced from 01.01.82 [*GOST 24737-81. Osnovnie normy vzaimozamenyaemosti. Rezba trapetseidalnaya odnozahodnaya. Osnovnie razmery. Vzamen GOST 24737-73; Vveden s 01.01.82*]. Moscow, Izd-vo standartov, 1981, 6 p.

15. Kravchuk, A. S., Chigarev, A. V. Mechanics of contact interaction of bodies with circular boundaries [*Mehanika kontaktnogo vzaimodeistviya tel s krugovymi granitsami*]. Minsk, Technoprint publ., 2000, 196 p. ●

Information about the authors:

Kravchuk, Alexander S. – D.Sc. (Physics and Mathematics), professor of Belarusian State University (BSU), Minsk, Belarus, a.kr4vchuk@yandex.ru, ask_Belarus@inbox.ru.

Kravchuk, Anzhelika I. – Ph.D. (Physics and Mathematics), associate professor of Belarusian State University (BSU), Minsk, Belarus, anzhelika.kravchuk@gmail.com.

Tarasyuk, Ivan A. – Master course student at the department of Bio and nanomechanics of Belarusian State University (BSU), Minsk, Belarus.

Article received 23.07.2018, accepted 14.09.2018.