Article citation info:
Bazhanova A, Nemchuk O, Lymarenko O, Piterska V, Sherstiuk O, Khamrai V. Diagnostics of stress and strained state of leaf springs of special purpose off-road vehicles. Diagnostyka. 2022;23(1):2022111. https://doi.org/10.29354/diag/147292.

DIAGNOSTYKA, 2022, Vol. 23, No. 1

e-ISSN 2449-5220
DOI: 10.29354/diag/147292

DIAGNOSTICS OF STRESS AND STRAINED STATE OF LEAF SPRINGS OF SPECIAL PURPOSE OFF-ROAD VEHICLES

Anastasiia BAZHANOVA [1], Oleksii NEMCHUK [2], Oleksandr LYMARENKO [1], Varvara PITERSKA [2*], Olha SHERSTIUK [2], Vadym KHAMRAI [1]

[1] Odessa Polytechnic National University, Ukraine
[2] Odessa National Maritime University, Ukraine
[*] Corresponding author, e-mail: varuwa@ukr.net

Abstract

The work is devoted to the diagnostics of the stress state of systems that soften the shock load on the vehicle body, the elastic element of the car suspension such as a multi-leaf spring. The construction of a mathematical model taking into account the geometric nonlinearity according to the finite element method theory is considered. Mathematical modelling was carried out to take into account the change in the stiffness matrix of the system when changing its shape. For research, a symmetrical semi-elliptical spring consisting of five leaves is used. All numerous numerical experiments were performed in two computer-aided design (CAD) systems: ANSYS, a heavy multipurpose package and SolidWorks, a middle-level multipurpose package. Computer-aided design algorithms have been developed to expand the capabilities of CAD. The analysis of the results obtained allows to conclude that the traditional models of nonlinearity in ANSYS and SolidWorks give approximately the same results, which at the maximum point differ by 20.6% from the data of a full-scale experiment. When using the proposed model, this difference is reduced to 7.95%.

Keywords: diagnostics, spring, mathematical model, geometric nonlinearity, computer-aided design (CAD), finite element method, ANSYS, SolidWorks

1. INTRODUCTION

Diagnostics of the load distribution in the damping elements of vehicles between the spring leaves is a complex problem in mechanical engineering, since strain leads to contact interaction forces that are applied at a number of isolated points. In this regard, the role of digitalization of the design diagram of the spring structure with the help of modern modelling and calculation methods, of which the finite element method is the most effective today, is significantly increasing. In this work, these problems are solved using the ANSYS software package, one of the more high-tech computer programs for modelling and calculating structures and SolidWorks software, which is a universal tool for modern engineering. Here it is possible to obtain a solution to the problem of the spring stress-strain state, taking into account the contact interaction of its leaves. This approach allows both to model a solid object of study and to carry out its strength analysis and optimization.

2. ANALYSIS OF RECENT RESEARCH AND PUBLICATIONS

Calculations of elastic and dissipative systems taking into account geometric and physical nonlinearity are traditionally considered as the “weakest link” of the FEM [1, 2]. Extensive research has been devoted to this issue. Many authors believe that currently used nonlinearity models are simple and unsuitable for use [3–5]. They provide proof with some simple test problems solved by other methods. A comparison with the results of calculations of the FEM shows the complete inconsistency of the latter. It is almost impossible to predict the results of nonlinear calculation of complex systems with their accuracy [6, 7]. The assessment of the stressed and strain state is possible only with a certain degree of error [8-10]. In this regard, the development of models using a comprehensive series of computer-aided design (CAD) tests on various elastic and dissipative systems is of particular importance [11-13]. Models and methods for accounting for contact interactions are different (and sometimes completely different) in different CAD systems; geometric models and methods for accounting for nonlinearity functionally inherent in CAD are practically identical [14-16]. The differences are due to the standard types of finite elements corresponding to a particular system and the number of methods used to implement nonlinearity algorithms.

3. THE PURPOSE OF THE RESEARCH

In this regard, the purpose of the work is to obtain the most accurate solutions for diagnostics of stresses and strains in damping devices of vehicles. This goal is achieved by solving the following tasks:

1. Performing a calculation based on the hypothesis of the transfer of forces along the end sections.
2. Modelling the spring structure using modern computer programs and its calculation by the finite element method.
3. Taking into account the contact interaction of the spring leaves.
4. Optimization of the springs in the SolidWorks complex.

4. MATERIALS OF THE RESEARCH

The object of the study is the off-road vehicle spring. This symmetrical semi-elliptical spring consists of 4 semi-elliptical leaves, the thickness of which is 6.5 mm, and a straight main leaf, the thickness of which is 10 mm, the width of the package is 70 mm, the length of the leaves is from 660 mm to 1252 mm, the spring deflection boom is 180 mm. (Fig. 1). The spring load is q = *1050 N/mm*^{2}. When modelling the boundary conditions, the real nature of attaching the leaf spring to the vehicle was taken into account.

Table 1. Spring leaf length

Leaf number	Length, mm
1	1252
2	1138
3	962
4	885
5	660

The model is executed by means of modelling of the ANSYS program. Considering that the structure and the load are symmetrical about the YOX plane (Fig. 1), the structure can be considered as flat in the modelling. This approach will not in any way affect the accuracy of the results, but at the same time the number of finite elements is significantly reduced, which means both the calculation time and computer resources. The spring finite element model is shown in Fig. 1. When approximating it, an ordered meshing is used. The average element length was 6.5 mm. The entire model consists of 3391 elements and 3654 nodes.

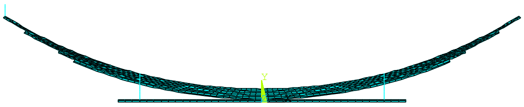

Fig. 1. Spring finite element model

When dividing the model into finite elements, two types of elements were used: for the spring leaves – the PLANE42 element, for the attachment fitting (Fig. 2) and for the clamps – the BEAM3 element.

PLANE42 is used for two-dimensional (2D) modelling of structures with volumetric stress-strain state. The element can be used as a flat (with a plane stressed or strain state) or as an axisymmetric one. An element is defined by four nodes with two degrees of freedom at each node: displacement in the X and Y directions of the nodal coordinate system. The geometry, location of nodes and the local coordinate system of the PLANE42 element are shown in Fig. 3.

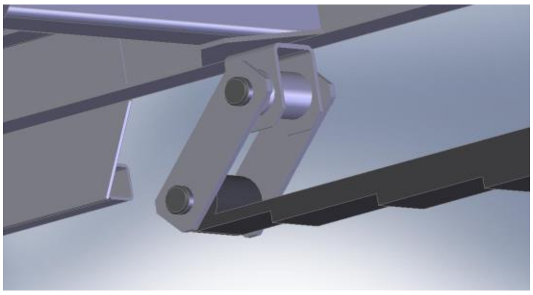

Fig. 2. Attachment fitting

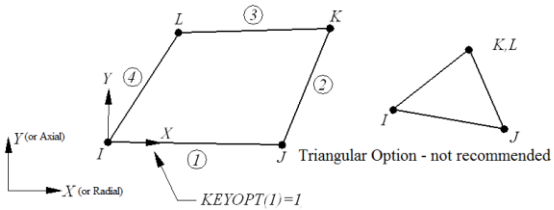

Fig. 3. Geometry of the PLANE 42 element

BEAM3 is a two-dimensional elastic beam. The element has three degrees of freedom at each node: movement in the X and Y directions and rotation around the Z-axis. The geometry, position of nodes and coordinate system of the BEAM3 element are shown in Fig. 4.

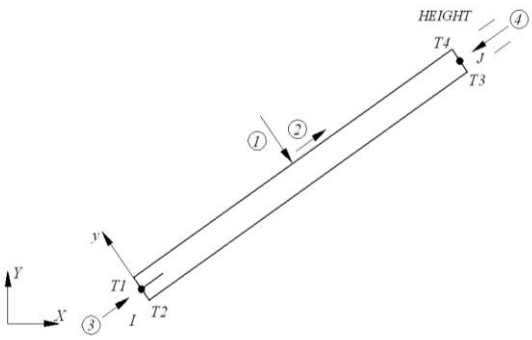

Fig. 4. BEAM3 flat element

When modelling the boundary conditions, the real nature of the fastening of most of the leaf springs was taken into account. One edge is fixed hinge-immovably, and the other has a hinge fastening, carried out by means of a steel bar (Fig. 2). Such a constructive solution allows the spring to perceive a certain part of the operating load during free movement of the spring end by the dimension determined by the length of the mounting plate.

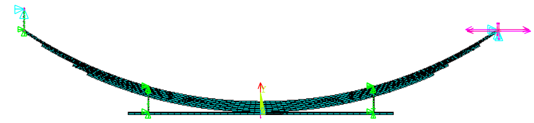

Fig. 5. Load and boundary conditions

The calculation was carried out with a load applied to the central node *F = 10000 N* (Fig. 5). The string strained shape is shown in fig. 6.

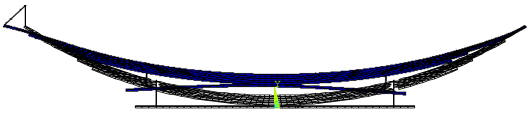

Fig. 6. The string strained shape

The capabilities of the ANSYS program make it possible to obtain as a result of the calculation a number of parameters of the stress-strain state: stress and displacement along the directions of the coordinate axes, principal stresses and displacement. In research practice, the analysis of the strength and stiffness of a structure is usually based on the total (equivalent) stresses and displacements. Fig. 7 shows the nature of the distribution of equivalent stresses in the spring, calculated according to the Huber-Mises strength hypothesis. The maximum stresses are 352.14 MPa. In this case, the vertical displacements (Fig. 8) were 7.102 mm. The values of displacements and stresses at the nodes of the mesh, generated by the program as a result of the calculation, are displayed in text files.

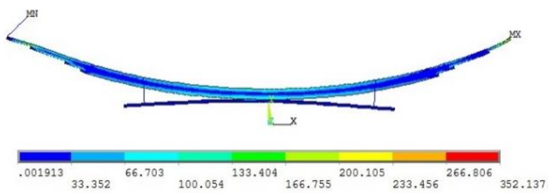

Fig. 7. Equivalent stresses

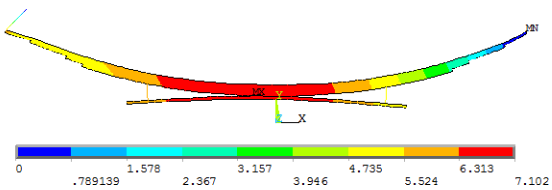

Fig. 8. Total displacements

4.1. Spring diagnostics taking into account contact stresses in the SolidWorks software package

Taking into account the contact interaction of the spring leaves leads to a change in the parameters of the stress-strain state. This fully confirms the calculation of the off-road vehicle spring, carried out taking into account the contact stresses. Linear static analysis assumes that the relationship between loads and induced reactions is linear (Fig. 9).

When the magnitude of the loads is doubled, the reactions (displacement, load, stress, support reaction, etc.) will also double.

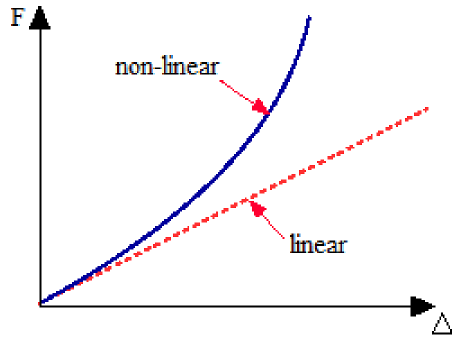

Fig. 9. Linear and nonlinear static analysis dependencies

All real structures behave non-linearly at different levels of overload. In certain cases, the linear analysis may be sufficient. In most cases, a linear solution can lead to erroneous results, since the assumptions and hypotheses on which it is based are violated. Non-linearity can be caused by material behaviour, large displacements and contact conditions, as in the case of spring design. To analyse the research object, the following actions were performed:

- on the basis of technical documentation, a spring assembly was built from individual parts such as leaves, clamps, plates to transfer the real nature of the load;
- the contact conditions between the spring leaves, the material, as well as its characteristics are set, similar to those specified in the Ansys program;
- some features of fixing were taken into account, so the package of leaves was fixed relative to the front plane, which makes it impossible to displace the leaves relative to it;
- the load on the model was transmitted through a steel plate, which fully corresponds to the transfer of the real load to the spring during its operation.

The spring finite element model in the SolidWorks software is shown in Fig. 10. Here, as in the calculation without taking into account contact stresses, an ordered meshing was used, but the average element length was 6.8 mm, and the model consists of 7647 elements and 6190 nodes.

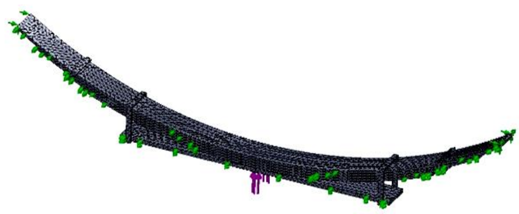

Fig. 10. Finite element model, loading and support fixing of the contact problem

The calculation of the spring was carried out with the same load $q = 1050\ N/mm^2$ applied to a plate with dimensions of 70*15 mm.

Fig. 11 shows the distribution of equivalent stresses according to the Huber-Mises strength hypothesis. The maximum stresses were 371.2 MPa.

The maximum displacements were 6.56 mm (Fig. 12).

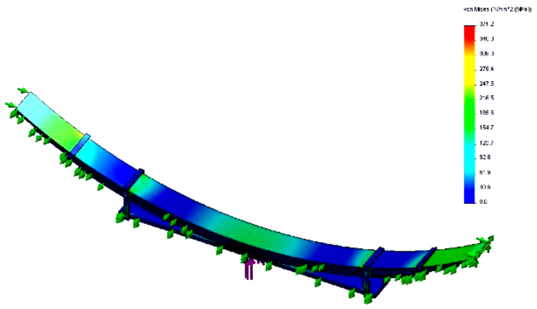

Fig. 11. Equivalent stresses in the contact problem

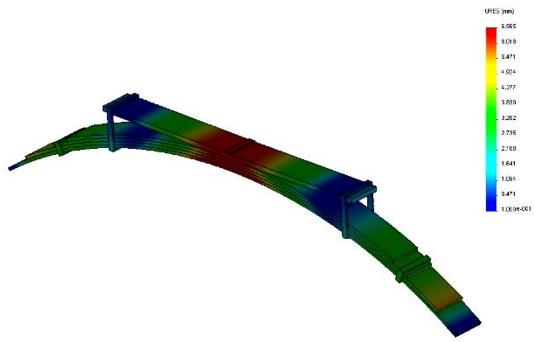

Fig. 12. Total displacements

The maximum contact stresses in the spring leaves were 56.1 MPa (Fig. 13).

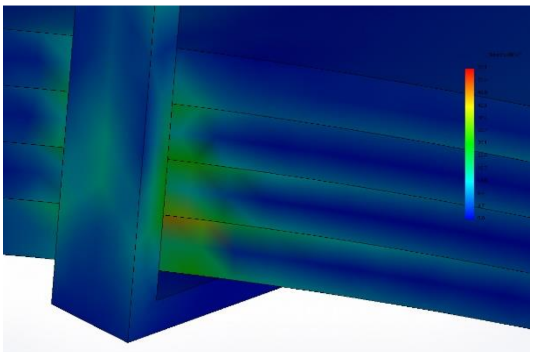

Fig. 13. Contact stresses in the spring leaves

The numerical values of the deflections and stresses at the mesh nodes are displayed in text files obtained as a result of the calculation. It should be noted that when taking into account the contact interaction of the spring leaves, the maximum stresses of 371.2 MPa are 7% higher than the stresses calculated without taking into account the contact of the leaves, but the nature of the stress distribution over the spring completely changes (Figs. 7 and 10). This indicates that the generally accepted model of contact interaction between leaves, according to which contact is carried out along the entire line of their message, does not correspond to reality; real contact between the leaves occurs at a number of points. This conclusion is confirmed by optimization calculations. The finite element analysis made it possible to take into account almost all those features that occur when calculating leaf springs. Ample opportunities have opened up for optimizing the design, reducing its material consumption and cost.

4.2. Optimization of leaf spring parameters. Analytical calculation of the number of spring leaves

The methodology proposed earlier to design a symmetrical spring of an off-road passenger car, with several semi-elliptical leaves is used. The working load is F = 10000 H. The distance between the end sections (ears) of the straightened spring is 1252 mm, the deflection under the working load is 180 mm. The total moment of inertia of the spring section is:

$$J = \eta \frac{Pl^3}{48Ef} = 1{,}35 \frac{10 \cdot 1{,}252^3}{48 \cdot 2{,}1 \cdot 10^8 \cdot 180 \cdot 10^{-3}} = . \quad (1)$$

$$= 1{,}08 \cdot 10^{-8} \ m^4 = 1{,}08 \ cm^4.$$

The sections of the spring leaves are different, in this regard, the moment of inertia of the main leaf with a section of 10 x 70 mm will be found:

$$J_1 = \frac{bh^3}{12} = \frac{7 \cdot 1^3}{12} = 0{,}5833 \ cm^4. \quad (2)$$

The moment of inertia of leaves with a cross section *6,5* x 70 *mm* is determined:

$$J_2 = \frac{bh^3}{12} = \frac{7 \cdot 0{,}65^3}{12} = 0{,}16 \ cm^4. \quad (3)$$

The moment of inertia of the package, excluding the main leaf is determined:

$$J - J_1 = 1{,}08 - 0{,}5833 = 0{,}497 \ cm^4. \quad (4)$$

In the case of the thickness of all remaining spring leaves, the required number of leaves is equal to:

$$m = \frac{J}{J_1} = \frac{0{,}497}{0{,}16} = 3{,}1 \quad (5)$$

Accepted *m* = 3.

That is, taking into account the main leaf, the designed spring should consist of four leaves. Note that the actually existing spring of an off-road vehicle consists of 5 leaves.

4.3. Spring Optimization in SolidWorks

SolidWorks software has the ability not only to calculate the structure, but also to optimize it. The purpose of the optimization calculation is to obtain the minimum values of such quantities as weight, surface area, volume, stress, natural frequencies, etc. It is necessary to distinguish between methods and means of optimization. Optimization methods optimize the objective function by varying the input parameters. Optimization tools provide one or more sets of initial values (objective function, state variables); when the input parameters change according to a given law, they do not optimize the objective function. Before optimization, it is necessary to perform a static calculation. To do this, a volumetric model of a leaf spring directly in the SolidWorks package is modelled (Fig. 14).

Fig. 14. Volumetric model of the vehicle spring

When solving optimization problems in SolidWorks, the following algorithm is used:

1. Creation of a geometric model of SolidWorks. The dimensional system is formed so that the geometric dimensions of the leaves are controlled by design parameters. The possibility of building a model when changing each of these parameters is checked.
2. Creation of a computational model of SolidWorks modelling, with the specification of fixings and loads necessary for the conditions.
3. Meshing and static calculation of the spring.
4. The variables on the basis of which the optimization of the research object will occur are set.
5. Let us set a limitation, in this problem the maximum allowable stress in the spring leaves should not exceed 600 MPa.
6. The aim of optimization is to reduce the weight of the structure.
7. As a result of optimization, a four-leaf spring was obtained with a 10 mm thick main leaf, a thickened leaf of 7 mm width, as well as two leaves with the same characteristic sections, shown on Fig. 15.

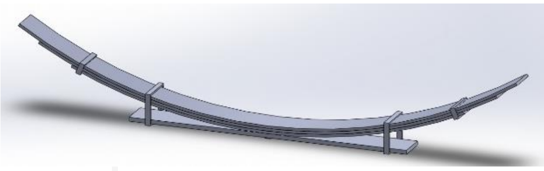

Fig. 15. Leaf spring model resulting from optimization

To confirm the reliability of the design obtained, this spring with the same fixings and loads will be calculated. The finite element model of a leaf spring is shown in Fig. 16.

Fig. 17 shows the distribution of equivalent stresses according to the Huber-Mises strength hypothesis. The maximum stresses are 598.7 MPa.

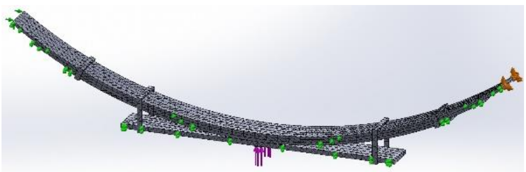

Fig. 16. Finite element model of a leaf spring

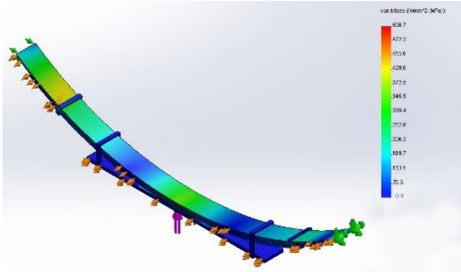

Fig. 17. Equivalent stresses in the contact problem

The maximum displacements were 7.972 mm (Fig. 18).

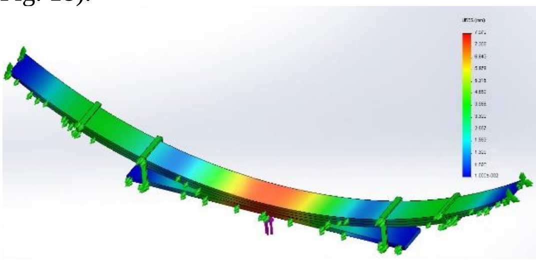

Fig. 18. Total displacements

5. CONCLUSIONS AND RECOMMENDATIONS FOR DIAGNOSTICS OF THE STRESS-STRAINED STATE OF DAMPING DEVICES OF VEHICLES

The calculation based on the hypothesis of the transfer of forces at the ends gives the results that are closer to reality than the calculation based on the hypothesis of equal curvature of the leaves, which gives underestimated values of stresses in the extreme short leaves. In view of this, it is recommended to apply a method based on the hypothesis of the transfer of forces at the ends of the leaves for a refined verification calculation of the spring, and for design calculations - a simplified method based on the assumption of the equality of the curvature of the leaves with corrections to the stress values in short leaves.

The exact solution of the problem of the load distribution between the leaves of the spring is difficult, since strain leads to contact forces of interaction applied at a number of isolated points. In this regard, the role of modelling the spring structure using modern computer programs and its calculation by numerous methods increases significantly, of which the most effective today is the finite element method, which is the basis of two modern programs ANSYS and SolidWorks.

Taking into account the contact interaction of the spring leaves leads to a change in the parameters of the stress-strain state. This is fully confirmed by the calculation of the off-road light motor vehicle spring, carried out taking into account the contact stresses in the SolidWorks package.

The studies have shown that the generally accepted model of contact interaction of leaves, according to which the contact is carried out along the entire line of their communication, does not correspond to reality, the real contact between the leaves occurs at a number of points.

Spring optimization in the SolidWorks complex resulted in a four-leaf spring, the stresses in which do not exceed the permissible values. This calculation is fully confirmed by comparing the results of the analytical determination of the number of spring leaves and the computer calculation by the finite element method.

Author contributions: *research concept and design, A.B., O.N., O.L., V.P.; Collection and/or assembly of data, A.B., O.L., V.K.; Data analysis and interpretation, O.L., V.P., V.K.; Writing the article, A.B., O.L., V.P.; Critical revision of the article, A.B., O.N.; Final approval of the article, O.N., V.P.*

Declaration of competing interest: *The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.*

REFERENCES

1. Bondar VN, Ashikhmina LA, Berezin IY, Khalturin VK. Directions of applied research in the field of improving the structures of road-building equipment. Actual problems of automobile, railway and pipeline transport in the Ural region 2005; 127(12): 243-249. Russian.
2. Staroverov NN. Modeling of viscoelastic properties and hysteresis damping of springs from composite materials. Mechanical engineering 2011; 8: 24-33. Russian.
3. Potapov ML. On the question of designing springs with a progressive elastic characteristic. Dynamics and strength of transport machines 1998; 7: 40-47. Russian.
4. Banerjee AK. Block-diagonal equations for multibody elasto-dynamics with geometric stiffness and constraints. Journal of Guidance, Control, and Dynamics 1993; 275(5): 1092-1100.
5. Dias MP, Pereira MS. Sensitivity analysis of rigid-flexible multibody systems. Multibody System Dynamics 1997; 3(1): 301-307.
6. Pedersen NL. On the formulation of flexible multibody systems with constant mass matrices. Multibody System Dynamics 1997; 3(1): 227-231.
7. Piterska V, Nemchuk O, Orobey V, et al. Diagnostics of the strength and stiffness of the loader carrier system structural elements in terms of thinning of walls by numerical methods. Diagnostyka 2021; 22(3):73-81.http://dx.doi.org/10.29354/diag/141313.
8. Orobey V, Nemchuk O, Lymarenko O, Piterska V, Lohinova L. Taking account of the shift and inertia of rotation in problems of diagnostics of the spectra of critical forces mechanical systems. Diagnostyka 2021;22(1):39-44. http://dx.doi.org/10.29354/diag/132555.
9. Nemchuk OO. Specific features of the diagnostics of technical state of steels of the port reloading equipment. Materials Science 2018; 53(6): 875-878. http://dx.doi.org/10.1007/S11003-018-0148-5.
10. Nemchuk O, Hredil M, Pustovoy V, Nesterov O. Role of in-service conditions in operational degradation of mechanical properties of portal cranes steel. Procedia Structural Integrity 2019;16: 245-251. http://dx.doi.org/10.1016/j.prostr.2019.07.048.
11. Nemchuk OO Influence of the working loads on the corrosion resistance of steel of a marine harbor crane. Materials Science 2019; 54(5): 743-747. https://doi.org/10.1007/S11003-019-00241-Y.
12. Ivankevich A, Piterska V, Shakhov A, Shakhov V, Yarovenko V. A Proactive Strategy of Ship Maintenance Operations. 2019 IEEE 14th International Conference on Computer Sciences and Information Technologies (CSIT) 2019: 126-129. http://dx.doi.org/10.1109/STC-CSIT.2019.8929741.
13. Nemchuk OO, Krechkovska HV. Fractographic substantiation of the loss of resistance to brittle fracture of steel after operation in the marine gantry crane elements. Metallofizika i Noveishie Tekhnologii 2019;41(6):825836. https://doi.org/10.15407/mfint.41.06.0825.
14. Shakhov A, Piterska V, Sherstiuk O, Rossomakha O, Rzheuskyi A. Management of the Technical System Operation Based on Forecasting its "Aging". Proceedings of the 1st ITPM 2020, CEUR Workshop Proceedings 2020; 2565: 130-141.
15. Varbanets RA, Zalozh VI, Shakhov AV, Savelieva IV, Piterska VM. Determination of top dead centre location based on the marine diesel engine indicator diagram analysis. Diagnostyka 2020; 21(1): 51-60. http://dx.doi.org/10.29354/diag/116585.
16. Nemchuk OO, Nesterov OA. In-service brittle fracture resistance degradation of steel in a ship-to-shore portal crane. Strength of Materials 2020; 52(2): 275–280. https://doi.org/10.1007/s11223-020-00175-w.

Received 2022-01-13
Accepted 2022-03-11
Available online 2022-03-14

Anastasiia BAZHANOVA – PhD (Engineering), Odessa Polytechnic National University, Associate Professor at the Department of Chair Dynamics, Capacity of Machines and Resistance of Materials, Odessa, Ukraine.

Oleksii NEMCHUK – Doctor of Sciences (Engineering), Docent, Odessa National Maritime University, Vice-Rector of Science, Odessa, Ukraine.

Oleksandr LYMARENKO – PhD (Engineering), Docent, Odessa Polytechnic National University, Head of the Department of Chair Dynamics, Capacity of Machines and Resistance of Materials, Odessa, Ukraine.

Varvara PITERSKA – Doctor of Sciences (Engineering), Professor, Odessa National Maritime University, Professor at the Department of Port Operation and Cargo Handling Technology, Odessa, Ukraine.

Olha SHERSTIUK – PhD (Engineering), Docent, Odessa National Maritime University, Associate Professor at the Department of Philology, Odessa, Ukraine.

Vadym KHAMRAI – Senior Lecturer of the Department of Dynamics, Machines Strenght and Material Resistance, Odessa National Polytechnic University, Odessa, Ukraine.