DOI 10.34185/1562-9945-4-159-2025-05
UDK 004.94

O.Yu. Rudyk, M.M. Poberezhnyi, P.V. Kaplun, V.A. Gonchar

# APPLICATION OF COMPUTER-INTEGRATED TECHNOLOGIES IN MODERN ENGINEERING SYSTEMS

*Abstract. In modern conditions of development, analysis of the strength of parts is impossible without the comprehensive use of modern computer-aided design systems, one of which is the SolidWorks CAD/CAE system and its application – SolidWorks Simulation. This work determines the dependence of the maximum stress acting on the gear block of the drum of the UVN-74 vacuum spraying installation at full load on the number of finite elements of its model. The first stage in analysing the stress-strain state of a gear blocking the SolidWorks Simulation environment is to select the type of study (static analysis) and its material. The next stages are the selection of attachment points (fixed geometry), the application of external loads, and the creation of a finite element model of the gear block. To solve the static problem, the parameter "at nodes" is set for the Jacobian check, and to obtain reliable results, the "curvature-based" mesh creation algorithm is activated. To avoid errors related to mesh density, the optimal number and size of finite elements were selected. SolidWorks Simulation software, taking into account the connections between the elements of the created mesh, created algebraic equations that relate the reaction to the material property of the gear block, the constraint, and the loads. After arranging the equations into a common system, diagrams of stresses, displacements, deformations and safety margin were created. Analysis of the stress diagram established that this loading mode occurs within permissible limits, but exceeding the total displacements by more than permissible ones can lead to unpredictable results during the operation of the gear block. To determine the exact values of the total displacements, their probing at critical points was used. The dependence of the maximum stress acting on the gear block at full load on the number of finite elements was determined.*

*Keywords: SolidWorks Simulation, MFE, FEM, gear block, maximum stress, number of finite elements.*

**Problem statement.** Modern Hi-Tech specialization combines classical engineering education with in-depth mastery of computer-integrated technologies and 3D modelling. In modern conditions of technological development, high-quality higher technical education is impossible without the comprehensive use of modern computer-aided design systems, one of which is the SolidWorks CAD/CAE system [1, 2]. Its solutions help in daily work by providing a convenient integrated three-dimensional design environment that covers all aspects of product development and helps increase the productivity of designing work [3].

---

Using the SolidWorks application – SolidWorks Simulation – for analysing the strength of parts has a number of advantages compared to traditional methods. In particular, allows for a comprehensive analysis, increases the accuracy of calculations, reduces the duration of the analysis, and allows for both design and verification calculations [4].

The analys is performed using the finite element method (MFE), which is currently the standard for solving problems in solid mechanics using numerical algorithms. MFE has taken a leading position due to its ability to model a wide range of objects and phenomena. In this case, it is necessary to take into account the errors inevitable in any numerical approximation. Therefore, the issue of correspondence between the calculation model and reality is the main one when using analysis programs. This means a certain unpredictability of the results and their interpretation.

The basic idea of MFE is that any continuous quantity can be approximated by a discrete model, which is built on a set of piecewise-continuous functions defined on a finite number of subdomains. These functions are defined using the values of the continuous quantity at a finite number of points in a given domain. In the general case, the continuous quantity is unknown in advance, and its value must be determined at some internal points of the domain. But a discrete model is easy to construct if we first assume that the numerical values of this quantity at each internal point of the domain are known. After that, we move on to the general case.

Therefore, when constructing a discrete model of continuous quantities, the following procedure is followed: a finite number of points (nodal points or nodes) are fixed in a given region; in the general case, the continuous quantity at each nodal point is taken as the variable to be determined; the domain of definition of a continuous quantity is divided into a finite number of subdomains (elements) that have common nodal points and collectively approximate the shape of the domain.

A continuous quantity is approximated at each element by a polynomial, which is defined using the nodal values of this quantity. A separate polynomial is defined for each element, but they are selected so that the continuity of the quantity along the boundaries of the element is preserved.

The calculation of the value of a continuous quantity at nodal points is described as follows:

– known coordinates of the element;

– node numbers in which the element is connected to other elements;

– material properties.

The information that allows obtaining a single solution to the problem and determining the global matrix is given in the form of boundary conditions and fixation conditions. The computational program, using data on the computational model, calculates the stiffness matrices of the elements and the loads on them, forms a vector of external loads and a global stiffness matrix, imposes specified constraints on the elements, and performs a triangular decomposition of the global matrix. Next, the program solves the system of linear equations and calculates the vector of linear displacements for the elements for which stresses, strains, displacements, and safety margins are calculated.

The set of elements into which the structure is divided is called a finite element mesh (FEM). Before it is generated on the model, the optimal values of the mesh resolution and the sizes of the finite elements (FE) are determined. The use of a mesh with a large resolution, consisting of large elements, contributes to a significant deviation of the shape of the FE model from the original geometry. A significant increase in the mesh density leads to a sharp increase in the number of FE; this leads to the fact that machine resources may not be enough to solve the problem. In addition, with the automatic method of generating FE, the location of element nodes often does not meet the necessary conditions [5, 6].

Controlling the size and density of the FEM, as well as manually specifying the number and location of element nodes, helps create a mesh with the optimal number and size of FE, which allows you to increase accuracy and reduce the time spent on calculations. The size of the generated mesh (number of nodes and elements) depends on various parameters. In the early stages of structural analysis, where approximate results may be appropriate, a larger element size is specified for faster solution. A smaller element size may be required for a more accurate solution [7].

For a solid mesh, the best accuracy is achieved with a mesh of identical ideal tetrahedral elements whose edges are equal in length. However, due to curved shapes, small edges, thin-walled elements, and sharp corners, the edge lengths vary. Therefore, it is impossible to create a mesh of ideal tetrahedral elements. And when the edges of an element differ in length, the accuracy of the results decreases: on very sharp or curved boundaries, placing middle nodes leads to the formation of distorted elements with intersecting edges. Therefore, to solve static problems, the parameter “at nodes” is set for the Jacobian check [8], and to obtain reliable results, the “curvature-based” mesh creation algorithm is activated [9].

Also, when analysing an object, errors may occur related to the mesh density, that is, it is necessary to choose the optimal number and size of FE.

**Analysis of recent research and publications.** The authors of [10] determined the dependence of the nodal stress on the number of FE of the earring model of the installation for electrocontact welding of powder materials.

The authors [8, 9] assessed the influence of mesh quality on the accuracy of calculations of the hydraulic distributor spool and the puller capture, using a high-quality mesh based on curvature, and for the Jacobian check, the parameter “in nodes” was set with the selection of the optimal number and size of the FE.

The authors [11] determined the grid parameters that determine the minimum permissible value of the safety factor ([n] = 1.5) of the driven gear of the gear unit of the drum of the UVN-74 vacuum spraying installation. To do this, they changed the maximum size of the grid elements and compared the experimental results: the minimum value of the safety factor, greater than the permissible one, corresponds to the maximum size of the mesh element of 5.54739077 mm. But in this work, the load is applied only to the driven gear – the forces acting on the driving gear are not taken into account.

**The purpose of the study.** It is necessary to determine the dependence of the maximum stress acting on the gear block at full load on the number of SE.

**Presentation of the main research material.** When modelling the gear unit in SolidWorks, its geometric model was first created. The first stage in analysing its stress-strain state in the SolidWorks Simulation environment was to select the type of research (static analysis) and the material (steel 1.4541 – X6CrNiTi18-10).

The next stage is the selection of mounting locations (fixed geometry) and the application of external loads to the gear unit (circular force: to the driving gear $Ft_6 = 367$ H, to the driven gear – $Ft_7 = 354$ H; radial force to the driving gear $Fr_6 = 134$ N, to the driven gear – $Fr_7 = 129$ N – Fig. 1).

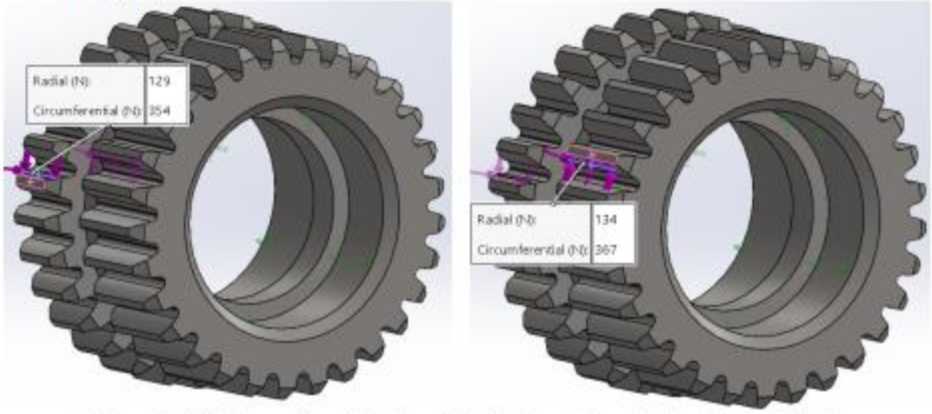
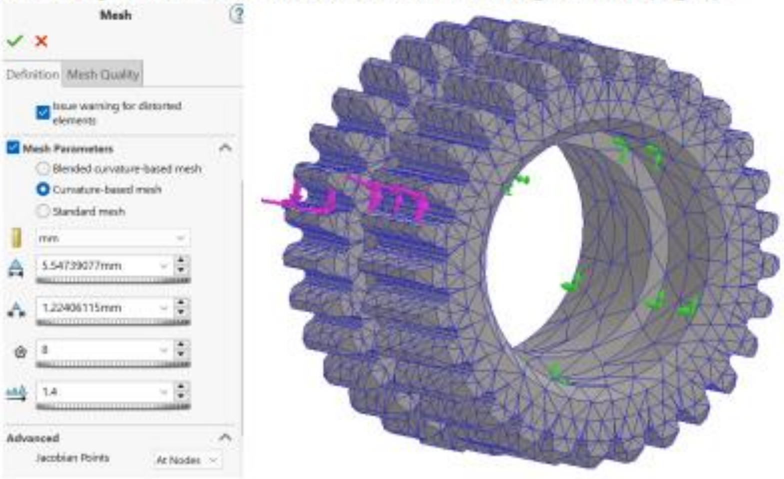


Figure 1 – Fixing and application of loads in static analysis of a gear block

The next step is to create a finite element model of the gear block (Fig. 2).



☑ Issue warning for distorted elements

☑ Mesh Parameters

○ Blended curvature-based mesh

● Curvature-based mesh

○ Standard mesh

mm

5.54739077mm

1.22406115mm

8

1.4

Advanced

Jacobian Points: At Nodes

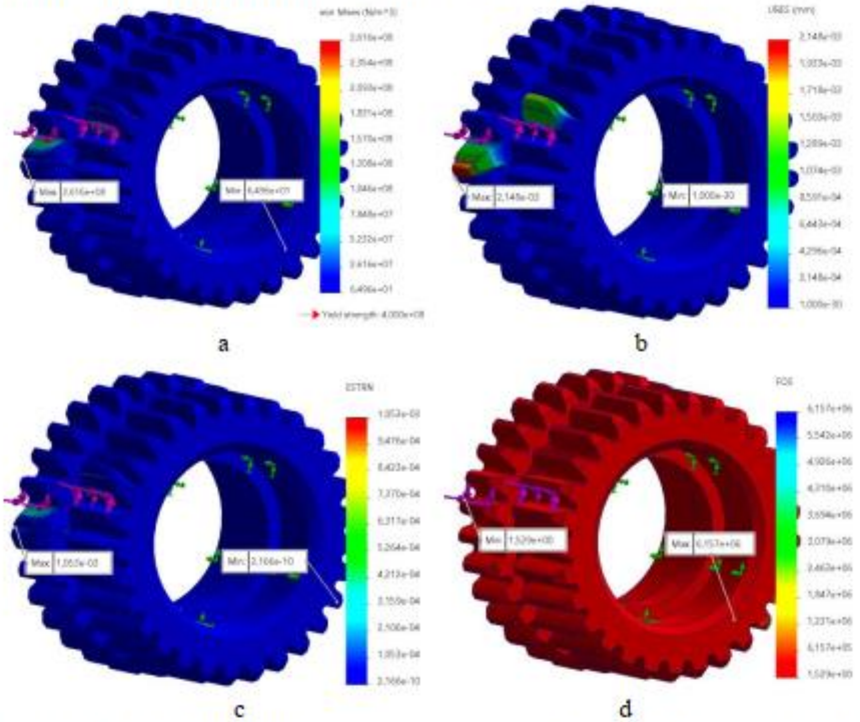
a b

Figure 2 – Parameters of the finite element mesh of the gear block (a)
and its reflection on the solid (b)

ISSN 1562-9945 (Print)
ISSN 2707-7977 (Online)

The instantaneously increasing magnitude of the force and moment of inertia causes a sharp increase in contact stress and bending in the gear pairs. In addition, the concentration of this stress occurs on a small area of contact between the gear teeth and the wheel. In this case, the main load is perceived by one tooth of the gear block.

SolidWorks Simulation software, taking into account the connections between the elements of the created mesh, creates algebraic equations. They relate the reaction to the material property of the gear block, the constraint and the loads [12]. After arranging the equations into a large common system, the unknowns are found (Fig. 3).



a b

c d

Figure 3 – Results of calculation of von Mises stresses (a), total displacements URES (b), total deformations ESTRN (c) and safety factor FOS (d) of the gear block

Fig. 3 shows the load distribution on the contact surface of the teeth: the colour scheme of the spectral diagram characterizes the contact area of the gear unit – the red colour indicates the meshing areas that operate with critical parameters. Since the red colour is absent on the total stress distribution diagram (Fig. 3, a), this loading mode occurs within permissible limits. But the red colour is present in Fig. 3, b. This indicates that the total displacements exceed the permissible ones, which can lead to unpredictable results during the operation of the gear block.

To determine the exact values of the total displacements, their probing at critical points was used (when probing the results plot, the software displays the node or element number, the value of the constructed result, and the global coordinates of the node or element centre (Fig. 4).

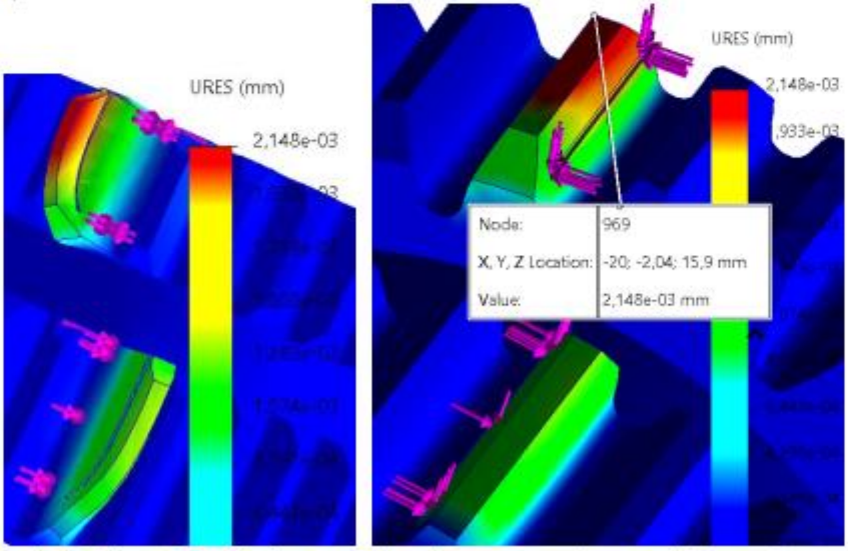


Figure 4 – Probing of total displacements in a dangerous section near their maximum values

The conducted research of the gear unit established: the maximum nodal von Mises stresses occur at node 22750 and are 261.6 MPa; the maximum resultant displacement URES occurs at node 969 and is 0.00215 mm; the maximum equivalent deformation ESTRN occurs at node 6109 and is 0.001053 mm. The experimental curve obtained as a result of changing the number of finite elements of the gear block model is presented in Fig. 5.

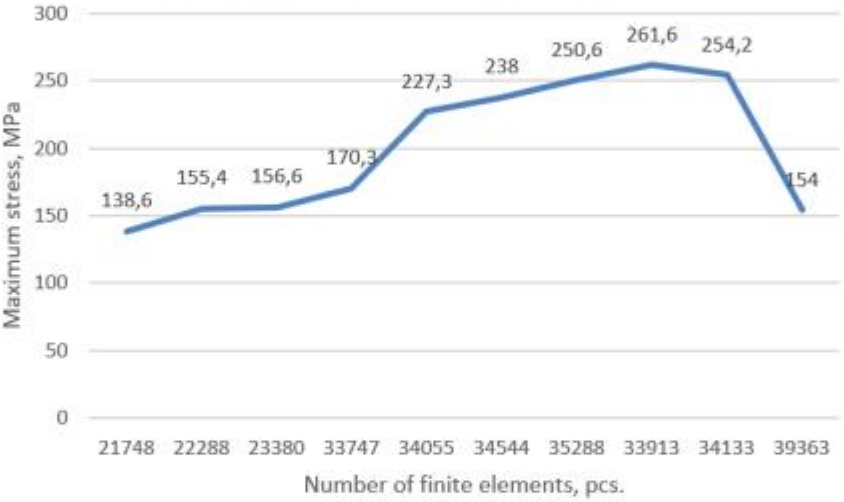


Figure 5 – Dependence of maximum stress on the number of finite elements gear block

Thus, since the constructed model of the gear block has a complex geometry, then:

– when constructing the mesh, an increase in the number of finite elements is required in places of large curvature and a significant change in the geometric characteristics of the elements of the connected structures;

– to construct a grid of optimal size, it is necessary to conduct a series of experiments, gradually increasing the number of finite elements and comparing the obtained research results;

– starting from a certain number of finite elements, the value of the maximum stress will change insignificantly, which will determine the optimal mesh size to obtain the minimum permissible value of the safety factor.

**Conclusions**. The dependence of the maximum stress acting on the gear unit at full load on the number of finite elements of the created model has been determined. But although the strength condition is satisfied, the total displacements are greater than the permissible ones, which can lead to unpredictable results during the operation of the gear block.

## LITERATURE/ REFERENCES

1. Rudyk, O.Yu., Dykha, O.V. (2020). SolidWorks as an innovative means for studying the disciplines of automobile profile. "System technologies" 3 (128), 21-35. DOI 10.34185/1562-9945-3-128-2020-03

2. Rudyk, O., Barnych, M., Humeniuk, O., Shchur, V. (2023). *SolidWorks as an innovative means of researching automotive engineering*. Current problems in the education system: general secondary education institution – pre-university training – higher education institution: collection of scientific works of the materials of the IX All-Ukrainian Scientific and Practical Conference, February 17, 2023, Kyiv, National Aviation University, 130-141. DOI 10.18372/2786-5487.1.17688

3. Rudyk, O.Yu., Kaplun P.V. (2020). *Integration of education, science and business based on the use of SolidWorks API.* Integration of Education, Science and Business in the Modern Environment: Winter Debates: abstracts of the 1st International Scientific and Practical Internet Conference, February 6-7 2020. Dnipro: WayScience, P.3, 127-131, URL: http://www.wayscience.com/wp-content/uploads/2020/02/TOM-3-Zbirnik-1-mizhnarodna-nauk-prakt-int.-konf-Winter-Debates-1.pdf

4. Rudyk, O., Kovalenko, R., Lavrenchuk, A., Hromov, D. (2024). *Application of innovative technologies when researching the performance of road transport*. Electronic International Scientific Journal «Actual problems in the system of education: general secondary education institution – pre-university training – higher education institution» (ISSN 2786-5487 online), 4, 78-90. DOI 10.18372/2786-5487.1.18723

5. Rudyk, O.Yu., Mariin, N.S., Yaroshchuk, S.I. (2020). *Application of SolidWorks in the educational process.* Proceedings of the IV All-Ukrainian scientific-practical Internet conference "Resource-oriented learning in "3D": accessibility, dialogue, dynamics ", incl. N. V. Kononets, V. O. Balyuk, Poltava: KUEP PDAA, 2020, 96-100, URL: http://acup.poltava.ua/wp-content/uploads/2020/02/Збірник-2020-0203.pdf

6. Rudyk, O., Kaplun, P., Gonchar, V. (2022). *Application of SolidWorks for the training of highly qualified specialists*. Current problems in the education system: general secondary ed-

ucation institution – pre-university training – higher education institution: collection of scientific works of the materials of the VIII All-Ukrainian Scientific and Practical Conference, February 17, 2022, Kyiv, National Aviation University. Kyiv: NAU, 699-706. DOI 10.18372/2786-5487.1.16657

7. Rudyk, O.Yu., Kaplun, P.V. (2020). *SolidWorks – CAD/CAE system of technical universities*. Science, society, education: topical issues and development prospects. Abstracts of the 2nd International scientific and practical conference. SPC "Sci-conf.com.ua". Kharkiv, 249-253, URL: http://sci-conf.com.ua/ii-mezhdunarodnaya-nauchno-prakticheskaya-konferencziya-science-society-education-topical-issues-and-development-prospects-20-21-yanvarya-2020-goda-harkov-ukraina-arhiv/

8. Rudyk, O.Yu., Gonchar, V.A. (2020). *The impact of the SolidWorks Simulation network quality on the accuracy of the calculations*. Eurasian scientific congress. Abstracts of the 1st International scientific and practical conference. Barca Academy Publishing. – Barcelona, Spain, 185-188, URL: http://sci-conf.com.ua/i-mezhdunarodnaya-nauchno-prakticheskaya-konferencziya-eurasian-scientific-congress-27-28-yanvarya-2020-goda-barselona-ispaniya-arhiv/

9. Dykha, O.V., Rudyk, O.Yu. (2021). *Application of SolidWorks Simulation for training automotive specialists*. Collection of abstracts of the II International Scientific and Technical Conference "Prospects for the Development of Mechanical Engineering and Transport – 2021", Collection of abstracts, Vinnytsia, VNTU, 484-485,
URL: http://elar.khmu.km.ua/jspui/handle/123456789/10992

10. Traskovetska, L.M., Rudyk, O.Yu., Biryukov, O.V. (2019). *Mathematical modelling of the performance of parts by the method of finite elements*. Informatics, information systems and technologies: abstracts of reports of the sixteenth all-Ukrainian conference of students and young scientists. Odesa, April 19, 2019, ONU, 96-97,
URL: http://elar.khmu.km.ua/jspui/handle/123456789/8467

11. Rudyk, O.Yu., Poberezhnyi, M.M., Kaplun, P.V., Gonchar, V.A. (2025). *Application of modern educational resources for calculations of details*. Resource-oriented learning in "3D": accessibility, dialogue, dynamics: collection of abstracts of the V International Scientific and Practical Internet Conference (Poltava, February 20–21, 2025), Poltava, PNPU, 84-87, URL: https://drive.google.com/drive/u/0/folders/1lB7BAXWSPHjrbKeCAy8DW4DDUf0PtGCs

12. Psol, S., Dykha, O., Rudyk, O., Golenko, K. (2023). Application of computer modeling for road transport calculations. Collection of scientific works of the National Academy of the State Border Service of Ukraine. Series: pedagogical sciences, CE. O. V. Didenko. Khmelnytskyi: NADPSU Publishing House, 1(32), 148-170.
DOI 10.32453/pedzbirnyk.v32i1.1322

Received 30.04.2025.
Accepted 02.05.2025.

## *Застосування комп'ютерно-інтегрованих технологій у сучасних інженерних системах*

*За допомогою SolidWorks Simulation визначена залежність максимального напруження, діючого на блок шестерень барабана установки вакуумного напилення УВН-74 при повному навантаженні, від числа скінченних елементів його моделі. Для розв'язку статичної задачі для Якобієвої перевірки встановлено параметр "у вузлах", а для отримання достовірних результатів активізовано алгоритм створення сітки "на основі кривизни". Для уникнення похибок, пов'язаних зі щільністю сітки, підібрано оптимальне число і розмір скінченних елементів. Аналізом епюри напружень встановлено – даний режим навантаження протікає у допустимих межах, але перевищення сумарних переміщень може призвести до непередбачуваних результатів при експлуатації блоку шестерень. Щоб визначити точні величини сумарних переміщень, використано їх зондування у критичних точках.*

**Рудик Олександр Юхимович** – доцент, кандидат технічних наук, кафедра «Трибології, автомобілів та матеріалознавства», Хмельницький національний університет.

**Побережний Михайло Михайлович** – доцент, к.т.н., кафедра «Трибології, автомобілів та матеріалознавства», Хмельницький національний університет.

**Каплун Павло Віталійович** – професор, доктор технічних наук, кафедра «Трибології, автомобілів та матеріалознавства», Хмельницький національний університет.

**Гончар Володимир Антонович** – доцент, кандидат технічних наук, кафедра «Трибології, автомобілів та матеріалознавства», Хмельницький національний університет.

**Rudyk Oleksandr Yuhymovych** – Ph.D., Associate Professors of Department of Tribology, Automobiles and Materials Science of Khmelnytskyi National University.

**Poberezhnyi Mykhailo Mykhailovych** – PhD student, of Department of Tribology, Automobiles and Materials Science of Khmelnytskyi National University.

**Kaplun Pavlo Vitaliyovych** – doctor of technical sciences, professor of Department of Tribology, Automobiles and Materials Science of Khmelnytskyi National University.

**Gonchar Volodymyr Antonovych** – Ph.D., Associate Professors of Department of Tribology, Automobiles and Materials Science of Khmelnytskyi National University.