UDC 629.113

APPLICATION OF ICT FOR THE DEVELOPMENT OF THE DIAGNOSTIC STAND OF GEAR-BOXES AND ANCHORMAN BRIDGES OF VEHICLES

Borovyk Oleh Vasyliovych,

Doctor of engineering sciences, Professor

Rudyk Oleksandr Yuhymovych,

Ph.D., Associate Professors

Pazii Maxym Olexandrovych

undergraduate

Khmelnytsky National University

Khmelnitsky, Ukraine

yuhymovych@gmail.com

Abstract: the possibility of using ICT in the learning process is determined on the basis of the application of the CAD/CAE system SolidWorks Simulation finite element method for calculating the shaft of the diagnostic stand of the transmission boxes and driving bridges of vehicles.

Keywords: SolidWorks Simulation, finite element method, diagnostic stand, shaft.

Professional modern engineering activities are inevitably linked to the use of systems of automatic design, production and engineering analysis, and the use of Information and Communication Technologies (ICT) in the educational process increases the ability to set learning tasks and control the process of their implementation. ICTs involve students in the learning process and contribute to the widespread disclosure of their abilities and the activation of mental activity.

The possibilities of using ICT in the educational process depend on a set of the following conditions:

- the teacher must not only have knowledge of ICT, but also be a specialist in their application in their professional activity;
- students should be prepared for the perception of an "electronic" learning path.

The purpose of the study is the static exploration of the shaft of the stand diagnostics of gearboxes and drive axles of vehicles (fig. 1, item 1) on the basis of CAD/CAE systems and finite element method (FEM). The design of the stand of diagnostics with the calculation of bearing support is given in [1]. Continuation of the study – the calculation of the shaft of a tension device of a chain transmission [2]. But the stand may not work due to insufficient mechanical characteristics of its other details.

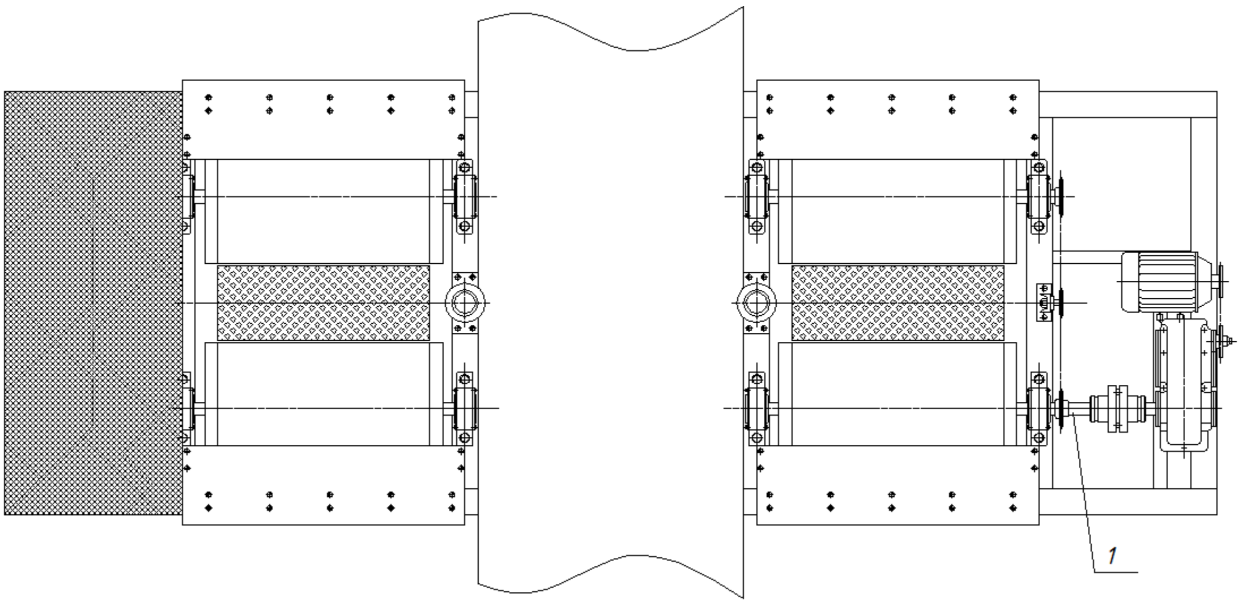


Fig. 1. General view of the diagnostic stand of gear-boxes and anchorman bridges of vehicles

The variety and complexity of the geometric shapes of automotive parts and their accessories require the use of numerical methods such as FEMs to analyze the stress-strain state of parts. It is now widely recognized as a common method of solving a wide range of problems in various fields of technology.

The essence of FEM is to approximate a continuous medium with an infinitely large number of degrees of freedom by a set of subregions (or elements) having a finite number of degrees of freedom [3]. There is a relationship between these elements. The recognition of the method is explained by the simplicity of its physical interpretation and mathematical form.

Finite Elements Analysis is widely used in solving the problems of deformed body mechanics, heat transfer, hydrodynamics, magnetostatics and other fields of physics. Software systems that implement FEMs allow with sufficient accuracy and efficiency to evaluate the load-bearing capacity of a workpiece: the ability of the design to withstand maximum load and to ensure the operation of a product with specified specifications.

In our study, they used the CAE system SolidWorks Simulation [4]. Orientation to its joint use with SolidWorks CAD system allows to maximize the benefits of the graphical environment: powerful parameterization, surface and solid geometric modeling, kinematics of assemblies. An important advantage of using the CAE system is the ability to obtain the results of static studies of the bearing structure of the workpiece (in this case, the shaft) at the stage of sketch design (technical proposal).

SolidWorks uses the principle of three-dimensional solid state and surface parametric design. This allows the designer to create bulky parts and build assemblies into three-dimensional models. They create two-dimensional drawings and specifications in accordance with the requirements of ESKD.

Finite element analysis provides a reliable numerical method of development. The process begins with the creation of a geometric model of the shaft (fig. 2).

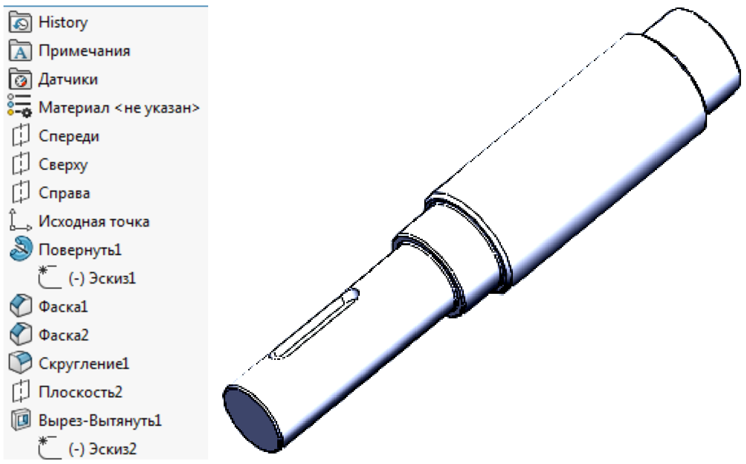
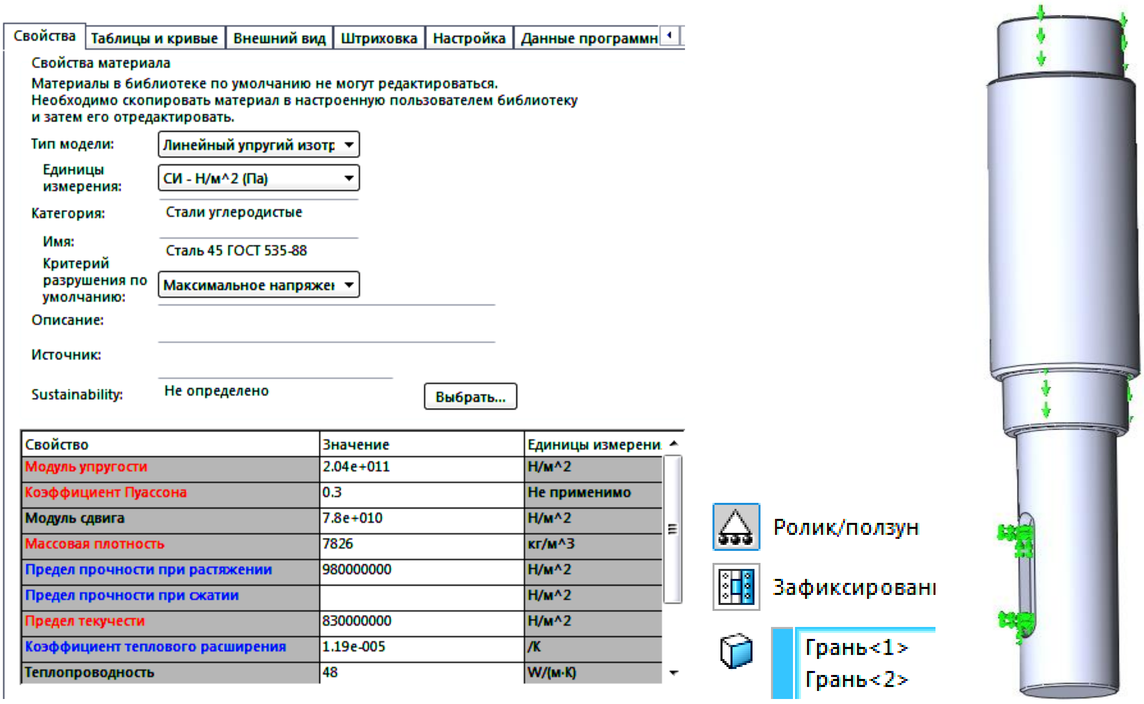


Fig. 2. Geometric model of the shaft

The next step is to select the shaft material (GOST 535-88 steel 45) from the SolidWorks library (fig. 3, a).



a b

Fig. 3. Selection of shaft material (a) and fixing of model (b)

To carry out the static analysis, the model was fixed (fig. 3, b). and load application (fig. 4).

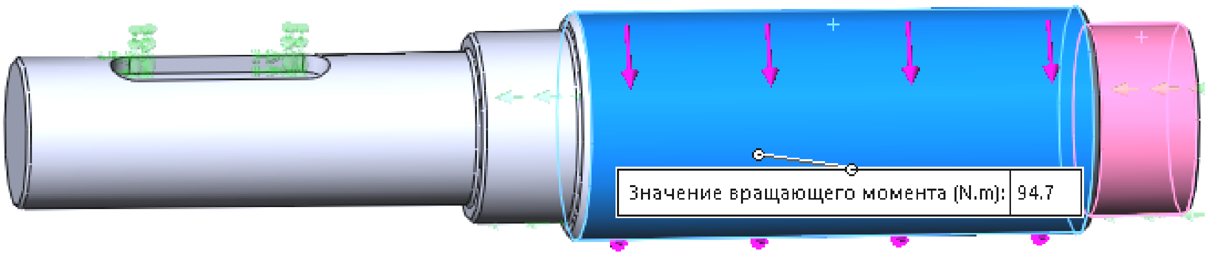


Fig. 4. Apply load to the shaft

The program then divides the model into small parts of a simple form (elements) connected at common points (nodes): the finite element analysis program views the model as a network of discrete connected elements (mesh). FEM predicts the behavior of a model by comparing the information obtained from all the elements that make up the model.

Creating a grid is a very important step in the analysis of structures. The grid is created based on the global element size, tolerance, and local grid management characteristics. It allows you to specify different element sizes for components, faces, edges, and vertices.

The program determines the size of the element for the model. This takes into account its volume, surface area and other geometric characteristics. The size of the created grid (number of nodes and elements) depends on the geometry and dimensions of the model, the tolerance of the grid, its control parameters and contact characteristics.

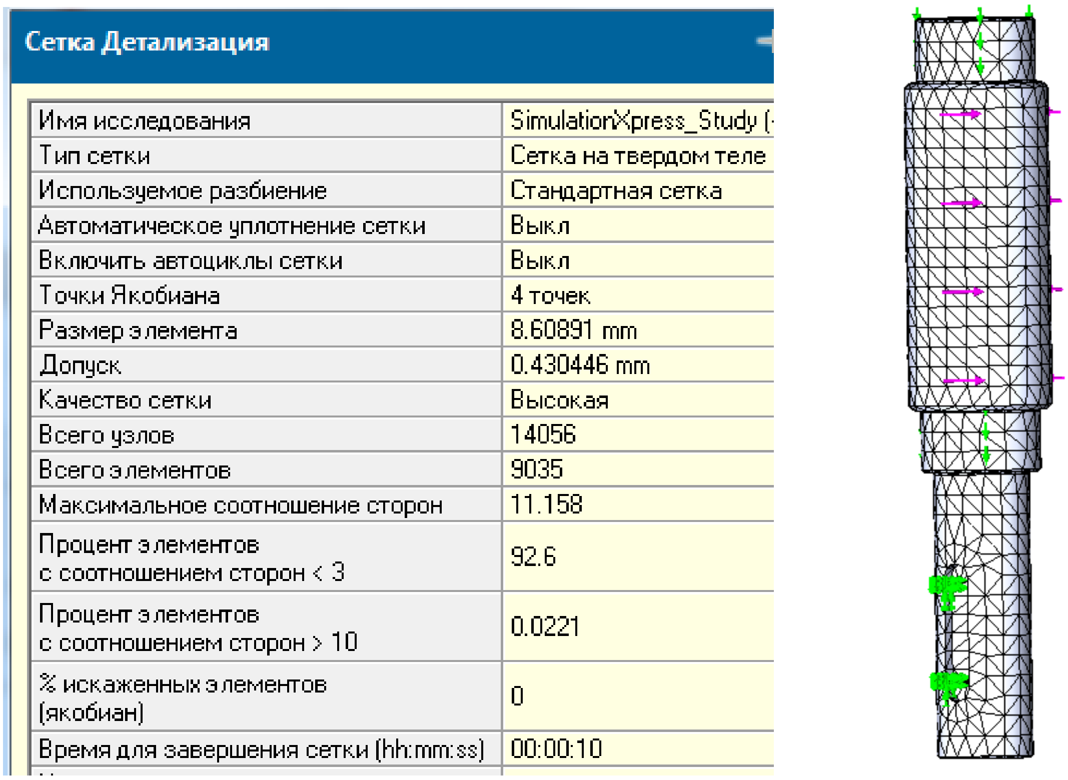
In the early stages of structural analysis, where approximate results may be appropriate, a larger element size can be set for faster resolution (smaller for more accurate solution).

When using shell elements, the program creates one of the following element types, depending on what grid creation options are enabled for the study:

– low quality mesh (linear triangular shell elements);

– high quality mesh (parabolic triangular shell elements).

In our study, the grid parameters are shown in fig. 5.



Сетка Детализация

Имя исследования	SimulationXpress_Study (-
Тип сетки	Сетка на твердом теле
Используемое разбиение	Стандартная сетка
Автоматическое уплотнение сетки	Выкл
Включить автоциклы сетки	Выкл
Точки Якобиана	4 точек
Размер элемента	8.60891 mm
Допуск	0.430446 mm
Качество сетки	Высокая
Всего узлов	14056
Всего элементов	9035
Максимальное соотношение сторон	11.158
Процент элементов с соотношением сторон < 3	92.6
Процент элементов с соотношением сторон > 10	0.0221
% искаженных элементов (якобиан)	0
Время для завершения сетки (hh:mm:ss)	00:00:10

a b

Fig. 5. Parameters of a finite element grid of a shaft and its reflection on a solid body

The results of the calculations are shown as a color gradient. It shows the color distribution of the calculated parameters: stresses in the model (fig. 6), its displacements (fig. 7), deformation, strength reserve.

The maximum Von Mises node stresses, URES displacements and ESTRN equivalent deformation are found to be 285.945 MPa, 0.131668 mm, and 0.000735392, respectively, ie, they do not exceed permissible values. The minimum factor of safety margin of FOS is 2,903: the margin of safety is sufficient.

The study of the stress-strain state of the shaft of the diagnostic stand of gear-boxes and anchorman bridges of vehicles with the help of FEM provides a qualitatively new approach to the determination of its basic parameters. The developed models and methodical positions of calculation allow to pass to an estimation of load of elements of a shaft in different calculation positions.

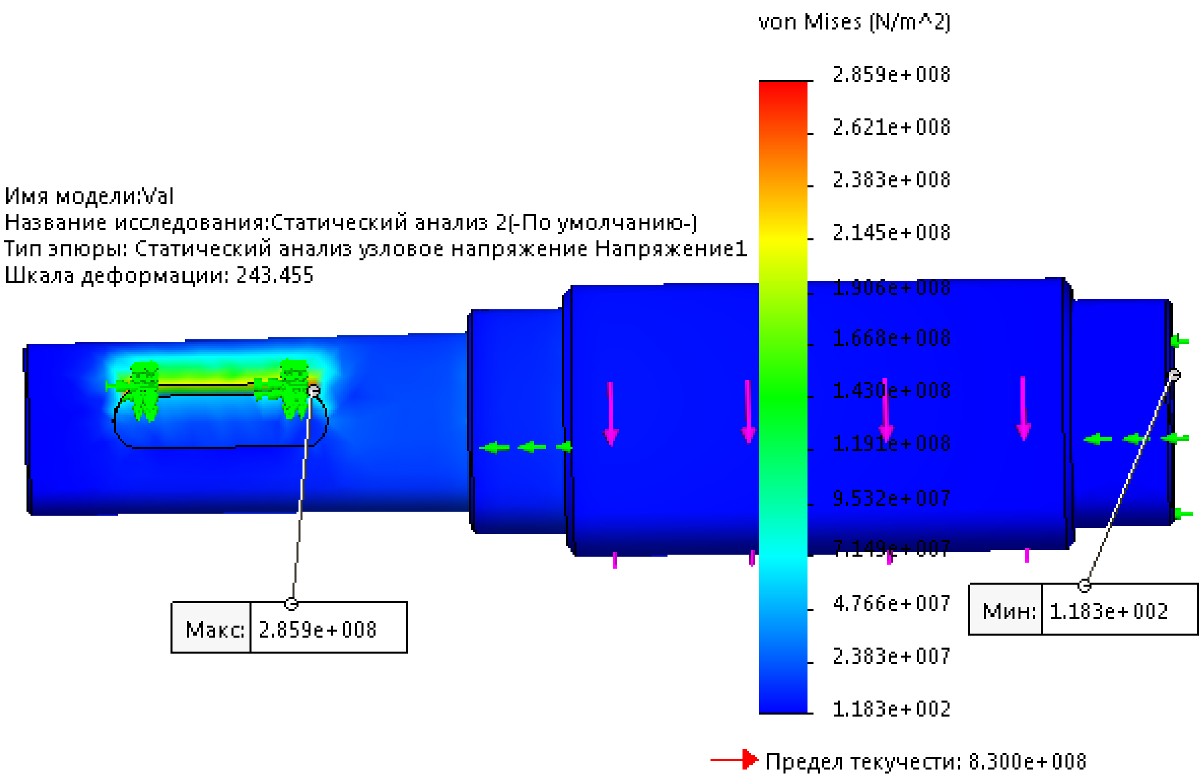


Fig. 6. Outline graph of total von Mises stresses

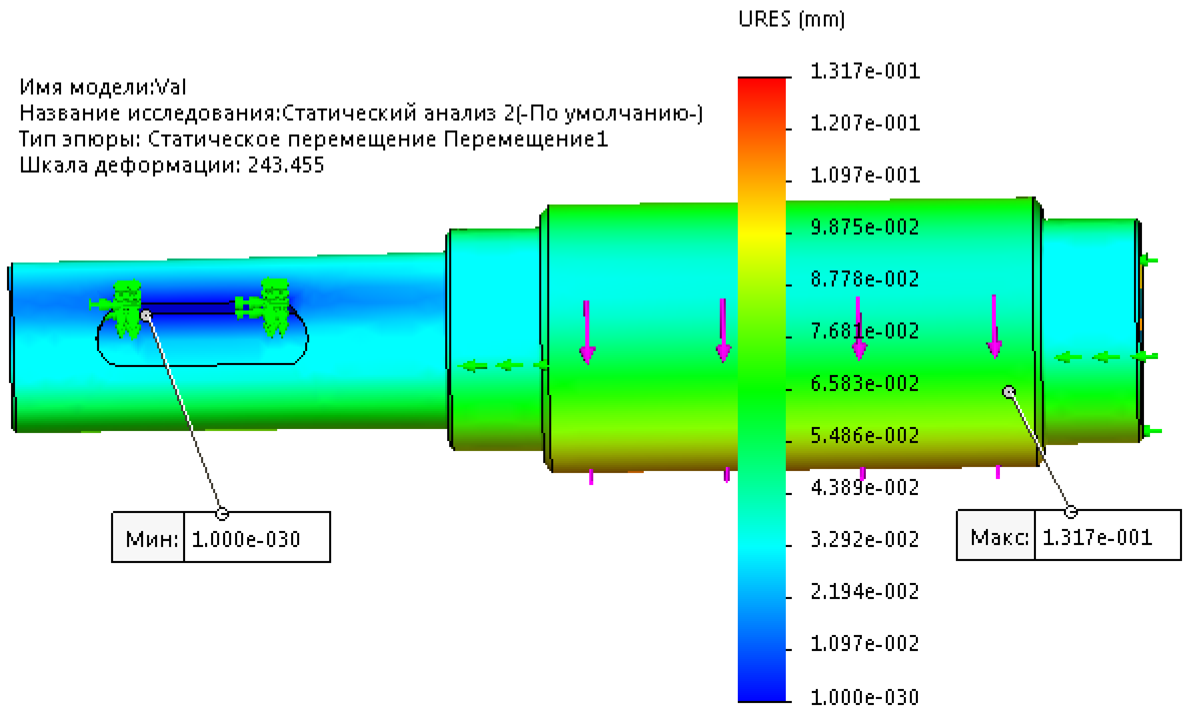


Fig. 7. Outline graph of total displacements of URES

REFERENCES

1. Рачок Р. В. Дослідження працездатності стенду діагностики автомобільної техніки за допомогою SolidWorks [Електронний ресурс] / Р. В. Рачок, О. Ю. Рудик, В. С. Єрмаков. – Режим доступу: http://elar.khnu.km.ua/jspui/handle/123456789/8431

2. Трасковецька Л. М. Застосування інформаційно-комп’ютерних технологій для міцнісних розрахунків [Електронний ресурс] / Л. М. Трасковецька, О. Ю. Рудик, С. П. Мазур. – Режим доступу: http://elar.khnu.km.ua/jspui/handle/123456789/8466

3. Rudyk O. Y. The impact of the SolidWorks Simulation network quality on the accuracy of the calculations [Electronic resource] / O. Y. Rudyk, V. A. Gonchar. – Access mode: http://elar.khnu.km.ua/jspui/handle/123456789/8681

4. Рудик О. Ю. SolidWorks – CAD/CAE-система технічних вузів [Електронний ресурс] / О. Ю. Рудик, П. В. Каплун. – Режим доступу: http://elar.khnu.km.ua/jspui/handle/123456789/8631