УДК 674.053:621.933.6 DOI 10.36910/775.24153966.2026.85.25

Д.Е. Селезньов[0000-0002-7729-9851], Е.Л. Селезньов[0000-0001-6470-0661], С.Ю. Грицюк

РЕЗУЛЬТАТИ ДОСЛІДЖЕНЬ НАПРУЖЕНОГО СТАНУ ВАЛА КРИВОШИПА RV РЕДУКТОРА

Луцький національний технічний університет

В роботі проведено дослідження статичного напруження стану валу кривошипу та розмір граничного значення напруження плинності, що необхідно для вдосконалення конструкції та матеріалу валу кривошипу.

Ключові слова: *вал кривошипа, статичне напруження, RV-редуктор, напружений стан, границя плинності матеріалу, максимальне напруження, результати моделювання.*

D.E. Seleznov, E.L. Seleznov, S.Y. Hrytsiuk

RESULTS OF RESEARCH ON THE STRESS STATE OF THE CRANKSHAFT OF THE RV GEARBOX

The paper presents research on the static stress state of a crankshaft and the limit value of yield stress, which is necessary for improving the design and material of a crankshaft.

Keywords: *crankshaft, static stress, RV-reducer, stress state, yield strength of material, maximum stress, simulation results.*

Introduction. The design of the RV reducer includes a whole process from determining geometric parameters, design to developing technological tolerances for dimensions. Due to the interaction of both stages of the gear speed switching mechanism, when choosing their geometric parameters, it is necessary to take into account the mutual limitation between them, and the impact on the transmission characteristics of the reducer and the accuracy of processing and manufacturing of the reducer. However, the RV reducer contains many key parts and assemblies whose design affects the characteristics of the reducer. The RV reducer has a two-stage transmission: the first stage is a conventional spur gear 2K-H, the second stage is a cycloidal KHV. The crankshaft is a key part, as it connects the two stages, and also supports and transmits motion to the two cycloidal wheels. Since a key connection is used between the crank shaft and the planetary wheel, the force perceived by the planetary wheel is transmitted to the crank shaft. All this requires targeted research to develop parameters and design the RV reducer. To ensure the accuracy of the calculation, the work uses the equivalent mode analysis method simultaneously with a simplified analysis of the perceived effort.

During continuous testing of the RV gearbox under a given load, the crankshaft experienced varying degrees of fracture. Therefore, in order to analyze the stress state of the crankshaft under a given load, and to provide a theoretical basis for improving the design of the structure and material of the crankshaft, the static stress state of the crankshaft and the size of the yield stress limit value were first analyzed. Through theoretical calculation and finite element analysis of ANSYS, the stress state of each section of the crankshaft, the bending stress limits and deformation were separately calculated, so that the analysis result was close to the actual state of the object.

Research methods. Finite Element Analysis (FEA) is based on the modeling of physical objects using mathematical algorithms [1, 2]. In this work, the modeling and analysis are performed using ANSYS software [3].

ANSYS has three main modules: 1 - Preprocessing module: Creates a mock-up model and performs finite element decomposition; 2 - Computational module: Analyzes structures, fields (fluid, magnetic) and performs coupled analysis between different physical fields (e.g. thermal coupling); 3 - Final processing module: Processes the results, including displaying cloud maps of stresses, displacements, and plotting diagrams.

Static analysis typically does not consider the effects of inertia and damping in finite element models. ANSYS is capable of performing linear analysis as well as complex nonlinear analysis, including thermal expansion, plastic deformation, and creep deformation of parts. These capabilities make ANSYS a leader in finite element analysis [3].

When creating finite elements [4], the main parameters for structural analysis are external force, stress, deformation and displacement. For this, a geometric formula is created and conditions for the associated contour are determined. The main equations and physical quantities of elastic mechanics are expressed in the corresponding formulas.

© Д.Е. Селезньов, Е.Л. Селезньов, С.Ю. Грицюк

Presentation of the main material. Using ANSYS Workbench's proprietary CAD connectivity interface, a three-dimensional model created in SolidWorks was loaded into Workbench.

The specific input process is as follows:

Launch the ANSYS Workbench software, open the "Toolbox" in AWB, find the "Analysis Systems" taskbar, right-click "Static Structural", and you can create a list of static structural analysis items on the software interface.

Double-click "Geometry" in the list of items to enter the ANSYS Workbench software, where you will find the "Design Modeler" - a geometric modeling module. Click the computer file on the menu bar, and then click "Import Geometry" in the open drop-down menu. After entering the external model, a file selection dialog box will appear, where you can select the object model created in SolidWorks and confirm it, thus completing the file selection. Clicking "Generate" will allow you to enter the model into ANSYS Workbench. Fig. 1a shows the geometric model created in Workbench after the specific input.

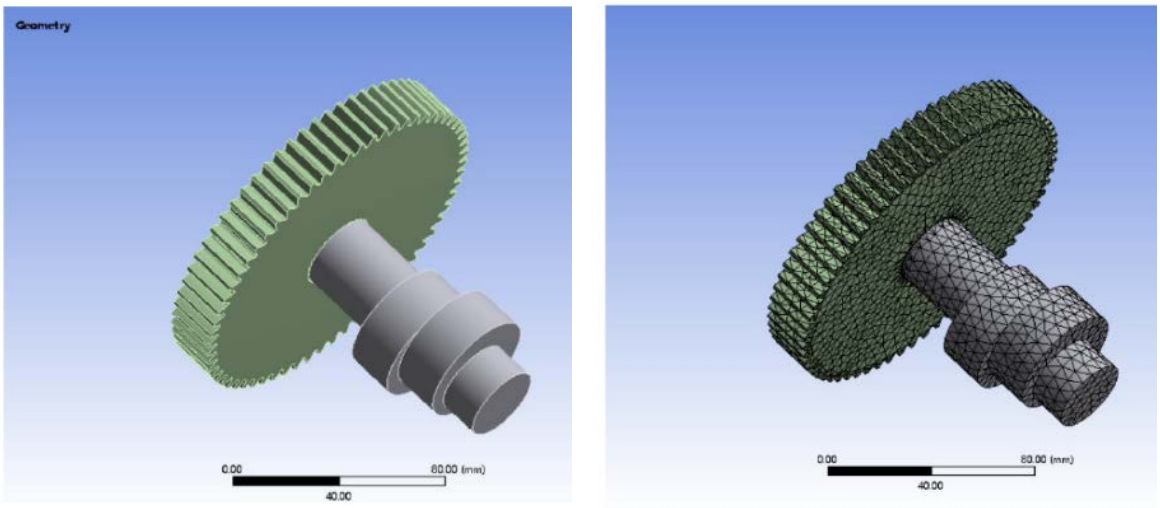
Setting material properties. First, set the material definitions of various elements. The AWB module has its own material database, which allows the user to directly call up material data and improves work efficiency. If necessary, the user can manually add data to the material database to enrich and expand it.

ANSYS Workbench contains one of the largest databases of materials and includes parameters such as material density, elastic modulus, Poisson's ratio, etc. By clicking on the name of the parameter, a window will appear for editing the material properties, then you need to enter specific values in the corresponding physical property field, and finally click the "Return to Project" button to return to to the main page of ANSYS Workbench, at which point the installation of material properties will be completed.

The quality of cell division is important in the analysis process and decides the accuracy of the final analysis results. Usually, cell division is performed in an adaptive automatic way. It is divided into three classes: coarse, medium and fine. The size of the cells is also one of the conditions that determines the accuracy of the analysis results.

The smaller the cells, the more they are, the better the accuracy, but the amount of calculation also increases. However, the results of the model analysis will be closer to the actual state, or vice versa.

Determine the cell type. Open the static structure list on the main interface of ANSYS Workbench - find and double-click "Model", enter "Mechanical", that is, the window for cellular decomposition and analysis of the structure, then find "Outline", expand the tree menu of the model "Model "under Project, and find the item " Mesh". Set the cell size of the tetrahedral object (Element size) to 5 mm, leave the other items by default. Right-click "Mesh", in the menu, then click the Generate mesh command. At this time, Workbench begins to develop a mesh for the geometric object. The geometric model after cellular decomposition is shown in Fig. 1 b. The total number of cells is 67,353, the total number of nodes is 32,136.



a b

Fig 1. **Crankshaft model: a) simplified model, b) model obtained after cellular decomposition**

Next, we conduct a stress state study. In order to better reflect the stress state of the crankshaft, it is necessary to securely fix both ends of the crankshaft to ensure the convergence of the simulation results, otherwise it is impossible to obtain the calculation results. Therefore, the stress or deformation response at both ends of the crankshaft should be ignored in the simulation results (Fis 2).

© *Д.Е. Селезньов, Е.Л. Селезньов, С.Ю. Грицюк*

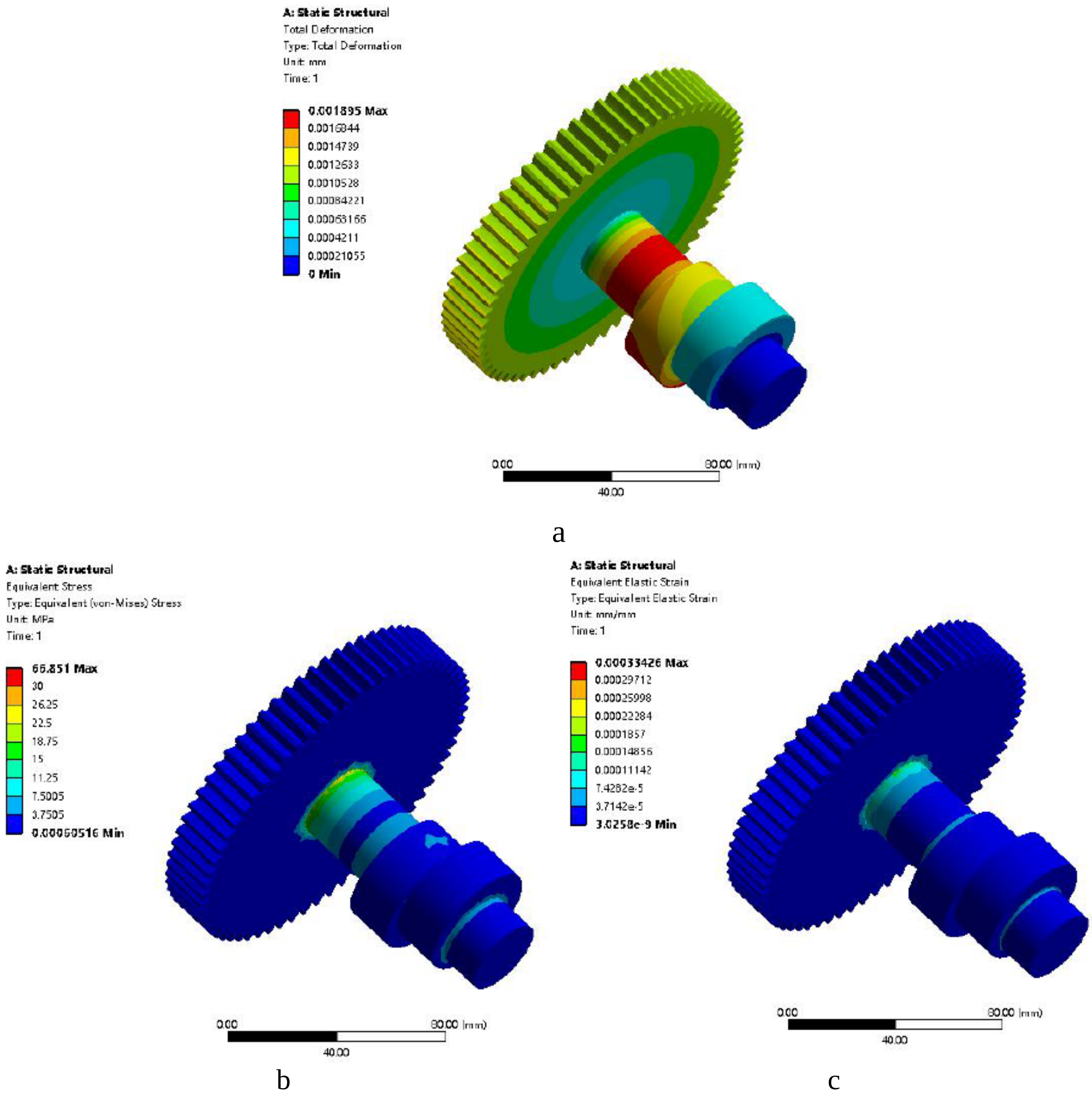


Fig. 2. **Simulation results: a) displacement diagram (w), b), c) cloud stress diagram**

Results. When the rotation angle (RV) of the crankshaft is equal $\theta = 15°$ to the nominal torque, the stress and strain values on it are as follows. According to the simulation results, it is established that the maximum load value is concentrated in the crank (corresponding to the area shown in Fig. 4-5). The maximum stress is 66 MPa. The crankshaft is made of GCr15 bearing steel, which after heat treatment (quenching and tempering) acquires high and uniform hardness, good wear resistance and high resistance to contact fatigue. The yield strength of this material is 608 MPa. Since the calculated maximum stress (66 MPa) is much lower than the yield strength (608 MPa), the material fully meets the operating requirements. According to the stress diagram, the maximum strain is 1,8 $\times 10^{-3}$ mm. This maximum strain is localized in the same zone of maximum stress - in the middle of the crankshaft.

References

1. Reddy J. N. An Introduction to the Finite Element Method. 4th ed. New York: McGraw-Hill Education, 2019. 768 p.
2. Bathe K. J. Finite element procedures. 2nd ed. Watertown, MA: Klaus-Jürgen Bathe, 2014. 1066 p.
3. Moaveni S. Finite element analysis: Theory and application with ANSYS. 5th ed. Upper Saddle River: Pearson, 2021. 704 p.
4. Logan D. L. A first course in the finite element method. 6th ed. Boston: Cengage Learning, 2016. 992 p.

Дата надходження статті до видання: 25.02.2026
Дата прийняття статті до друку після рецензування: 24.03.2026
Дата оприлюднення 14.04.2026

© Д.Е. Селезньов, Е.Л. Селезньов, С.Ю. Грицюк