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COMPUTER SIMULATION OF DRIVEN COG-WHEEL OF BACK BRIDGE MAZ CAR

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Abstract: SolidWorks Simulation was used to determine the performance of the MAZ-509 driven rear axle gear. The material for its production is 18HGT steel, which is not always present in repair shops. Therefore, a determination was made as to the possibility of replacing this material with more affordable material – steel 45 or even steel 20.

Keywords: back bridge, driven cog-wheel, SolidWorks Simulation, engineering analysis, finite element method, modeling, material replacement.

Extreme working conditions of elements of modern structures, complexity of their shape and large dimensions make it extremely difficult and expensive to carry out a full-scale or semi-natural experiment, especially when it comes to the establishment of boundary (destructive) loads. Creating structures of this type is impossible without improving and automating the design process, the use of new materials and technologies.

The need to introduce sophisticated engineering into production in the short term leads to the creation of computer aided design systems. Strength calculation plays an important role in these systems.

Nowadays, CAD/CAE-systems are increasingly used in the educational process of engineering colleges. They provide a fast and accurate solution of computer problems in three-dimensional space. The fact is that the methods of 3D modeling (solid state, surface, hybrid) implemented by modern CAD/CAE-systems radically change the methodology of production design and preparation. In this case, the 3D model is the main and primary carrier of information about the projected object, and the drawings created by this model are a secondary form of object display.

One of such CAD/CAE-systems is SolidWorks – a software complex for automation of work of the industrial enterprise at the stages of design and technological preparation of production [1]. SolidWorks can work equally well with solids and surfaces (as a rule, the workpiece is a solid, a surface, or a combination of a solid and a set of surfaces).

SolidWorks Simulation – SolidWorks application designed to solve the problems of mechanics of a deformed solid by the finite element method. It is a software for calculations of static strength and stability in linear and nonlinear formulation, eigenfrequency, optimization of the form of parts and assemblies in linear formulation, fatigue analysis and design behavior when falling [2].

The program uses a geometric part model or SolidWorks assembly to form a design model. Integration with SolidWorks minimizes operations related to the specific features of the finite element approximation. The boundary conditions are assigned in conjunction with the geometric model. Procedures for presenting results are the same.

SolidWorks Simulation is based on the finite element method. Note the tips of its implementation in this program.

SolidWorks Simulation uses three basic types of finite elements: three-dimensional isoparametric tetrahedra, triangular shell elements, and beam elements. The first two types of finite elements can have a linear or parabolic displacement field (permanent deformation or linear deformation field). Tetrahedrons contain, respectively, 4 or 10 nodes, shells 3 or 6, and beams/rods – 2.

The program allows for coexistence in one model of solid and shell finite elements. Moreover, hybrid grids work in both linear and nonlinear calculation models. However, the beams/rod elements do not combine with any other types of finite elements.

There are no contact finite elements, at least explicitly, in the program. Based on indirect observations, it can be argued that taking into account the corresponding boundary conditions is made by changing the global stiffness matrix of the system. Virtual objects such as bolts, rods/pins, springs implemented on the basis of the elements of beams/rods.

Some other types of kinematic boundary conditions are realized by directly changing the stiffness matrix of the system (in the early versions of the program, penalty functions were used – in fact, "very" rigid auxiliary elements, which led to program errors.

Within one addition, arbitrary combinations of contact boundary conditions such as entering and exiting a contact are allowed.

The boundary conditions, which are grouped into the “Connectors” group, are implemented in the program for calculations of assemblies. The implementation of these conditions (or some of their varieties) implies such changes in the stiffness matrix of the system, which in fact lead to the appearance of an absolutely rigid virtual object in the model. As a consequence, at the point where this object interacts with the "real" details of the assembly (in fact, in the area of application of the described boundary conditions), theoretically infinite deformations (stresses) may occur. In practice, this is reflected in the lack of convergence of the solution when compaction of the mesh and, most likely, incorrect results.

SolidWorks Simulation has a p-adaptive finite element mesh method. This means that in areas with a high gradient of the deformation energy, the program increases the order of the polynomial, which approximates the displacement field in the finite element. In case of incorrect formulation of kinematic boundary conditions, the appearance of features (theoretically infinite deformations and stresses) is possible. Applying this option to such computational models leads to absurd results.

SolidWorks Simulation also has an h-adaptive finite element grid method. It consists in compaction of the grid in zones where the magnitude of the strain energy density is relatively large compared to its average value.

In the framework of elastic analysis, the use of orthotropic materials is possible. Orthogonal-orthotropic and transversal-isotropic materials are available. The appointment of cylindrical orthotropy is possible. There is no curvilinear orthotropy. These properties can be assigned to both solids and shells.

When evaluating the strength of assemblies using the SolidWorks Simulation function, the “Design Check Wizard” uses the same type of strength criterion for all materials. Thus, using this feature to analyze assemblies that contain parts made of brittle and viscous materials is problematic if you want to display results for all parts at once.

SolidWorks Simulation allows you to perform the following types of simulation:

– static analysis in elastic formulation with calculations of individual parts by spatial or shell model, as well as assemblies in three-dimensional formulation taking into account the interaction of details;

– calculations of natural frequencies, values of critical loads of loss of stability and their corresponding forms for details in solid or shell representation, as well as assemblies with non-moving parts;

– thermal calculations taking into account the phenomena of thermal conductivity, convection, radiation, but without taking into account the movement of media;

– thermoelastic analysis based on the results of thermal calculations;

– parametric optimization by the criterion of minimization/maximization of mass, volume, natural frequencies and critical force;

– simulation of deformation of a structure taking into account physical and geometrical nonlinearity, as well as due to changes of loads and temperature in time;

– modeling of effect of falling of a structure on a firm or elastic surface;

– fatigue calculations taking into account the fatigue curves, the shape of the load curve, as well as the linear hypothesis of summation of damage.

All of these types of analysis can be related to the same SolidWorks object. SolidWorks Simulation requires that the finite element method algorithm be observed, giving some freedom within each step in the steps of model preparation and review. For calculations in the elastic formulation for models in solid state, the predicted chain of events is described below.

1. Create an analysis of a particular type and determine its settings. The latter can be changed at any time before calculations.

2. Fill in, if necessary, a parameter table that defines a set of variables that can be changed (specifically, to which lists of values can be assigned) during the calculations.

3. Preparation of raw data within a given analysis:

– the appointment of the material of the part or parts;

– assignment of kinematic boundary conditions;

– designation of static boundary conditions;

– assignment of contact boundary conditions if a multi-body assembly or part is calculated;

– creating a grid.

4. Associate, if necessary, parameters from the parameter table with the appropriate analyzes.

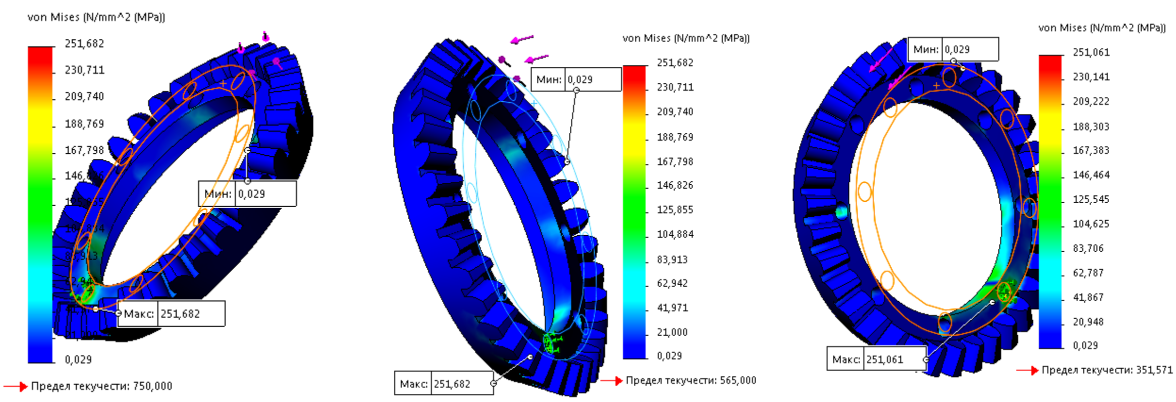
5. Perform calculations.

6. Results processing: creation of necessary diagrams; diagram analysis; export of results.

The optimization procedure is based on the results of calculations in linear formulation (static analysis, natural frequency calculations and stability). Fatigue analysis also requires at least one static calculation.

As an example, SolidWorks Simulation was used to determine the performance of driven cog-wheel of back bridge MAZ-509 car. The material used for its production is 18HGT steel (DIN 1.7147 steel is selected from the SolidWorks library). It is established [3] that the minimum safety factor k = 2,821, which is greater than the permissible [k] = 1,5.

But 18HGT steel is not always present in repair shops. Therefore, the purpose of the work was to replace this material with more affordable - steel 45 (DIN 1.1191 – C45E) or even steel 20 (AISI 1020). To do this, in the simulation of stresses, displacements, deformations, the coefficient of safety margin of the driven gear made of 18HGT steel, a corresponding replacement of the material was carried out with the following re-calculations. The results obtained are reflected in fig. 1 and in table. 1.



a b c

Fig. 1. Plot of equivalent normal stresses von Mises of driven cog-wheel made of: a – 18HGT steel, b – steel 45, c – steel 20

Table 1

The results of the study of the strength characteristics of the driven cog-wheel

Steel	Tension (maximum), MPa	Moving (maximum), mm	Deformation (maximum), mm	Stock strength (minimum)
18HGT (DIN 1.7147)	265,836	0,573077	0,00080085	2,821
45 (DIN 1.1191)	251,682	0,5761	0,0007535	2,245
20 (AISI 1020)	251,061	0,6062	0,0007953	1,400

Conclusion: in the case of replacement of steel 18HGT with steel 20 for the manufacture of driven gear strength is insufficient (allowable margin of safety is less than acceptable). And a suitable replacement for steel 45 is possible.

Thus, the results confirm the relevance of the study using SolidWorks Simulation: in terms of providing strength for the manufacture of driven cog-wheel the replacement of its material is possible. However, given the working conditions of this part, thermal or chemical-thermal treatment is recommended to increase its durability.

REFERENCES

1. Рудик О. Ю. SolidWorks – CAD/CAE-система технічних вузів / О. Ю. Рудик, П. В. Каплун // Science, society, education: topical issues and development prospects. Abstracts of the 2nd International scientific and practical conference. SPC "Sci-conf.com.ua". Kharkiv, Ukraine. 2020. Pp. 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/

2. Borovyk O. V. Implementation in the educational process of the SolidWorks Simulation / O. V. Borovyk, O. Yu. Rudyk, V. M. Ganovskyi // Priority directions of science development. Abstracts of the 4th International scientific and practical conference. SPC "Sci-conf.com.ua". Lviv, Ukraine. 2020. – Pp. 169-173. URL: http://sci-conf.com.ua/iv-mezhdunarodnaya-nauchno-prakticheskaya-konferencziya-priority-directions-of-science-development-3-4-fevralya-2020-goda-lvov-ukraina-arhiv/?utm_source=eSputnik-promo&utm_medium=email&utm_campaign=MATERIALY_L_VOV&utm_content=761175902

3. Рачок Р. В. Застосування SolidWorks для підготовки фахівців у галузі автоматизації та інформаційних технологій / Р. В. Рачок, О. Ю. Рудик, В. М. Волошин // Автоматизація та комп'ютерно-інтегровані технології у виробництві та освіті: стан, досягнення, перспективи розвитку: матеріали

Всеукраїнської науково-практичної Internet-конференції. – Черкаси, 2018. – С. 198-200. URL: http://elar.khnu.km.ua/jspui/handle/123456789/8414