УДК 629.113

PROJECT DEVELOPMENT AND PERFORMANCE STUDY OF THE REPAIR STAND FOR REDUCERS OF REAR AXLES OF CARS USING SOLIDWORKS

Rudyk Oleksandr Yuhymovych,
Ph.D., Associate Professors
Korzun Vadym Viktorovych,
Antonov Anatoliy Arkadiyovych,
Nechyporov Vadym Viktorovych,
graduate students
Khmelnytskyi National University
Khmelnitskyi, Ukraine

Abstract. The application of SolidWorks Simulation is considered for calculations on the static strength of the lock of the repair stand for reducers automobile. The results of calculations of the dependence of stresses, displacements, deformation and safety margin on static loads according to the theory of specific energy of deformation are given.

Key words: car, reducer, repair, stand, lock, SolidWorks Simulation, static strength, stress, displacement, deformations, margin of safety.

The extreme operating conditions of elements of modern structures, the complexity of their shape and large dimensions make it extremely difficult and expensive to carry out full-scale or semi-full-scale experiments, especially when it comes to establishing limit (destructive) loads. The creation of structures of this type is impossible without improvement and automation of the design process, the use of new materials and technologies [1, 2].

At the moment, stands for disassembling and assembling reducers of automobiles are not used in auto repair shops, and the existing designs are not unified (they are used only for a certain model and its modification). All operations of disassembling and assembling reducers are carried out on metalworking machines. In this connection, a stand for repairing reducers of rear axles of cars was developed (fig. 1).

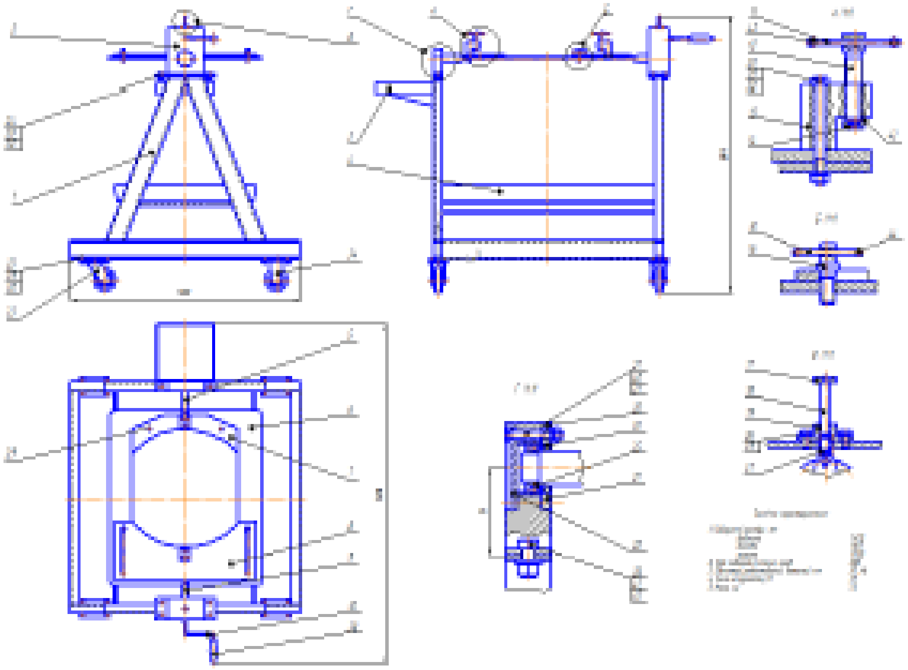


Fig. 1. Stand for repairing reducers of rear axles of cars

This stand is intended for faster and more convenient disassembly and assembly of reducers of rear axles of cars into assemblies and parts. Compared with a rack for disassembly and assembly, the mechanization and automation of the process increases, the work becomes easier, and the convenience and quality of unit repair increases.

The need to introduce complex equipment into production in a short period of time leads to the creation of automated design systems. Strength calculation plays an important role in these systems.

Nowadays, CAD/CAE systems are increasingly used in the educational process of engineering universities. They provide a quick and accurate solution of technical problems in three-dimensional space on a computer. The fact is that the methods of three-dimensional modeling (solid, surface, hybrid) implemented by modern CAD/CAE systems fundamentally change the methodology of design and production preparation. At the same time, the main and primary carrier of information about the designed object is its 3D model, and the drawings created according to this model are a secondary form of displaying the object.

One of these CAD/CAE systems is SolidWorks (SW) - a software complex for automating the work of an industrial enterprise at the stages of design and technological preparation of production. In SW, you can work with both solids and surfaces equally well (as a rule, a part is a solid, a surface, or a combination of a solid and a set of surfaces).

SW Simulation is a SW application designed for solving problems of the mechanics of a deformed solid body using the finite element method. This is software for calculations of static strength and stability in linear and non-linear construction, selection of natural frequencies, optimization of the shape of parts and assemblies in linear construction, analysis of fatigue and behavior of the structure when falling.

The program uses the geometric model of the part or SW assembly to form the calculation model. Integration with SW makes it possible to minimize operations associated with specific features of finite-element approximation. Boundary conditions are assigned in relation to the geometric model. Procedures for presenting results have the same features.

SW Simulation is built on the basis of the finite element method. Let us note some features of its implementation in this program.

SW Simulation uses three basic types of finite elements: volumetric isoparametric tetrahedra, triangular shell elements, and beam elements. The first two types of finite elements can have a linear or parabolic displacement field (constant strain or linear strain field).

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

Contact finite elements, at least in an explicit form, are absent in the program. Based on indirect observations, it can be stated that the consideration of the relevant boundary conditions is carried out by changing the global stiffness matrix of the system. Virtual objects such as bolts, rods/pins, and springs are implemented on the basis of beam/rod elements.

Some other types of kinematic boundary conditions are implemented by directly changing the stiffness matrix of the system (in early versions of the program, penalty functions were used for this - actually "very" stiff auxiliary elements, which led to errors in the program).

Arbitrary combinations of contact boundary conditions such as entry into contact and exit from it are allowed within the limits of one assembly.

Boundary conditions combined in the "Connectors" group (Connectors) 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 actually lead to the appearance of a completely rigid virtual object in the model. As a result, in the place where this object interacts with "real" parts of the assembly (actually, in the zone of application of the described boundary conditions), the appearance of theoretically infinite deformations (stresses) is possible. In practice, this is expressed in the lack of convergence of the solution when the mesh is compacted and, most likely, in incorrect results.

In SW Simulation, there is a p-adaptive method of constructing a mesh of finite elements. This means that in zones with a high strain energy gradient, the program increases the order of the polynomial that approximates the displacement field in the finite element. If the kinematic boundary conditions are incorrectly set, features (theoretically infinite deformations and stresses) may appear. Application of this option for such calculation models leads to absurd results.

In SW Simulation, there is also an h-adaptive method of constructing a mesh of finite elements. It consists in compacting the grid in zones where the value of the strain energy density is relatively large compared to its average value.

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

When assessing the strength of assemblies using the SW Simulation "Strength Check" function (Design Check Wizard), the same type of strength criterion is used for all materials. Thus, the use of this function for the analysis of assemblies that contain parts made of brittle and viscous materials is problematic if it is necessary to display the results for all parts at once.

SW Simulation requires compliance with the basic canvas of the finite element method algorithm, giving within each stage a certain freedom in the sequence of steps of model preparation and consideration of results.

For calculations in elastic formulation for models in solid representation, the assumed chain of events is described below.

1. Creating an analysis of a certain type and defining its settings. The latter can be changed at any time before the calculations are made.

2. Filling, if necessary, a table of parameters, which defines a set of values that can change (specifically, for which lists of values can be assigned) during calculations.

3. Preparation of initial data within the given analysis:

– designation of the material of the part or parts;

– assignment of kinematic boundary conditions;

– designation of static boundary conditions;

– designation of contact boundary conditions, if an assembly or part from several bodies is calculated;

– creating a grid.

4. Linking, if necessary, the parameters from the parameter table with the corresponding analyses.

5. Making calculations.

6. Processing of results:

– creating the necessary diagrams;

– diagram analysis;

– export results.

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

The principle of operation of the wall: the reducer is installed in the seat of the table and fixed with the help of locks that are installed on leaves 7 and 8 (intended for the versatility of the stand and allow fixing reducers of different sizes). But it is necessary to investigate the operability of its details. Therefore, the following studies were carried out with the help of SW Simulation:

– the authors of [3] analyzed the model of the movable sash (position 7 in fig. 1) to obtain a picture of its stress-strain state;

– in work [4], a similar study of a fixed sash (position 8 in fig. 1) was performed.

According to the received calculations, the considered parts have sufficient performance. But similar calculations of other components of the stand are required. Therefore, a static calculation of the stand lock was carried out (position 13 in fig. 1). At the same time, we will complicate the calculations: we will determine the maximum load it can withstand in the event of a violation of safety techniques during the operation of the stand with the planned safety factor n = 2.5.

Steel C35E (DIN 1.1181) was selected from the SolidWorks library - an analogue of the material of the lock (steel 35), for which σ_t = 580 MPa (fig. 2).

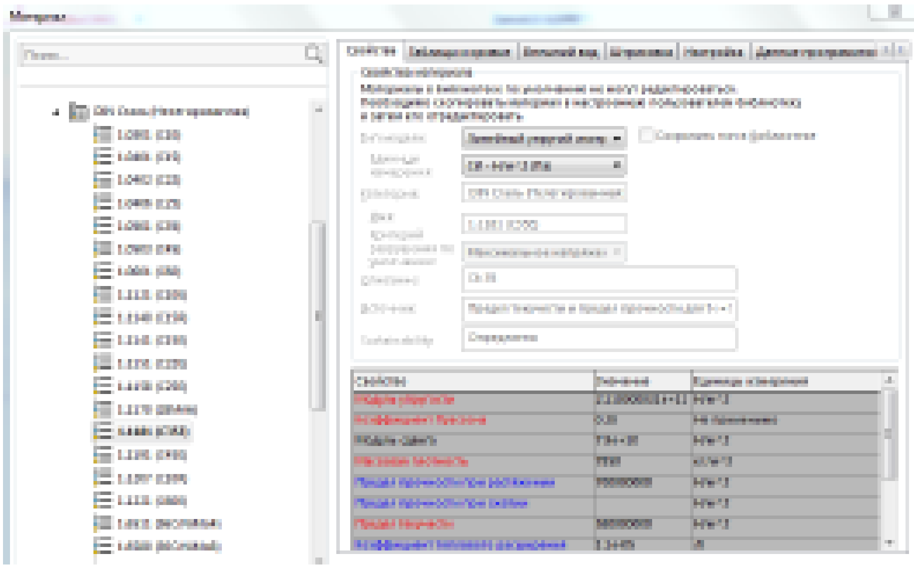


Fig. 2. Information about the analogue of steel 35

To carry out a static analysis, the lock model was fixed (fig. 3) and a load was applied to it (fig. 4).

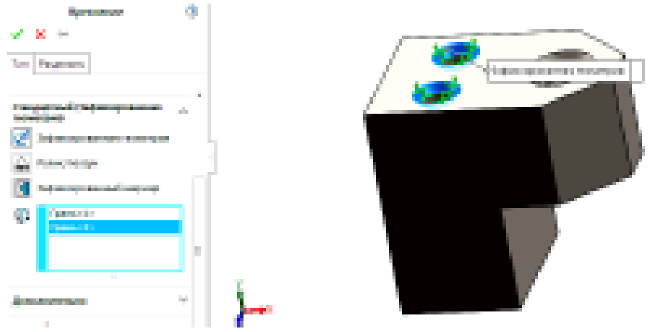


Fig. 3. Fixing the lock model

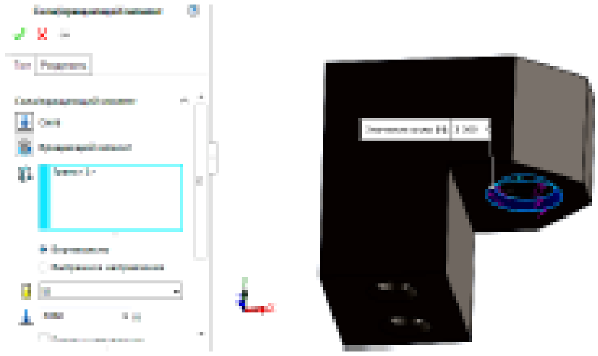
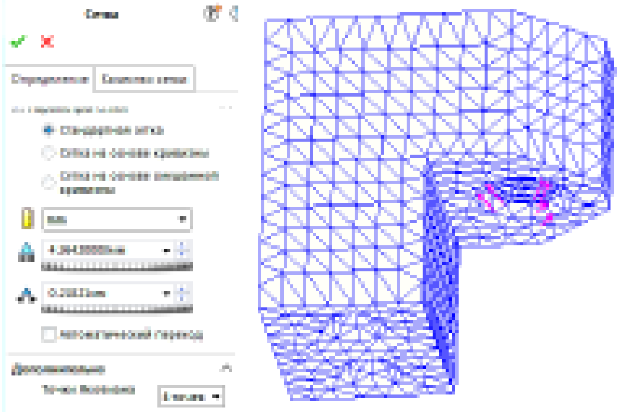


Fig. 4. Applying load to the lock model

The next stage in the analysis of the lock model is the creation of a grid (fig. 5).



a **b**

Fig. 5. Parameters (a) and finite element mesh (b) of the lock model

Calculation results – fig. 6.

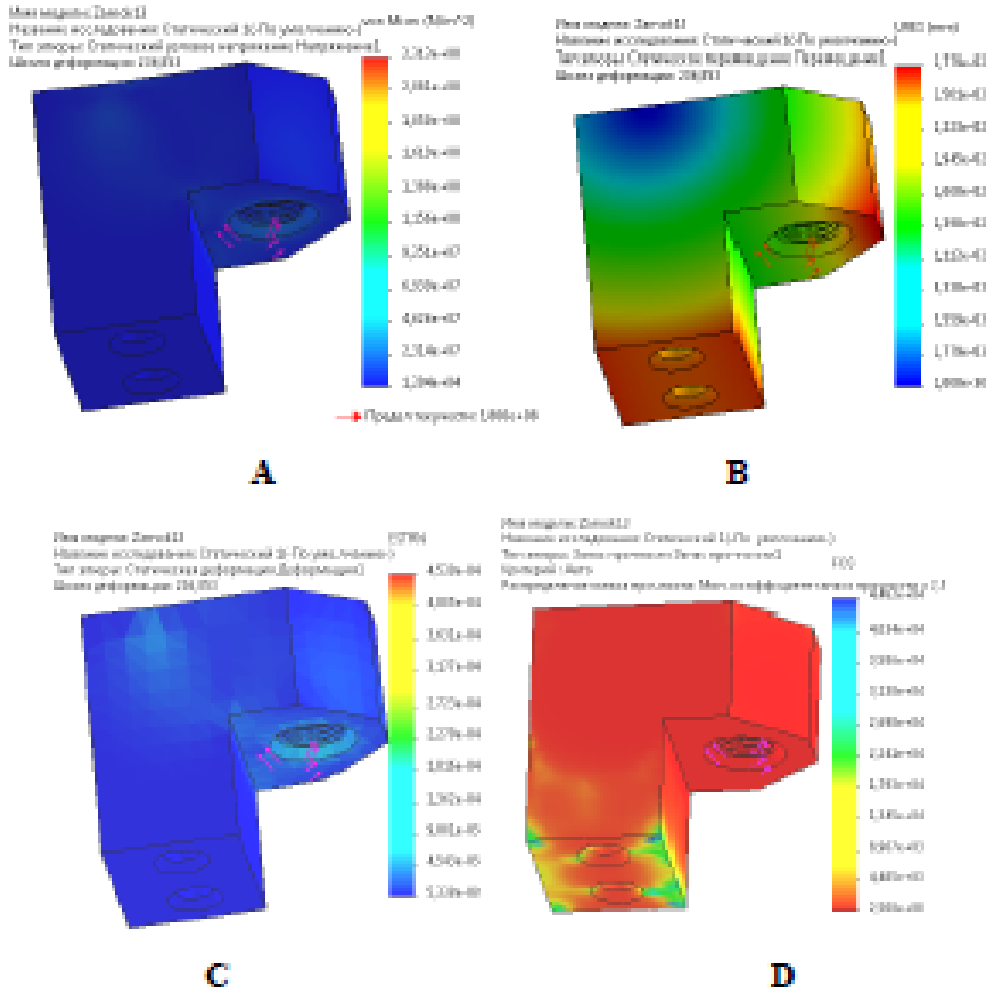


Fig. 6. The results of the lock calculations (a – Von Mises nodal stress distribution diagram, b – URES displacement distribution diagram; c – ESTRN deformation distribution diagram; d – safety margin distribution diagram in the lock model)

According to the calculations, with the planned safety factor n = 2.5, the lock will withstand a load of 3360 N, which is much more than the permissible one (the weight of the reducer is 1400 N).

Therefore, the capabilities of modern computers provide, with the help of CAD/CAE modeling systems (in particular, SolidWorks Simulation), various processes of obtaining an acceptable result without time-consuming field experiments.

LIST OF REFERENCES

1. Rudyk O. Yu. Using of SolidWorks for simulation of screw puller of bearings / O. Yu. Rudyk, P. V. Kaplun, R. V. Solovyov // World science: problems, prospects and innovations. Abstracts of the 5th International scientific and practical conference. Perfect Publishing. – Toronto, Canada. 2021. – Pp. 185-191. – URL: https://sci-conf.com.ua/v-mezhdunarodnaya-nauchno-prakticheskaya-konferentsiya-world-science-problems-prospects-and-innovations-27-29-yanvarya-2021-goda-toronto-kanada-arhiv/

2. Rudyk O. Yu. Application of SolidWorks is for professional preparation of specialists / O. Yu. Rudyk, P. V. Kaplun, V. A. Gonchar // ІІ Міжнародна науково-практична інтернет-конференція «Ресурсно-орієнтоване навчання в «3D»: доступність, діалог, динаміка» : збірник тез доповідей (електронне видання) (м. Полтава, 22–23 лютого 2022 року). – Полтава : ПУЕТ, 2022. – С. 140-146. – Режим доступу: http://elar.khmnu.edu.ua/jspui/handle/123456789/12794

3. Блінніков Г. П. Дослідженні працездатності стенду для ремонту вузлів автомобілів / Г. П. Блінніков, О. Ю. Рудик, М. Є. Топалян // Тези Всеукраїнської науково-практичної on-line конференції здобувачів вищої освіти і молодих учених, присвяченої Дню науки. – Житомир : ЖДТУ, 2019. – С. 196. – Режим доступу: http://elar.khnu.km.ua/jspui/handle/123456789/8464

4. Рудик О. Ю. SolidWorks Simulation у дослідженні працездатності стендів ремонту автомобільної техніки / О. Ю. Рудик, Д. А. Барчишин // Інформатика, інформаційні системи та технології: тези доповідей шістнадцятої всеукраїнської конференції студентів і молодих науковців. Одеса, 23 квітня 2021 р. – Одеса: ОНУ, 2021. – С. 75-77. – Режим доступу: http://elar.khnu.km.ua/jspui/handle/123456789/10224