THE USAGE OF SOLIDWORKS SIMULATION FOR THE DESIGN OF WELDED METAL STRUCTURES WITH SHEET METAL ELEMENTS ON THE EXAMPLE OF WAREHOUSE RACKS

Viacheslav KHARZHEVSKYI, Khmelnytskyi National University, email: kharzhevskyi@khmnu@edu.ua
Mykhailo SVIATUSHENKO, Khmelnytskyi National University, email: sviatushenkom@khmnu.edu.ua

Summary

The paper considers the problem of the strength calculations using the numerical method of finite element analysis (FEA) that is implemented in Computer-Aided Engineering system SOLIDWORKS Simulation. It is shown that the usage of welded metal structures and sheet metal elements in the model has a number of peculiarities that are shown in the paper on the example of strength calculating of a rack for a warehouse. In particular, the division of the model into the finite elements was carried out for the channel elements and for the rectangular pipe – as beam finite elements, sheet metal – as shell finite elements, the contact of metal sheets to beams – was considered as welded joints. The load was simulated by placing two standard Euro pallets (1200x800) in the center on each shelf of the rack. According to the results of the calculations, changes to the structure of the rack were proposed, which provided the required safety factor.

Keywords: sheet metal, weldments, factor of safety, finite elements, simulation.

Introduction

The problem of optimal design of parts of machines, mechanisms and various machine-building structures is important in modern mechanical engineering, as it allows to reduce the material consumption of designed parts and constructions, to use their optimal geometric shapes, in particular the minimum possible cross-section, and at the same time to ensure the specified operational characteristics: load capacity, minimum or predefined geometric dimensions, etc.

With the development of modern Computer-Aided Design (CAD) and Computer-Aided Engineering (CAE) software, it became possible to solve the problems of optimal design of parts in mechanical engineering more accurately, using the modern engineering software, by means of integrated numerical methods. Moreover, it is also important to be able to perform not only calculations using modern methods, but also to perform multi-criteria optimization according to various constraints and goal functions.

It is known that the use of methods of mechanics of materials allows to optimally use the shape of the cross-section of parts, which provides better strength even in the case when the metal capacity of such a structure may be less.

In this work, the problem of optimal design of a welded metal structure of warehouse rack is considered that is used in a warehouse, which also includes sheet metal elements.

Research aim: to research the carrying capacity of the warehouse racks with the given dimensions of the beams by means of the Computer-Aided Engineering (CAE) methods in SOLIDWORKS Simulation and to make necessary changes to ensure the specified load capacity with the given factor of safety (FOS).

The following **objectives** have been set to achieve the aim: 1) to make a solid model of the warehouse rack using the given dimensions and create a calculation model using the finite elements analysis method; 2) to calculate the stresses in the parts of the model, maximum displacements and to calculate the factor of safety. By means of optimization methods, to calculate the optimum load for the construction; 3) to define the optimum dimensions of the cross section of the beams to ensure the specified load capacity.

Research object and methods

The solid model (assembly) of the warehouse rack that was created in SOLIDWORKS CAD system and then researched in this paper is shown in the Fig. 1. The model was made using the weldment and sheet metal tools (Fig.1).

The appropriate methods for creation 3D solid geometry in SOLIDWORKS are described by Howard (2022). The created solid models can be used for carrying out the various types of calculations in machine building. For example, to calculate the main kinematical and dynamical characteristics of assemblies the SOLIDWORKS Motion module of SOLIDWORKS can be used, as described by Kuang-Hua Chang (2021). Moreover, to carry out the necessary calculations of the warehouse racks, a calculation model was created in SOLIDWORKS Simulation and it was used to define the main characteristics, such as stress distribution, displacements of the points of parts and the factor of safety of the construction. To carry out strength calculations, the *Finite Element Analysis* method (FEA) is used, which is implemented in many Computer-Aided Engineering software systems, in SOLIDWORKS Simulation engineering software. The fundamentals of the FEA method that is implemented in the SOLIDWORKS Simulation engineering software, are described in the works of Kurowski P. (2022), Mustapha K. (2022), Nudehi S. (2022) and Verma G. (2020).

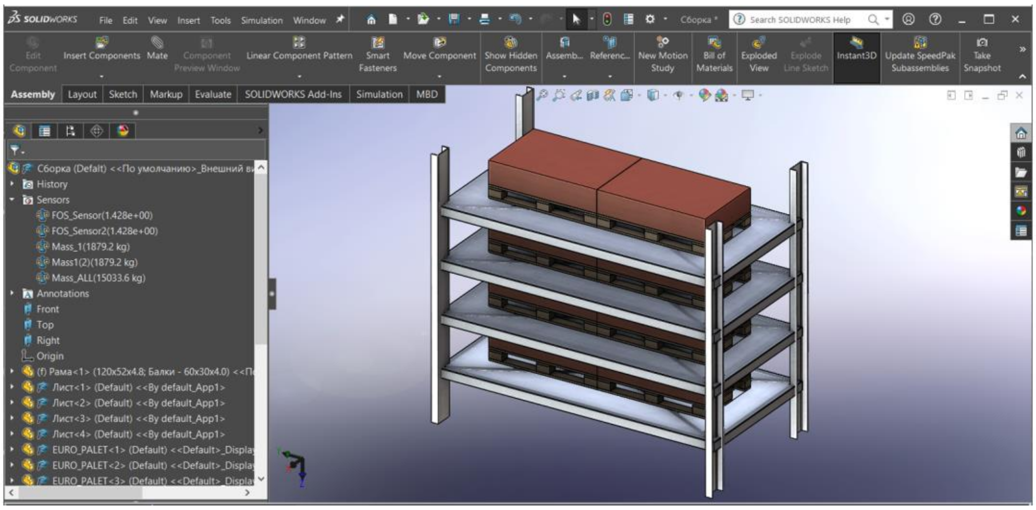
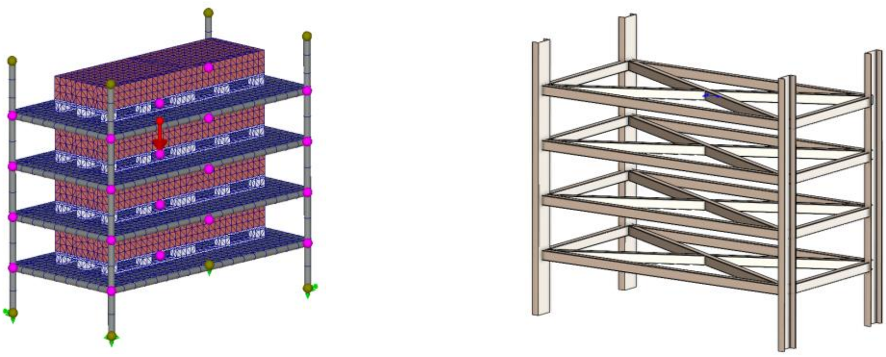


Fig. 1. The model of the warehouse rack that was made in SOLIDWORKS

The idea of FEA method is that a three-dimensional solid model of parts, assemblies or structures is divided into a mesh of finite elements, after which a stiffness matrices are formed and then the values of stresses and displacements at all nodal points of the finite elements are calculated. When creating a calculation model, it should be taken into account that several types of finite elements can be used in SOLIDWORKS Simulation, such as the following: 1) volume tetrahedrons (for solid models); 2) beam elements – for elements of welded metal structures (channel, rectangular pipes, etc.); 3) thin-walled elements (in particular, for sheet metal parts). As can be seen from the Fig. 2, *a*, all the specified types of finite elements were used in the developed three-dimensional model: beam elements for vertical supports and beams that form the rack shelves; shell elements for metal sheets that are welded to shelf beams, as well as finite elements in the form of three-dimensional tetrahedrons for standard Euro pallets and cargo on them.



a *b*

Fig. 2. The developed model of the rack: *a*) finite element meshing; *b*) welded rack frame.

The rack that is intended for a warehouse, contains 4 shelves measuring 3 x 1.5 m, the total height of the rack is 2.5 m, the distances between the shelves in height are 560 mm. The rack is designed for storing materials on standard Euro pallets measuring 1200x800 mm. As can be seen from the Fig. 1, a, it is planned to place 2 such pallets on each shelf, so totally – 8. It was proposed to use a standard channel measuring 120x52x4.8 mm for the manufacture of rack vertical supports, for the manufacture of shelves – standard rectangular pipes with a cross section of 60x30 mm, with a wall thickness of 4 mm. The material of the supports and beams is Steel 3, which is characterized by the parameters as shown below in the Table 1.

Table 1. Parameters of Steel 3

Elastic Modulus, MPa	Poisson's Ratio	Shear Modulus, MPa	Mass Density, kg/m^3	Tensile Strength, MPa	Yield Strength, MPa	Model Type
2.1×10^5	0.29	7.9×10^4	7830	440	225	Linear Elastic Isotropic

As shown, the material for making the rack will be considered as elastic linear isotropic, which means that it obeys Hooke's law, until the yield point is reached (225 MPa). When making calculations, we will assume that the default failure criterion will be – “Maximum von Mises Stress”. The ratio of the value of the yield point stress of the material to the value of the maximum working stress in the model will determine the *Factor of Safety* (FOS): to ensure the sufficient strength of the construction, the value of FOS should be greater than 1, but it is recommended that it lies within such range as: $FOS \geq 1.25...1.5$.

Research results and discussion

To carry out all the necessary strength calculations for the given model of the warehouse rack, we used a numerical FEA method that is implemented in SOLIDWORKS Simulation, as described above. As it was also mentioned, when creating the calculation model in SOLIDWORKS Simulation, different types of finite elements were set for different elements of the warehouse rack: solid elements, thin-walled and beam elements. In addition, the mass of the rack itself was also taken into account during the calculations (as the force of gravity). As a result of the calculation that were carried out for the developed model, the following parameters were obtained: the stress distribution in the parts of the model (Fig. 3); the maximum displacement of the parts of the model (Fig. 4, *a*), which can characterize the stiffness of the welded rack frame; the distribution of the factor of safety of the model (Fig. 4, *b*).

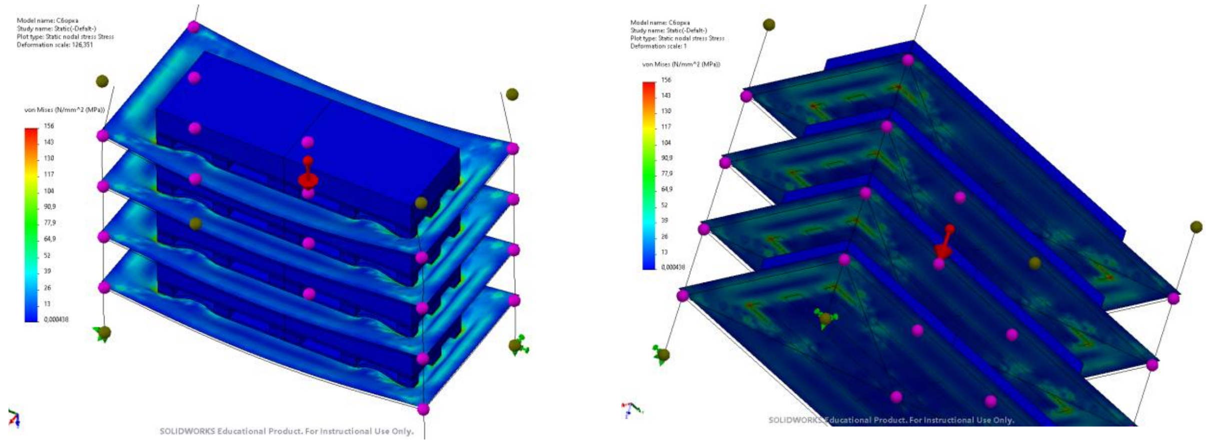
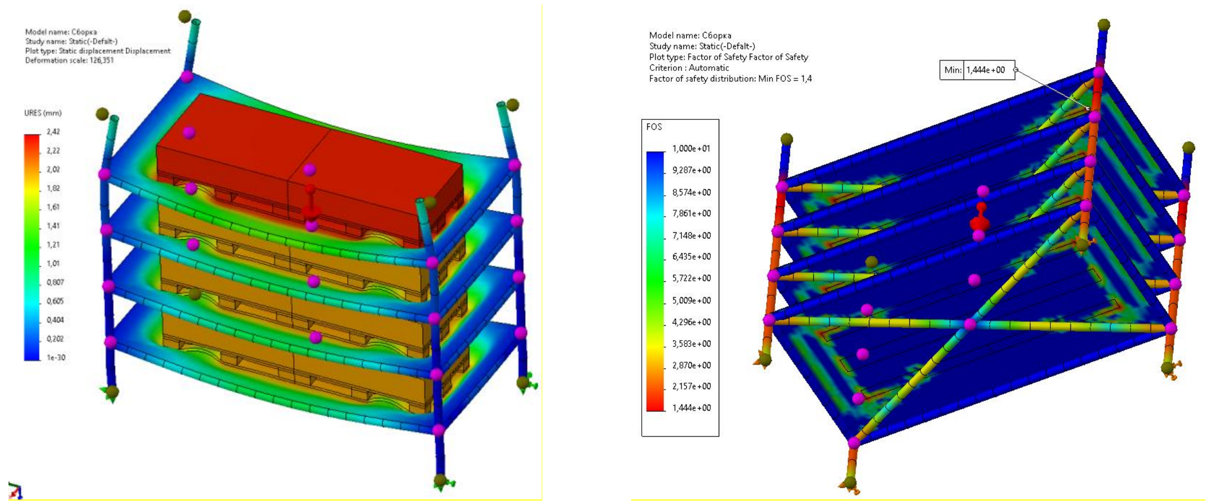


Fig. 3. The results of the calculations: values of stress distribution of the model (von Mises)



a *b*

Fig. 4. The results of the calculations: *a*) values of displacements (mm); *b*) the distribution of Factor of Safety of the model

The obtained results can be used for prediction of the load capacity of the rack. As it was mentioned above, it is recommended that the Factor of Safety of the model should be in range $FOS > 1.25...1.5$, so the problem is to find such optimum value of load that will satisfy that criterion. For that purpose, an optimization procedure can be used, which is implemented in SOLIDWORKS Simulation software as “Design Study”. During the optimization, one of the variable parameters was the height of the load on the pallet, which was varied form the 50 mm to 250 mm, so the total mass of the load was changed accordingly from 3 to 15 tons (material of the cargo is Steel 3 – sheets of metal). It was also constrained that FOS must be greater than 1, and the goal function was maximizing the mass of the load on the pallet. The results of the optimization procedure for the warehouse rack of initial dimensions of the construction are shown on the Table 2. It should be noted that unacceptable scenarios are highlighted in red, recommended – in green color.

For that type of the rack, we used the cross sections of the beams as 60x30 mm (thickness of the wall – 4 mm), the dimensions of the channels (vertical supports) – 120x52x4.8 mm. So, as in can be seen from the Table 2, it was established that for the existing construction of the rack, the optimal safe load is 6000 kg for the entire rack (1500 kg for each shelf), with a safety factor of 1.29. In this case, as established, the maximum height of the cargo on the pallet should be no more than 100 mm. However, the existing geometric dimensions of the rack allow much larger values (up to 250 mm), thus there is a need to strengthen the structure in order to increase its carrying capacity and reliability. In addition, as can be seen from the obtained results (Table 2), a significant increase in the thickness of the metal sheet for the shelf (from 1 to

4 mm) does not lead to a significant increase in the safety factor of the rack (there is an increase in the own weight of the structure), therefore, a sheet metal thickness of 1 mm will be considered further as sufficient.

Table 2. Results of the optimization for the initial construction

Parameter of optimization	Scenario 1	Scenario 2	Scenario 3	Scenario 4	Scenario 5	Scenario 6	Scenario 7	Scenario 8	Scenario 9
Height of the load, mm	50	75	100	125	150	175	200	225	250
FOS, at sheet metal thickness – 1 mm.	2.459	1.69	1.29	1.04	0.871	0.7493	0.6577	0.586	0.5284
FOS, at sheet metal thickness – 4 mm.	2.53	1.785	1.375	1.118	0.9419	0.8140	0.7168	0.6404	0.5787
Total mass of the load on the rack, kg	3006.72	4510.08	6013.44	7516.8	9020.16	10523.52	12026.88	13530.6	15033.6

For that purpose, the dimensions of the cross sections of the beams (rectangular pipes) were changed to 100x70 mm (thickness of the wall – 5 mm), the dimensions of the channels (vertical supports) – to 200x76x5.2. The optimization procedure for the construction of the rack with the specified sizes of elements was carried out under the same conditions that were used in the previous case. The results of the calculation are shown in the Table 3, where the recommended variant (scenario) is highlighted with green color. As it can be seen from the obtained results, this type of the rack can carry 15 tons (instead of 3 tons in the previous case), with the maximum height of the load on the pallet – 250 mm. In this case, the minimum factor of safety (FOS) will be sufficient – 1.36 for the recommended thickness of the sheet metal for the shelves – 1 mm.

Table 3. Results of the optimization for the improved construction

Parameter of optimization	Scenario 1	Scenario 2	Scenario 3	Scenario 4	Scenario 5	Scenario 6	Scenario 7	Scenario 8	Scenario 9
Height of the load, mm	50	75	100	125	150	175	200	225	250
FOS, at sheet metal thickness – 1 mm.	5.58	4.039	3.155	2.586	2.189	1.899	1.675	1.5	1.36
FOS, at sheet metal thickness – 4 mm.	5.991	4.345	3.383	2.765	2.337	2.024	1.784	1.6	1.44
Total mass of the load on the rack, kg	3006.72	4510.08	6013.44	7516.8	9020.16	10523.52	12026.88	13530.6	15033.6

It should be noted that increasing of the load capacity of the rack in 2.5 times is possible only due the increase in the cross-sectional dimensions of the beams and vertical supports, which also increases the weight of the structure, its metal content, which accordingly increases the cost of its manufacture. For comparison: in the first case, the weight of the structure is 563 kg, and in the version of the rack design with reinforced beams – 1082 kg. After analyzing the prices of metal, it was established that the cost of metal for the manufacture of the first version of the rack is about $450, and in the second case – $970, which is quite proportional to the mass of both structures.

However, it should be noted that the usage of a structure with reinforced beams is beneficial for the following reasons: 1) with an increase in the mass of the structure by 1.92 times, the increase in carrying capacity is more than proportional – by 2.5 times; 2) saving space in the warehouse as a result of placing a larger mass of cargo per unit area; 3) exclusion of the possibility of loading a cargo of such a mass on the rack that would cause the entire structure to break, since the reinforced structure of the rack will withstand the weight of the load that can geometrically fit there, unlike the initial version of the design. However, we also have the urgent task to increase the rack's load capacity without significantly increasing its metal content, which would reduce its mass and, accordingly, the cost of manufacturing. To solve this problem, we will use the following laws of mechanics of materials:

1) for more efficient operation of the bending beams, in the case of its rectangular cross section, it is necessary to orient them vertically;

2) in order to effectively reduce stresses in the model arising under the action of bending forces or loads, it is necessary to limit the maximum movements of the part by introducing additional elements into the design that will work in compression. So, it allows to replace the bending stress in the part with compressive stress, which is advisable.

3) in places of stress concentration, it is advisable to introduce additional elements into the structure, in order to distribute them more evenly in the model.

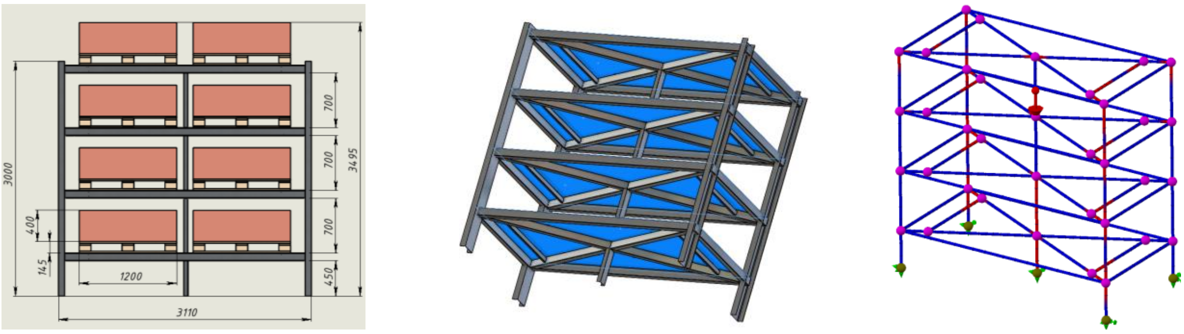
Having analyzed the results of previous calculations, which are shown in the Fig. 3 and Fig. 4, it can be noted that each shelf of the rack can be considered as a beam on two supports, on which a distributed load acts, and in this case, the largest movements under the action of the force will be in the center of the beam (at the intersection of the diagonal beams of the shelf), therefore, introducing another additional support in the center of the rack will limit the maximum movements under the load, replacing bending stress with compression stress. It should be noted that such an additional support can be introduced only in the case if the width of the cargo does not exceed the width of the Euro pallet on which it is placed. Otherwise, the only possible option for increasing the carrying capacity of the rack is the option of strengthening the beams, which was considered above.

Analysis of the results of strength calculation shows that the maximum stresses in the model act along the left and right edges of the pallets on which the load is placed (Fig. 3). Furthermore, the lower values of factor of safety are observed in the same areas (Fig. 4, *b*). Therefore, by placing an additional two beams on each shelf, we can strengthen their structure and distribute the stresses more evenly. The proposed design of the rack with an additional support and beams is shown on the Fig. 5. To fit the cargo with the increased height (up to 500 mm), the distance between shelves was enlarged to 700 mm, and the height of the vertical supports was changed to 3 m. The results of the optimization procedure are shown in the Table 4.

Table 4. Results of the optimization for the improved construction

Optimization parameter	Scenario											
	1	2	3	4	5	6	7	8	9	10	11	12
Height of the load, mm	225	250	275	300	325	350	375	400	425	450	475	500
FOS, at sheet metal thickness – 1 mm.	2.301	2.075	1.889	1.734	1.603	1.489	1.391	1.305	1.229	1.162	1.101	1.046
Total mass of the load on the rack, kg	13530	15033	16536	18040	19543	21047	22550	24053	25557	27060	28563	30067

Thus, the received results of optimization show that the optimal height of the load is 400 mm, it corresponds to the total mass of the load on the rack – 24 tons (3 tons on each pallet), which is 4 times greater than the load capacity of the initial construction (see Table 2). An important parameter of the design is its metal capacity, which is directly proportional to the cost of its manufacture. It was mentioned above that the mass of the initial variant of the rack is 563 kg (optimal variant in the Table 2), and the optimal variant with the larger cross sections of the beams is 1082 kg (Table 3). For the comparison, it was also determined that the optimal variant of the rack with additional support and beams (Table 4) is 657 kg. So, the usage of additional support and beams makes is possible to increase the load capacity of the rack in 1.6 times (in comparison with the second variant), and at the same time the overall metal capacity of the optimized structure will be 39% smaller.



a *b* *c*

Fig. 5. The improved construction of the rack: *a*) the overall dimensions; *b*) additional support in the center and additional beams of the shelves; *c*) the distribution of the FOS in the beams of the model: the elements with FOS < 3 are marked with red color, the other elements – with blue color.

Conclusions

As a result of the conducted research, the carrying capacity of the warehouse rack was determined by means of Finite Elements Analysis method (FEA) that is implemented in the SOLIDWORKS Simulation engineering software. It was established that for the initial construction of the rack, the optimal safe load is 6000 kg for the entire construction (1500 kg for each shelf), which ensure a safety factor of 1.29. For the design of a new rack, in order to increase its carrying capacity compared to the initial variant, it is recommended to use a channel 200x76x5.2 for supports (instead of 120x52x4.8), and it is also recommended to use beams in the form of a pipe with a larger rectangular cross-section 100x70x5 (instead of 60x30x4), which will increase the total load capacity of the designed structure is 2.5 times – up to 15 tons, while a sufficient safety factor of 1.36 will be provided. Therefore, the use of a proposed design will allow much more efficient usage of the warehouse area and will prevent critical rack overload. It was also established that in the case when the width of the cargo does not exceed the width of the pallet, it is possible to use the initial variant of the rack with the additional support in the centre and two additional beams on each shelf: in this case the load capacity can be increased in 4 times (up to 24 tons) in comparison with the initial variant, and the factor of safety of 1.305 is satisfied.

References

1. Chang, K. H. 2024. *Motion Simulation and Mechanism Design with SOLIDWORKS Motion 2024*. SDC publications.
2. Kurowski, P. 2013. *Engineering Analysis with SolidWorks Simulation 2013*. SDC publications.
3. Howard, E., & Musto, J. 2005. *Introduction to solid modeling using Solidworks*. McGraw-Hill, Inc..

4. Mustapha, K. B. 2022. *Practical Finite Element Simulations with SOLIDWORKS 2022: An illustrated guide to performing static analysis with SOLIDWORKS Simulation*. Packt Publishing Ltd..
5. Nudehi, S., Steffen, J. 2018. *Analysis of machine elements using SolidWorks simulation 2018*. SDC Publications.
6. Verma G.2020. *SolidWorks Simulation 2021 Black Book*. CADCAMCAE Works. 2020. 436 p.