International Journal of Research in Engineering and Science (IJRES)
ISSN (Online): 2320-9364, ISSN (Print): 2320-9356
www.ijres.org Volume 10 Issue 12 ǁ December 2022 ǁ PP. 578-587

Application of topology optimization in the modification of a robotic gripper

Lukáš Hanko[1], Peter Križan[*1], Ján Kijovský[1], Stanislav Strigáč[1]
[1]Institute of Manufacturing Systems, Environmental Technology and Quality Management, Faculty of Mechanical Engineering, Slovak University of Technology in Bratislava, Slovak Republic
Corresponding Author[*]:Peter Križan, peter.krizan@stuba.sk

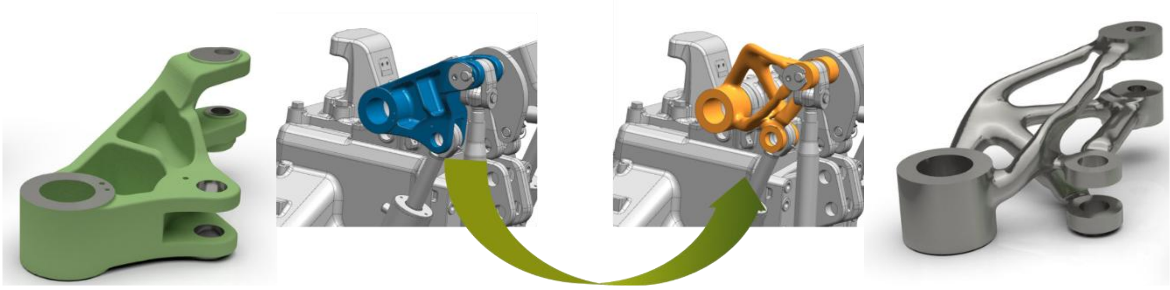
Abstract
Optimizing the shapes and volumes of a part through topological optimization is a relatively new method. This field is expanding extremely rapidly in research sectors, which has interesting theoretical implications in mathematics, mechanics, metaphysics and computer science. Currently, topological optimization is successfully used in many industrial areas, such as the automotive industry, the aerospace industry, but it is most likely to have a significant role in micro- and nano-technologies. Topological optimization has been a popular design method among CAD designers in recent decades. Topological optimization optimizes a given design domain by minimizing or maximizing one or more objective functions such as the stiffness of the entire structure. The intention of this paper is to describe the method of topological optimization in engineering dimensioning. The main aim of this paper is to present topology optimization in real part dimensioning, in the modification of robotic gripper.

Keywords:*Topology optimization, SolidWorks, Robotic gripper, Deformation, Stiffness/weight ratio*.

Date of Submission: 15-12-2022 Date of acceptance: 31-12-2022

I. INTRODUCTION

Topological optimization (TO)have recently been a popular design method among CAD designers and engineers. This method optimizes a specific part of the model design by minimizing or maximizing one or more objective functions, such as the stiffness of the entire model structure. This method respects given constraints such as weight or volume reduction. Thus, the given 3D model (structure) can be materially saved while its mechanical strength can be preserved or increased. The topological optimization process ensures that shapes are created by removing material from the area or locations where the model exhibits low stress levels under load conditions (Figure 1). It may be difficult to produce such shapes using conventional technologies and they are thus suitable for use in additive manufacturing. Hybrid production processes combine conventional and additive manufacturing technologies, where additive technologies create the final shape (created by topological optimization) by adding material layer by layer, and conventional technologies complete the manufactured work piece (using CAM module) from the point of view of the resulting surface quality and shape accuracy.This field is expanding extremely rapidly in research sectors, which has interesting theoretical implications in mathematics, mechanics, metaphysics and computer science [1]. Since the pioneering work of Bendsoe and Kikuchi [2], topology optimization, which focuses on the optimal distribution of available material in a prescribed design area, has undergone tremendous development.



a.) b.) c.)

Figure1: An example of using topological optimization – part before TO (a.), example of using (b.), part after TO (c.)

Topological optimization optimizes a particular design domain by minimizing or maximizing one or more objective functions such as the stiffness of the entire structure. This method respects given constraints such as weight or volume reduction [3]. This can lead to savings in the material of the structure while the mechanical strength remains preserved or can be increased. The topological optimization process ensures that shapes are created by removing material from the area or locations where the part exhibits low stress levels under load conditions. Such faces can be complex and therefore difficult to produce with the help of conventional machines.

However, with the current capabilities provided by additive manufacturing, the complexity of manufacturing complex shapes is not a problem, and therefore a topology-optimized design can be realized. Since additive manufacturing is a technology that is not yet sufficiently mature, studies of several aspects need to be carried out. This is in order to properly understand the behaviour of the material under load conditions [4]. Topological optimization together with size and shape optimization form three categories of so-called structural optimization.

II. METHOD OF TOPOLOGICAL OPTIMIZATION

Technologists use computer-aided manufacturing (CAM) as a tool for designing and simulating manufacturing processes that use machining technologies. To optimize and modify 3D models, the topology optimization method is also used nowadays. A benefit of this method is that it allows the 3D model to be modified (perfect for additive manufacturing), even though the model is complex, which cannot be produced by machining technologies. However, with the current possibilities provided by additive manufacturing, the difficulty of manufacturing complex shapes is not a problem, and therefore a topologically optimized design can be realized. Since additive manufacturing is a technology that is not yet sufficiently mature, studies of several aspects need to be carried out. This is in order to properly understand the behaviour of the material under load conditions. Topology optimization together with size and shape optimization form three categories of so-called structural optimization. The traditional and still dominant way of performing structural optimization is characterized by:

• Design variables that define parameters, properties or elements that will be the subject and can be changed during optimization. These variables form the design space.

• State variables that represent the response of structures. Structural responses are, for example, stress, displacement, force, deformation, weight, and volume. This factor can be global or related to the load condition and is calculated by a finite element analysis [5].

• Limiting functions represent the boundaries that must be reached for answers.

• Objective functions for classifying designs and returning the correctness of designs. This function is associated with the response and is a function of the design variables. It must be minimized or maximized.

Based on the design variable that is parameterized, design optimization can be classified as topology size and shape optimization. Figure 2 shows the difference between size, shape, and topology optimization. The differences between these three categories of structural optimization mainly consist of the definition of design variables [6].

Optimization of size: In optimization of size, the structural elements are changed. It follows that before performing such an optimization, the structure to be analysed must be defined. One of the main disadvantages of size optimization is that the structure topology remains fixed during the optimization procedure. Therefore, if a sub-optimal topology is chosen when formulating the optimization problem, the resulting structure will also be sub-optimal. The optimal size optimization design is the most optimal design that can come out of the predetermined design geometry definition. No elements are added or removed during size optimization. This means that the topology of the design is preserved throughout the optimization process.

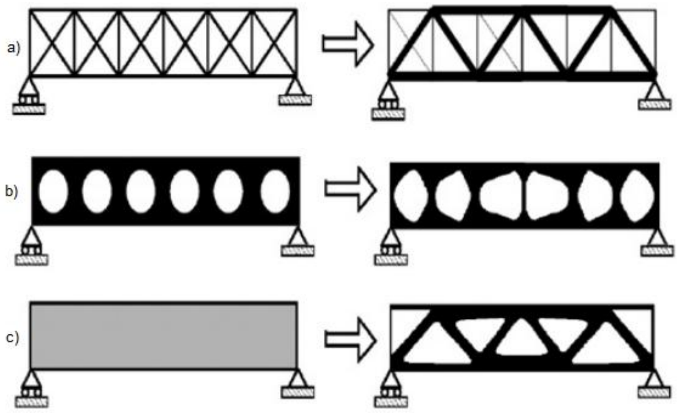


Figure2: Categories of structural optimization: a) size optimization; b) shape optimization; c) topology optimization

Shape optimization: Shape optimization means defining the geometric boundaries of the outer edges or faces and inner openings of the structure. These are so-called structural variables. Design variables are dimensions that describe the geometry of the product and are usually discrete variables. This approach is mostly used in the design phase and the variables are generally obtained from the designer. Shape optimization requires the finite element model to change during the optimization process. The geometry of the product can change during the process, as long as changes in the boundaries of the geometry of the product are allowed. Because of these geometry changes, automatic finite element meshing is usually needed. This optimization can be classified into direct geometry manipulation and indirect geometry manipulation. Before optimizing the shape, the geometric configuration of the structure is required. The geometric configuration used for shape optimization is not included in the predefined geometric modeling set if the optimization is based on a geometric configuration. Therefore, shape optimization converges to different optimal shapes for different starting topologies.

Topology Optimization: Topology optimization is the most common structural optimization to determine not only the size and shape of elements, but also how they are connected. When optimizing a structure's topology, it is natural to require that the solution consists of clearly separated material and voids, preferably with a material distribution that can be fabricated. This means using a discrete variable for each element state, either material or void. On the other hand, the use of continuous variables opens up the possibility of using effective mathematical programming schemes [5-8].

III. THE PROCESS OF OPTIMIZING THE DESIGN OF A PART

In general, structural parts are designed and improved on the basis of previously created designs of real parts. Therefore, the design is already predetermined and has defined parameters. In the absence of a previous design or an actual manufactured part, designers typically start with a few conceptual designs. They would define their parameters according to the environment, and use existing product development methods. Another method of working and developing a new design is to start with a material block and use shape optimization. As is obvious, this method is called topological optimization. Topological optimization can be used whenever we want to design a completely new part or improve an existing one, which must fit into a certain space, be light and withstand certain external loads. Topological optimization works on the principle that a block of material is taken, on which material is removed based on minimizing/maximizing its weight, displacement or compliance in order to simultaneously satisfy constraint conditions such as displacement and minimum element size[9].

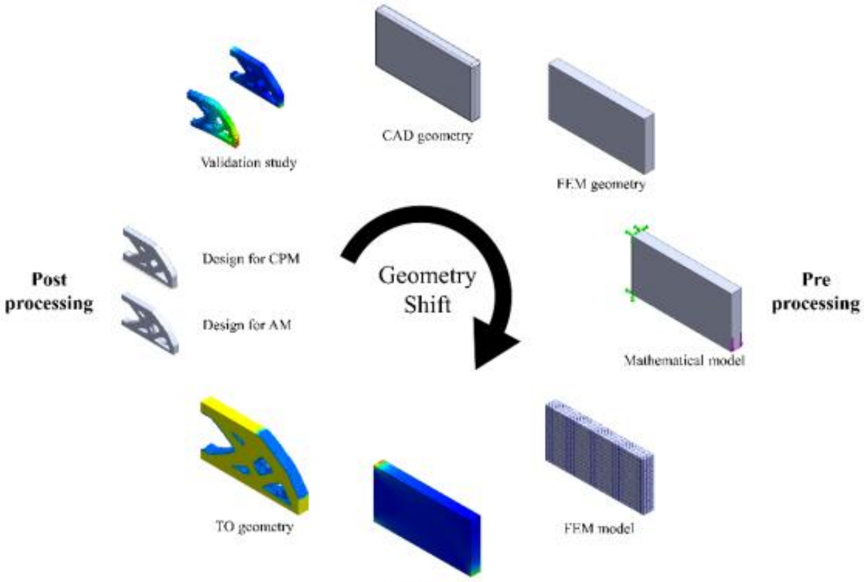


Figure3: Topological optimization workflow

We can see in Figure 3, the topological optimization is divided into two major steps, namely the pre-processing and the processing (post processing). Pre-processing consists of the design phase, implementation of the finite element method, and definition of the mathematical model. The last part of the pre-processing task is completed by evaluating the results from the analysis of the finite element method. At this point, it is necessary for the designer to consider whether there is room for topological optimization. In addition, it is necessary to decide which methods of topological optimization to use and which software is suitable for the given process. On the other hand, processing is required. In this task, optimized designs are prepared for production either by conventional methods, if the optimized shape allows, or by additive manufacturing (Figure4).

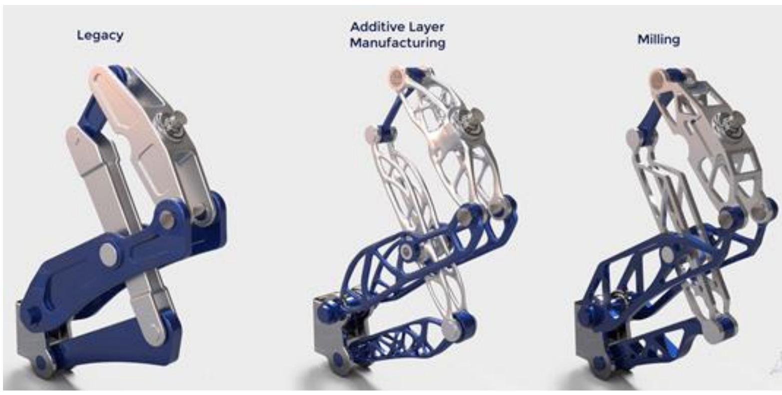


Figure4: Topological optimization for additive and conventional manufacturing

Several decisions must be made during the workflow, referred to as topology optimization inputs. These inputs can be divided into several groups: design constraints, supports and connections, external loads, objectives and constraints, and geometric constraints caused by manufacturing constraints [9].Design constraints are all geometric dimensions that create the final dimension and shape of the given part. A connection is a mechanism that defines how an entity (vertex, edge, face) is connected to another part entity or to a ground. We simplify modelling by using coupling because in many cases we can simulate the desired behaviour without having to create detailed geometry or specify contact conditions. External loads are all external forces acting on a given component. The designer must correctly define these external loads during topological optimization. It is imperative that these forces act in the right place and have the right direction and orientation. Objectives and constraints govern the mathematical formulation of the optimization algorithm. When specifying goals and constraints, the designer selects the most appropriate optimization conditions, such as the appropriate stiffness-to-weight ratio, minimizing weight, and minimizing maximum displacement [10].

IV. THE TOPOLOGY OPTIMIZATION OF ROBOTIC GRIPPER

In our case, the SolidWorks CAD software was used. Topological optimization (TO) be one of the possibilities that software SolidWorks offers directly in its simulation module. Thanks to the fact that we have a component divided into a finite number of elements and the load is known, the software can remove "unnecessary" material and preserve the mechanical properties of the component according to the boundary conditions. Through such optimization, we save material and extend the life of the equipment that works with the tool.In order to be able to start the topological optimization, we have to add boundary conditions in addition to entering the acting force and the anchor point. These boundary conditions were entered during the static FEM analysis. Optimizing the stiffness-to-weight ratio is the first option to choose. The second method is volume reduction at the expense of stiffness, and the third method is maximum dynamic stiffness while reducing volume. In addition to loading and anchoring, we have defined the preservation of functional surfaces to a depth of 1 mm in the boundary conditions. In the case of screw holes, we have also preserved the cylindrical part of the hole to a thickness of 1 mm.Our original idea was to design a robotic tool for automating the tightening of screws on the underside of the fender (Figure 5). The maximum tightening moment is 38Nm and the maximum force with which the screw attachment presses against the screw head is 250N. Input material and volumetric properties of robotic gripper can be seen in following Table 1.

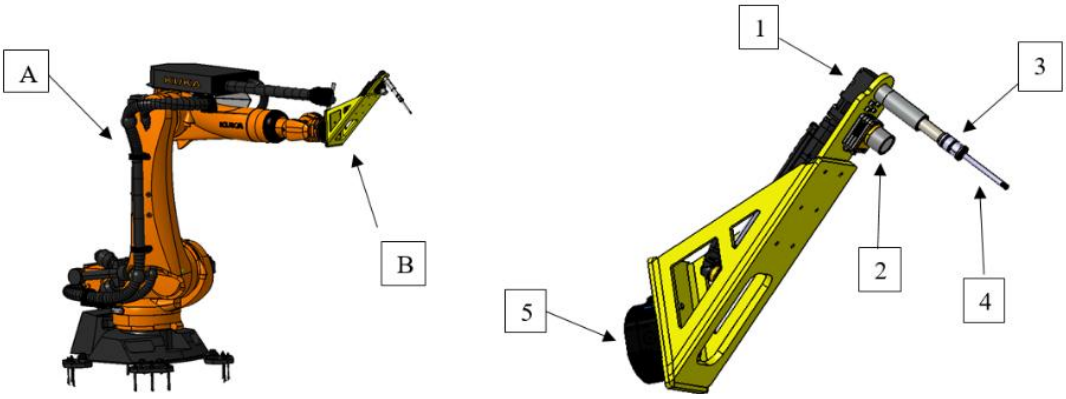
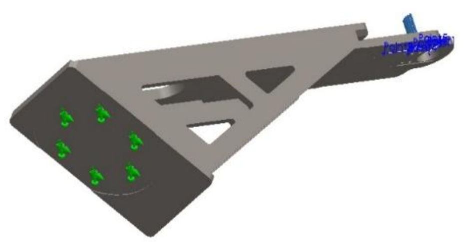


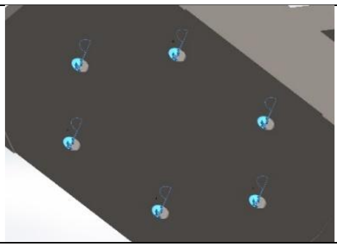
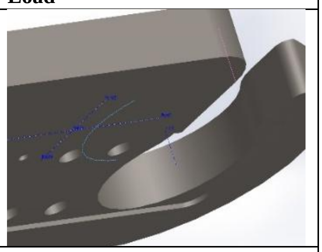
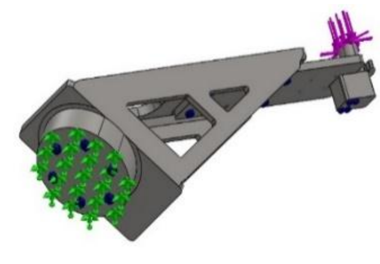
Figure5: Robotic gripper used for TO (description: A – robotKUKA KR 210, B – gripper, 1 – tightening equipment Atlas Copco, 2 – camera Kognex 7000, 3 – reductionpart, 4 – screwattachment, 5 – quick coupler Schunk SHK 125)

Table 1: Input properties of optimizing robotic gripper.

	Material properties	Volumetric properties
	Material: ASTM A36 Steel Yield strength: 250MPa Tensile strength: 400MPa Elastic modulus: 200GPa Poisson´s ratio: 0.26 Mass density: 7850 kg.m^{-3} Shear modulus: 79.3 GPa	Mass: 23.694 kg Volume: 0.00301834 m^{3} Density: 7 850 kg.m^{-3} Weight: 232.201 N

The next step after creating a 3D CAD model and defining the model's properties is FEM analysis of the initial shape of the robotic gripper. For the purpose of FEM simulation, it is necessary to define the load of the component and fixtures. These are listed in the following Table 2.

Table 2: Definition of fixtures and loads.

	Fixture	**Load**	
Schematic image			
Type	Fixed through holes	Apply force	

Details	Entities: 6 faces; fixed geometry; Reaction force: X = -191.709 N, Y =142.662 N, Z = -73.3125 N, Resultant force = 249.959 N	Entities: edge Force value: 250 N	

Creating and defining the volume elements of the model is a necessary condition for starting the FEM analysis. This is realized through the meshing of the model, where the relevant volume elements and node points are created. Figure 6 shows the created mesh model of robotic gripper needed for FEM. Basic information and definition of created mesh are follows: mesh type – solid mesh; mesher used – standard mesh; Jacobian points for high quality mesh – 16 points; element sizes – 14.4546 mm; tolerance – 0.72273 mm; mesh quality – high; total nodes – 110068; total elements – 66636; maximum aspect ratio – 55.94; percentage of elements with aspect ratio <3 – 98.2; percentage of elements with aspect ratio >10 – 0.0015.

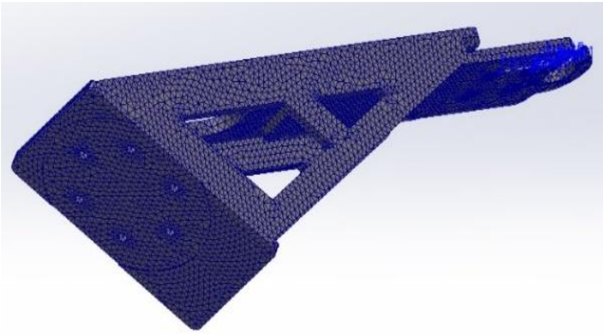


Figure6: Created 3D mesh model for FEM analysis

In the following figures the results of FEM analysis are displayed. Figure 7 shows the result of stress analysis, where the maximal stress value was recorded at 34.02 MPa. For this analysis the von Misses method was used. Figure 8 shows the result of displacement analysis, where the maximal displacement value of 0.2592 mm was recorded. Figure 9 displays the result of FEM analysis from the factor of safety (FOS) point of view. The lowest value of FOS is 7.348. On the basis of these results we can say that the robotic gripper meets the load conditions, but this gripper is unnecessarily oversized at its weight of 23.694 kg. That is the main reason why it is so useful to use the TO for gripper optimization.

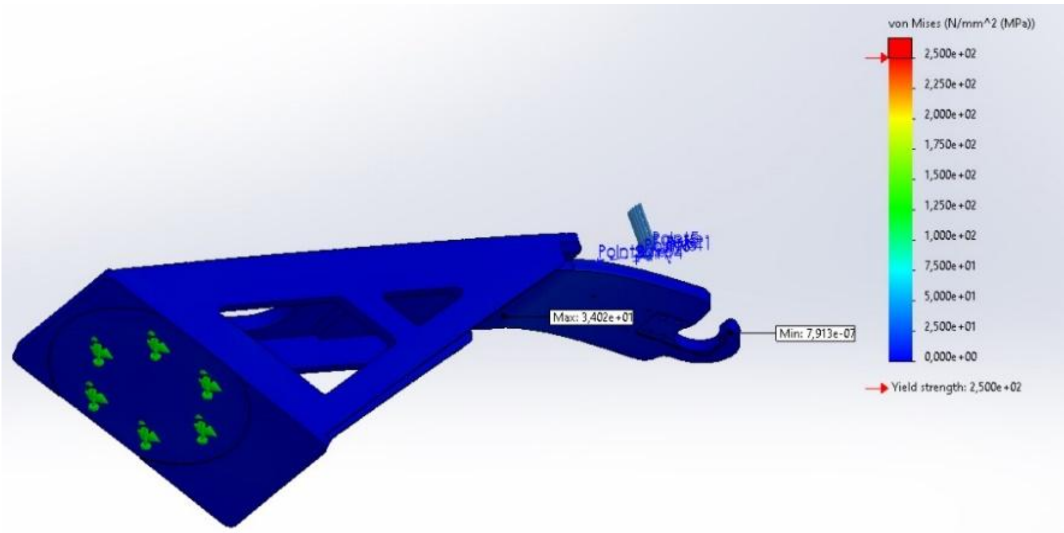


Figure7: Graphical interpretation of stress analysis

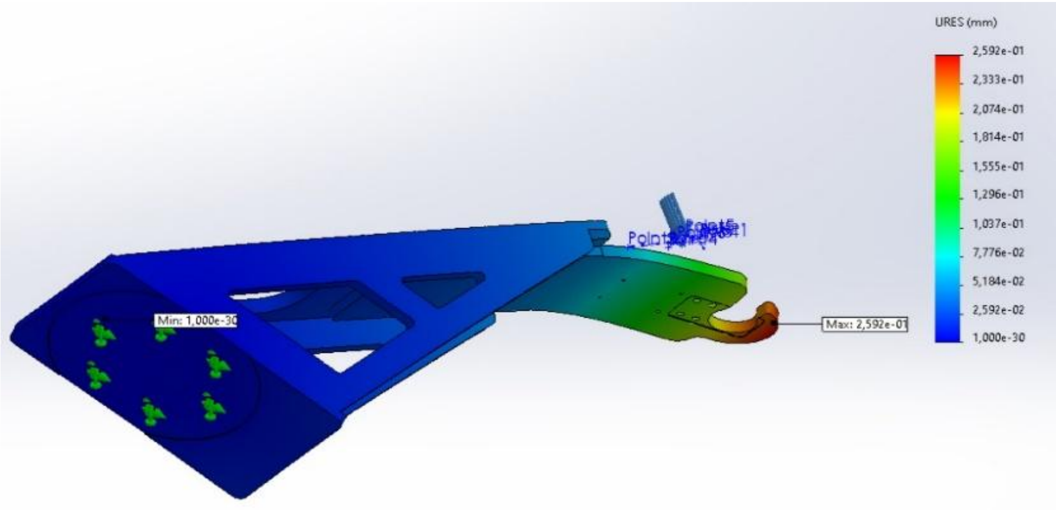


Figure8: Graphical interpretation of displacement analysis

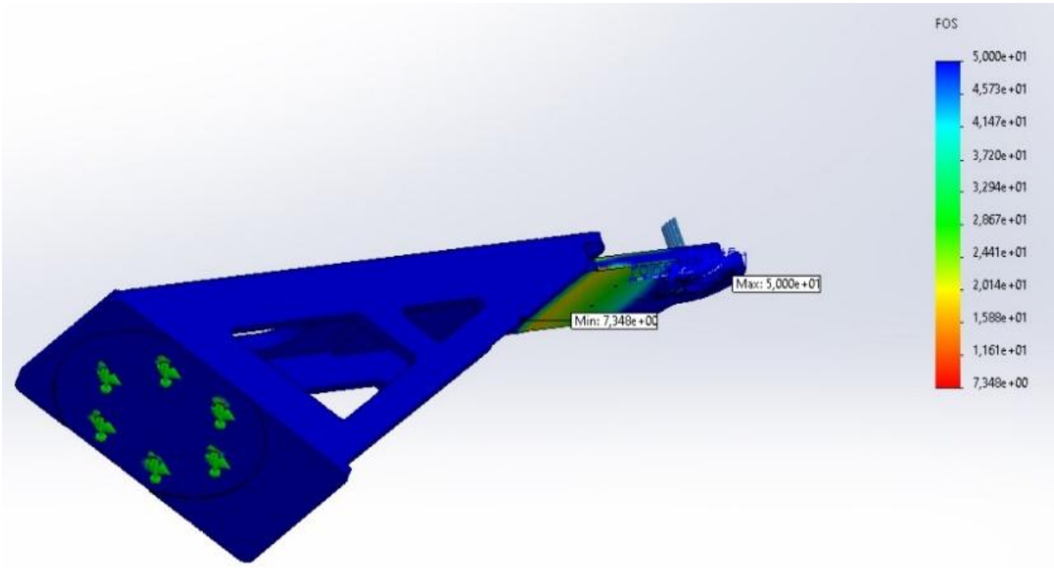


Figure9: Graphical interpretation of FEM analysis from factor of safety point of view

The steel part of the structure that we optimized consists of a welded parts at the end of which there is a coupling. This connects the gripper to the robot head and a removable steel plate on which a screwdriver with camera guidance is screwed. We have simplified the model intended for topological optimization compared to the model from FEM analysis. We got rid of the screw connections and created a one-piece part. We can subject the model prepared in this way to topological optimization.The goal of upcoming topological optimization was to reduce the weight of the gripper by 60% while maintaining the highest possible stiffness-to-weight ratio (Figure 10). We used the same input conditions as for the original FEM analysis of the gripper in terms of loads and fixtures.First of all is to important set up the goal of the topological optimization.

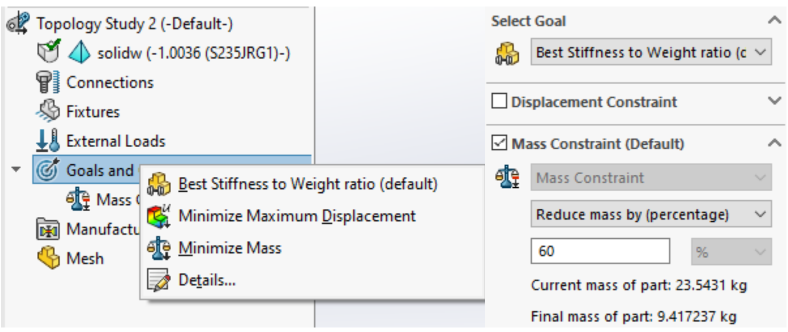


Figure10: Setting the goal of topological optimization in the SolidWorks environment

The next step is to define the manufacturing conditions within the topological optimization, marked as "Manufacturing Controls" in the SolidWorks software.In our case, it is critical to maintain accurate milled surfaces, contact surfaces and screw holes (Figure 11), that affect the final accuracy of the assembled tool. We kept these surfaces at a distance of 1 mm.

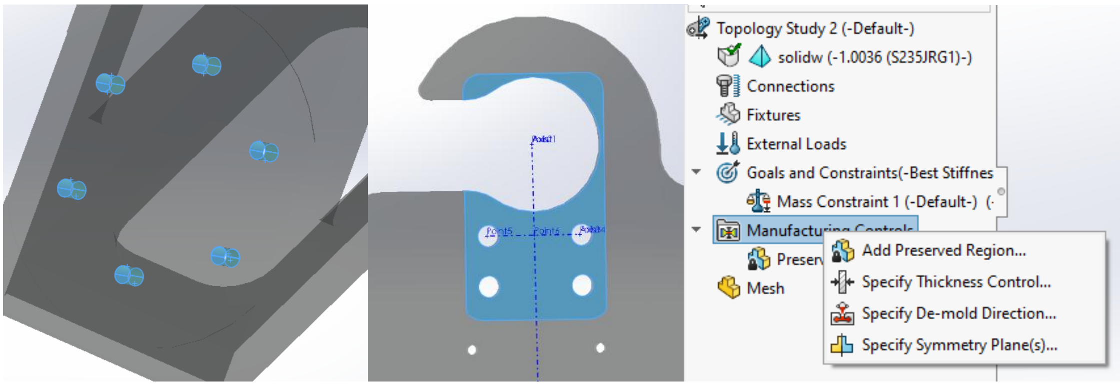


Figure11: Setting the preserved regions of model in manufacturing control

Manufacturing controls also offer us options such as specifying a minimum thickness. In the defined area, we can define the axis of symmetry if we require the result to be symmetrical. The last tool is the de-mold direction, and it is used for casting purposes. By using this method correctly, it ensures that castings can be manufactured according to the standard method.

The result of the study is "Smooth mesh" (Figure 12) which we saved as a model with which we can continue to work.The results had to be analysed and compared with the original model of the gripper. The analysis will show whether the proposed product is strong enough and suitable for our use. It is necessary to re-evaluate the defined boundary conditions if the stresses in a newly designed product are too high. For this purposes one again the FEM analysis was done, but no with the newly developed shape of the gripper. The results and comparison with the first FEM analysis can be seen in Table 3.

Table 3: Comparison of the results of the original and optimized part.

	Maximal stress (MPa)	Maximal displacement (mm)	Minimal factor of safety	Weight (kg)
Original shape of gripper	34.02	0.2592	7.348	23.694
Optimized shape of gripper	92.13	0.3522	2.714	9.41

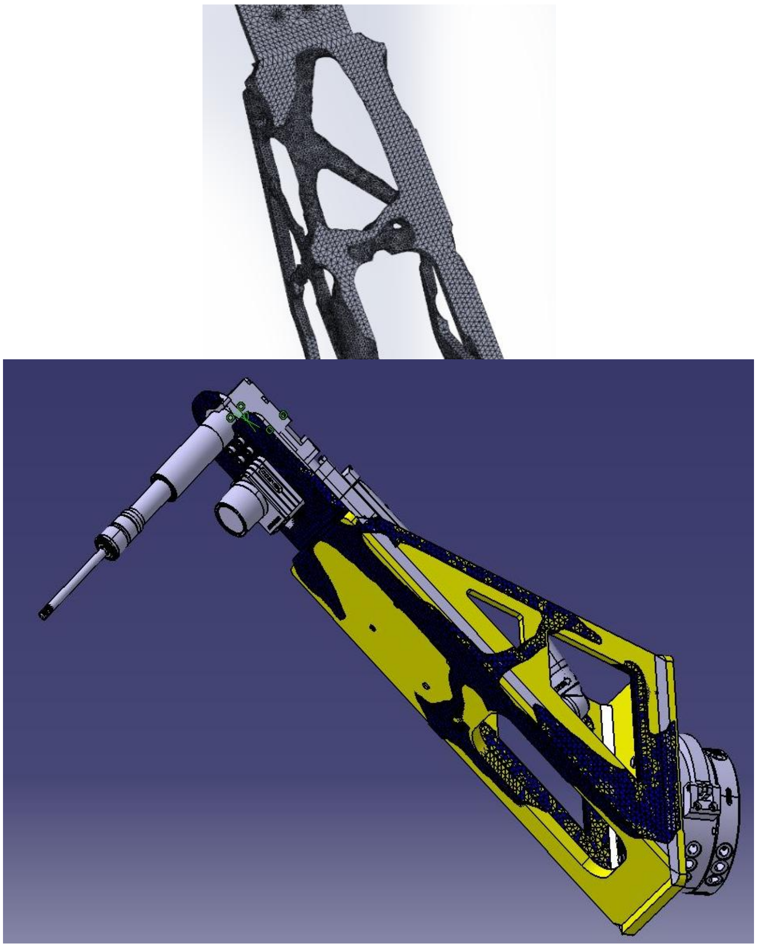


Figure12: Resulting shape after TO (left) and comparison of both shapes of gripper (right)

In Table 3, we can see that after optimization, there is a stress peak with a value of 92.13MPa, which is satisfactory. We are still achieving a minimum safety factor of 2.7. The highest displacement in the nodal point is approx. about 1 mm larger than before optimization. We managed to save 60% of the material and the optimized gripper weighs 9.41 kg.

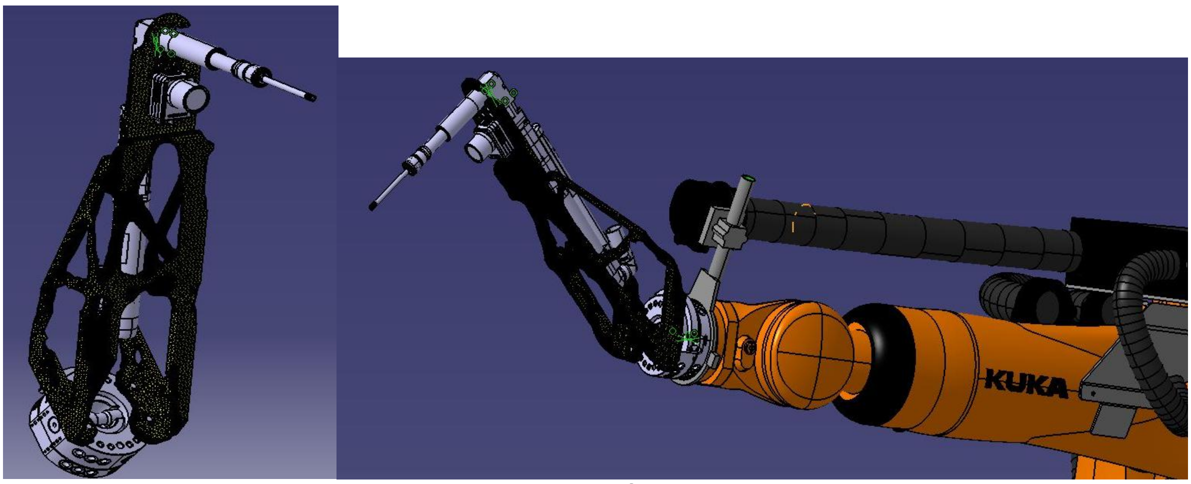


Figure13: Attaching the modified gripper to the robot

V. CONCLUSION

The decisive factor in the solution of topological optimization is not only the definition of the boundary conditions of the solved part. This is the choice of a suitable numerical topology method, but also the choice of the software solver itself. By default, SolidWorks uses the FFEPlus iterative solver unless certain conditions are met. FFEPlus uses advanced matrix reordering techniques. These techniques are approximate. A solution is assumed at each iteration, and iterations continue until the error becomes acceptable.

The purpose of this paper was to present the application of topology optimization in engineering practice. The TO method was demonstrated during the modification of the robotic gripper. Author´s intention was to present a step-by-step process for optimization of a robotic gripper. Acceptable results were achieved in the optimization of the topology.

The topological optimization process ensures that shapes are created only by additive manufacturing. . It may be difficult to produce such shape using conventional technologies and they are thus suitable for use in additive manufacturing. Also the price of production is limited factor.Hybrid production processes combine conventional and additive manufacturing technologies, where additive technologies create the final shape (created by topological optimization) by adding material layer by layer, and conventional technologies complete the manufactured work piece (using CAM module) from the point of view of the resulting surface quality and shape accuracy. Therefore, a necessary prerequisite is to modify the topologically modified shape of the manufactured work piece for the needs of CNC machining (Figure 4). For the sake of production efficiency, it is therefore necessary to combine CAM, topological optimization and robot simulation.Based on the mentioned input information and knowledge from practice, we came up with an idea and innovation (for the further research) in the form of integration of additive technology, CAM module, topological optimization and monitoring of the conventionalmanufacturing process.

Acknowledgments

The paper is a part of the research done within the project KEGA 033STU-4/2022 „Creation and implementation of a certified course for CAx systems with elements of artificial intelligence in the teaching of mechanical engineering“ funded by the Ministry of Education of Slovak Republic and to the Slovak Academy of Sciences, and the STU grant to support young researchers „Topological optimization in the robotic machining process“, acronym: Torsion, funded by the Slovak University of Technology in Bratislava.

REFERENCES

[1]. RozvanyG.I.N. (2009) "A critical review of established methods of structural topology optimization",Structural and Multidisciplinary Optimalization, Vol. 37, pp. 217-237.

[2]. Gao J., Gao L. and Xiao M. (2022) "Isogeometric topology optimization. Methods, applications and implementations",Springer Nature Singapure, pp. 222.

[3]. Tyflopoulos E., Lien M. andSteinert, M. (2022) „Optimization of Brake Calipers Using Topology Optimization for Additive Manufacturing“, Applied Sciences, Vol. 11, pp. 2-21.

[4]. Garcia-Granada A. A., Catafal-Pedragosa J. andLemu H. G. (2019) „Topology optimization through stiffness/weight ratio analysis for a three-point bending test of additive manufacturing parts“, IOP Conference Series: Materials Science and Engineering, Vol. 700, pp. 1-11.

[5]. Christensen P. W. and Klarbring A. (2009) „An introduction to structural optimization“, Springer, Nederland, ISBN 978-1-4020-8666-3.

[6]. Huang X. and Xie Y. M. (2008) „A new look at ESO and BESO optimization methods“, Structural and Multidisciplinary Optimalization, Vol. 35, pp. 89-92.

[7]. Bhaumikkumar, A.K. (2014) "Topology optimalization of Continuum Structure Genetic Algorithm Optimization Tool",Master Thesis, School of Engineering, R.K. University.

[8]. Grujicic M., Arakere P., Pisu P., et al. (2008)„Application of Topology, Size and Shape Optimization Methods in Polymer Metal Hybrid Structural Lightweight Engineering“, Multidiscipline Modeling in Materials and Structures, Vol.4, pp. 305-330.

[9]. FolgerO. (2021) "Topology optimalization of a mounting bracket", Degree Thesis,Massachusetts, online https://www.solidworks.cz/, cited 2022-11-02.

[10]. "SIMP method for topology optimization" (2021), Massachusetts, online at: https://help.solidworks.com/2022/czech/SolidWorks/cworks/c_simp_method_topology.htm?verRedirect=1, cited 2022-11-02.