The International Journal of Engineering and Science (IJES)
|| Volume || 11 || Issue || 6 || Series I || Pages || PP 09-15 || 2022 ||
ISSN (e): 2319-1813 ISSN (p): 20-24-1805

Topology Optimization Of The Plastic Holder By Computational Methods

Samuel Vilkovský, Janka Majerníková[1]
[1]*Institute of Technology and Material Engineering,*
Faculty of Mechanical Engineering, Technical University of Košice, Košice, Slovakia

ABSTRACT
Component weight reduction is one of the biggest trends today, and not just in the automotive and aerospace industries. Topological optimization is a method by which we can obtain structures with significantly reduced weight with unchanged function and component safety. This article deals with the topological optimization of a plastic bracket made by additive manufacturing. Topology optimization and subsequent verification of the suitability of the new optimized model will be performed based on three independent static structural simulations using Ansys. The holder is made of ABS plastic.
Keywords *–topology optimization, optimization, simulation, ANSYS*

Date of Submission: 24-06-2022 Date of Acceptance: 06-07-2022

I. INTRODUCTION

Topology optimization has been remarkably developed over the last few decades in theoretical studies as well as in practical applications. Topology optimization has been developed as an advanced structural design methodology for generating innovative lightweight and high-performance structures that are difficult to obtain with conventional ideas. [1,2,3]

Topology optimization provides answers to the basic engineering question - how to redistribute the material within the prescribed design area to achieve the best design performance? This concept was initiated for mechanical design problems but has spread to a wide range of other physical disciplines, including fluids, acoustics, electromagnetics, optics, and combinations thereof. By redistributing the material distribution and correspondingly supporting parts, topology optimization has been recognized as one of the most promising techniques in structural design. [4,8]

Topology optimization has been used by civil and mechanical engineers for several years, for instance to minimize the use of materials for structures while retaining their mechanical strength. Topology optimization is a mathematical method that spatially optimizes the distribution of material within a defined area by meeting predetermined constraints and minimizing a predefined cost function. For such an optimization process, the three key elements are the design variables, the cost function, and the area constraints within the simulated object. [5,6,7,9]

Primary objective of topology optimization is to find the optimal distribution of material within a specified region. Sole known parameters for resolving the problem are the applied loads, the volume of the structure, the support conditions and possibly some design limitations, such as the size and location of some fixed surfaces. Conventional construction methods often do not make the most of structures, as opposed to the new production methods, such as additive production, which removes design constraints and allows to design lighter and stronger constructions. [5]

Additive production techniques have become very popular in recent years. Topology optimization has primarily gained popularity as a computational design tool for reducing the weight of components in the automotive and aerospace industries. Additive manufacturing (AM), also known as 3D printing, produces parts via joining material layer-by-layer. [10,11,12]

Additive manufacturing opens the opportunity to manufacture complex structures especially for topology optimization constructions. Without extra tools, molds and complicated procedures, AM is flexible for any complex structures, which not only saves cost of production but shortens the manufacturing cycle especially in rapid prototyping and small batch production. In addition, AM’s potential for complex structures also contributes to the design of integral structures that reduces the number of parts and assembly processes. [12,13,14,15]

For topology optimized structures, AM enables engineers to get rid of the limitations of conventional manufacturing techniques and focus attention to design high-performance and lightweight structures. In turn, topology optimization is an effective approach for additively manufactured products with innovative and lightweight configuration. The integration of topology optimization and AM is an essential way to achieve matching of structural design and manufacturing. [3]

Structure optimization methods are divided into three categories: shape optimization, sizing optimization, and topology optimization. In characteristic sizing optimization, the aim is to find the optimal thickness of a structure. On the other hand, the objective of shape optimization is to find optimum shape of the structure. Topology optimization is associated with the determination of features such as shape, location, size, and number of the holes in the structure. [5,7]

II. COMPUTER PROCESS SIMULATION

Due to its requirements relating to high speed and large computer capacity, computer modeling of forming processes using the finite element method was not so long ago available only for specialized workplaces and groups of qualified users. With the development of technology, this method is currently widely used. Computer simulation provides support that allows to modify or enhance the original design and quickly evaluate these changes before starting to develop an expensive prototype. With sufficient accuracy it also provides the possibility of designing the production process before the actual production. With the use of suitable programs, it is possible to simulate forming processes in all their complexity, while computer simulation using the finite element method gives great opportunities to monitor their progress. Currently, it is possible to use programs such as ANSYS, Abaqus, Fusion 360 and others.

ANSYS simulation program

For modeling and simulation of more complex, multi-dimensional engineering tasks in the areas of heat transfer, fluids, statics, dynamics, electromagnetism, or other scientific disciplines, where the computational area consists of many nodes, a solution without computer support is long-lasting and, in some cases, unrealistic. In the recent past, several software products have been developed that eliminate the numerical complexity of such processes.

One such system is ANSYS software (Figure 1), which can be regarded as one of the most widely used computing software. The term ANSYS is generally considered to be a whole package of tools for solving engineering tasks in the field of static and modal analysis, dynamics, mechanics, thermodynamics, flow, or topology optimization. After creating the model in external software, or directly in the ANSYS module and defining the material properties, the topology optimization simulation consists of three parts. Initial static simulation on the original model, topology optimization and subsequent validation of the modified geometry. In the static structural simulation environment, we define the boundary conditions in both cases (size, force, supports, etc.). When optimizing the topology, it is necessary to define areas that cannot be subject to optimization in terms of component function, as well as the main objective of optimization, such as mass reduction.

III. EXPERIMENTAL MODEL AND MATERIAL

The plastic holder model is shown in Figure 1. The plastic holder is made of ABS material. In experimental research, we will use Ansys software to simulate three different load scenarios in which the plastic holder is loaded. In three static simulations the holder will be stressed by different forces in different directions.

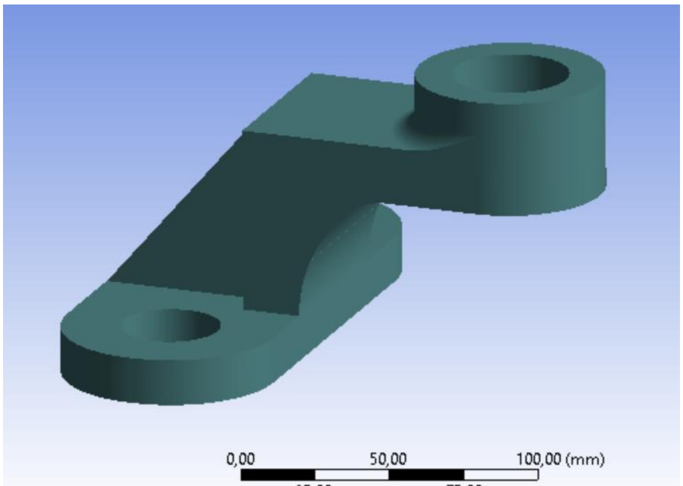

Figure 1 CAD model of the plastic holder

The mechanical properties of the used material are shown in Table 1.

Table 1Mechanical properties of the material

ABS material	
Properties	**Value**
Density [kg/m^3]	1030
Yield strength [MPa]	27.44
Ultimate strength [MPa]	36.26
Young´s modulus [MPa]	1628
Poisson´s ratio [-]	0.4089

The topology optimization of the plastic holder will be based on three independent static structural simulations performed in the Ansys simulation program. For all three load scenarios, we will use the boundary condition fixed support to prevent a selected geometry from moving or deforming. Figure 2 shows an area which is fixed.

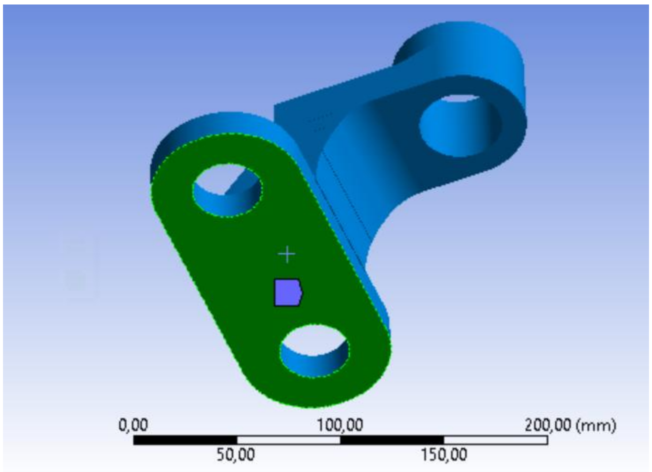

Figure 2 The surface which is prevented from deforming

The first load scenario represents the action of force on the upper part of the holder with the force F_1 = - 650N. Graphic display of the force action is illustrated in the Figure 3.

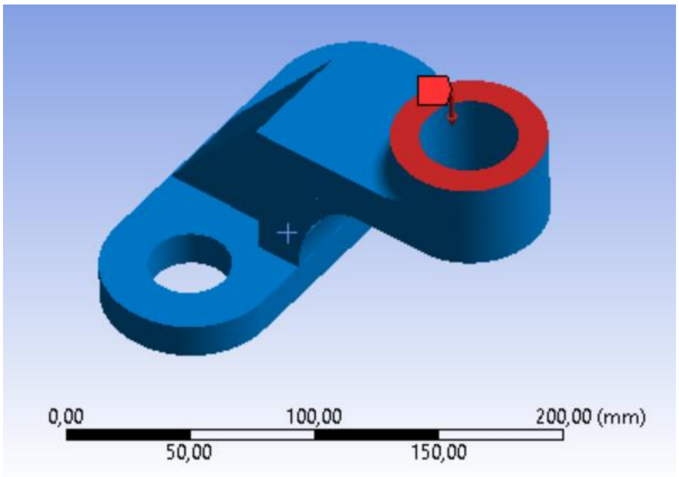

Figure 3 Graphical representation of the load case 1

In the second simulation, a force of magnitude F_2 = 1300 N acts on the inner surface of the hole in the direction of the X-axis. The representation force action area and its direction are shown in Figure 4.

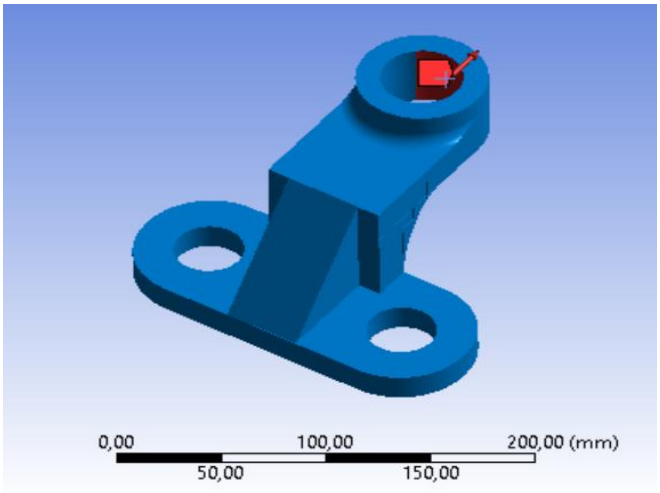

Figure 4 Graphical representation of the load case 2

The third simulation combines the previous two cases and considers the simultaneous action of the force on the upper part of the holder, like load scenario 1 and the force in the X-axis direction, like load scenario 2. The magnitude of the applied forces is presented in the Table 2.

Table 2 Magnitude of the applied forces

Model	Force [N]		
	X	Y	Z
Load case 1	-	-650	-
Load case 2	1300	-	-
Load case 3	500	-500	-

In the simulation we used tetrahedral elements (Tet10) as well as hexagonal elements (Hex20) The simulation results are the values of maximum stress (von Misses), maximum total deformation, maximum directional deformation, and safety factory. The obtained values of these metrics for all three load scenarios are in Table 3.

Table 3 Simulation results of the model

Model 1	Max. stress (von-Misses) [MPa]	Max. total deformation [mm]	Max. directional deformation [mm]	Safety factor [-]
Load case 1	7.278	1.390	-1.255 (Y-axis)	4.014
Load case 2	6.028	1.115	0.682 (X-axis)	4.654
Load case 3	7.203	1.492	-1.308 (Y-axis)	3.894

The maximum value of stress in the plastic holder was at the first load case, when the part was stressed with a force F_1 = -650 N. In this case, the value of the total deformation was 1.390 mm, and the safety factor was 4.014. In the second simulation, under the action of force F_2 = 1300 N the plastic holder was the least stressed. The measured tension reached 6.028 MPa and the safety factor was 4.654.

With the simultaneous action of forces in the X and Y direction the stress on the holder reaches a maximum value of 7.203 MPa. The holder reaches the largest total deformation of 1.492 mm while the value of the safety factor reached the value of 3.894.

Topology optimization will be performed based on three static structural simulations. The topology optimization simulation will be made to reduce the weight of the plastic holder while maintaining the functionality and safety of the part. Figure 5 shows tension gradient for all three load scenarios along with a maximum and minimum tension location indicator.

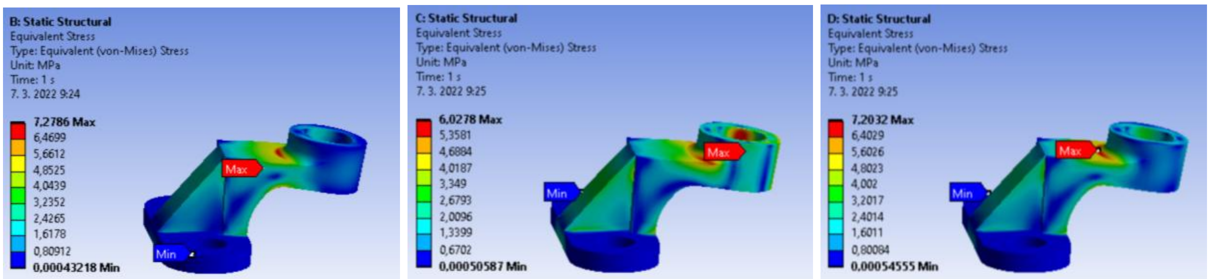

Figure 5 Stress gradient for all three load cases

Due to the function of the holder, we must define the areas that are needed for the proper functioning of the part, and which will not be subject to optimization. These areas are displayed with red colour in the Figure 6.

Figure 6 Excluded regions

Our objective is to optimize the shape and weight of the part. In the topology optimization module, we defined that we want to retain at least 40% of the mass from the original design with respect to the results of all three static structural simulations. Topology optimization results are shown in Figure 7.

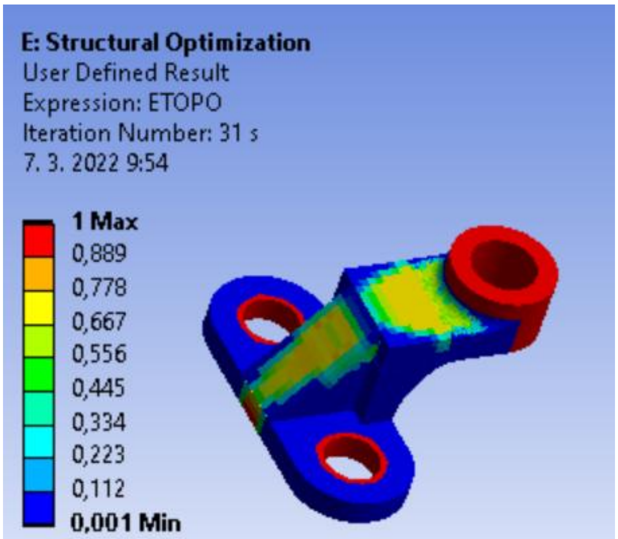

Figure 7 Topology optimization results

Excluded and necessary surfaces of the component are shown with red colour (values from 0.6-1.0). Surfaces in the range of 0.0-0.4 can be removed for given boundary conditions. Regions with range values 0.4- 0.6 are marginal areas and their need may or may not be necessary. Based on these data, Ansys automatically created a new bracket model (Figure 8).

Figure 8 Bracket model created by Ansys topology optimization module

Based on the data obtained from the simulation, a new optimized model was created (Figure 9).

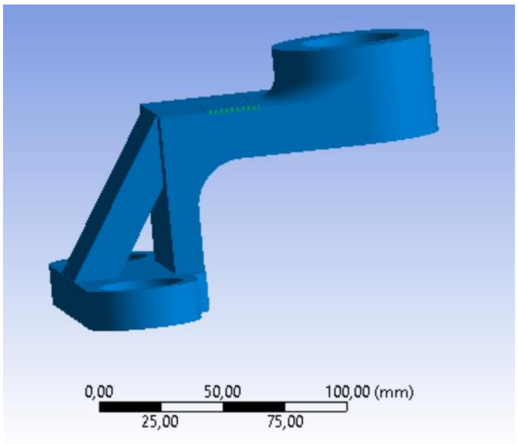

Figure 9 Optimized bracket model

The new shape of the holder was verified by same load cases as we loaded the original model.

Table 4 Simulation results of the new model

Model 2	Max. stress (von-Misses) [MPa]	Max. total deformation [mm]	Max. directional deformation [MPa]	Safety factor [-]
Load case 1	15.593	3.6234	3.3496	1.7989
Load case 2	15.251	2.3284	1.2451	1.8392
Load case 3	14.47	3.6731	-3.3354	1.9385

Compared to the original model, we can see an increase in the values of maximal stress, total deformation, and directional deformation. The safety factor dropped significantly in all load cases, but its value is still acceptable. Table 5 compares the key indicators values of the original model (Model 1) and the optimized model (Model 2) by topological optimization in the Ansys environment.

Table 5Results comparison of the models

Properties	Model 1	Model 2
Volume [cm^3]	471.8	318.28
Mass [g]	493.03	332.6
Safety factor (load case 1)	4.014	1.7989
Safety factor (load case 2)	4.654	1.8392
Safety factor (load case 3)	3.894	1.9385

IV. CONCLUSION

In this paper, a new plastic bracket model was designed based on the results of three independent static structural simulations with the use of topological optimization in the ANSYS simulation software. The main objective was to reduce the weight of the part while maintaining the safe functioning of the part. The new model design created by the topological optimization module shows a weight reduction of 153.52 g.

By reducing the weight of the component, there was a significant increase in stress values in the component. In the first load scenario, the stress increases by 114.25% from 7.278 MPa to 15.593 MPa. In the second load scenario, we register even greater increase in von-Misses stress, specifically by 153% from 6.028 MPa to 15.251 MPa. We observe the smallest increase in stress in the third case where the value increased by 100.89%.

As expected, the value of the safety factor also decreased, but in all three cases it reached the permissible value.

ACKNOWLEDGEMENTS

The authors are grateful to VEGA 1/0384/20 and KEGA 036TUKE-4/2021.

REFERENCES

[1]. G. Allaire, F. Jouve, H. Maillot.: Topology optimization for minimum stress design with the homogenization method. In: Structural and Multidisciplinary Optimization. Volume 28, 2004, p. 87–98. ISSN: 1615-1488.
[2]. X. Huang, Yi Min Xie.: Evolutionary Topology Optimization of Continuum Structures. 2010, ISBN: 978-0-470-74653-0.
[3]. J. ZHU et al.: A review of topology optimization for additive manufacturing: Status and challenges. In: Chinese Journal of Aeronautics. Volume 34, Issue 1, 2021, p. 91-110. ISSN: 1000-9361.
[4]. J. Deaton, R. Grandhi: A survey of structural and multidisciplinary continuum topology optimization: post 2000. In: Structural and Multidisciplinary Optimization. Volume 49, 2014, p. 1-38. ISSN: 1615-1488
[5]. W. Zhang, J. ZHU, T. GAO.: Topology optimization in engineering structure design. London: Oxford, UK: ISTE Press Ltd.; Elsevier, 2016. ISBN: 978-1-78548-224-3.
[6]. Y. Rong et.al.: Structural topology optimization with an adaptive design domain. In: Computer Methods in Applied Mechanics and Engineering. Volume 389, 2022, ISSN: 0045-7825.
[7]. G. Tejani et al.: Size, shape, and topology optimization of planar and space trusses using mutation-based improved metaheuristics. In: Journal of Computational Design and Engineering. Volume. 5, Issue 2. 2018, p. 198-214. ISSN: 2288-4300.
[8]. O. Sigmund, K. Maute: Topology optimization approaches. In: Structural and Multidisciplinary Optimization. Volume 48, 2013, p. 1031–1055. ISSN: 1615-1488.
[9]. I. Rosinha et al.: Topology optimization for biocatalytic microreactor configurations. In: Topology optimization for biocatalytic microreactor configurations. Volume 37, 2015, p. 1463-1468. ISSN: 1570-7946.
[10]. G. Vantyghem et al.: 3D printing of a post-tensioned concrete girder designed by topology optimization. In: Automation in Construction. Volume 112, 2020, ISSN: 0926-5805.
[11]. J. Hong Zhu, Wei-Hong Zhang, L. Xia: Topology optimization in aircraft and aerospace structures design. In: Archives of Computational Methods in Engineering. Volume 23, 2015, p. 595- 622. ISSN: 1134-3060.
[12]. Y. Zheng et al.: Tissue transformation mold design and stereolithography fabrication. In: Rapid Prototyping Journal. Volume 23, 2017, p.162-168. ISSN: 1355-2546.
[13]. E. Spišák, J. Majerníková, M. Tomáš, J. Varga: Teória konvenčných technológií I. 1. vyd. Košice: Technická univerzita v Košiciach, 2019, p. 169. ISBN 9788055334158.
[14]. P. Mulidrán, E. Spišák, M. Tomáš, V. Rohaľ, S. Vilkovský: The Influence of Numerical Parameters on the Springback Prediction of U-Shaped Part. In: International Journal of Engineering and Science. Volume 10, Issue 4, 2021, p. 27-30. ISSN 2319-1805.
[15]. Ľ. Dulebová, J. Majerníková: Progresívne výrobné technológie, 1. vyd. Košice: Technická univerzita v Košiciach, 2021, p. 169. ISBN: 978-80-553-4003-6.

Samuel Vilkovský, et. al. "Topology Optimization Of The Plastic Holder By Computational Methods." *The International Journal of Engineering and Science (IJES)*, 11(6), (2022): pp. 09-15.