https://doi.org/10.32056/KOMAG2024.1.3

Optimization of the topology of the structural component of a magnetic robot

Received: 01.02.2024

Accepted: 29.02.2024

Published online: 28.03.2024

Paweł PAC [1*], Dariusz PROSTAŃSKI [2]

Author's affiliations and addresses:

[1] PONAR Wadowice S.A., Św. Jana Pawła II 10, 43-170 Łaziska Górne

[2] KOMAG Institute of Mining Technology, Pszczyńska 37, 44-101 Gliwice, Poland

*** Correspondence:**

e-mail: pawel.pac@ponar-wadowice.pl

Abstract:

The article focuses on topology optimization using the Finite Element Method (FEM) in the context of mechanical design and structural engineering. The analysis was based on a robot arm component, which is important for minimising its mass due to the way it moves on steel structures such as mining shaft towers. FEA simulations of the non-optimized workpiece were carried out, followed by iterative mass reduction of the workpiece by editing its geometry. Two approaches were compared: manual weight reduction by the designer and topology optimization using the Shape Generator. The presented results answer the question posed in the introduction whether topology optimization using the Shape Generator can yield better results than manual optimization based on the engineering intuition of an experienced designer. The paper also answers the question of at which stage of design it is better to use tools such as the shape generator. It is confirmed that topology optimization can significantly reduce the weight of the designed component, which is important especially for structures subject to special requirements, such as in the case of equipment used in mining. The authors describe what the effectiveness of the optimization may depend on.

Keywords: FEM, CAD systems, topology optimization, shape generator, Autodesk Inventor

Publisher: **KOMAG Institute of Mining Technology, Poland**

1. Introduction

Topology analysis, topology optimization using FEA (Finite Element Analysis), genetic algorithms, gradient-based shape optimization are all related techniques used in engineering, particularly in the fields of mechanical design and structural engineering [1-5].

Topology analysis involves examination and evaluating the optimum form of a structure's geometry in order to achieve the best strength performance or to meet other specified criteria. By 'topology' here we mean the overall form and distribution of the material in the component structure. Topological analysis aims to find the optimum distribution of material in an object in order to minimise its mass while maintaining the required strength properties. In other words, the aim is to find a structure with optimum material performance [6, 7].

The finite element analysis (FEA) is a numerical technique used to model and analyse the mechanical behaviour of structures. FEA divides the structure under study into a finite number of elements and then calculates the reactions of the structure based on physical equations such as equilibrium equations and elasticity equations of materials. FEA is widely used in the analysis of stress, deformation, temperature, heat flow, etc. [5, 8, 9].

The optimization of topology using FEM (Finite Element Method) is a technique that combines topology analysis with the application of finite element analysis. The process of topology optimization involves iteratively changing the material distribution within the structure and evaluating performance based on predefined criteria (Fig. 1). Based on the results of FEM analysis, it is possible to determine which areas of the structure are subjected to the highest loads and where material reduction can be achieved without compromising strength parameters. Various algorithms, such as genetic algorithms or gradient methods, are employed in the process of topology optimization to find the best material distribution that satisfies specific optimization criteria, such as minimal mass, minimal stress, and minimal deformation [6, 10-13].

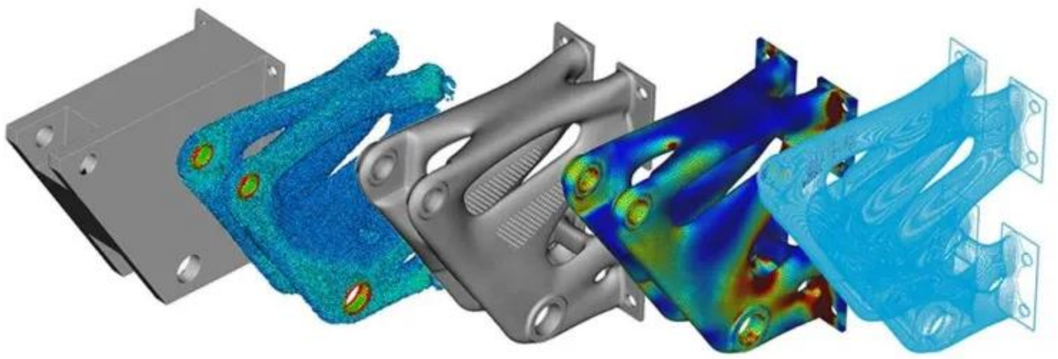

Fig. 1. Example of part optimization [14]

Optimising topologies using FEA can bring many benefits, such as reducing the weight of a structure, increasing its strength performance, and reducing manufacturing costs. This process is used in many fields such as the design of mechanical components, vehicles, aircraft, etc. [1, 15-17].

In this article, a comparison of optimization results using the FEM environment has been proposed for a modified structural element subjected to various methods.

2. Introduction

To illustrate the potential application of topology optimization, a passive wheel element of a prototype device, namely a magnetic robot (Fig. 2), was chosen for the maintenance of ferromagnetic surface structures of mining shaft towers. The selected object is an arm element, as depicted in Fig. 3. For the purpose of this consideration, the same robot element was chosen for comparison of alternative methods of FEM simulation results analysis in CAD software [18]. Due to the robot's mode of movement on steel constructions, which involves the use of magnet arrays for adherence at various angles, minimizing the robot's mass is crucial.

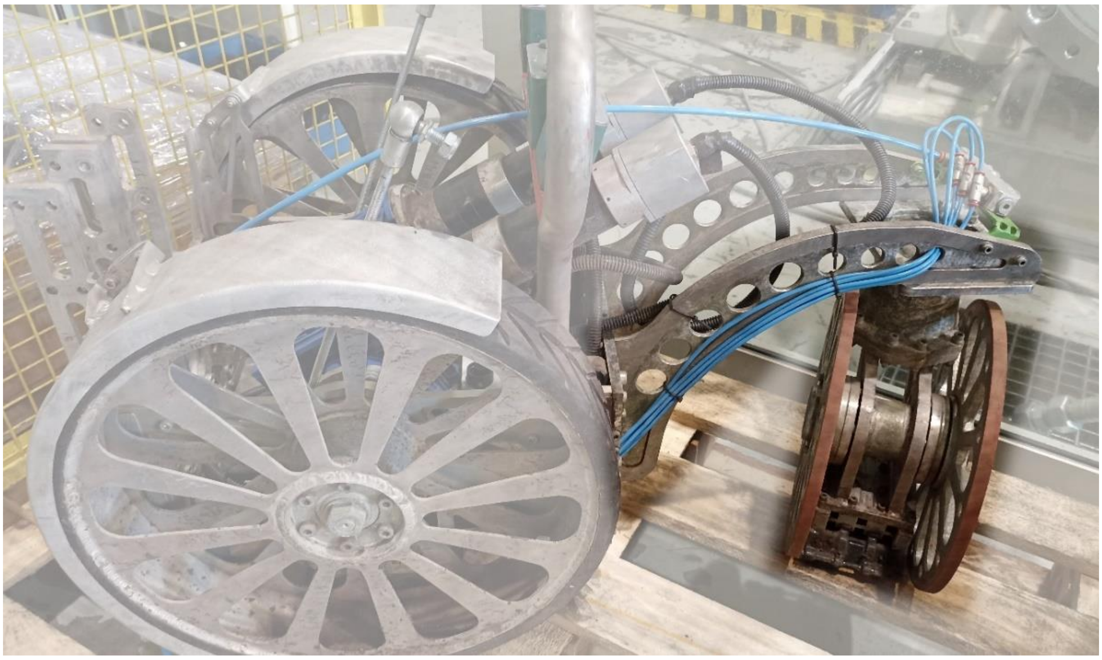

Fig. 2. Magnetic Robot Chassis

The most unfavorable boundary conditions were adopted, i.e., the longest arm of the cleaning module with three attached nozzles (Fig. 4). During the operation of the nozzles, the recoil force presses the passive wheel with a rounded value of 1 kN acting on the analyzed bracket.

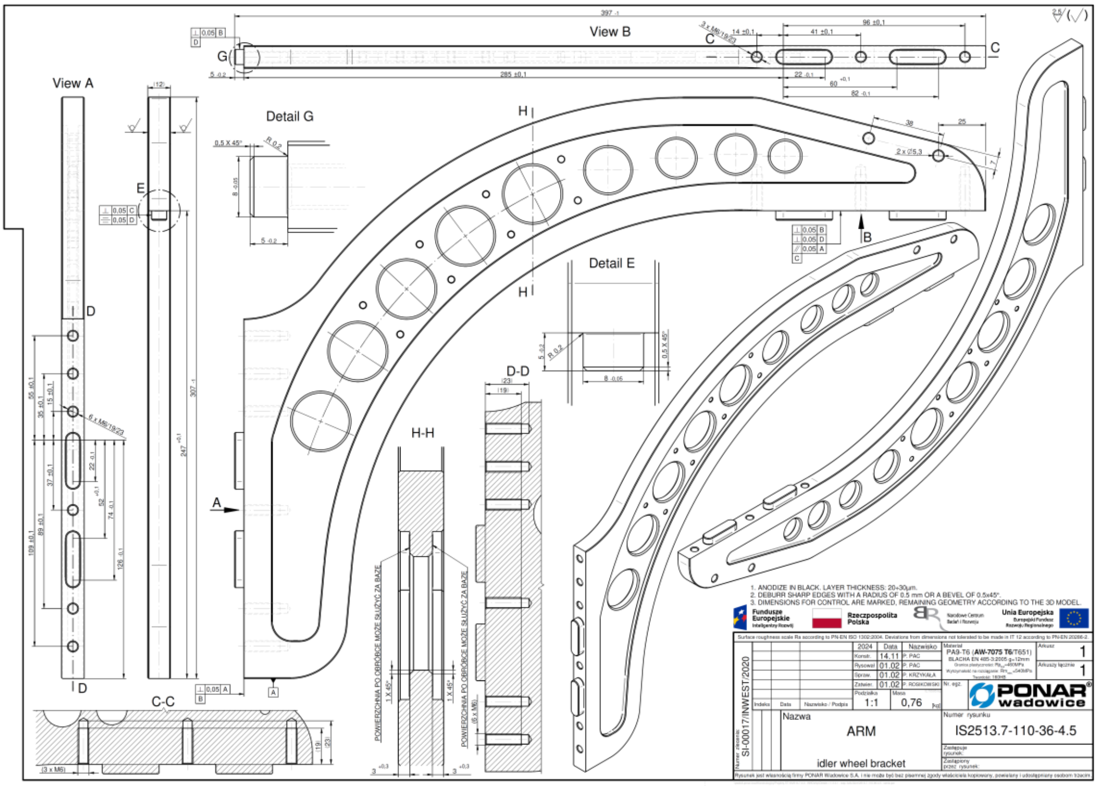

Fig. 3. Bracket of the passive wheel of the magnetic robot

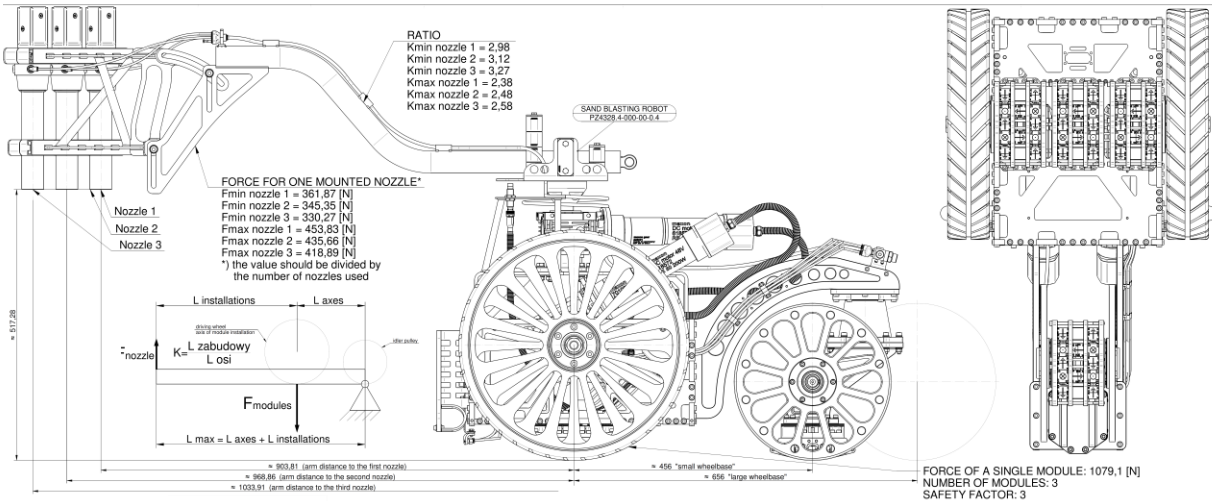

Fig. 4. The force exerted on the robot and the passive wheel due to recoil

The magnetic robot, developed as part of the project POIR.01.02.00-00-0105/19, was created using Solid Edge 2020 software. The strength analyses and optimization of the magnetic robot components are performed using Autodesk Inventor 2017 Professional software.

These are software programs used at PONAR Wadowice S.A., which is significant due to the potential for utilizing the results for the company's needs even after the completion of the Implementation Doctorate Programme.

3. FEM simulation results (non-optimized component)

The general principles in design are known, which pertain to identifying areas in components that do not bear load. Based on experience and these well-known principles, a designer is able to create a component with reduced mass, for example, by incorporating holes in the part of the element that does not carry the load (presumption made during the design stage). An example developed according to the mentioned principle is presented in Fig. 5, with a component mass of 0.792 kg.

Fig. 5. The designed component based on the experience of the designer

Typically, in businesses (custom production, small batches), the first step is to reduce the mass of structural elements. Subsequently, FEM simulations are conducted to verify the design. If the component meets the simulation criteria (usually permissible displacement or load), detailed drawings are created, and the component is released for production.

Due to the nature of this specific component, it is crucial, from the perspective of the device, for the detail to retain appropriate rigidity. The FEM simulation results are presented in Fig. 6 and Fig. 7, with a maximum displacement value of 0.74 mm, which is below the established criterion of 1 mm. Additionally, the reduced stresses remain well below the yield strength limit ($Rp_{0.2} \geq 230$ MPa) for the 6061 series aluminum alloy.

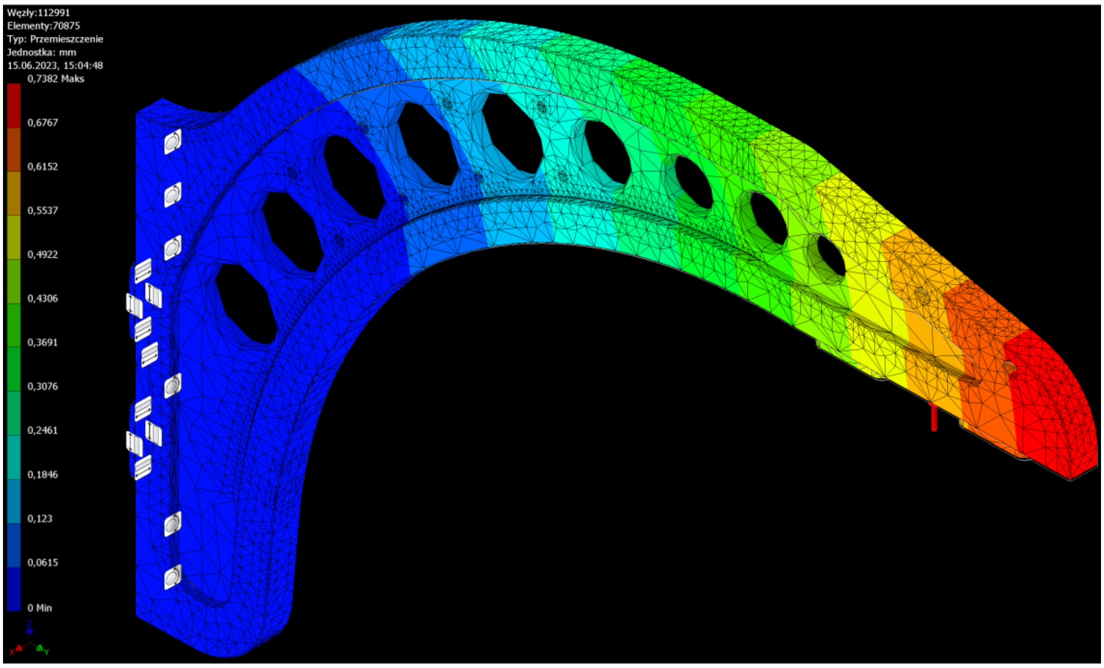

Fig. 6. Simulation result - displacement

Using a simple geometry editing tool, namely the face offset (Fig. 8), the model's geometry was modified by 1 mm, resulting in a reduced mass of 0.695 kg. Subsequent simulations were conducted, yielding an increased displacement value of 0.85 mm (Fig. 9), and the stress distribution is depicted in Fig. 10.

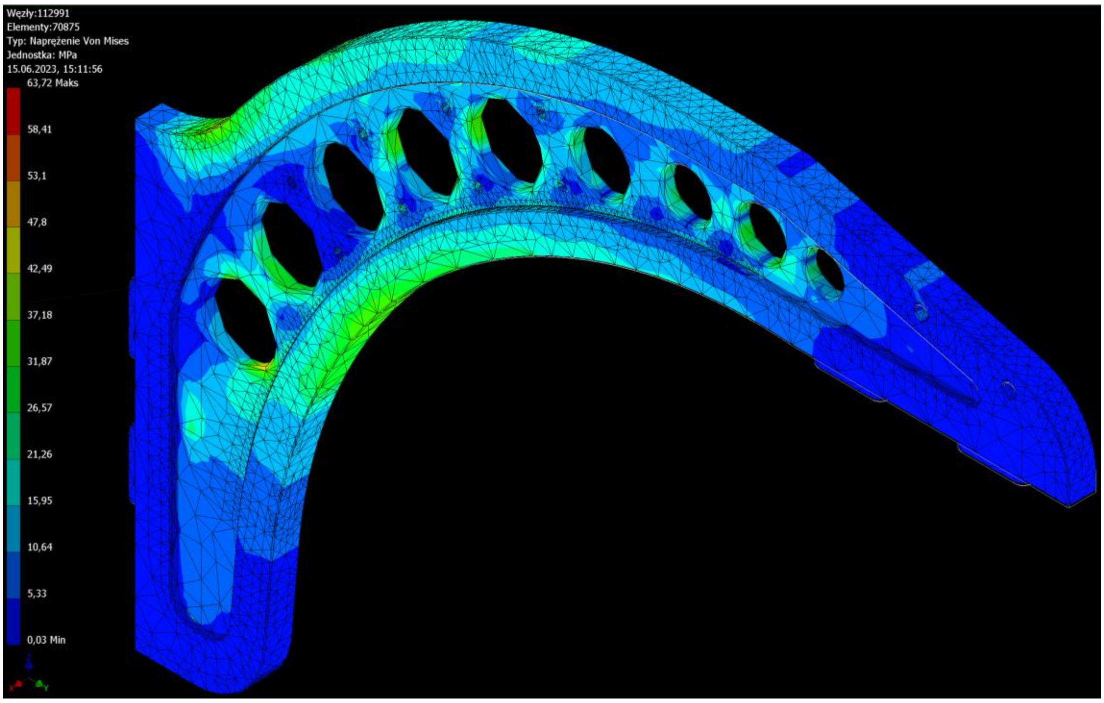

Fig. 7. Simulation result - reduced stress

The displacement of the modified element by 1 mm still falls within the acceptable range. In the next step, the offset value was increased from 1 mm to 2 mm (Fig. 11). Not all faces could be

Publisher: **KOMAG Institute of Mining Technology, Poland**

offset by 2 mm (Fig. 11), and the software reported an error. Achieving a 2 mm offset required dividing the process into three operations, which extended the time spent working on the detail.

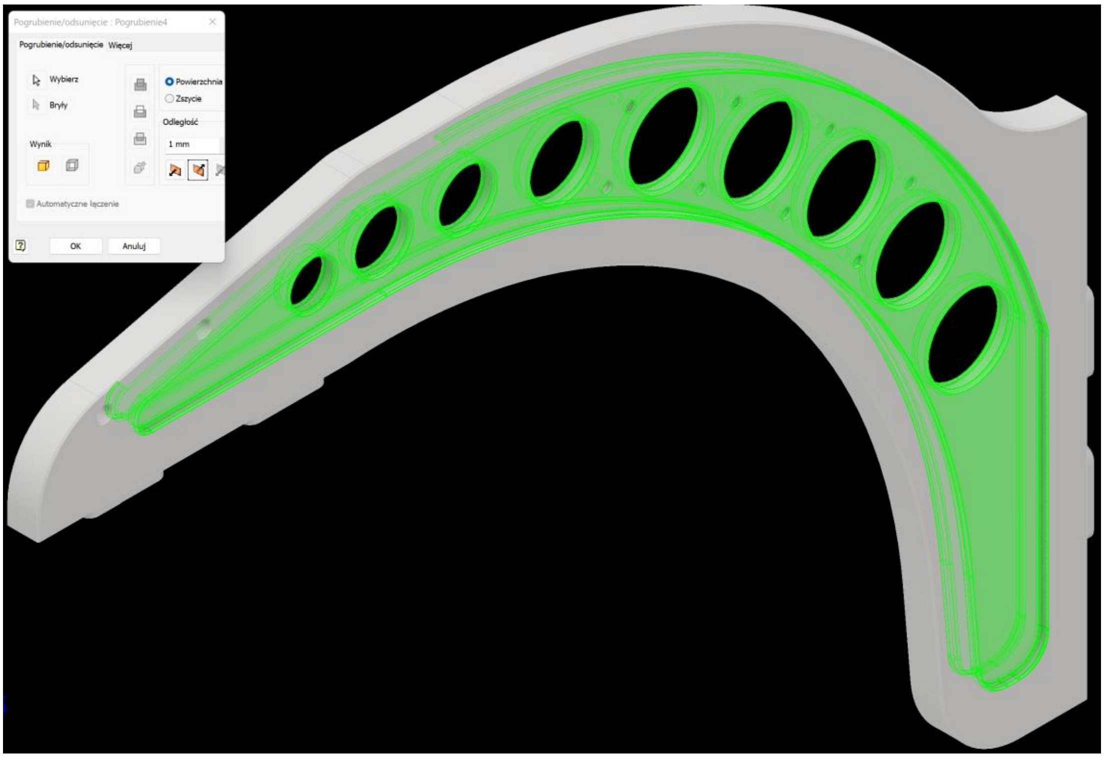

Fig. 8. Editing the element by offsetting the face by 1 mm

The FEM simulation result (Fig. 12) indicated an increase in displacement to a level of 1.05 mm, which slightly exceeded the adopted criterion. The geometry was not modified, and no further strength simulations were conducted.

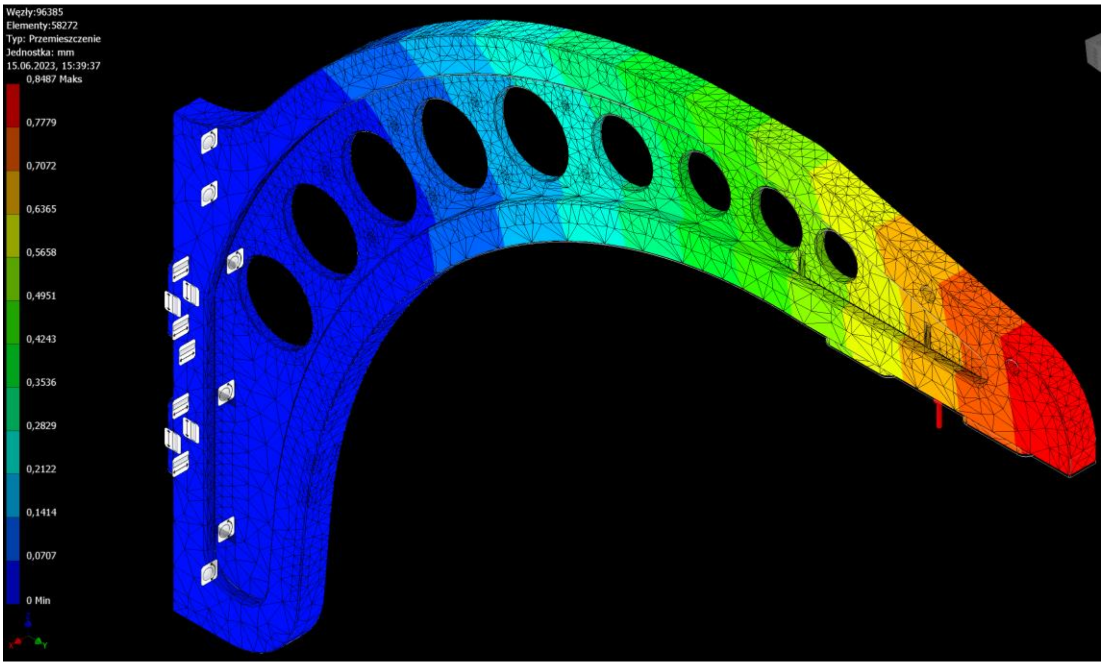

Fig. 9. Result of the simulation - displacement

The mass reduction process concluded with a result of 0.594 kg. While the displacement levels were slightly exceeded, the reduced stresses remained within a safe range (Fig. 13 and Fig. 14).

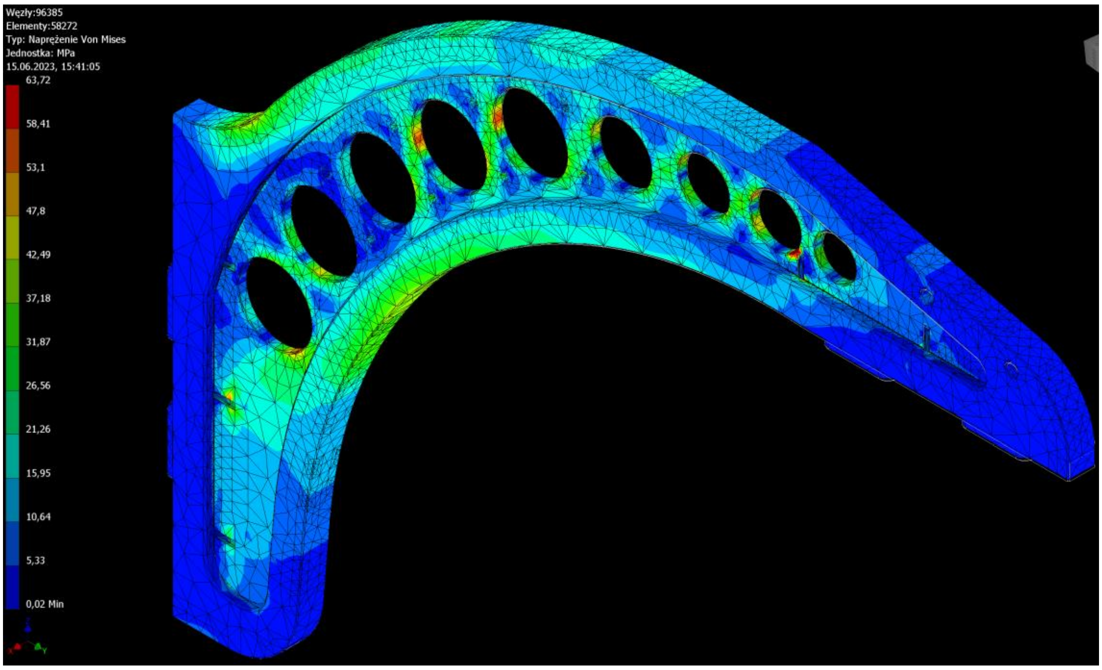

Fig. 10. Result of the simulation - reduced stress (scale limited to 63.72 MPa)

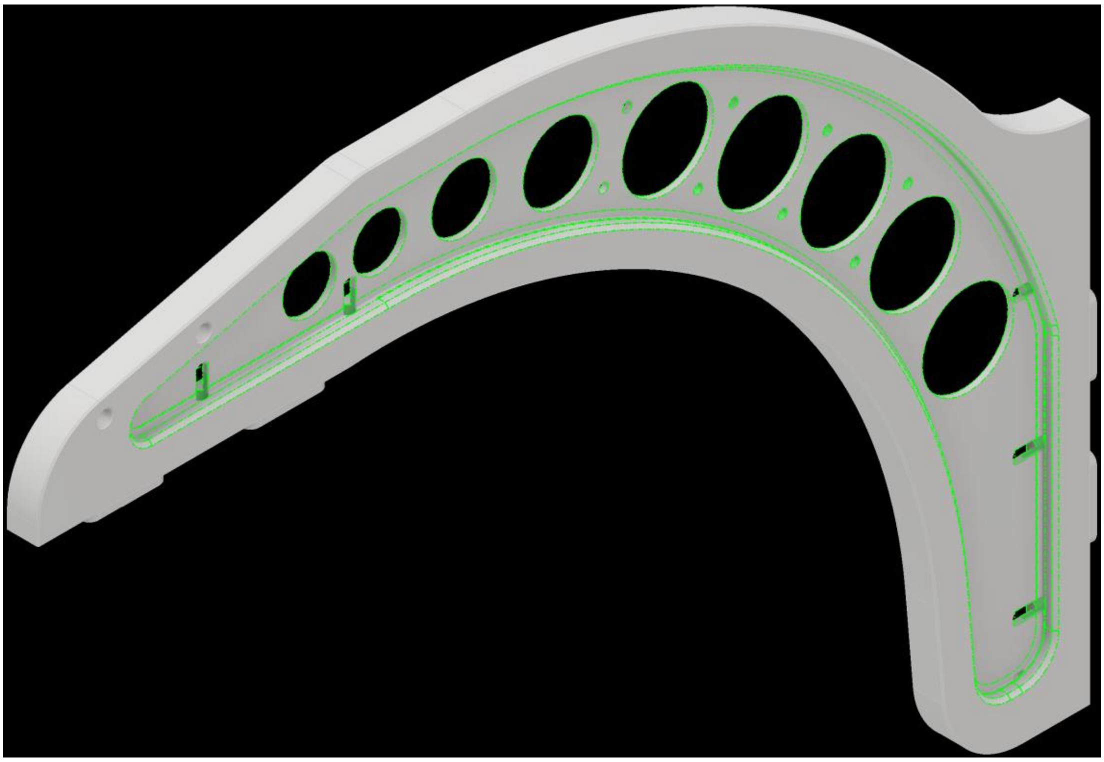

Fig. 11. Editing the element by offsetting the face by 2 mm

While not the most favorable method for mass reduction, in the design and engineering process, especially for custom production, the time invested by the designer on individual components can often carry greater significance.

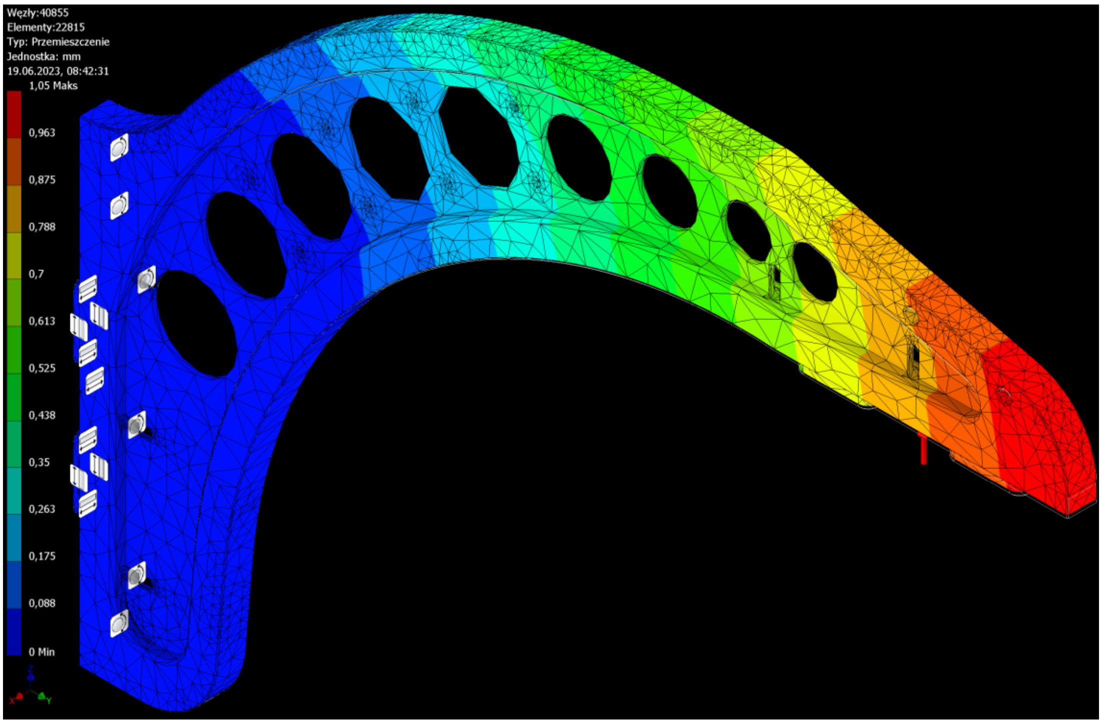

Fig. 12. Result of the simulation – displacement

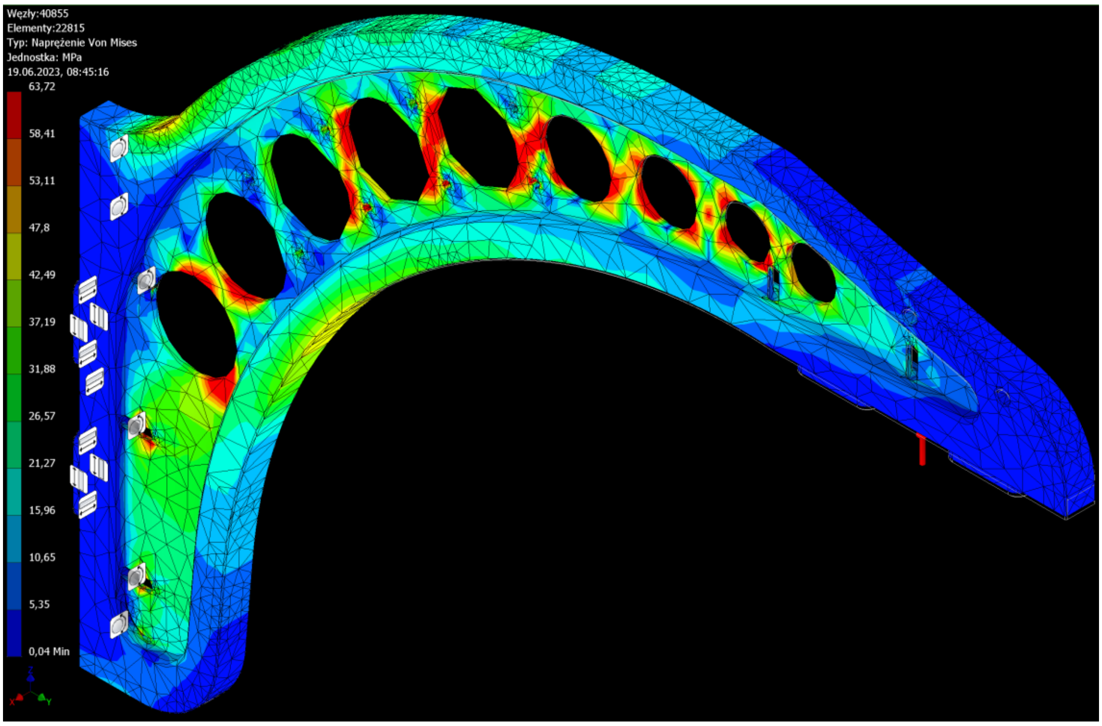

Fig. 13. Result of the simulation - reduced stress (scale limited to 63.72 MPa)

It is also worth noting that in such a mode of operation, a lot depends on individual factors (knowledge, experience, engineering intuition), and the results achieved by several employees can vary significantly from one another.

It is also important to design the component in a way that allows for easy dimension editing. This will have a favorable impact on reducing the time spent working on the detail.

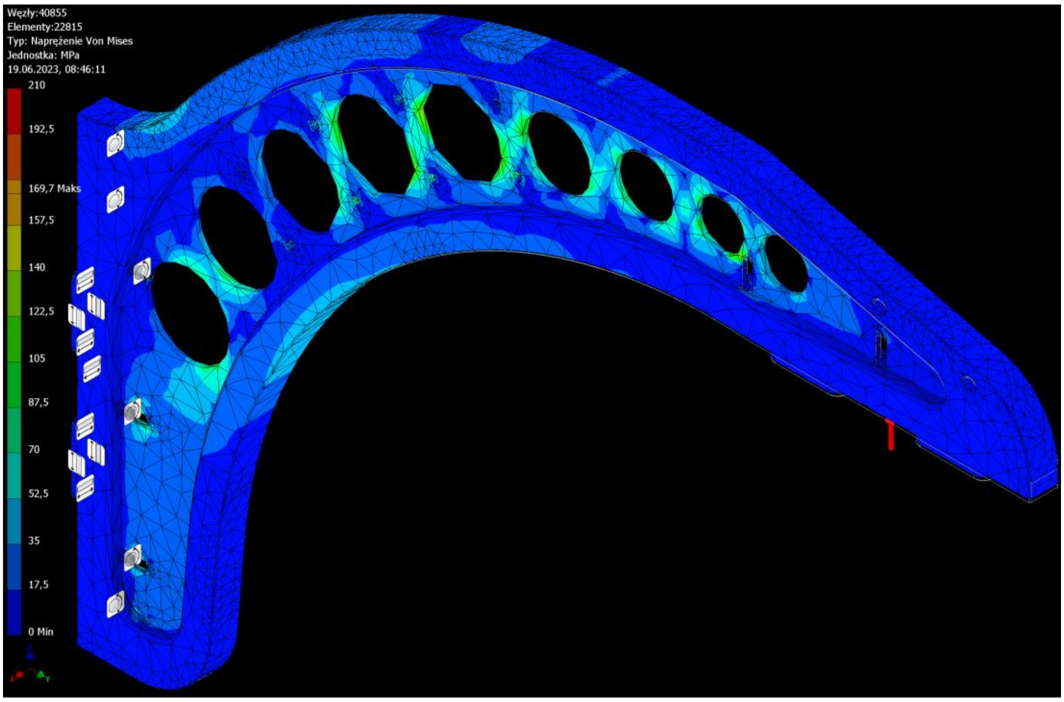

Fig. 14. Result of the simulation - reduced stress (scale limited to 210 MPa)

Contemporary Computer-Aided Design (CAD) programs now offer tools for editing solids even in the absence of operation history, for instance, when importing a model from a universal file format like STEP (Standard for the Exchange of Product model data) or IGS (Initial Graphics Exchange Specification).

4. Shape Generator in Inventor software

The Shape Generator provides an intelligent strategy for maximizing the rigidity of a component based on specified constraints. It generates a 3D mesh that can serve as a reference during design refinement. Consequently, it is best employed in the early stages of conceptual design [19].

The subsequent steps executed in the program are depicted graphically in Fig. 15. These steps can be outlined as follows:

1) Creating a basic working shape, approximating the model roughly, which includes the required contact points, fixations, etc.

2) Defining zones to be preserved, those which will not be modified during the shape generation process, as well as specifying boundary conditions (constraints and loads),

3) Establishing design criteria, generating the FEM mesh, and initiating the Shape Generator,

4) Exporting the obtained mesh to an STL model or the part environment, and subsequently modifying the model based on this.

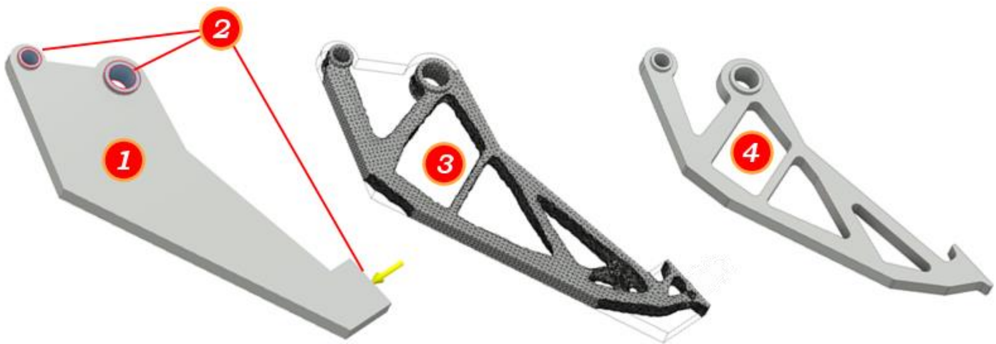

Fig. 15. The sequence of operations in Inventor software [19]

The interface of the environment used for topology optimization is presented in Fig. 16. It closely resembles the strength simulation environment, and the initial steps (assigning material, fixing, and applying loads) in the project are identical in both environments.

Among the activities specific to topology optimization modules will be the determination of regions to be preserved, i.e., areas of the component where it interacts with other details.

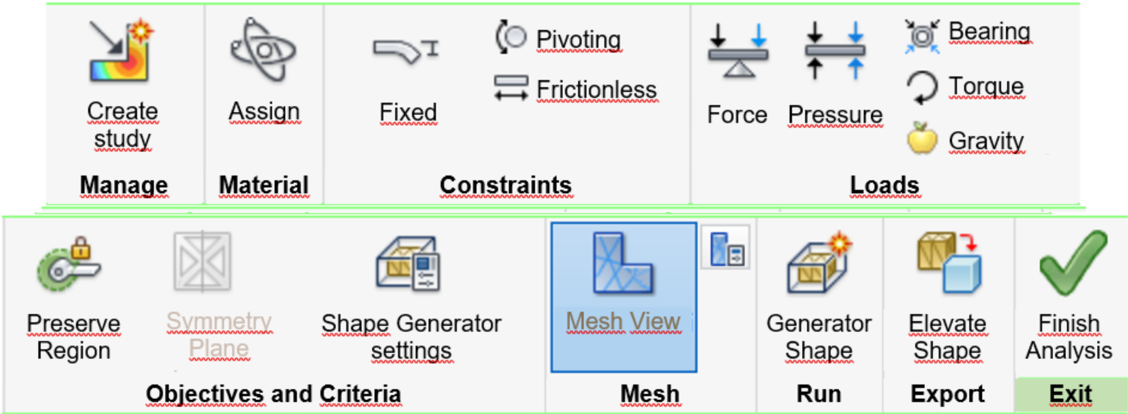

Fig. 16. Topology optimization environment interface

The project tree after configuration is shown in Fig. 17. Due to the fact that the load is applied in the plane of symmetry, it was possible to expedite numerical computations by dividing the model into two identical parts. It is important to note that Shape Generator calculations take several times longer than a regular strength simulation.

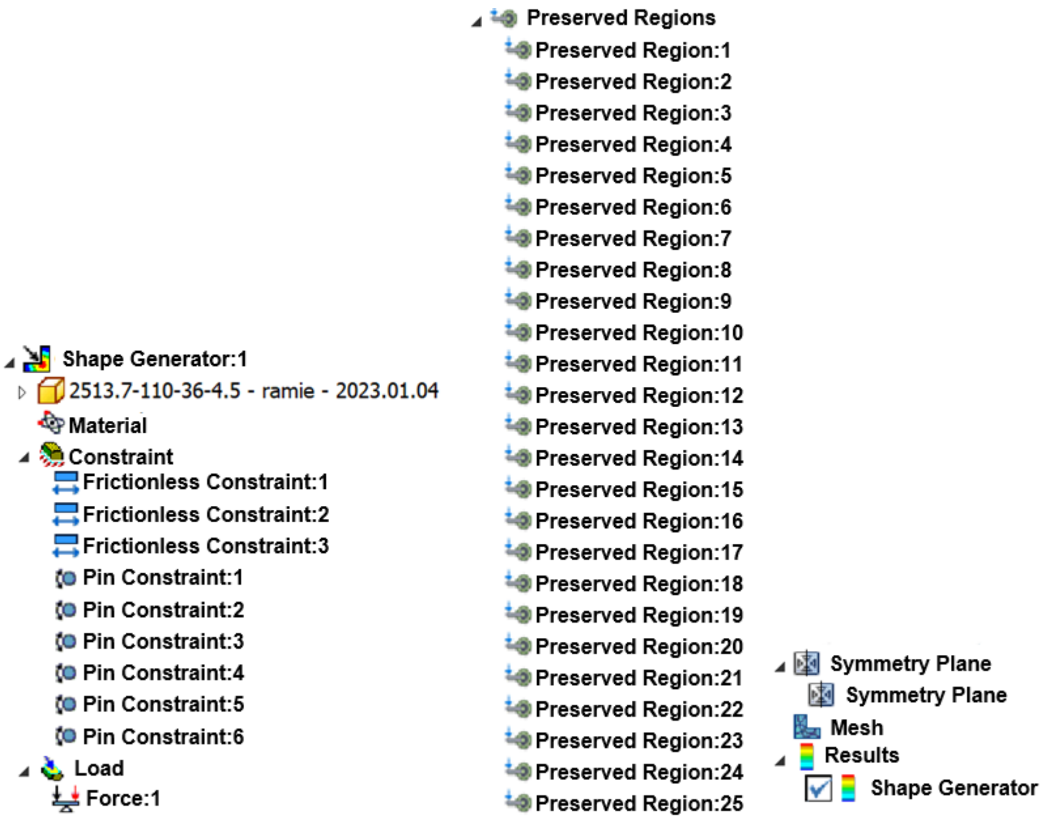

Fig. 17. Simulation configuration tree

The boundary conditions adopted for the Shape Generator and strength calculations are identical in all discussed examples within the study.

Publisher: **KOMAG Institute of Mining Technology, Poland**

The study aims to compare the results of different approaches in designing elements using various methods. Additionally, in order to compare the effects of optimization at different stages of the design process, it was decided to conduct two simulations. The first one was based on a preliminary optimization by the designer, but prior to verification in the FEM environment (4.1). The second calculation was performed in relation to a non-optimized solid (4.2).

4.1. Applying the shape generator to a pre-optimized component

Using the Shape Generator to optimize the geometry of a finished product is not a recommended approach; however, such an attempt was made for research purposes.

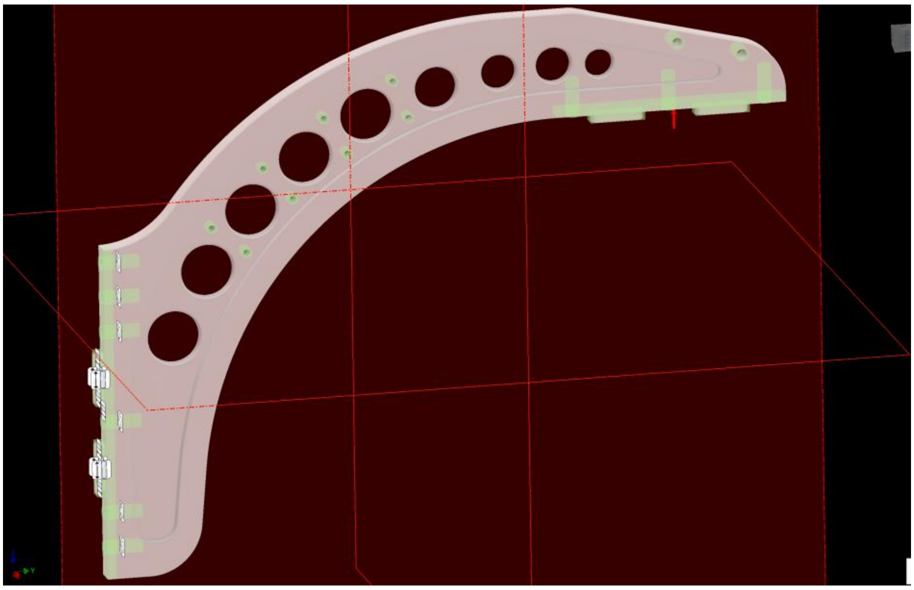

Fig. 18. Boundary conditions, regions to be left unchanged

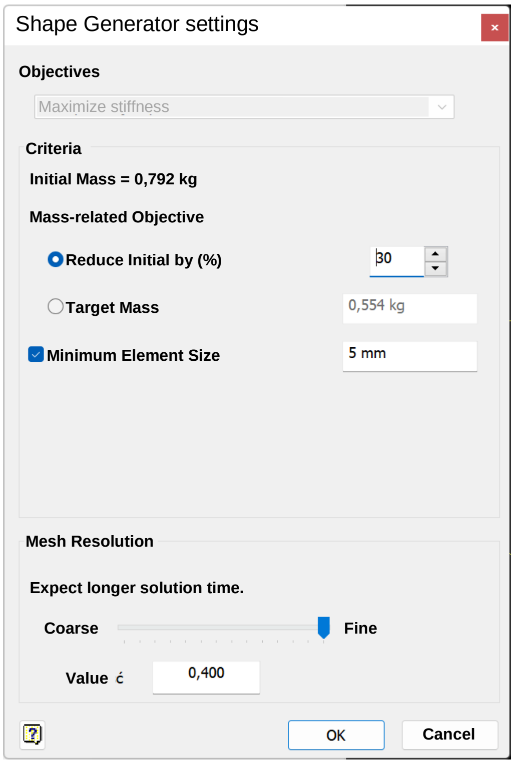

Fig. 19. Generator shapes settings in Inventor software

The question is whether the above approach is not recommended due to time-saving for the designer, or if the achieved results will be significantly worse? The answer to this question is crucial for planning activities in future projects. In many cases, a prototype is only required to function correctly, perform the intended function, and optimization is carried out in the production version.

The prepared model in the version after the initial mass reduction by the designer is shown in Fig. 18. The following elements are visible:

– Areas to remain unchanged are marked in green,
– A red arrow indicates the direction and point of force application,
– Constraint symbols specify the method and location of fixation.

The adopted topology optimization criteria in the Shape Generator settings window are illustrated in Fig. 19. The aim was to achieve a weight reduction of approximately 30%. This criterion was based on the assumptions made regarding the entire robot project.

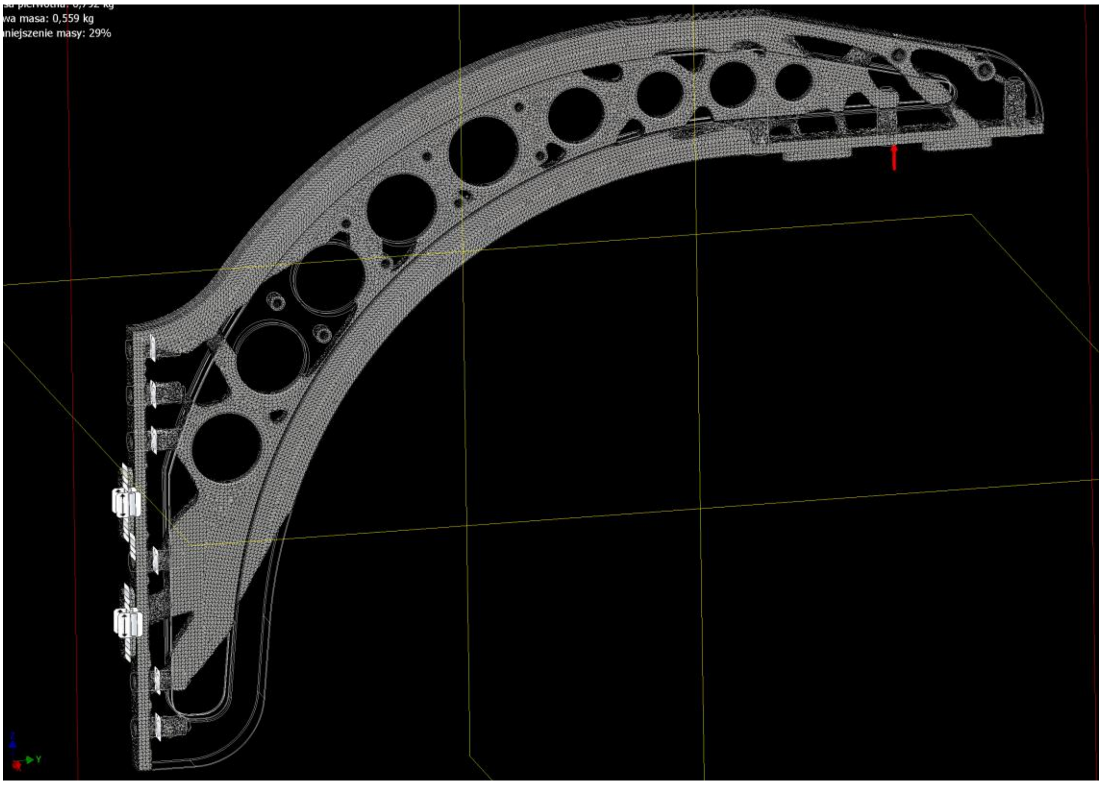

Fig. 20. Result of the shape generator

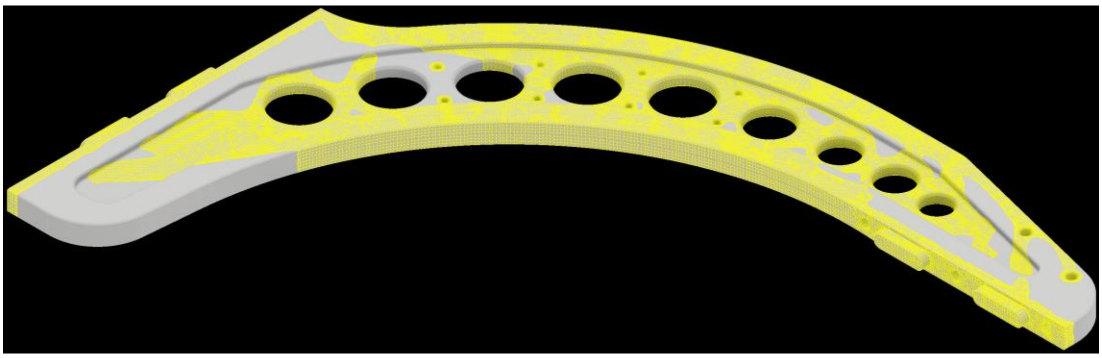

Fig. 21. Transfer of the obtained shape to the part environment

The result of the conducted calculations (Fig. 20) led to a reduction in weight from 0.792 kg to 0.559 kg, which is approximately 29%. However, in practice, reproducing such a shape is only feasible in the case of casting or 3D printing. Castings are not cost-effective for small-batch production, and 3D printing technology can be costly. Additionally, the program did not connect the mounting areas with the load-bearing structure.

Next, the step involves what is known as 'Shape Elevation', which means exporting the obtained mesh to the STL format or transferring it to the Part environment (Fig. 21). By using the Trim and Extend tools, both internal and external radii were added, achieving a shape similar to the mesh (Fig. 22), with a weight of 0.619 kg.

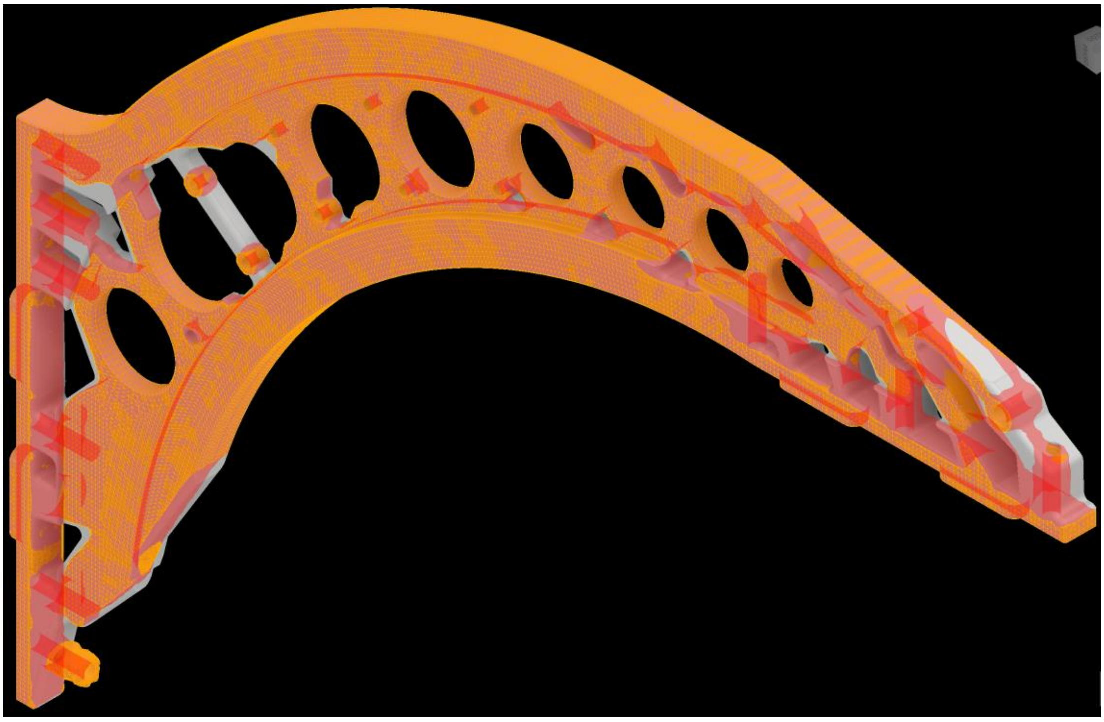

Fig. 22. Editing the working volume to approximate the shape of the obtained mesh

With respect to the geometry prepared in this manner, a finite element analysis was conducted, yielding a displacement result of 0.79 mm (Fig. 23) and a distribution of reduced stresses within a safe range (Fig. 24).

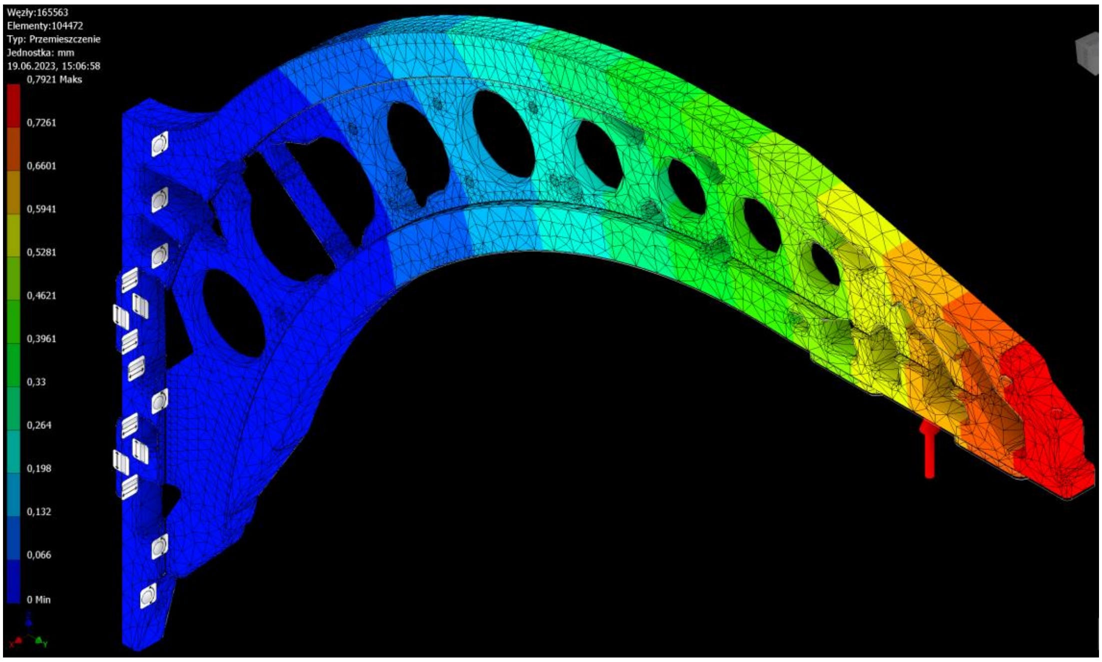

Fig. 23. Simulation result – displacement

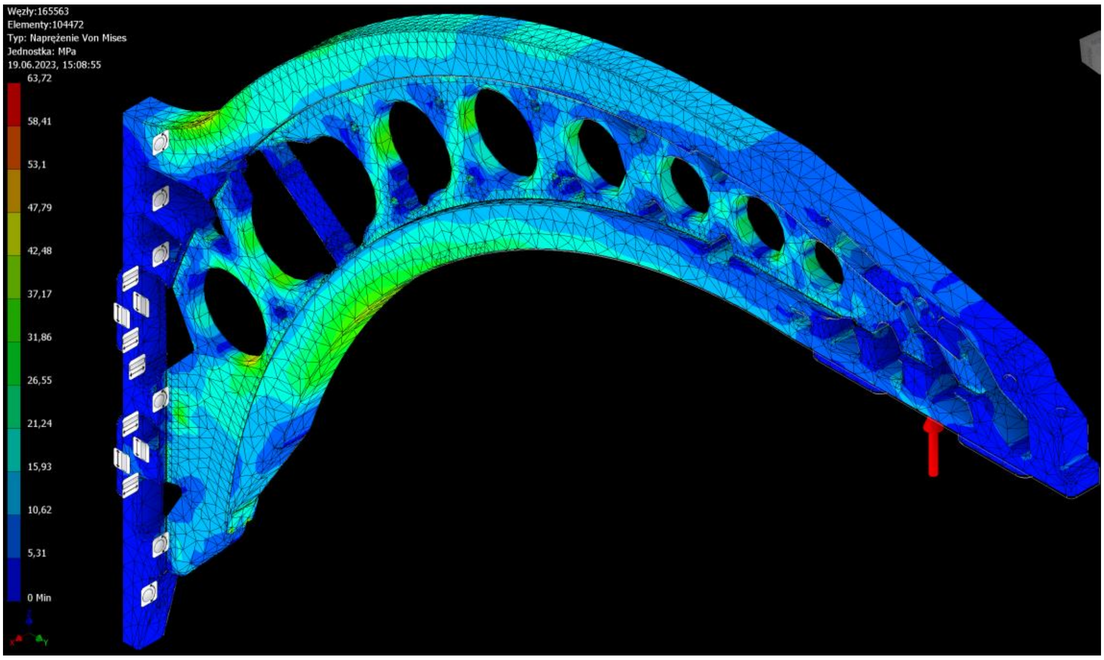

Fig. 24. Simulation result – reduced stress (scale limited to 63.72 MPa)

As a result of the straightforward operations, the FEM mesh was not precisely replicated, but that was not the goal. The aim was to apply a relatively simple geometry on the sketch, making it possible to manufacture the component through laser or water cutting and process it on three-axis CNC machines. The results of the conducted simulation indicate that the process of material removal can be continued without reaching dangerous stress distribution limits, and displacements will remain below the permissible criterion of 1 mm. The time required for replicating the shape increases exponentially with precision. For the purposes of analysis, the obtained result is sufficient, which is why further iterations were not pursued.

4.2. Topology optimization of the entire solid

In the analyzed example, to obtain the working volume of the solid, it was necessary to modify the model by filling the empty spaces/cutouts inside the solid.

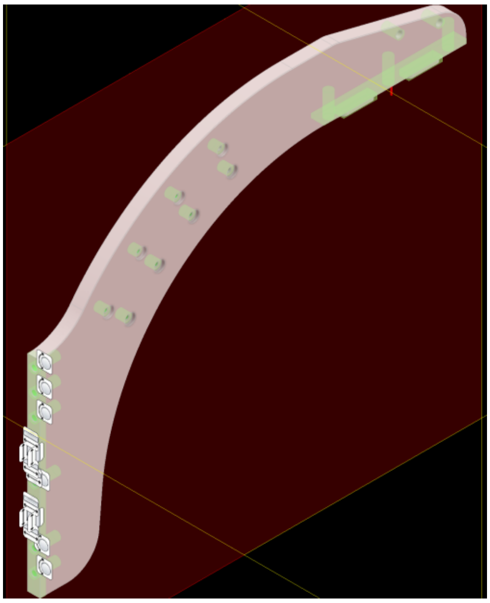

Fig. 25. Boundary conditions, regions to leave unchanged

Publisher: **KOMAG Institute of Mining Technology, Poland**

These actions would not typically be taken and were only conducted for research project purposes. This necessity arose from the fact that an element of the finished product (magnetic robot) was adopted for analysis, for which a 3D model and documentation were created. According to the assumptions, the total weight of the device could not exceed 100 kg. The result of the operation and the prepared design, achieved by adding areas to be left (identically to 4.1), boundary conditions, and the location of applied load, are shown in Fig. 25. The same effect could also be achieved by removing the operation in the Solid Edge program, or using other tools in Inventor. However, this does not matter for the purposes of this analysis.

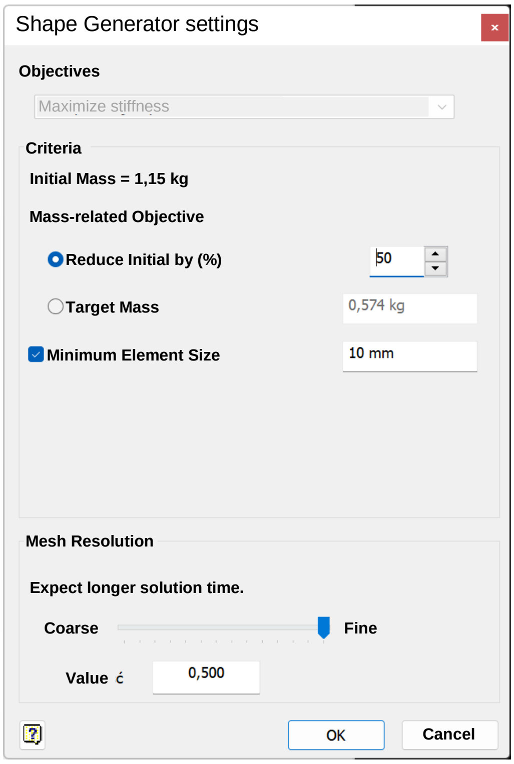

Fig. 26. Shape Generator settings in Inventor software

As the optimization criterion, the reduction of weight to a value similar to the first simulation was adopted. This required setting a goal in the Shape Generator to reduce the weight by approximately 50%, or to a value of 0.574 kg (Fig. 26). The percentage value is different because the finished component had a different initial weight compared to the full component. The justification for the chosen optimization criterion lies in the aim of this study, which is to compare the shapes obtained using different methods, and subsequently, to compare the results of the finite element simulations, with a focus on displacements.

The result of the conducted calculations yielded a reduction in weight from 1.15 kg to 0.588 kg, which is approximately 49%. The achieved weight of the component is close to the result from the first simulation (Fig. 27). The obtained mesh was transferred to the part environment (Fig. 28), and in the next step, through cutting and rounding operations, a roughly shaped form corresponding to the result of the Shape Generator was achieved (Fig. 29). It was assumed that the manufacturing technology would still be based on laser or water cutting of the rough shape, followed by machining selected surfaces on three-axis CNC machines. The displacement for the obtained body with a mass of 0.692 kg was 0.76 mm in the first simulation (Fig. 30), which falls comfortably within the accepted deformation criterion of 1 mm. Reduced stresses were also maintained within a safe range (Fig. 31), indicating that the body allows for further reduction of structural material.

For the purpose of result analysis, the achieved outcome is deemed sufficient, and further iterations were not pursued.

Publisher: **KOMAG Institute of Mining Technology, Poland**

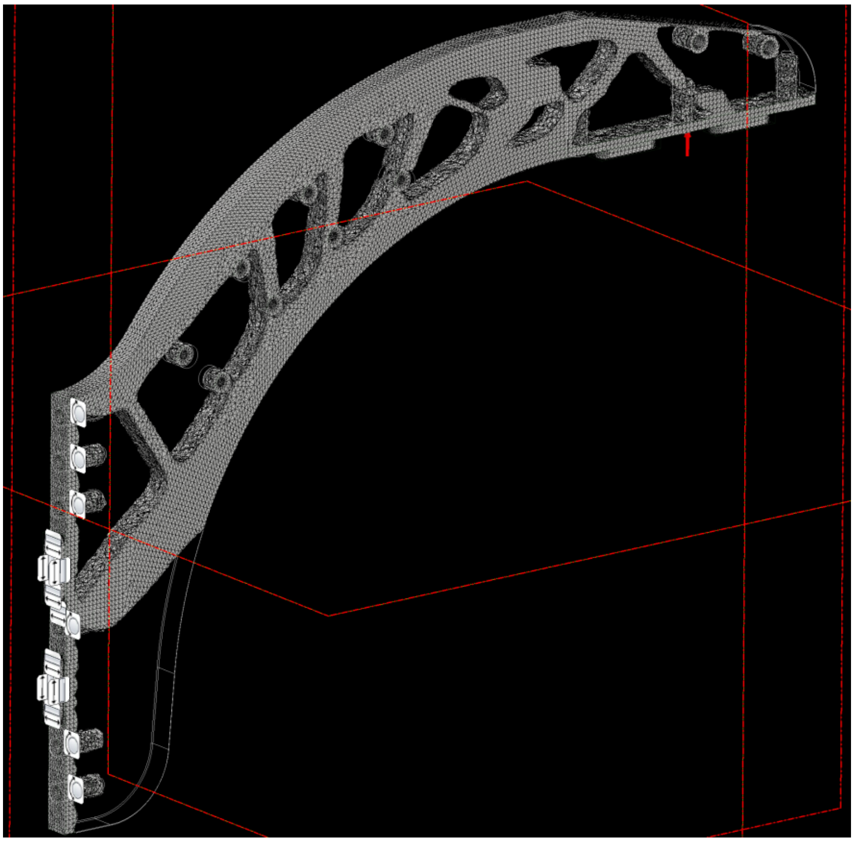

Fig. 27. Shape Generator result

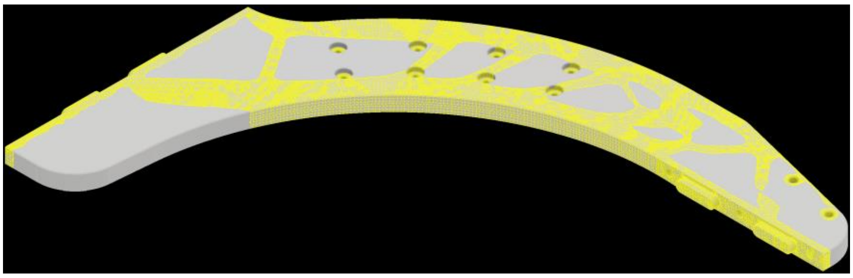

Fig. 28. Transferring the obtained shape to the part environment

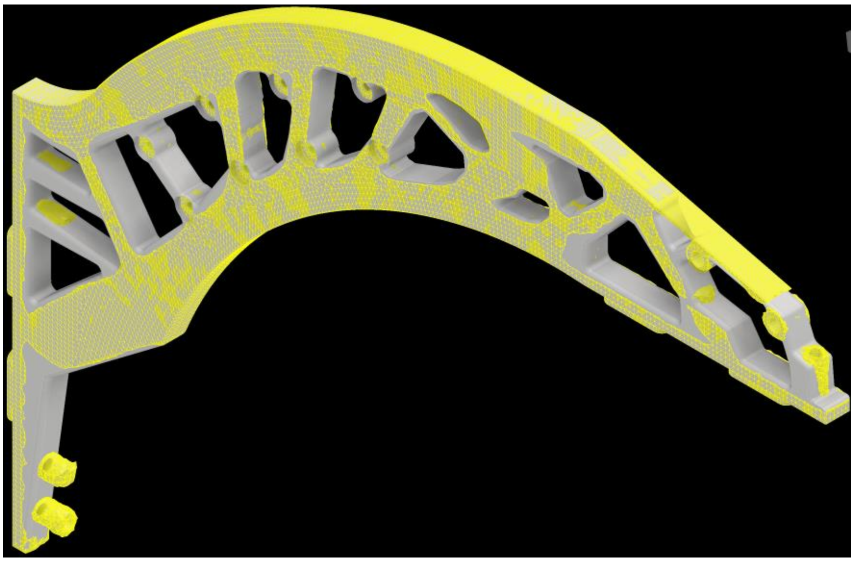

Fig. 29. Editing the working volume to approximate the shape of the obtained mesh

Publisher: **KOMAG Institute of Mining Technology, Poland**

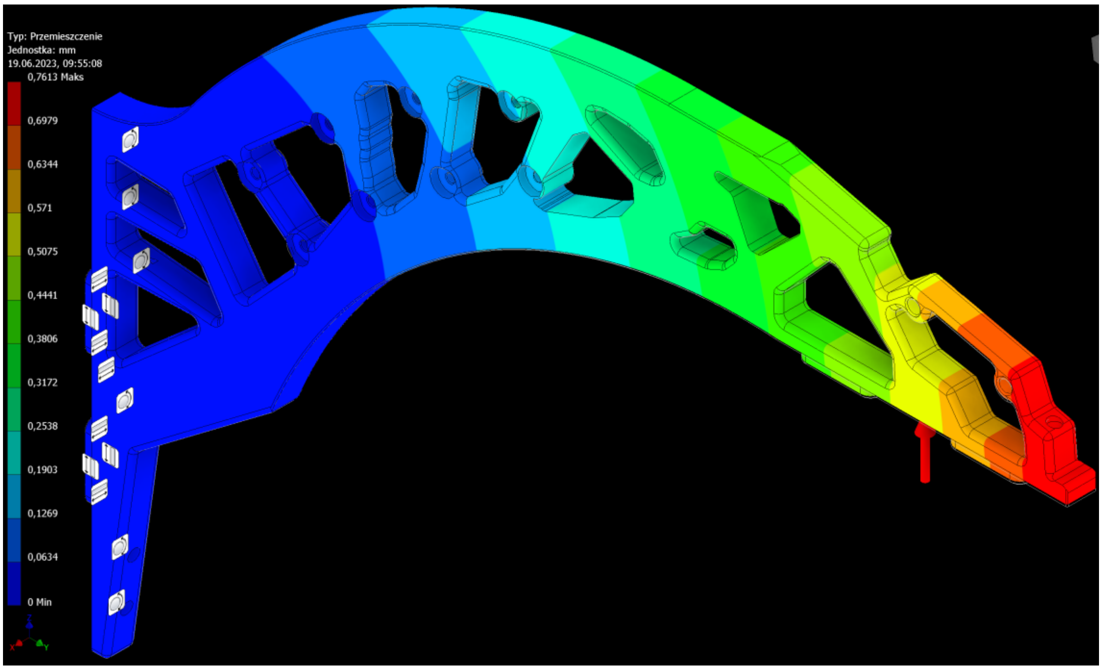

Fig. 30. Simulation result – displacement

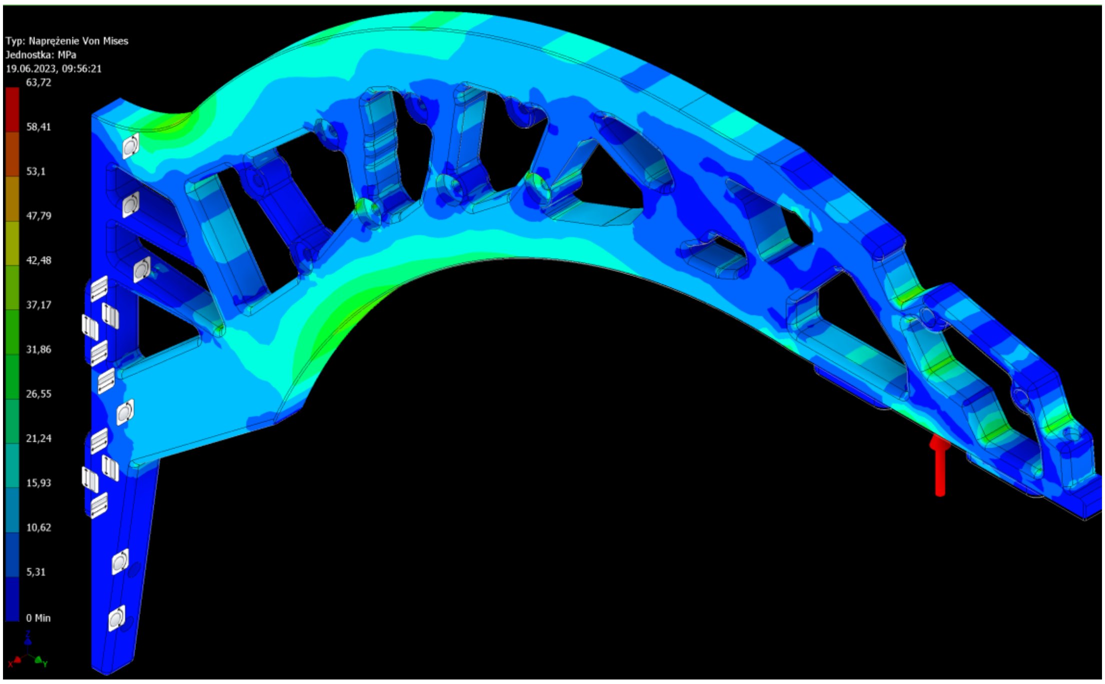

Fig. 31. Simulation result – reduced stress (scale limited to 63.72 MPa)

The mass reduction method using the Shape Generator took more time, but the savings come in at the stage of creating the working geometry, especially if topology optimization is considered from the very beginning. The process of reproducing the mesh shape is labor-intensive, and the effectiveness of the optimization process depends on the quality and precision of this task.

5. Summary

In Table 1, the results of the conducted studies are presented. It is clearly demonstrated that even an imprecise reproduction of the topology-optimized mesh yields better results than manual editing of the solid.

Table 1. Results Comparison

Lp.	Method	Initial Shape State	Mass (in kg)	Displacement (in mm)	Iteration
1	Manual mass reduction, designer's experience, generally accepted principles	prototype	0.792	0.738	0
2		prototype -1 mm	0.695	0.848	1
3		prototype -2 mm	0.594	1.05	2
4	Topology optimization, shape generator	prototype	0.619	0.792	1
5		working shape	0.692	0.762	1

In this specific case (pos. 4 and 5), it is difficult to conclusively determine whether it is more advantageous to optimize the finished product or the working solid. It would be necessary to conduct a similar analysis on a larger number of components, preferably with diverse shapes.

6. Conclusions

Topology optimization can assist in identifying areas of a component where the amount of structural material can be reduced while still maintaining the required strength properties. This leads to a reduction in the weight of the designed component compared to the originally developed model. The benefit of topology optimization will be more significant (in comparison to manual optimization) for designers with less experience.

However, these actions do not always lead to a reduction in production costs and can sometimes even contribute to an increase. A lot depends on the scale of production and the technologies employed. In the case of the component discussed in the publication and low-volume production, it is most advantageous to use a flat semi-finished product made of aluminum sheet with a pre-cut laser outline of the component and through-holes. CNC machining (simple 3-axis machines) is limited to surfaces requiring high tolerances.

The overall shape of the designed component also results from the connection point with the frame and the necessity to bypass the wheel set. Mounting points for accessories were provided on the component. The generator was unable to connect these areas with the sections subjected to load. Therefore, the result of topology optimization can be applied in the analyzed case to a limited extent.

The analysis of the problem has shown that it is justified to use the shape generator on a component with a pre-modified geometry in order to reduce its mass.

In the subsequent steps of the analysis, it is worth considering whether in projects developed for one-off production, the following modified procedure would be more advantageous:

1) Creating a working geometry of the component,
2) Performing calculations in the Shape Generator,
3) Based on the obtained results, selecting areas where a maximum volume can be removed with simple shapes in the flat sketch geometry, such as circles, rectangles, triangles,
4) Conducting a finite element analysis (FEA) to verify the strength or stiffness of the element,
5) Making modifications to the geometry in the sketch (changing hole diameters, radii, lengths of rectangle sides, and triangle sides).

These issues will be one of the topics for the author's future research.

References

[1] Kendibilir A., Kefal A.: Enhanced ship cross-section design methodology using peridynamics topology optimization. Elsevier, Ocean Engineering, 2023, Vol. 286. DOI: 10.1016/j.oceaneng.2023.115531

[2] Krog L., Tucker A., Kemp M., Boyd R.: Topology Optimisation of Aircraft Wing Box Ribs. MAO-30: 10th AIAA/ISSMO Multidisciplinary Analysis and Optimization Conferences, 2004. DOI: 10.2514/6.2004-4481

Publisher: **KOMAG Institute of Mining Technology, Poland**

[3] Mancuso A., Saporito A., Tumino D.: Designing the internal reinforcements of a sailing boat using a topology optimization approach. Elsevier, Applied Ocean Research, 2022, Vol. 129. DOI: 10.1016/j.apor.2022.103384

[4] Nguyen N. P., Maghsoudi E., Roberts S. N., Kwon B.: Shape optimization of pin fin array in a cooling channel using genetic algorithm and machine learning. Elsevier, International Journal of Heat and Mass Transfer, 2023, Vol. 202. DOI: 10.1016/j.ijheatmasstransfer.2022.123769

[5] Tiago P. Ribeiro, Luís F. A. Bernardo, Jorge M. A. Andrade: Topology Optimisation in Structural Steel Design for Additive Manufacturing. Applied Sciences, 2021, Vol 11, Issue 5. DOI: 10.3390/app1105211

[6] Alfouneh M., Keshtegar B.: STO-DAMV: Sequential topology optimization and dynamical accelerated mean value for reliability-based topology optimization of continuous structures. Elsevier, Computer Methods in Applied Mechanics and Engineering, 2023, Vol. 417. DOI: 10.1016/j.cma.2023.116429

[7] Yang X.Y., Xie Y.M., Steven G.P.: Evolutionary methods for topology optimisation of continuous structures with design dependent loads. Elsevier, 2004, Vol 83, Issues 12-13, Pages 956-963. DOI: 10.1016/j.compstruc.2004.10.011

[8] Barkanov E.: Introduction The Finite Elements Method. Institute of Materials and Structures Faculty of Civil Engineering Riga Technical University, 2001.

[9] Zienkiewicz O. C., Taylor R. L., ZHU J. Z.: The Finite Element Method. Vol 1: Its Basis & Fundamentals. Vol 2: For Solid and structural mechanics. Sixth edition. Elsevier Butterworth – Heinemann, Oxford 2005.

[10] Bureerat S., Limtragool J.: Performance enhancement of evolutionary search for structural topology optimisation. Elsevier, 2006, Vol 42, Issues 6, Pages 547-566. DOI: 10.1016/j.finel.2005.10.011

[11] Kiyono C. Y., Picelli R., Sivapuram R., De Lean D. M., Silva E. C. N.: Stress-based topology optimization approach using binary variables and geometry trimming. Elsevier, Finite Elements in Analysis and Design, 2023, Vol. 227. DOI: 10.1016/j.finel.2023.104044

[12] Qiu W., Wang Q., Gao L., Xio Z.: Stress-based evolutionary topology optimization via XIGA with explicit geometric boundaries. Elsevier, International Journal of Mechanical Sciences, 2023, Vol. 256. DOI: 10.1016/j.ijmecsci.2023.108512

[13] Vertonghen L., Irisarri F-X., Bettebghor D., Desmorat B.: Gradient-based concurrent topology and anisotropy optimization for mechanical structures. Elsavier, Computer Methods in Applied Mechanics and Engineering, 2023, Vol. 412. DOI: 10.1016/j.cma.2023.116069

[14] https://3d.edu.pl/co-to-jest-optymalizacja-topologii-proste-wyjasnienie/ [accessed: 31.01.2024]

[15] Chiandussi G., Gaviglio I., Ibba A.: Topology optimisation of an automotive component without final volume constraint specification. Elsevier, 2004, Vol 35, Issues 10-11, Pages 609-617. DOI: 10.1016/j.advengsoft.2003.07.002

[16] Kendibilir A., Kefal A., Sohouli A., Yildiz M., Koc B., Suleman A.: Peridynamics topology optimization of three-dimensional structures with surface cracks for additive manufacturing. Elsavier, Computer Methods in Applied Mechanics and Engineering, 2022, Vol. 401, Part B. DOI: 10.1016/j.cma.2022.115665

[17] Korus K., Salamak M., Jasiński M.: Optimization of geometric parameters of arch bridges using visual programming FEM components and genetic algorithm. Elsevier, Engineering Structures, 2021, Vol. 241. DOI: 10.1016/j.engstruct.2021.112465

[18] Pac P., Tokarczyk J., Prostański D., Rosikowski P.: Alternatywna metoda analizy wyników symulacji MES w oprogramowaniu CAD bez zaawansowanych narzędzi postprocesingu. Interdyscyplinarne badania młodych naukowców, InterTechDOC 2023. ISBN 978-83-7880-905-0

[19] https://help.autodesk.com/view/INVNTOR/2021/ENU/?guid=GUID-D74F47F3-FE22-44EF-85BE-7C6B1F56DCF9 [accessed: 31.01.2024]