



Design and Analysis of Additively Manufactured Skeletal Structures

Mariusz Dębski*, Grzegorz Budzik, Łukasz Przeszlowski

Abstract: Additive technologies enable the production of elements with complex geometry, which opens up new possibilities in the design of lightweight and strong structures. However, the additive manufacturing process is associated with limitations that must be taken into account at the design stage. This article presents the results of research on the design and analysis of skeletal structures intended for additive manufacturing. CAD models of the robot gripper were developed and then optimized using three different approaches: the use of a shape generator, generative design, and finite element analysis (FEM). The aim of the study was to compare the strength of a robot gripper that was lightened using three different design approaches, with a fixed mass reduction of 40%, in order to determine which method provides the greatest mechanical strength. Experimental studies included numerical analysis and real tests conducted on a testing machine. The test results showed that generative design allows for the most optimal material distribution, which translates into a more favorable strength-to-weight ratio compared to other methods of reducing the weight of the structure.

Keywords: additive technologies; design optimisation; generative design; polymeric materials; strength tests

1 INTRODUCTION

Modern additive manufacturing technologies play an increasingly important role in industry, engineering and science. Due to their flexibility and design capabilities, they enable the creation of geometries that would be difficult or even impossible to achieve using conventional machining or injection molding methods [1].

In the era of increasing requirements for energy efficiency, sustainable development and cost optimization, it is crucial to design elements with the lowest possible mass while maintaining their mechanical properties [2, 3]. Skeletal structures using advanced topological optimization algorithms and inspired by biological structures allow for a significant reduction in the amount of material used, which translates into lower raw material consumption and lower production and operating costs [4]. Such solutions work particularly well in sectors where minimizing mass affects system efficiency, e.g. in the aviation industry, where each additional gram of structure translates into higher fuel consumption [5].

Designing skeletal elements requires an interdisciplinary approach, including materials engineering, structural mechanics, computer modeling, and numerical simulations [6]. Using methods such as finite element analysis (FEM) and generative design, it is possible to obtain optimized components that are characterized by high strength while reducing weight [7]. Another important aspect is the selection of appropriate printing technologies and materials that must meet specific requirements regarding durability, resistance to dynamic loads, and thermal properties.

Reducing the mass of robot grippers has a significant impact on their efficiency, precision and energy consumption. Lighter grippers allow faster manipulator movements, which increases the efficiency of industrial operations and reduces the load on motors and actuators [8]. This allows the use of smaller, more energy-efficient drive components, which leads to reduced operating costs and increased durability of the device [8, 9]. Additionally, the smaller mass of the gripper reduces the moments of inertia,

which allows for more precise control and reduction of vibrations when gripping and moving objects [10]. 3D printing enables the creation of weight-optimized structures that maintain high strength while eliminating unnecessary materials and using intelligent internal structures. This allows the design of grippers with high functionality, low weight and increased work efficiency [11].

2 DESIGN OF SKELETAL STRUCTURES

Designing additively manufactured skeletal elements requires taking into account a number of key factors that affect their strength, functionality and manufacturing efficiency. An appropriate approach to these aspects allows for the creation of optimal structures with high strength while minimizing material consumption.

One of the most important factors is the selection of material, which must be adapted to the operational requirements of a given element. The selection of the appropriate filament in FDM technology (Fused Deposition Modeling), powder in the SLS method (Selective Laser Sintering), or resin in the SLA method (Stereolithography) depends on such parameters as mechanical strength, flexibility, and chemical and thermal resistance. The material should be selected in such a way as to ensure the durability of the structure while maintaining low weight.

An important aspect is also considering the printing direction, as the layer-by-layer structure of the models affects the material anisotropy. Proper orientation of the component on the build platform can significantly improve its load resistance and reduce the risk of fractures at layer junctions.

When designing skeletal structures, it is also necessary to take into account the limitations related to the additive manufacturing process. The geometric conditions of the designed models depend primarily on the type of material used, the technological limitations of the selected method and the printing device.

The process of designing additively manufactured skeletal components can be divided into seven key stages:

- 1) Requirements analysis – defining project goals, working conditions and technological constraints.
- 2) CAD modeling – creating a digital model.
- 3) Mass reduction (optional) – depending on the design requirements, various methods of structural lightening can be applied, such as topology optimization (TO), generative design (GD), or manual modification of the model based on load analysis. The purpose of this stage is to reduce the weight of the part while maintaining the necessary mechanical strength; however, it is not a mandatory step for every additively manufactured component.
- 4) Numerical simulations – conducting strength analysis (e.g. finite element method – FEM) to identify weak points.
- 5) Adaptation to printing technology – modification of the model to suit printer limitations (e.g. wall thickness).
- 6) Manufacturing and testing – test printing and analysis of the mechanical and functional properties of the element.
- 7) Modifications and finalization – improving the design based on test results and developing the final version of the item.

When designing skeletal samples, it is possible to use various tools to support the process of creating optimal structures. The selection of appropriate software and methods depends on the design requirements, such as mechanical strength, mass minimization, or manufacturing method. The basic tools supporting the design include FEM analysis, shape generators, generative design, and programs for creating openwork structures.

One of the tools used in the design of skeletal structures is the FEM analysis. It allows for the simulation of the actual working conditions of the element and the assessment of its behavior under the influence of various loads. Based on the numerical analysis, it is possible to determine which areas of the structure are most heavily loaded and which can be slimmed down without losing strength (Fig. 1). The analysis of the obtained results also allows for the detection of potential critical points that may lead to damage to the given element, which allows for the introduction of corrections before the start of production.

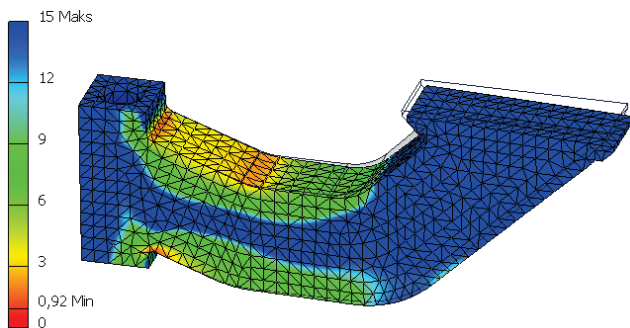


Figure 1 Numerical analysis of robot gripper - safety factor

Shape generators are tools that allow you to automatically adjust the geometry of a model to specific criteria, such as stiffness, mass or structural constraints. They

operate on the principle of algorithmic optimization, removing unnecessary material where it is not necessary, while maintaining the required strength. The stages of part design are presented using the Shape Generator tool in Inventor Professional as an example. The first step is to determine the boundary conditions, the load acting on the element and generate a triangle mesh. Additionally, the software allows for the exclusion of specific regions from the shape generation process and the definition of symmetry planes. Based on the selected material, the shape generator creates a conceptual 3D mesh (conceptual shape – Fig. 2), which can be used as a reference in the design process. Figure 3 shows a mass-reduced gripper whose geometry was designed using the Shape Generator tool.



Figure 2 Conceptual shape of the robot gripper



Figure 3 Reduced-mass robot gripper

Generative design is a more advanced form of optimization that uses artificial intelligence and evolutionary algorithms to create optimized geometry. Such programs can generate dozens or even hundreds of design variants that meet specific requirements for strength, weight, and production costs. This method allows for obtaining unique, organic shapes that are impossible to achieve using traditional design techniques. One of the tools for generative design is Generative Design in Autodesk Fusion 360. The design process is similar to a shape generator. You need to define boundary conditions, which are the load and material acting on the element (different materials can be defined depending on the type of processing). The difference is that in the case of generative design, you indicate the preserved geometry, i.e. solids that will not change during the generation of results. Additionally, you can assign obstacle geometry representing empty spaces in which no material will be placed during the generation of results (Fig. 4). Before generating the results, you can use the preview tool to see how the settings affect the results. Fig. 5 shows the selected robot gripper geometry generated by the program.

Openwork structures play a key role in designing lightweight but strong skeleton elements. Programs for their creation allow for generating lattice meshes with a specific geometry (e.g. gyroidal, hexagonal, tetragonal). One of the

tools for creating openwork structures is the Meshmixer program. This program allows for easy generation of lightweight structures with complex geometry, but such structures work best for elements that are not exposed to significant mechanical loads. In situations where high strength parameters are required, it may be necessary to use more advanced optimization methods or reinforced internal structures. The first step in creating such a structure in the Meshmixer program is to set a regular triangle grid on the surface of the element. This grid is the basis for later modification of the geometry and allows for uniform distribution of the openwork structure pattern. Then, the type and geometry parameters of the structure are determined, such as wall thickness, cell shape or filling degree (Fig. 6).

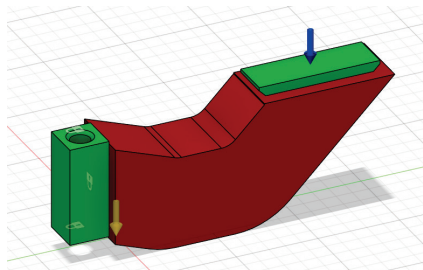


Figure 4 View of the solids defining the preserved geometry and the obstacle geometry

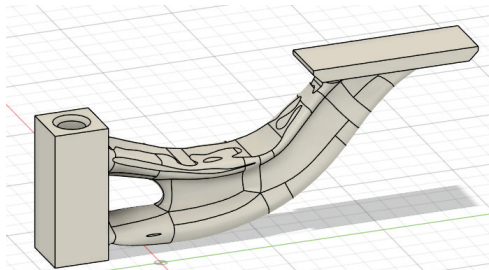


Figure 5 Robot gripper geometry created in generative study

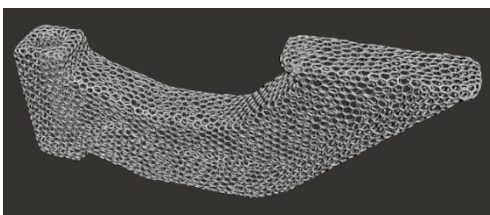


Figure 6 Robot gripper geometry created in Meshmixer

The purpose of the work was to compare the effectiveness of three different approaches to designing lightweight gripper structures in terms of their mechanical strength, while maintaining a consistent 40% reduction in mass. In contrast to typical optimization strategies, where the goal is to minimize mass while satisfying defined boundary conditions, this study adopted a fixed level of mass reduction for all design variants. This approach enabled a direct comparison of the performance of three design methods: shape generation, generative design, and manual design based on results from finite element analysis (FEA). The safety factor was used as the main evaluation criterion, as it allows for the identification of the structural margin of

strength with respect to the failure criterion and indicates potential areas at risk of damage, which is particularly important in the context of the anisotropic structure of additively manufactured materials.

3 RESEARCH MODELS

Based on the analysis of available design solutions on the market, a robot gripper model was developed in the form of a three-dimensional CAD model (reference sample K1 – Fig. 7). In order to optimize the structure in terms of weight reduction, the model was slimmed down using three different methods: shape generator, generative design and optimization based on finite element analysis (FEM). For all analyses, the construction material polylactic acid (PLA) and a load of 100 N were assumed. As a result of the design work, three optimization variants were obtained:

- one version of the gripper created using the shape generator (K2 – Fig. 8),
- one version created as a result of using generative design (K3 – Fig. 9),
- two versions developed on the basis of FEM analysis, taking into account different material reduction strategies (K4, K5 – Figs. 10 - 11).

For variants K4 and K5, the geometry was designed manually based on FEA results obtained for the reference model (Fig. 14). This analysis enabled the identification of low-stress areas, which were considered safe for removal or partial lightening without a significant deterioration in the mechanical properties of the structure.

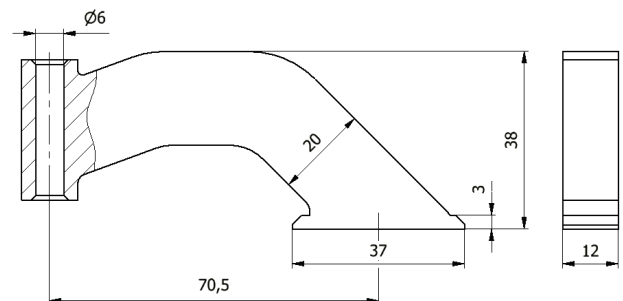


Figure 7 Shape and basic dimensions of the K1 reference gripper

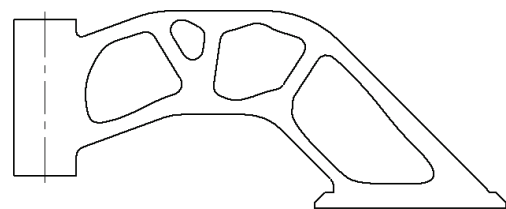


Figure 8 Shape of the K2 gripper

For each of the developed design concepts, a 40% reduction in the gripper mass was achieved, while maintaining the appropriate strength parameters required for the correct functioning of the component in the robot manipulation system.

The test samples were made using the layer extrusion method on a Prusa i3 MK3 printer. The printing process for

all models was characterized by the following basic parameters:

- alternating paths at angles of 45° and 135°,
- internal structure filling density – 100%,
- two contour lines,
- plastic layer height 0,2 mm.

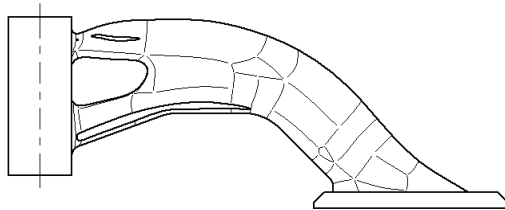


Figure 9 Shape of the K3 gripper

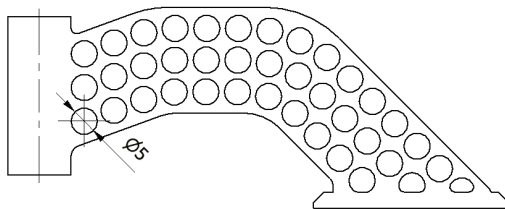


Figure 10 Shape of the K4 gripper

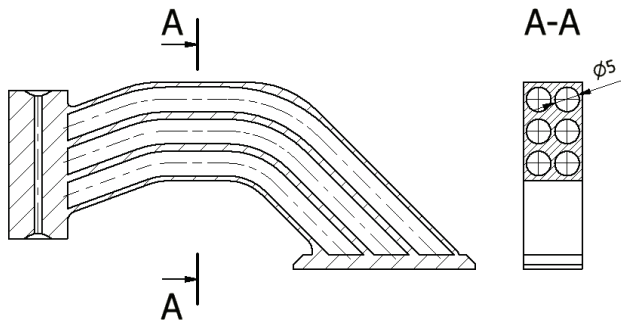


Figure 11 Shape of the K5 gripper

4 EXPERIMENTAL RESEARCH AND DISCUSSION

The main objective of the conducted tests was to verify the gripper's ability to transfer the given loads in static conditions. The verification included the analysis of the safety factor, which allows for the assessment of the structure's strength margin in relation to the assumed destruction criterion. As part of the conducted static analysis, a variant was considered in which a load of 100 N acts on the end part of the gripper in a direction perpendicular to its surface. The numerical analysis was performed using Autodesk Inventor Professional software. The initial conditions of the analysis are shown in Fig. 12. One of the significant problems that may occur during the use of the gripper is its deflection and permanent deformation, especially in the case of using materials susceptible to bending. If the load acts on the tip of the gripper, it generates a bending moment, which reaches the highest value at the base of the structure, at the point of attachment. This can lead to undesirable deformations, reduced precision of the gripper's operation, and in extreme cases, to its damage.

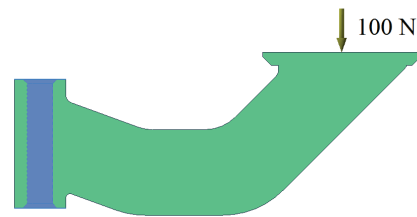


Figure 12 Constraints and load

The next stage of the research involved verifying the results of the numerical analysis through experimental strength tests conducted on a MultiTest-dV machine (Fig. 13). The aim of these tests was to confirm the consistency of the computer simulation results with the actual behavior of the gripper under load. To ensure the coherence of the test conditions, the same loading configuration as in the numerical analysis was applied. The tests were carried out until the samples reached failure.



Figure 13 MultiTest-dV testing machine

4.1 Strength Analysis using the Finite Element Method

After generating and optimizing the finite element mesh, an analysis was performed for each gripper variant, the results of which in the form of a safety factor are presented in Tab. 1 and Figs. 14 - 18.

Table 1 Summary of the minimum safety factor for each variant tested

Sample type	K1	K2	K3	K4	K5
Minimum safety factor	2,46	1,42	1,88	1,44	1,65

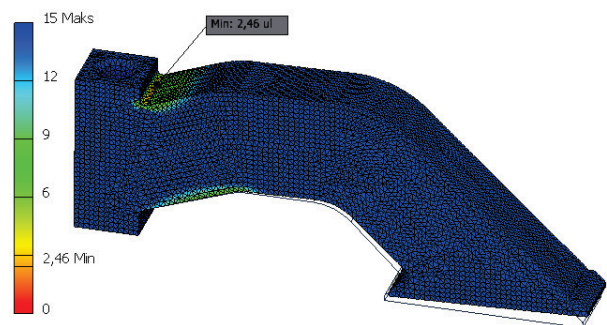


Figure 14 Analysis results for gripper K1

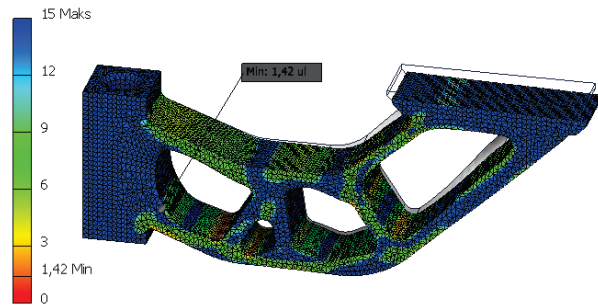


Figure 15 Analysis results for gripper K2

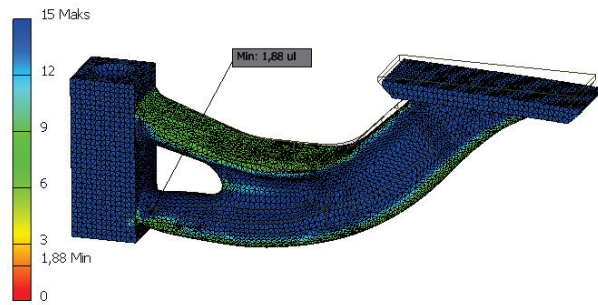


Figure 16 Analysis results for gripper K3

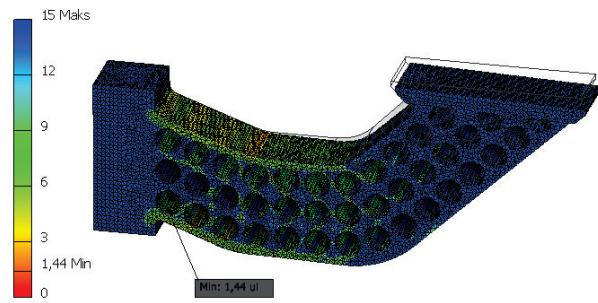


Figure 17 Analysis results for gripper K4

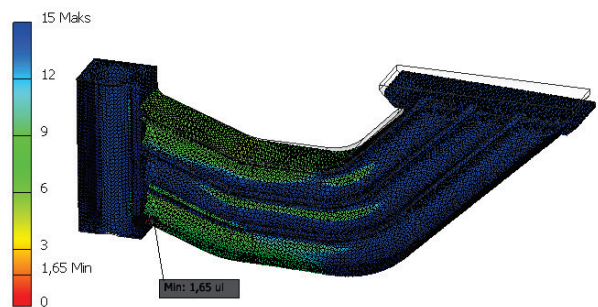


Figure 18 Analysis results for gripper K5

The analysis of the results carried out using the finite element method included an assessment of the safety factor for different design variants of the robot gripper. The highest value of the safety factor was obtained for the reference sample, which resulted from its full structure and lack of mass reduction. The next in terms of strength was the sample designed using the generative design method, which, despite significant mass reduction, retained optimal strength properties thanks to intelligent material distribution in key areas. In the case of variants K2, K4 and K5, the analysis showed the possibility of local overloads in the mounting

areas and at the end of the gripper, where the highest bending moments are generated. These zones can be potential critical points, especially in dynamic operating conditions. Additionally, the analysis allowed for the identification of areas at risk of weakening the structure, which is particularly important in the case of additive manufacturing due to the layered structure. This is due to the fact that the FEM analysis assumes an isotropic material model, while the materials used in additive technology are characterized by mechanical anisotropy. This means that the mechanical properties of the model depend on the direction of the layers, which can lead to local weakening and discrepancies between the simulation results and the actual strength of the element. Additionally, the way the model paths are arranged at a specific angle means that certain areas of the structure may be more susceptible to damage than the numerical analysis would suggest.

4.2 Strength Analysis on a Real Model

The experimental results are presented in Fig. 19.

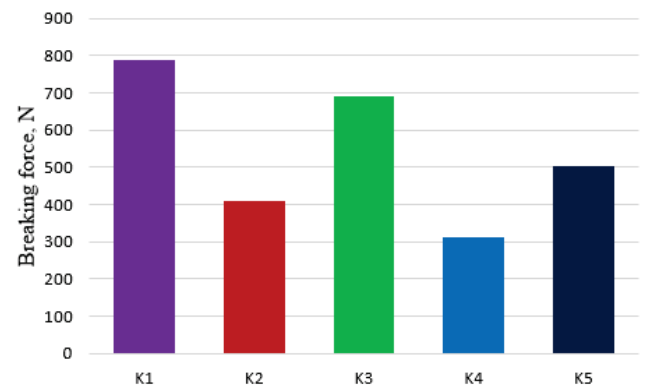


Figure 19 Average values of the breaking force for each tested variant

Based on the obtained results, it can be seen that the highest strength was achieved by the reference sample (K1), which was not subjected to mass reduction. Among the optimized variants, the best results were achieved by sample K3, which was designed using the generative design method. Despite the 40% mass reduction, its strength was the closest to the full sample, which indicates the effectiveness of this approach in the distribution of material according to the loads acting on the element. The average value of the force required to destroy the sample is about 88% of the reference gripper.

The remaining conventionally optimized variants, including sample K2, showed significantly lower values of the ultimate force. This means that manual mass reduction and geometry shaping based on FEM analysis can lead to a weakening of the structure if the actual load directions are not properly taken into account.

5 CONCLUSIONS

Designing skeleton samples manufactured using additive technologies requires the use of advanced engineering tools. FEM analysis allows for the assessment of the strength of the

structure, shape generators help in the optimization of geometry, and generative design allows for the creation of innovative and optimized forms. Proper use of these tools allows for the creation of lightweight and durable elements.

The study confirmed that generative design is a more effective tool for optimizing the structure of the robot gripper compared to shape generation and manual design based on FEA. This is due to the fact that generative algorithms can better adapt the structure to real working conditions, which is difficult to achieve with conventional methods.

In addition, the research results indicate a significant influence of material anisotropy in 3D printing technology, which means that the actual strength of the component may be lower than it would result from the FEM analysis for an isotropic material. Therefore, in the design of components intended for additive manufacturing, both the optimization of geometry and taking into account the specificity of the technological process are of key importance.

When designing skeleton elements for additive manufacturing, it is crucial not only to perform numerical analysis, such as the finite element method (FEM), but also to conduct experimental studies. This is due to the fact that in the additive manufacturing process there are a number of factors that can significantly affect the mechanical properties of the final element, and which are difficult to fully include in simulation analyses.

Acknowledgements

Scientific work/Publication co-financed from the state budget under the program of the Ministry of Science called "Polska Metrologia II" (Polish Metrology II) project number PM-II/SP/0085/2024/02 titled: "Methodology for measuring and analyzing the geometry of skeleton and shell structures of additively manufactured objects", Poland.

6 REFERENCES

- [1] Yalçın, M. M. (2024). Flexural Behavior of 3D-Printed Carbon Fiber-Reinforced Nylon Lattice Beams. *Polymers*, 16(21), 2991. <https://doi.org/10.3390/polym16212991>
- [2] Tyflopoulos, E., & Steinert, M. (2020). Topology and Parametric Optimization-Based Design Processes for Lightweight Structures. *Applied Sciences*, 10(13), 4496. <https://doi.org/10.3390/app10134496>
- [3] Tang, T., Wang, L., Zhu, M., Zhang, H., Dong, J., Yue, W., & Xia, H. (2024). Topology Optimization: A Review for Structural Designs under Statics Problems. *Materials*, 17(23), 5970. <https://doi.org/10.3390/ma17235970>
- [4] Ryan-Johnson, W. P., Wolfe, L. C., Byron, C. R., Nagel, J. K., & Zhang, H. (2021). A Systems Approach of Topology Optimization for Bioinspired Material Structures Design Using Additive Manufacturing. *Sustainability*, 13(14), 8013. <https://doi.org/10.3390/su13148013>
- [5] Iftekar, S. F., Aabid, A., Amir, A., & Baig, M. (2023). Advancements and Limitations in 3D Printing Materials and Technologies: A Critical Review. *Polymers*, 15(11), 2519. <https://doi.org/10.3390/polym15112519>
- [6] N. Musthafa, H.-S., Walker, J., & Domagala, M. (2024). Computational Modelling and Simulation of Scaffolds for Bone Tissue Engineering. *Computation*, 12(4), 74. <https://doi.org/10.3390/computation12040074>
- [7] Balayan, A., Mallick, R., Dwivedi, S., Saxena, S., Haorongbam, B., & Sharma, A. (2024). Optimal Design of Quadcopter Chassis Using Generative Design and Lightweight Materials to Advance Precision Agriculture. *Machines*, 12(3), 187. <https://doi.org/10.3390/machines12030187>
- [8] Francalanza, E., Fenech, A., & Cutajar, P. (2018). Generative design in the development of a robotic manipulator. *Procedia CIRP*, 67, 244–249. <https://doi.org/10.1016/j.procir.2017.12.207>
- [9] Sha, L., Lin, A., Zhao, X., & Kuang, S. (2020). A topology optimization method of robot lightweight design based on the finite element model of assembly and its applications. *Sci. Prog.*, 103. <https://doi.org/10.1177/0036850420936482>
- [10] Liu, B., Sha, L., Huang, K., Zhang, W., & Yang, H. (2022). A topology optimization method for collaborative robot lightweight design based on orthogonal experiment and its applications. *Int. J. Adv. Robot. Syst.*, 19. <https://doi.org/10.1177/17298814211056143>
- [11] Curkovic, P. (2024). Comparative Analysis of Topology Optimization Platforms for Additive Manufacturing of Robot Arms. *Designs*, 8(5), 98. <https://doi.org/10.3390/designs8050098>

Authors' contacts:

Mariusz Dębski, DSc. Eng.
(Corresponding author)
Rzeszów University of Technology,
Al. Powstańców Warszawy 12, 35-959 Rzeszów, Poland
m.debski@prz.edu.pl

Grzegorz Budzik, Prof. PhD, DSc, Eng.
Rzeszów University of Technology,
Al. Powstańców Warszawy 12, 35-959 Rzeszów, Poland
gbudzik@prz.edu.pl

Łukasz Przeszlowski, DSc. Eng.
Rzeszów University of Technology,
Al. Powstańców Warszawy 12, 35-959 Rzeszów, Poland
lprzeszl@prz.edu.pl