

Topology Optimization of Snake-like Robot Limb Using Ansys

Yusuf Coşkun^{1,*}, Muhammet Aydın², Muhammed Tayyip Koçak³

¹Istanbul Health and Technology University, Faculty of Engineering and Natural Sciences, Department of Mechatronics Engineering, Istanbul, Turkiye

²Firat University, Faculty of Engineering, Department of Mechatronics Engineering, Elazig, Turkiye

³Istanbul Health and Technology University, Faculty of Engineering and Natural Sciences, Department of Software Engineering, Istanbul, Turkiye

Recent natural disasters lead to the formation of large amounts of debris and consequently to the trapping of living creatures among the debris. A search and rescue robot inspired by the morphology of a snake has been designed to effectively reach the trapped creatures in these challenging conditions. Optimization of the body limb is of great importance for the robot to perform fast, agile, and durable in narrow and complex areas under debris. In this study, a topology optimization analysis of the robot's limb is performed using Ansys software. The optimization is focused on the body limb, which is the basis of the snake's mobility. The main objective of this study is to achieve significant optimization of the mass and volume parameters for the robot's body limb while maintaining structural robustness. Thus, the overall design of the robot is aimed to be more compact and efficient, while achieving more effective mobility in narrow and difficult areas under debris. As a result of detailed analysis, a significant decrease in stress and strain values was found. However, an increase in deformation values was observed. The fact that this increase occurs in deformation indicates that some parameters of the model deviate from the expected results. Furthermore, the investigation revealed that the mass and volume of the model decreased by half. This shows the potential of topology optimization in terms of material utilization and structural efficiency. However, the increase in deformation indicates that the current parameters of the model should be reviewed. Consequently, it has become imperative to revise the selected parameters to further improve the performance of the model and achieve the expected mechanical properties. This study successfully demonstrates how lighter and more durable structures can be achieved through topology optimization with efficient use of resources.

Keywords: Snake-like robot, Topology optimization, Ansys, Mass optimization, Remodelling

Submission Date: 09 October 2023

Acceptance Date: 01 December 2023

*Corresponding author: yusuf.coskun@istun.edu.tr

1. Introduction

Topology optimization plays a role, as a design tool that finds applications in domains, such as engineering and robotics [1]. It empowers designers and engineers to convert their creations into optimized designs that are not only lighter but also more appropriate and efficient [2]. Through topology optimization, the structural suitability and

performance of a design can be enhanced by rearranging the distribution of materials, within it [3].

One of the most important benefits of topology structure is the ability to create optimal load possibilities in the design space [4]. This allows designers and engineers to determine the most effective methods for load distribution, leading to increased structural integrity and performance [5]. It facilitates more effective and useful design iterations by

providing an organized design process where the best possible design is known in advance [2].

Topology optimization has been successfully applied in a wide variety of fields such as structural engineering, aerospace, automotive, and packaging [1,3,4,6]. In the automotive and aerospace sectors, topology optimization has been used to achieve low-mass and high-performance designs, with impressive structural performance achievements [4,6]. In packaging, topology optimization has been used to design structural elements with reduced weight and increased stiffness, leading to more efficient and aesthetically pleasing designs [2].

Innovations in additive manufacturing have advanced topology optimization as well as product applications. Additive manufacturing has brought design and manufacturing closer together by enabling complex models resulting from topology optimization to be transformed into real parts [1].

Topology optimization plays a critical role in the design and development of lightweight, high-performance robotic components. This practice optimizes the master design of components, enabling manufacturers to optimally distribute material. In the field of robotics, one of the benefits of this optimization is that robotic limbs move with the same or better performance despite the reduced weight [7]. By determining the best material distribution within the design, this process allows for the removal of unneeded material and the creation of lightweight designs [8]. Overall, the reduced weight increases both the robot's efficiency and payload capacity, saving energy. In addition, this optimization leads to the design of sophisticated complex products tailored to the needs of robotic projects [9]. Furthermore, topology optimization has been applied to some robotic components to increase usability and power [10]. With the optimization applied, the material quantity and shapes of these components have been updated, improving the grip and softness of the limbs. The combination of additive manufacturing and topology optimization has led to the optimized emergence of complex robotic designs [11].

The coming together of topology optimization and additive manufacturing could bring about a transformation, in the way robotic systems are manufactured. It enables minimizes material wastage and offers greater design flexibility. This makes topology optimization an essential element in developing lightweight and high-performance components for robots. Engineers can employ topology optimization to optimize the design and distribution of materials resulting in efficient robotic systems. With its ability to reduce weight enhance functionality and integrate with manufacturing techniques topology optimization proves to be an asset, in the field of robotics.

In applications, there is a growing need, for robotic systems that are highly efficient, adaptable, and lightweight. In this

context, the application of topology optimization emerges as a solution to fulfill these demands. Unlike design approaches topology optimization not ensures meeting specific performance standards but also optimizes material utilization. As a result, it leads to cost energy production processes that contribute to sustainability.

Additive manufacturing brings attention to the benefits of topology optimization. It enables the creation of shapes that would be challenging to produce using manufacturing techniques. This newfound capability grants designers. Engineers have a level of design freedom that was previously unattainable. Additionally, by speeding up prototyping processes, it promotes innovation cycles and time, to market for robotic systems.

The exciting collaboration, between topology optimization and additive manufacturing is paving the way for possibilities in the field of robotics. In the coming years, we can anticipate an expansion in the applications of systems thanks, to this powerful combination. This advancement will undoubtedly lead to ground-breaking solutions. Transform the landscape of robotics design and manufacturing.

Xinke Song et al. used the topology optimization method to reduce the weight of the designed robotic arm. After performing static analysis on the robotic arm manipulator through the Ansys Workbench program, topology optimization was performed based on the results obtained. The topology optimization results of the analysis, which had a weight reduction target of 30%, were reduced by 29.65% [12].

Yilun Sun's students implemented a 3D topology optimization method using the Solid Geometry (SG) library in a MATLAB-based design module. With this method, soft robotic grippers were designed and 3D printed. The work is implemented in MATLAB and two design examples are presented to illustrate the automatic synthesis process. With this framework, soft robotic grippers can be designed with different motion mechanisms and task-specific grasping behaviors. The study concludes that the proposed framework provides a promising approach for the design of soft robotic grippers using topology optimization [13].

Topology optimization was used in this study on the optimization of lower actuator robots designed for stroke patients. In this study, SolidWorks, GIM, Ansys, and Matlab software were used. SolidWorks and GIM software are specifically designed for robotic analysis. Ansys software was used for finite element modeling. Matlab software was used for optimization processes. Optimization processes were performed using MATLAB R2022b academic version. The Genetic Algorithm (GA) function of MATLAB Optimization Toolbox was used for the optimization process. Ansys software was used for the structural analysis of the robot. Ansys is a software used for finite element analysis (FEA) and is widely used in structural analysis. In

this study, the 3D CAD model of the robot was imported into Ansys Mechanical Workbench software and a static structural analysis was performed. Mazars states that optimization approaches result in a significant reduction in the weight of the robot and maintain its structural integrity and performance [14].

Yilun Sun et al. wanted to improve these legs by optimizing the legs designed for quadruped robots. They especially considered the standing and walking positions. The experiment was organized using Arduino Uno and a servo motor. After 58 iterations, the optimal design was remodeled and printed on a 3D printer. The experiment also showed that the optimized legs reduce energy consumption and vibrations during walking [15].

Bodog and Grebenişan emphasize that topological optimization is an effective tool to reduce costs and optimize material usage of tools such as cutting tools or plastic deformation (molds) used in industrial engineering applications. The paper examines the use of ANSYS as a tool and the effectiveness of topological optimization. Furthermore, a method for improving the accuracy of the design using parametric validation and response surfaces is presented. Before optimization, the boundary conditions are given by static analysis the total displacement is measured as 0.173 mm and the maximum equivalent stress is 142.85 MPa. After optimization, the total displacement is 0.198mm and the maximum equivalent stress is 216.22 MPa. In conclusion, this paper highlights the importance of topological optimization in the design process and provides a valuable guide on how this approach can be applied using ANSYS [16].

Xishun Yang et al. performed finite element simulation and 3D parametric CAD analysis for the optimal design of torsional elastic element. The paper emphasizes that the complex topology design of the elastic element is important for miniaturization and lightweight. In the static analysis of the part subject to the article, the moment of rotation was used as a boundary condition, and topology optimization was performed. As a result of the optimization, 3 different optimum designs were obtained. According to the analysis of simulation results, the most optimum design will be used [17].

Junbin LOU and Yaping GONG present the analysis of the structural dynamic characteristics of the end joint of a robot and its topological optimization design. According to the results of topological optimization, the structural weight of the end joint is significantly reduced and a lightweight structural design is realized. The structural weight was reduced by 42.16% and the mass of the end joint was reduced from 16.43 kg to 9.50 kg. However, the maximum deformation displacement increased by 11.2% and the maximum equivalent stress increased by 19.82%. The

results show that a significant reduction of structural weight and a lightweight structural design can be realized [18].

Yilun Sun et al. optimized the design of a lightweight robotic gripper. The design of the gripper consists of two main parts: the adaptive fingers and the lightweight base. Both parts are designed using 3-D topology optimization methods. The design flow of these methods starts with the definition of the size of the 3-D design space and the load cases. Then, a 3-D finite element analysis (FEA) is performed and the value of the objective function is calculated. Sensitivity analysis is performed and the design variables (densities) are updated accordingly. The update process is repeated until the objective function converges. By optimizing the density of a design space, this method optimizes the design to best meet a given objective function. This method can be used in many different application areas and is particularly used in the design of lightweight and robust structures. This innovative gripper is a robotic gripper that can easily grasp objects of different shapes and materials and has a maximum gripping load of up to 8.8 kg, about 49 times its weight [19].

2. Material and Methods

In this study, topology optimization is applied to one of the motion limbs of a snake-like robot. Firstly, the drawing of the model was made in SolidWorks, a computer-aided drawing program. Then, one of the body limbs of the assembled model was converted into a step file and sent to Ansys, which is used as an analysis program. After these steps, a static analysis was performed to examine the static and strength properties of the part by adhering to the boundary conditions. Then topology optimization was performed according to the static analysis results. With the results of topology optimization, the reconstructed design was redrawn and static analysis was performed. The analysis results of the optimized part and the old part were compared. The flow diagram of the work done is shown in Figure 1.

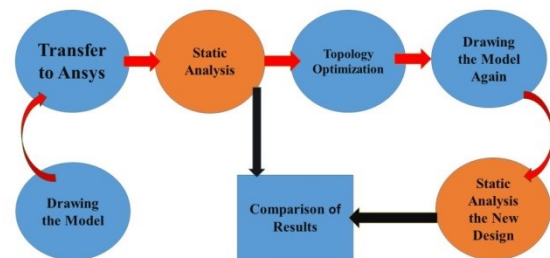


Fig.1. Flow chart of the study

2.1. Pre-Modeling and static analysis

2.1.1. Modeling of the snake robot limb

The snake robot was modeled and assembled in SolidWorks with head, tail, and body limbs, 7 limbs in total. The number of body limbs is 5. The body was formed by duplicating and combining a designed body limb [20]. To transfer the main part to the Ansys interface, the file format was changed to step and transferred to the Ansys Static Analysis window. The main part will be printed on a 3D printer and PLA will be used as the material. The properties of the PLA material have been entered into the material properties to analyze correctly. The image of the designed model is given in Figure 2.

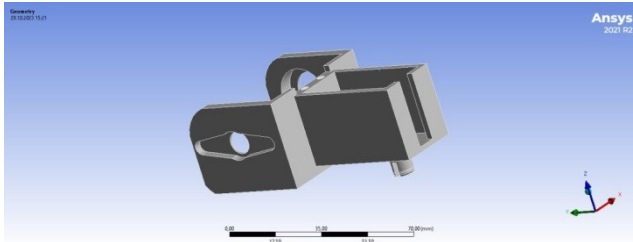


Fig.2. Snake-like robot body part

2.1.2. Identification of static loads

To perform static analysis, the part must be meshed. Meshing is a fundamental step in finite element analysis. This process allows a complex object to be divided into simple parts so that the analysis can be performed more efficiently. A denser mesh usually gives more accurate results but increases the computation time. These simple elements more closely follow the response of the material and allow the results to be localized. The mesh is also essential for defining boundary conditions and external loads. According to the mesh result of the part, the number of nodes and elements was calculated mathematically. In total, 640.627 nodes and 429.295 elements were formed. The mesh processed part is given in Figure 3.

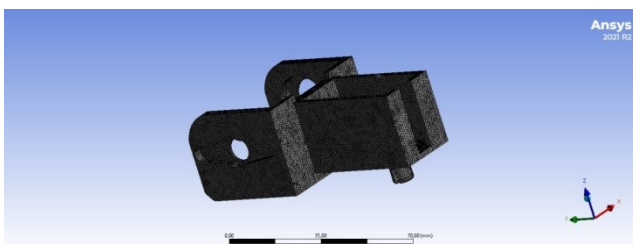


Fig.3. Static meshing of the part

For the static analysis to take place, it is necessary to give boundary conditions to the part. It is necessary to give the place where the part will be fixed and the places where force will be applied. Since the part is a rotary joint, moment, force, and fixing loads are given. The image of the part with boundary conditions is given in Figure 4.

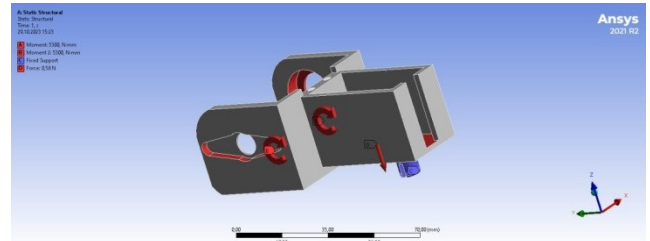


Fig.4. Determination of static boundary conditions

2.1.3. Realization of static analysis

The analysis was concluded according to the given boundary conditions. The analysis measures the mechanical resistance of the part to the given forces. In engineering, the term "stress" refers to the force acting on a unit area of a material. This helps us to understand how much load a material is under and how this load is distributed in the material. This is important for determining whether materials fracture, deform, or otherwise fail. Figure 5 shows the stress distribution.

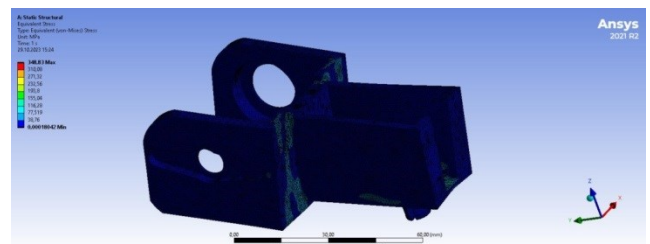


Fig.5. Stress results

Total deformation is a quantity that expresses how much a material or structural element is deformed as a result of applied loads and boundary conditions. As seen in Figure 6, the total displacement of the part is 19,347 mm. This displacement is the total displacement of the part according to the applied boundary conditions.

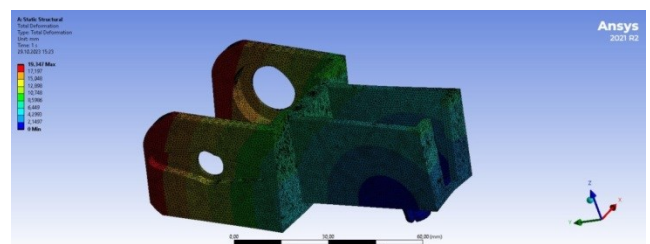


Fig.6. Total deformation results

Elastic strain refers to the deformation of a material within its elastic limit under an applied load or stress. Such deformations disappear completely when the load applied to the material is removed and the material returns to its original shape. Elastic strain results are shown in Figure 7.

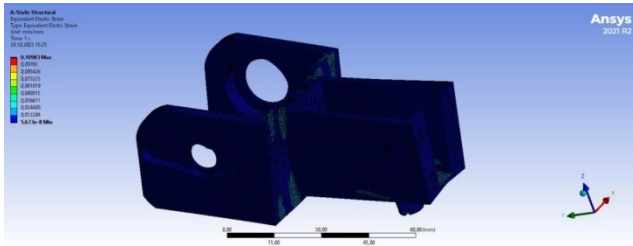


Fig.7. Elastic strain results

2.2. Topology optimization

2.2.1. Defining optimization parameters

Topology optimization is a structural optimization method used to maximize specified performance criteria by optimizing material distribution in a given design space, and this process aims to achieve lightweight yet functional and durable designs by minimizing material usage while aiming to maintain structural performance. The analysis was performed in the Ansys program with 48 iterations. It is desired to reduce the volume and mass ratio by 50%. The volume of the part before the analysis was measured in the program as 25.115 mm³ and the mass as 31,394 grams. The new model resulting from the optimization is shown in Figure 8.

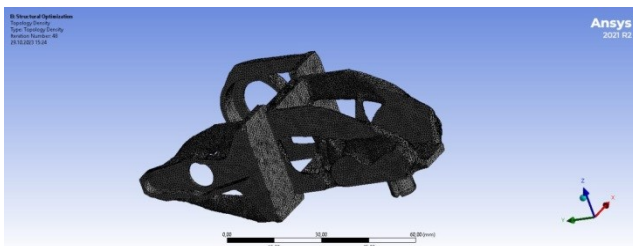


Fig.8. New optimization model

2.2.2. Evaluation of optimization results

According to the optimization result, the mass and volume of the part were almost halved by 49.86%. The new volume of the part was measured as 12.524 mm³ and its mass as 15,655 grams.

2.3. Remodeling and analysis of optimized model

2.3.1. Modeling of the optimized limb

Since the new design resulting from the optimization was a distorted design, the modeling was redone manually. The new design was made to resemble the optimized part. It was redrawn in Space Claim, a sub-window of the Ansys program. The new model is shown in Figure 9.

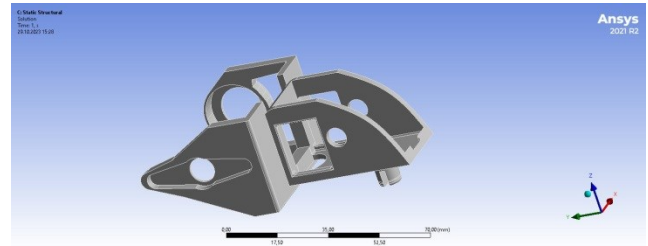


Fig.9. New model after optimisation

2.3.2. Static analysis of optimized design

The final version of the model was evaluated for accuracy and reliability. In this verification process, the model was subjected to static analysis in the Ansys simulation program with a standardized mesh structure by the specific boundary conditions. The stress distributions obtained as a result of this comprehensive analysis are presented in Figure 10, strain rates in Figure 11, and total deformation results in Figure 12. These results are critical for evaluating the overall performance and application potential of the model.

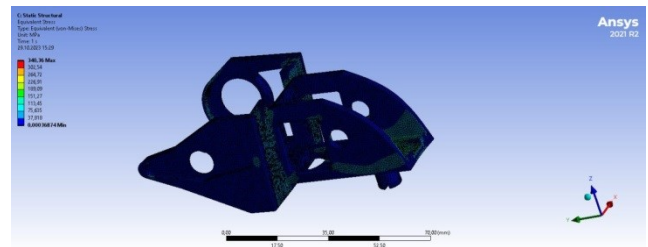


Fig.1. Stress results of the new model

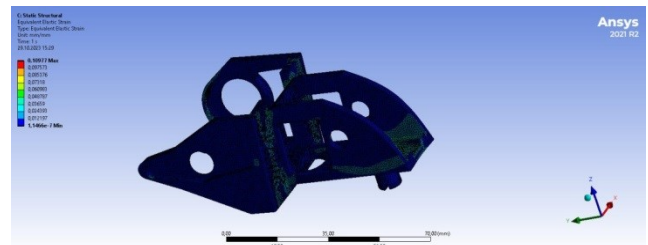


Fig.11. Elastic strain results of the new model

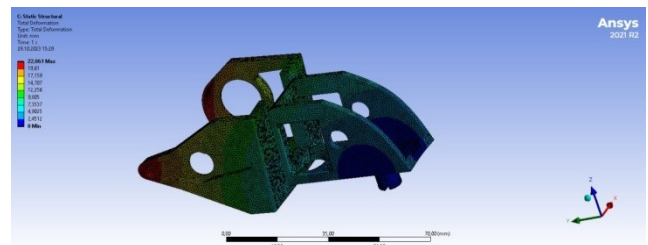


Fig.12. Total deformation results of the new model

Tab.1. The Comparison of Each Model

| Analysis Results | Results Before Optimization | Results After Optimization |
|------------------|-----------------------------|----------------------------|
|------------------|-----------------------------|----------------------------|

| | | |
|--|---------|---------|
| Total Deformation (mm) | 19,347 | 22,061 |
| Equivalent Elastic Strain (mm) | 0,10983 | 0,10977 |
| Equivalent(von-Mises) Stress(MPa) | 348,83 | 340,36 |
| Weight (g) | 31,394 | 15,655 |
| Volume (mm³) | 25115 | 12524 |

The results found after the optimization study are compared in a table. The table shows the results found before optimization and the results found after optimization. The maximum deformation increased by 14,028%. This increase harmed the part. The maximum strain decreased by 0.055%. On the other hand, a decrease of 2.43% was observed in the maximum stress. The part of 31,394 grams decreased to 15,655 grams as a result of optimization, resulting in a reduction of 50.133%. The same reduction was achieved in volume as in mass. The volume value, which was 25115 mm³, decreased by 50.133% and became 12524 mm³.

3. Results and Discussion

The limb of the snake robot was analyzed in a computer environment and the results were compared. The part was printed on a 3D printer to see the accuracy of optimization using PLA material in real life. First, the non-optimized part was printed. Then the optimized part was printed under the same conditions. The mass of the first printed part was 23.50 grams and the mass of the second printed part was 12.00 grams. Weighing was done using a precision balance. Experimentally, a mass reduction of 48.936% was determined. The measurement results are given in Figure 13 and Figure 14.

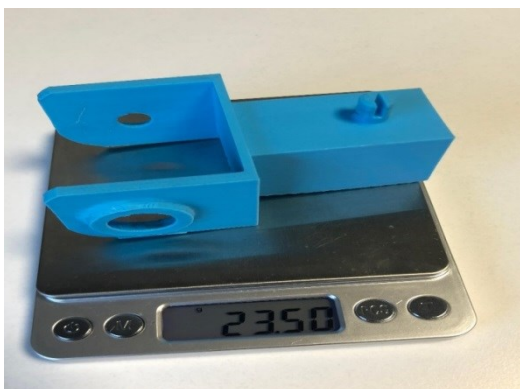


Fig.13. Precision balance results before optimization

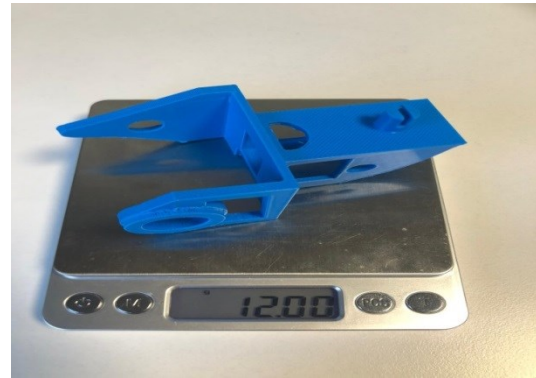


Fig.14. Precision balance results after optimization

4. Conclusion

Topology optimization enables the optimal distribution of material, maximizing structural strength while achieving significant increases in functionality and efficiency. This methodology allows the creation of lightweight yet robust designs with the economy of materials and promotes the improvement of mechanical properties. In this study, the topology optimization of the main part structure of a snake-like robot is investigated. As a result of the analysis, critical evaluations and conclusions about the model and thematic area are presented. A detailed investigation of the main part region of the snake-like robot is carried out. In the first step, a static analysis of the selected main part was performed. The results of this initial analysis were used to determine the basic parameters of the topology optimization process. Upon completion of the optimization, the geometry and structural features of the resulting model were taken into account and revised for a more consistent and functional design. This revised model was again subjected to static analysis under the same boundary conditions and loadings used initially. Comparative analysis showed a mass reduction of 50,133% in the post-optimization model. To validate these findings, the post-optimization model was converted into a physical prototype using 3D printing technology. Measurements performed on a precision balance confirmed a 48.936% reduction in the mass of the prototype. A significant improvement in stress and strain values was also observed. These improvements, together with a significant weight reduction, allow for sustainable resource management as well as effective optimization of raw material use. This is a great advantage not only in terms of mechanical performance enhancement but also in terms of environmental sustainability. Inconsistencies were detected between the mass value obtained as a result of the calculations made in Ansys software and the mass of the part produced by the 3D printing method. This is an unexpected finding encountered during the study process. Although the reasons underlying this discrepancy require detailed research, it can be predicted that some main factors play a

role in this difference. The element mesh density and distribution chosen during the meshing phase can have a critical impact on the results of the simulation. In regions with complex geometry, suboptimal mesh density may adversely affect the accuracy of the results. In addition, the homogeneous assumption of the material considered in the simulation ignores the variations and irregularities in the microstructure of the real material. This may reflect the differences in homogeneity of the material used in practice. Furthermore, all simulation and manufacturing processes have a certain range of tolerances within them, and the cumulative effect of these tolerances can lead to variances in the mass results obtained.

The core objective of this research is to maximize the performance and effectiveness of parts and engineering modeling by minimizing material usage. Structural improvements achieved through optimization methods can extend the lifetime of robotic components by significantly reducing stress and strain values. This not only improves the overall performance of the components but also ensures longer and more reliable operation. Such an optimization approach not only offers technical advantages but also positively improves environmental impacts. In particular, energy consumption is optimized along with weight reduction, contributing to the wider adoption of sustainable engineering practices. The potential improvements achieved in the current research are of critical value. However, the identification of adverse side effects occurring during the modeling process is essential to provide direction for future studies. The reduction in weight, stress, and strain parameters represents positive progress in terms of technical development. However, the marked increase in deformation is an issue that requires further investigation. Therefore, minimization of deformation and further refinement of the model should be critical for future steps.

In future work, it is suggested to explore alternative material options to improve the performance, durability, and potential of the optimized model. In conclusion, this research documents that topology optimization can contribute to efficiency gains and sustainable resource utilization.

References

- [1] Chuang, C., Chen, S., Yang, R. and Vogiatzis, P., Topology optimization with additive manufacturing consideration for vehicle load path development. *International Journal for Numerical Methods in Engineering*, 113:8 (2017) 1434-1445. <https://doi.org/10.1002/nme.5549>
- [2] Bahlau, J. and Lee, E., Designing moulded pulp packaging using a topology optimization and superimpose method. *Packaging Technology and Science*, 35:5 (2022) 415-423. <https://doi.org/10.1002/pts.2639>
- [3] Zhu, J. and Chen, X., An engineering constraint method for continuum structural topology optimization. *Advances in Mechanical Engineering*, 9:12 (2017). <https://doi.org/10.1177/1687814017743387>
- [4] Hu, Z., Vambol, O., Sun, S. and Zeng, Q., Development of a topology optimization method for the design of composite lattice ring structures. *Eastern-European Journal of Enterprise Technologies*, 4:1:112 (2021) 6-13. <https://doi.org/10.15587/1729-4061.2021.238266>
- [5] Kentli, A., Topology optimization applications on engineering structures. *Truss and Frames - Recent Advances and New Perspectives*, (2023) 1-23. <https://doi.org/10.5772/intechopen.90474>
- [6] Tsavdaridis, K. D., Applications of topology optimization in structural engineering : high-rise buildings and steel components. *Jordan Journal of Civil Engineering*, 9:3 (2015) 335-357. <https://doi.org/10.14525/jjce.9.3.3076>
- [7] Eschenauer, H. and Olhoff, N., Topology optimization of continuum structures: a review. *Applied Mechanics Reviews*, 54:4 (2001) 331-390. <https://doi.org/10.1115/1.1388075>
- [8] Sha, L., Lin, A., Zhao, X., and Kuang, S., A topology optimization method of robot lightweight design based on the finite element model of assembly and its applications. *Science Progress*, 103:3 (2020). <https://doi.org/10.1177/0036850420936482>
- [9] Yin, H., Liu, J., and Yang, F., Hybrid structure design of lightweight robotic arms based on carbon fiber reinforced plastic and aluminum alloy. *IEEE Access*, 7 (2019) 64932-64945. <https://doi.org/10.1109/access.2019.2915363>
- [10] Zhang, H., Kumar, A., Chen, F., Fuh, J. Y. H., and Wang, M. Y., Topology-optimized multimaterial soft fingers for applications on grippers, rehabilitation, and artificial hands. *IEEE/ASME Transactions on Mechatronics*, 24:1 (2019) 120-131. <https://doi.org/10.1109/tmech.2018.2874067>
- [11] Junk, S., Klerch, B., Nasdala, L., and Hochberg, U., Topology optimization for additive manufacturing using a component of a humanoid robot. *Procedia CIRP*, 70 (2018) 102-107. <https://doi.org/10.1016/j.procir.2018.03.270>
- [12] Song, X., Chen, D., and Qu, B., Topology optimization of robotic arm for gas drainage pipeline installation. In *Journal of Physics: Conference Series*, 2383:1 (2022, December) 012014, IOP Publishing. <https://doi.org/10.1088/1742-6596/2383/1/012014>
- [13] Sun, Y., Liu, Y., Zhou, N., and Lueth, T. C., A matlab-based framework for designing 3D topology optimized soft robotic grippers. In *2021 IEEE/ASME International Conference on Advanced Intelligent Mechatronics (AIM)*, (2021, July) 1283-1289. <https://doi.org/10.1109/AIM46487.2021.9517350>
- [14] Hussain, F., Mohammadian, M., and Goecke, R.,

- Topology and material optimization of an underactuated robot for gait rehabilitation. *Mechanics Based Design of Structures and Machines*, (2023) 1-18. <https://doi.org/10.1080/15397734.2023.2252492>
- [15] Sun, Y., Zong, C., Pancheri, F., Chen, T., and Lueth, T. C., Design of topology-optimized compliant legs for bio-inspired quadruped robots. *Scientific Reports*, 13:1 (2023) 4875. <https://doi.org/10.1038/s41598-023-32106-5>
- [16] Bodog, A. B., and Grebenişan, G., An integrated approach of topological optimization with parametric validation and response surfaces, using ANSYS®. In *IOP Conference Series: Materials Science and Engineering*, 1256:1 (2022, October) 012014. IOP Publishing. <https://doi.org/10.1088/1757-899X/1256/1/012014>
- [17] Yang, X., Luo, W., Liu, H., Zhang, S., and Ge, W., Optimal Design for the Torsional Elastic Element Topology of a Rotary Series Elastic Actuator. In *2022 7th Asia-Pacific Conference on Intelligent Robot Systems (ACIRS)*, (2022, July) 13-17. <https://doi.org/10.1109/ACIRS55390.2022.9845652>
- [18] Lou, J., and Gong, Y. Rigid-flexible Coupling Dynamic Analysis and Topological Optimization Design of Robot End-effector, *U.P.B. Sci. Bull., Series D*, 84:2 (2022) 17-30.
- [19] Sun, Y., Liu, Y., Pancheri, F., and Lueth, T. C., Larg: A lightweight robotic gripper with 3-d topology optimized adaptive fingers. *IEEE/ASME Transactions on Mechatronics*, 27:4 (2022) 2026-2034. <https://doi.org/10.1109/TMECH.2022.3170800>
- [20] Coşkun, Y. and Aydın, M. (2023). Yılansı robotun tasarımı, statik, dinamik ve modal analizi. B. Akdemir (Ed.), *Mühendislikte Yenilikçi Yaklaşımlar*, İzmir: Duvar Yayınları. (2023) 602-615.