DESIGN and STRUCTURAL OPTIMIZATION of a ROBOTIC END-EFFECTOR USING SIMP-BASED TOPOLOGY OPTIMIZATION

Hilmi Saygin SUCUOGLU
Aydın Adnan Menderes University, Engineering Faculty, Mechanical Engineering Department

Abstract— **In this study, a robotic end-effector was designed, analyzed, and optimized using the Solid Isotropic Material with Penalization (SIMP) topology optimization technique. The design process began with the CAD modeling of a robotic arm and its end-effector using SolidWorks, followed by structural evaluation through Finite Element Analysis (FEA) to assess mechanical feasibility. The SIMP-based optimization was then applied under identical boundary and loading conditions. As a result, the end-effector's weight was reduced by approximately 47%—from 530.7 g to 280.1 g—while maintaining acceptable levels of stress and displacement. The von Mises stress increased slightly from 3.4 MPa to 4.3 MPa, and the maximum displacement rose from 1.2 mm to 1.9 mm, both remaining within safe operational limits. These findings demonstrate the effectiveness of SIMP-based topology optimization in reducing material usage and enhancing structural efficiency in robotic systems.**

Index Terms— **Evolutionary Structural Optimization, Robot Arm, Robot End Effector, Solid Isotropic Material with Penalization, Topology Optimization**

I. INTRODUCTION

The topology is the main branch of mathematics. Topology can be used in the context of location, surface and space. It examines the qualitative properties and relative positions of objects independent of shape and size. Topology optimization has been developed within the field of applied mathematics. Among structural optimization methods, it is a relatively recent and rapidly evolving area of research. There are numerous application examples of topology optimization in engineering disciplines such as mechanical, automotive, civil, and aerospace engineering [1].

Topology optimization is one of the most powerful design tools for determining the most efficient material distribution within a given design domain. The core principle of topology optimization is to strategically remove material from specific regions of a component without altering its outer dimensions or compromising its structural rigidity [2]. The aim of topology optimization is to find a conceptual layout of a design by distributing a given amount of material in a domain thereby achieving the lightest and stiffest structure while satisfying certain specified design constraints [3,4]. Within the selected design domain, the optimal material distribution that satisfies constraints while achieving objectives such as minimum mass or volume is obtained through multiple iterations [5]. Different constraints can be included to get to the target distribution. Until the constraints define mass, volume, weight, and displacement, the optimization program removes materials on the component model [6].

Topology optimization includes various techniques that can be distinguished into four subfields. The commonly used methods are SIMP (Solid Isotropic Material with Penalization) and ESO (Evolutionary Structural Optimization). SIMP is a density-based topology optimization technique. A key feature of the SIMP method is that the densities of the finite elements composing the structure are driven toward either 0 or 1. Intermediate densities are penalized to encourage a binary distribution, which aids in producing a manufacturable geometry, as regions with intermediate densities are typically difficult to fabricate. However, in some cases, including such regions may enhance the overall performance of part [7].

In the density based SIMP technique, although microstructural details are not considered, this method can establish a useful relationship between density and material properties [8].

Many numerical challenges related to topology optimization have been addressed and handled through the usage of the SIMP technique. The SIMP has been improved significantly with the contribution of researchers such as Rozvany [9]. Today design life is the most widely adopted and used approach.

SIMP technique is also implemented by various design software as it is easy to use and can provide extremely fast solutions for topology problems. It provides an alternative and easier solution to power-law approaches which is relatively more complex. The structural compliance or strain energy is implemented as the objective function of volume with applied constraint. In SIMP technique, two important parameters, material properties and element thickness (porosity) are assumed as uniform through each finite element [10].

The main purpose of the SIMP method is to eliminate intermediate pseudo-density values (ηi). In finite element analysis (FEA), the design domain is discretized into N finite elements, each assigned a pseudo-density value (ηi). This value ranges between 0 and 1, where $\eta i = 1$ represents a solid element and $\eta i = 0$ corresponds to a void. These values are directly related to the stiffness of the structure. Intermediate pseudo-density values between 0 and 1 are considered undesirable, as they represent partially solid regions that are difficult to manufacture and may reduce structural clarity. To address this issue, the SIMP method introduces a penalization factor (p) to modify material properties and discourage intermediate densities. The penalization factor, which must be greater than 1, forces the density values toward either 0 or 1. In this approach, the Young's modulus of each element is expressed as a function of its pseudo-density raised to the power of p, typically in the form:

$E_i = x_i^P E_0$ (1)

Where E_i is the effective Young's modulus of the i-th element, x_i is the pseudo-density, E_0 is the Young's modulus s of the solid material, and p is the penalization factor. This penalization ensures that intermediate densities contribute less to stiffness, promoting a clearer and more manufacturable topology [10-12].

The optimization problem is then formulated as:

$$\min \mathrm{C}(\eta) = u^T \mathrm{K}(\eta)\mathrm{u} \text{ subject to } = \sum_{i=1}^{N} \eta i v i \leq \mathrm{V} \quad (2)$$
$$0 < \eta\min \leq \eta i \leq 1$$

Where:

$\mathrm{K}(\eta)$ is the global stiffness matrix, which depends on the material distribution,

u is the displacement vector obtained from the finite element analysis,

ηi represents the pseudo-density assigned to the i[th] finite element,

vi is the volume of the i[th] element,

V denotes the maximum allowed total material volume in the design domain,

ηmin is a small positive lower bound for the density values to avoid singular stiffness matrices.

This formula (2) ensures that elements with intermediate density values contribute less to the stiffness of the structure. The penalization scheme guides the solution toward a binary (0 or 1) material distribution, which is more manufacturable and structurally clear.

The Evolutionary Structural Optimization (ESO) method was first introduced by Xie and Steven in 1993 for numerical structural topology optimization problems [13]. ESO is an effective heuristic approach for topology optimization problems. It uses discrete finite elements. This approach utilizes the finite element method (FEM). The iterative removal of inefficient material from a structure is the main principle of ESO is. "Inefficiency" in this context is a general term referring to the sensitivity of a finite element's contribution to the defined optimality criterion. The sensitivity can be considered based on various performance parameters. The optimality criteria may be a composite of multiple physical parameters such as stress, strain energy, or compliance [14]. The ESO method evolves structure toward an improved topology. In most initial designs, element removal in the ESO method proceeds without issues; however, under certain loading and support conditions, the process may lead to structural instability. This is because the elements located near load application or support regions which may exhibit low stress values, to prevent those failures, such elements should be excluded from the removal process through using protective rules [15]. Following each finite element analysis, the stress distribution across the structure is evaluated. Portions of the material contribute minimally to structural performance and can be considered inefficient. A commonly used rejection criterion is the von Mises stress. The elements: von Mises stress is below a threshold, defined as a rejection ratio (RR) time the maximum von Mises stress in the structure is deemed inefficient and removed. This analysis–removal cycle is repeated iteratively using the same RR value until the system reaches a steady state. Once this steady state is achieved, an evolution rate (ER) is introduced and added to the RR to gradually increase material removal intensity. The process continues with updated RR values until a new steady state is obtained. This evolutionary optimization process proceeds until a desired structural condition is met. As an example, until all stress levels within 25% of the maximum stress. Although the result may not represent a mathematically absolute optimum, the ESO method provides a step-by-step evolution path that can provide valuable insights into the progression of shape and topology. Ultimately, the final design obtained through ESO often approaches a fully stressed configuration, in which all retained material is utilized efficiently and stressed close to its capacity [16]. Based on the findings of some publications, it is assumed that the ESO method effectively minimizes the compliance volume product of a structure or a finite element model. The mathematical formulation of the minimum compliance problem can be expressed as [17]:

Minimize $C(x) \; x \, \Omega$ (3)

Subject to $V(x) \, / \, V_0 \; \leq \; f$ (4)

Here, $C(x)$ represents the compliance of the topology, $V(x)$ is the material volume, V_0 is the total volume of the design domain, and f denotes the prescribed volume fraction, defined as $f = \frac{V(x)}{V_0}$. (5)

The stress-based version of the ESO method typically uses the von Mises stress as the optimality criterion to guide element removal. This initial concept removes elements with low stress levels. It could be interpreted as equivalent to using a sensitivity-based approach, where the sensitivity is defined as the change in compliance with respect to the removal of material [18].

In the initial setup, a sufficiently large piece of material is discretized into a fine mesh of finite elements. Loads and boundary conditions are then applied, and a stress analysis is performed using a finite element solver. Because the structure is divided into numerous small elements, the removal of material can be systematically implemented based on chosen criteria. The stress level in each element is typically evaluated by calculating an average measure of all stress components. For isotropic materials, the von Mises stress is one of the most widely used and reliable criteria for this purpose. It effectively captures the combined effect of multi-axial stresses and serves as a robust indicator for identifying low-stress regions that are candidates for removal during the optimization process [17].

Robot manipulators are equipped with end-effectors to perform various tasks. Among these, grippers are devices designed to grasp and handle workpieces. As one of the most widely used types of end-effectors, grippers have become the most extensively developed and diversified due to their broad range of applications.

Multi-purpose grippers have emerged with advancements in actuators, mechanisms, and sensor technologies. The design of those grippers has been inspired by nature, especially modeled like a human hand. End-effectors can provide the ability to robotic structures to physically interact with the environment. Those grippers are the critical components, as they are crucial to the performance of robotic systems.

End-effectors can take various forms. They can be mechanical grippers, vacuum systems, magnetic holders, and special-purpose machining tools. Each type of end-effector is selected based on the requirements of the application. They are designed to perform specific functions. The selection of an appropriate end-effector depends on the requirements of the specific task and the operational environment.

The advancements in structural optimization have led to increased interest in the application of topology optimization methods to design robotic components such as end-effectors. Several studies have shown that important mass reductions can be achieved. For instance, Degirmencioglu et al. reported that applying topology optimization to a robotic end-effector resulted in approximately 20% weight reduction [19].Their findings underscore the potential of optimization-based design approaches in improving the efficiency, responsiveness, and energy consumption of robotic systems, In addition to these developments, recent studies have applied topology optimization to multi-end-effector robotic arms using commercial software platforms. In one such investigation, a robotic arm equipped with four end-effectors was optimized using Fusion 360, resulting in a substantial weight reduction of approximately 43%. Despite this reduction, the safety factor for each end-effector remained above 3, and a minimal displacement of only 0.0067 mm was observed. [20].

This study focuses on the design, analysis, and optimization of a robotic end-effector. The primary objective is to reduce the overall mass of the end effector by applying SIMP based topology optimization, while keeping its structural integrity. The robotic structure was designed using SolidWorks. The FEA (Finite element analysis) was conducted to evaluate structural behavior under specified loading conditions. The SIMP-based optimization approach was applied to create a lighter structure. The obtained results were validated through comparative stress and displacement analyses. This study presents a practical workflow that integrates CAD modeling, structural analysis, and optimization for the development of high-performance robotic components.

Recent advancements in structural optimization have enabled the design of lightweight, efficient robotic components, particularly end-effectors that directly impact system performance. While several studies have demonstrated the utility of topology optimization in mechanical and aerospace structures, the application of SIMP-based methods to robotic grippers and end-effectors remains relatively limited. Existing works, such as those by Degirmencioğlu et al. [19] and Monteiro et al. [20], highlight the potential for mass reduction and structural efficiency, but are often focused on multi-component systems or specific industrial applications. This study contributes to the field by integrating parametric CAD modeling, finite element analysis, and SIMP-based topology optimization in the context of a single-function robotic end-effector. The achieved 47% weight reduction, while maintaining acceptable stress and displacement levels, demonstrates the practical value of this method for enhancing the energy efficiency and dynamic responsiveness of robotic systems. Therefore, the proposed approach serves as a repeatable and accessible optimization framework for robotic component designers.

II. MATERIAL AND METHOD

A. Designs of the Robot Arm and End Effector

The mechanical design of the robotic arm and end-effector was conducted using SolidWorks Software, which is commonly used computer-aided design (CAD) software. In the design process, individual modeling of each component, including servo motors, connecting brackets, structural arms, and the gripping mechanism were created. Geometric compatibility between parts and ease of assembly, as well as preserving the functional integrity of the system during motion were taken into consideration.

Once the components were modeled, an assembly model was created to verify mechanical fit, joint constraints, and overall motion capabilities. All the structures were designed using parametric modeling to enable rapid iterations and dimensional adjustments. Servo motors were positioned to ensure the necessary degrees of freedom and to provide effective manipulation tasks by the end-effector.

After the 3D Cad modelling process, detailed engineering drawings of the robotic structure and its end-effector were created. Those technical drawings were prepared to define precise dimensions, tolerances, and geometric relationships. The engineering drawings of both the robotic structure and the end-effector are presented in Figures 1 and 2, respectively.

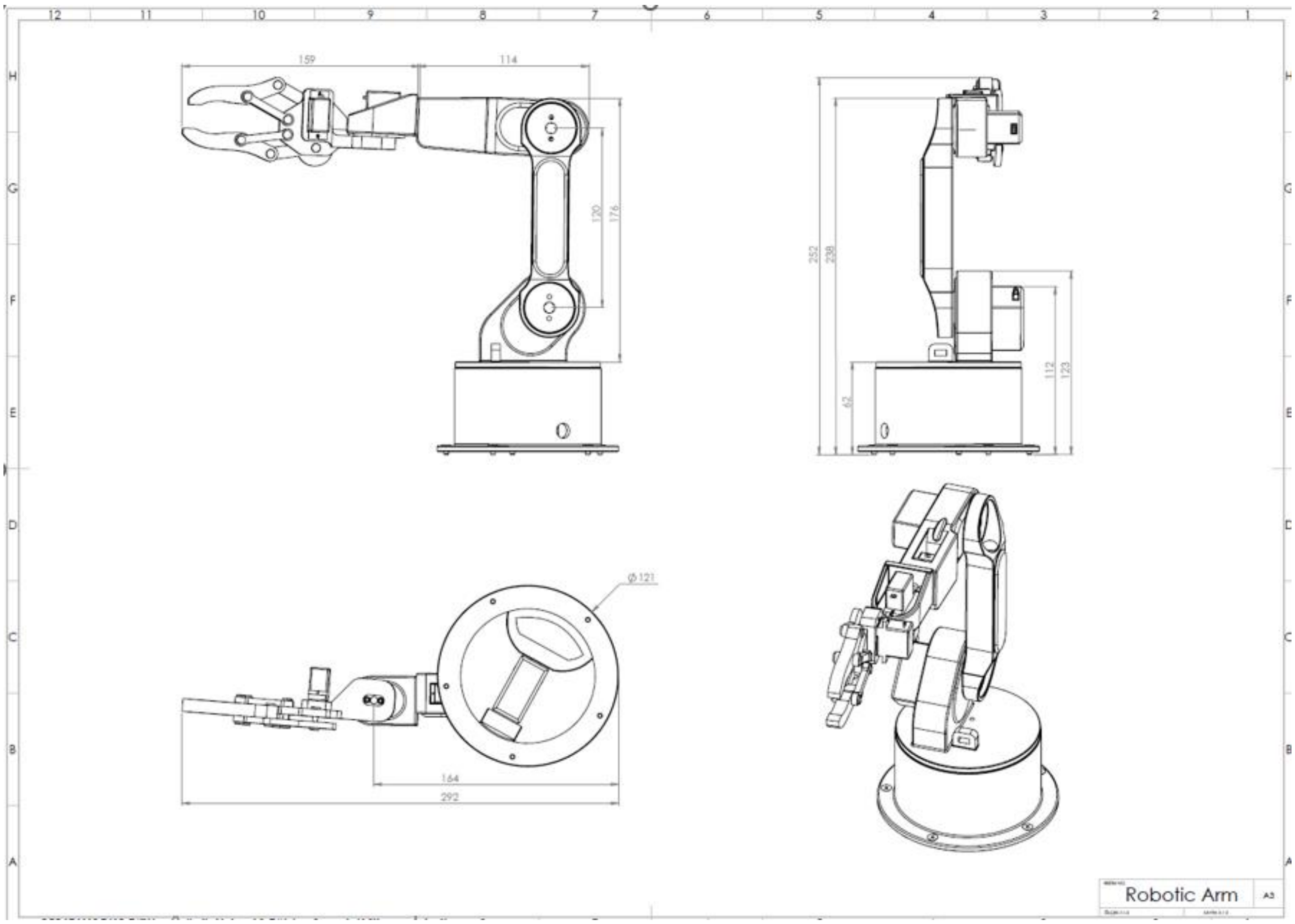

Fig. 1. Engineering drawing of robotic structure

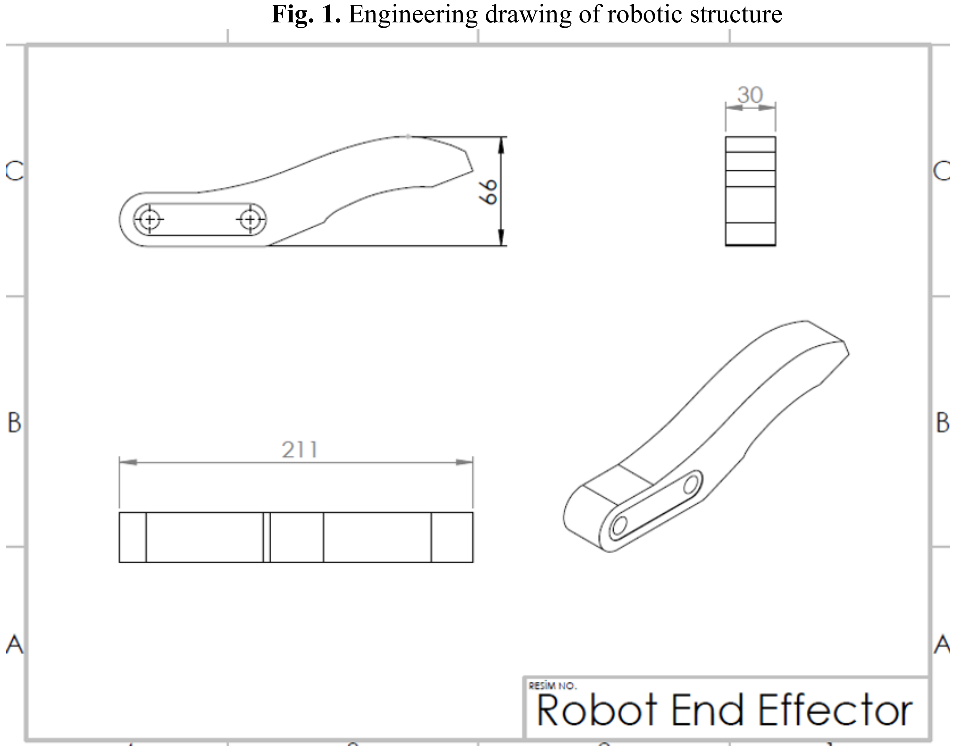

Fig. 2. Engineering drawing of end effector

B. Engineering Analysis of Preliminary (Non-Optimized) End Effector

The finite element analysis (FEA) was performed using the structural analysis module of SolidWorks. The selected material for the structural components was aluminum alloy 6061, chosen for its favorable strength-to-weight ratio and machinability. As shown in Figure 4, the model was constrained at the points indicated by green arrows to simulate fixed boundary conditions. A load of 1,000 N was applied to each end-effector component at the locations marked with purple arrows to represent operational loading conditions. The details of the preliminary engineering analysis of the end-effector, including boundary conditions and loading configuration, are illustrated in Figure 3.

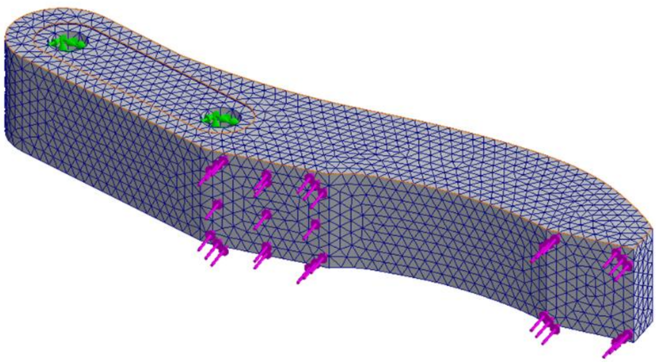

Fig. 3. Engineering analysis of Preliminary structure

C. Topology Optimization Process

Topology optimization was performed using the SolidWorks Optimization environment based on the SIMP (Solid Isotropic Material with Penalization) method. The same loading and boundary conditions applied in the preliminary engineering analysis were maintained throughout the optimization process to ensure consistency (Figure 4).

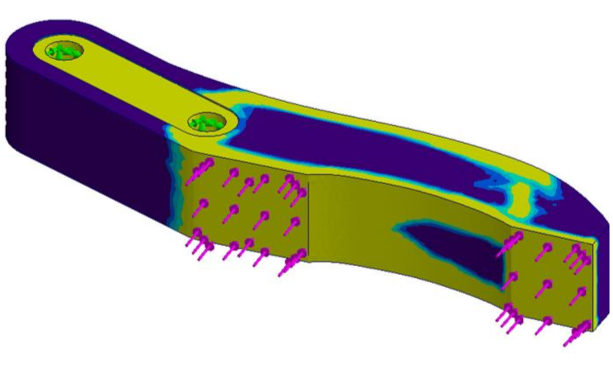

Fig. 4 Optimization process

Through multiple iterations, the optimization algorithm generated several design alternatives aimed at reducing mass while preserving structural integrity. Based on the results obtained and design file outputs, a new optimized end-effector geometry was developed, as shown in Figure 5.

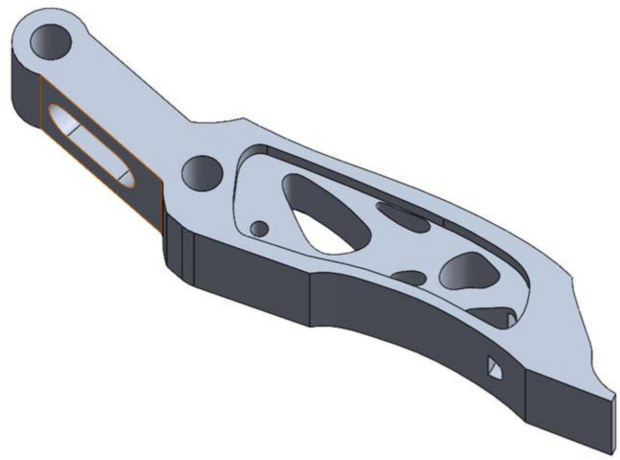

Fig. 5. Optimized end-effector

As a result of topology optimization, the weight of the end-effector was reduced from 530 grams to 280 grams, representing a significant decrease in mass. To validate the structural performance of the newly optimized design, finite element analysis was conducted under the same loading conditions used for the non-optimized structure. This ensured a direct comparison in terms of structural integrity and mechanical behavior. The results of the structural analysis for the optimized end-effector are presented in Figure 6.

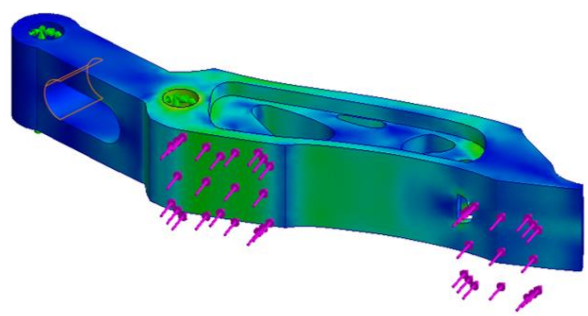

Fig. 6. Engineering analysis of optimized end-effector

III. RESULTS AND DISCUSSION

The results of the structural analyses were examined to assess the necessity of applying topology optimization. In the initial, non-optimized design, the weight of the end-effector was approximately 530 grams, and the observed von Mises stresses were around 3.4 MPa under the defined loading conditions (Figure 7). Based on these findings, it was concluded that the structure contained regions with excess material that did not significantly contribute to load-bearing capacity. Therefore, the application of topology optimization was deemed both feasible and beneficial for improving structural efficiency.

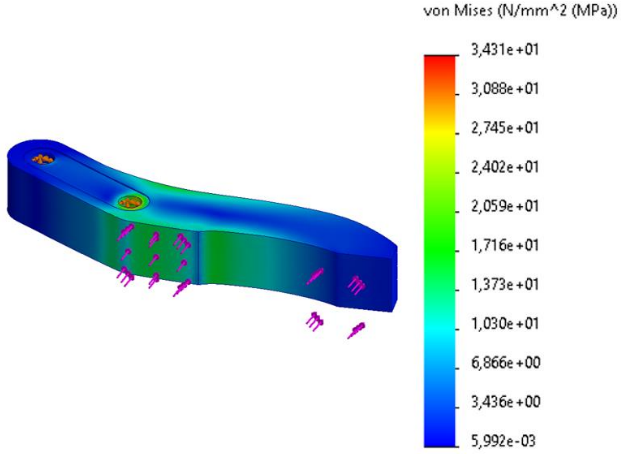

Fig. 7. von Misses stress value of non-optimized end effector

Following the optimization process, engineering analysis was repeated on the newly optimized end-effector under the same loading and boundary conditions. The von Mises stress observed in the optimized structure increased to 4.3 MPa, while the maximum displacement was measured as 1.9 mm. The von Mises stress distribution for the optimized end-effector is presented in Figure 8. Despite the slight increase in stress and displacement, both values remained within acceptable engineering limits, confirming the structural integrity of the optimized design.

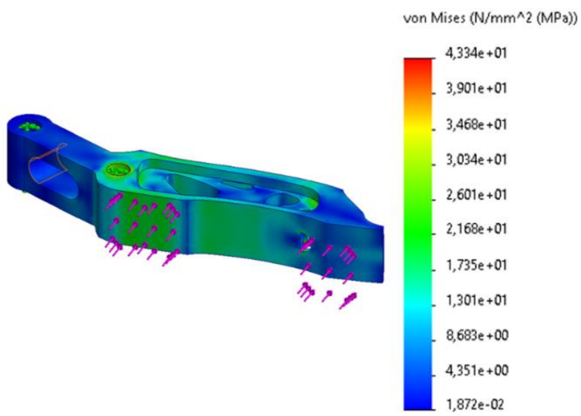

Fig. 8. von Misses stress value of optimized end effector

A detailed comparison table of non-optimized and optimized end effectors are given in Table 1.

TABLE I

COMPARISON TABLE of STRUCTURES of END EFFECTORS

Results	Non-optimized Structure	Optimized Structure
von Mises stresses of Robot End Effector (MPa)	3,4	4,3
Weight of Robot End Effector (g)	530,7	280,1
Displacement of Robot End Effector (mm)	1,2	1,9

As shown in Table 1, topology optimization of the robotic end-effector successfully reduced its weight from 530.7 to 280.1 grams. It was an approximate 47% decrease, while keeping structural integrity. Throughout the process, von Mises stress increased slightly from 3.4 MPa to 4.3 MPa, and displacement from 1.2 to 1.9 mm. However, both remained within acceptable engineering limits, confirming the validity of the SIMP-based optimization.

These results are consistent with prior studies. For instance, Değirmencioğlu et al. reported a 20% mass reduction with retained structural performance [19], while Monteiro et al. achieved a 43% decrease in a multi-end-effector robotic arm while maintaining safety factors above 3 [20]. The 47% reduction achieved in this study surpasses those benchmarks, demonstrating the effectiveness of the integrated design and optimization approach using SolidWorks.

Moreover, reduced mass could enhance system level efficiency by lowering energy consumption and improving the dynamic response of robotic structure. These findings emphasize the value of incorporating topology optimization early in the design phase, particularly for mobile and terrain-adaptive robotic applications.

IV. CONCLUSION

This study demonstrates that advanced computational tools can be used to successfully design, analyze and optimize the topology of a robotic end-effector. The structural analysis of the initial design revealed regions of low stress that contributed minimally to the overall performance, indicating a clear potential for mass reduction. Applying the SIMP-based topology optimization method in SolidWorks reduced the end-effector's weight by 47%, from 530.7 g to 280.1 g. This was achieved without compromising mechanical integrity, as validated by the von Mises stress and displacement values, which remained within acceptable engineering limits.

The optimized structure clearly improved material efficiency and will undoubtedly enhance the energy performance and agility of the robotic system. The results of this study are consistent with similar findings in the literature. This confirms the reliability and practical value of integrating topology optimization into early-stage robotic design workflows. The study definitively shows that topology optimization is an effective tool for improving the structural and operational performance of robotic components in engineering applications.

REFERENCES

[1] Y. Düzcan, "Yapısal optimizasyon teknikleri ile taşıt süspansiyon bileşenlerinin geliştirilmesi", Master's thesis, Bursa Uludag University (Turkey), 2019.

[2] S. Dinçel, "Araç fren pedalının polimer kompozit malzemeden eklemeli üretim yöntemine uygun optimum tasarımı ve seri parça ikamesi olarak üretimi", Master's thesis, Bursa Uludag University (Turkey), 2022.

[3] H. E. Enginar, "Ağır taşıt jantının topoloji optimizasyonu yardımıyla optimum tasarımı", Master's thesis, Dokuz Eylul Universitesi (Turkey), 2014.

[4] H. Guan, Y. J. Chen, Y. C. Loo, Y. M. Xie, and G. P. Steven, "Bridge topology optimisation with stress, displacement and frequency constraints", Computers & Structures, vol. 81, no. 3, pp. 131–145, 2003.

[5] B. Erol, "Uçak komponentlerinin bilgisayar destekli malzeme seçimi ve topoloji optimizasyonu ile tasarımı", Master's thesis, Fen Bilimleri Enstitüsü, 2019.

[6] N. Demir, H. S. Sucuoglu, I. Bogrekci, and P. Demircioglu, "Topology optimization of mobile transportation robot", International Journal of 3D Printing Technologies and Digital Industry, vol. 6, no. 3, pp. 210–219, 2021.

[7] D. Ö. Helvacı, "Tiling of Cellular Structures Into the Parts According to the Density Values of SIMP Topology Optimization", Master's thesis, Middle East Technical University (Turkey), 2020.

[8] M. P. Bendsøe and N. Kikuchi, "Generating optimal topologies in structural design using a homogenization method", Computer Methods in Applied Mechanics and Engineering, vol. 71, no. 2, pp. 197–224, 1988.

[9] G. I. Rozvany and T. Lewiński (Eds.), Topology Optimization in Structural and Continuum Mechanics, Springer, 2014.

[10] S. Liu, Q. Li, J. Liu, W. Chen, and Y. Zhang, "A realization method for transforming a topology optimization design into additive manufacturing structures", Engineering, vol. 4, no. 2, pp. 277–285, 2018.

[11] A. Olason and D. Tidman, "Methodology for topology and shape optimization in the design process", 2010.

[12] G. I. Rozvany, "Aims, scope, methods, history and unified terminology of computer-aided topology optimization in structural mechanics", Structural and Multidisciplinary Optimization, vol. 21, pp. 90–108, 2001.

[13] K. A. Proos, "Evolutionary structural optimisation as a robust and reliable design tool", 2002.

[14] M. Zhou and G. I. N. Rozvany, "On the validity of ESO type methods in topology optimization", Structural and Multidisciplinary Optimization, vol. 21, no. 1, pp. 80–83, 2001.

[15] Y. M. Xie, P. Felicetti, J. W. Tang, and M. C. Burry, "Form finding for complex structures using evolutionary structural optimization method", Design Studies, vol. 26, no. 1, pp. 55–72, 2005.

[16] S. Y. Wang, K. Tai, and M. Y. Wang, "An enhanced genetic algorithm for structural topology optimization", International Journal for Numerical Methods in Engineering, vol. 65, no. 1, pp. 18–44, 2006.

[17] F. M. Özkal and H. Uysal, "General Aspects of Evolutionary Structural Optimization: A Review", Pamukkale University Journal of Engineering Sciences, vol. 15, no. 3, 2009.

[18] G. P. Steven, Q. Li, and Y. M. Xie, "Evolutionary topology and shape design for general physical field problems", Computational Mechanics, vol. 26, no. 2, pp. 129–139, 2000.

[19] F. Değirmencioğlu, Y. C. Koşer, and A. Yıldız, "Endüstriyel Robotlarda Kullanılan Çoklu Tutucunun Yapısal Analizi ve Optimizasyonu", Uluslararası Bilim Teknoloji ve Tasarım Dergisi, vol. 4, no. 2, pp. 103–112, 2023.

[20] B. Monteiro, F. Rocha, and J. Costa, "Topology optimization of a robot gripper with nTopology", U. Porto Journal of Engineering, vol. 10, no. 1, pp. 11–19, 2024.