



RESEARCH ARTICLE

OPEN ACCESS

PARAMETRIC TOPOLOGY OPTIMIZATION OF CYLINDER MOUNTING BRACKET USING PTC CREO

Edosa Kefyalew^{*1} Adugna Fikadu², Mahesh Gopal³¹Technion – Israel Institute of Technology, Technion City, Haifa, 3200003, Israel^{2,3}Department of Mechanical Engineering, College of Engineering and Technology, Wollega University, Nekemte, Ethiopia¹<http://orcid.org/0009-0008-5407-3797>, ²<http://orcid.org/0009-0004-4322-7683>, ³<http://orcid.org/0000-0001-7672-5399>E-mail: *edosa.tesema@campus.technion.ac.il

ARTICLE INFO

Article History

Received on: March 30, 2026

Reviewed on: April 30, 2026

Accepted on: July 01, 2026

Published on: July 31, 2026

Keywords:Structural Optimization,
Additive Manufacturing,
Creo Parametric,
Weight Reduction.

ABSTRACT

Topology optimization is an emerging design technique for producing extremely lightweight, highly efficient structural components without compromising structural integrity. In the current investigation, the topology of a circular mounting bracket is optimized using the integrated structural optimization capabilities of PTC Creo Parametric software. The objective is to minimize the structure's weight without compromising its strength and rigidity. From an initial weight of 1.189 kg, the design was optimized through six phases to achieve the required weights of 70%, 50%, 35%, 25%, 20%, and 17.5%. The actual weight reductions were 71.2%, 52.3%, 37.2%, 27.1%, 22%, and 19.5%. As weight decreased, relative strength increased substantially, reaching a maximum of 100.833 MPa/kg. After analyzing failure modes and strength characteristics, particularly bending failures, the 20% weight-retention design, weighing 0.261 kg (22% of the original weight), was found to be optimal. The results of the study demonstrate that it is possible to create lightweight, high-strength mounting brackets suitable for industrial and automotive applications using parametric topology optimization in PTC Creo.



Copyright ©2026 by authors and Galileo Institute of Technology and Education of the Amazon (ITEGAM). This work is licensed under the Creative Commons Attribution International License (CC BY 4.0).

I. INTRODUCTION

Using the Generative Topology Optimization (GTO) extension in PTC Creo, designers can specify functional design requirements, loads, constraints, and material properties to generate 3D CAD models that are optimized for lightweight manufacturing.

a. Topology optimization

Topology optimization methods for the design of multi-scale structures have advanced rapidly in recent years. A computational design method, topology optimization, can automatically design a structural form with optimal performance for given design parameters [1]. Since it is challenging to produce these novel structures using traditional methods, additional steps are often required to improve manufacturability, which falls short of the full potential of topology optimization. Topology optimization is a sophisticated structural design process that produces novel, high-performance, lightweight configurations that are challenging to achieve using traditional concepts [2]. In the aerospace industry, a small reduction in engine weight results in substantial fuel savings and reduced pollution. For this reason, topology optimization is employed to make aircraft components lighter while maintaining the functionality and lifespan specified by [3]. Without making assumptions about the final geometry, the topology approach optimizes the material layout within a given design region. Consequently, it rapidly gained prominence in the industrial sector and was applied to the design of aircraft components [4]. It has been shown that the topology approach is effective for finding the initial geometric configuration in structural design. The main drawback of the approach, however, is that topology optimization will always yield a non-smooth geometric configuration of the structure.

In contrast, most engineering applications, especially in manufacturing [5], require a smooth configuration. Although topology optimization may not effectively control microstructure, there is still considerable scope to reduce deviations from the desired microstructure [6]. Topology optimization is a numerical process that aims to optimize the shape and material layout within a design space under certain constraints. The solution obtained from topology optimization need not be optimal, but it serves as a basis for optimal design [7]. In the vast majority of engineering applications, topology optimization is among the most effective techniques for the conceptual design phase. Given the boundary conditions and constraints of the design, it is feasible to obtain an optimal structural material distribution that represents the most efficient path in the design domain [2], within certain loads and boundary conditions, topological optimization allows for the estimation of the optimal distribution of material within a design region [8], to enhance the performance of the system, topology optimization is a mathematical method that aims at optimizing the material distribution within a predefined design space for specific loads and constraints [9]. The research work involved optimizing the topology and comparing analytical results before and after the design optimization of the connecting rod. Since the optimized model will be less stressed and lighter than the existing model, it will be more efficient [10]. Within the framework of density-based topology optimization, the behavior of microscopic stress was examined to develop topology optimization techniques capable of handling stress constraints. In aircraft component structural design, topology optimization reduces weight while retaining the required stiffness and strength. In complex aerodynamic and operational conditions, topology optimization enhances loading capacity by optimally distributing materials in the design domain. This helps increase fuel efficiency and reduce emissions [11]. The aim of topology optimization for a lightweight diesel engine alternator support bracket is to minimize material usage while maintaining its stiffness and strength. Based on the applied loads and boundary conditions within a specified design domain, any extra material is reduced throughout the design process [12].

To withstand dynamic loads and vibrations, as well as engine and belt forces, the material distribution is carried out through topological and structural analyses. Stress and deformation behavior under real operating conditions are validated using finite element analysis. An optimized design of a passenger car's compressor mounting bracket improves overall vehicle economy and reduces material costs [13]. The objectives of performing FEA and topology optimization for an engine mounting bracket are to achieve lightweight and optimize structural performance. The deformation, stress distribution, and safety features of an engine mounting bracket are analyzed. The newly designed engine mounting bracket is stronger, more vibration-resistant, reduces material usage, and improves performance without compromising its strength [14]. Topology optimization in Creo Simulate 4.0 enables the reduction of low-stress material in the gripper, thereby reducing structural weight, improving performance, and increasing strength [15]. The aim of the topology optimization of the compressor bracket is to reduce its weight without compromising the required strength and stiffness. Optimized compressor brackets will have better performance, reduced material usage, and longer life. To reduce weight while satisfying the required stress and displacement constraints, the integrated particle swarm optimizer efficiently searches the design space. By considering the structural and geometric effects simultaneously, the topology optimization method improves the global optimality. Optimized truss configurations achieve better material utilization and improved structural performance [16]. A Comparative Study of Topology Optimization Methods for 3D Structural Problems focuses on several numerical methods with respect to their convergence characteristics, efficiency, and correctness of solutions for material distribution in three-dimensional domains. The results help select the most appropriate methods for complex 3D design problems [17].

b. Additive Manufacturing

Additive manufacturing is a process of making parts directly from a digital 3D model. This process differs from subtractive manufacturing in that it reduces waste. This process is also used to make complex parts. This process produces lightweight, topology-optimized, and customized parts. This process is used to make parts for automotive, aerospace, medical, and tool-die industries. The objective is to optimize the Piston's weight while maintaining its strength and thermal stability, thereby enhancing its performance. This can be done using ANSYS and PTC Creo Parametric software. For the design of a lightweight, high-performance device based on this innovative method, the results show improved stress distribution compared with traditional solid orthotics. There is a 30-50% weight reduction compared to traditional solid orthotics. The research article by [18] has made a significant discovery. Under thermo-mechanical stress conditions, the topology of the parametric 3D piston model is optimized to reduce compliance without sacrificing strength. The ANSYS Workbench FEA tools are then employed to simulate the optimized topology. The factors influencing the metal additive manufacturing process include material wall thickness, overhang angle, and orientation [19]. FEA tools are then used to evaluate the material's deformation. The next step is to use the additive manufacturing process to create the required geometry. The results show a significant reduction of 30-60% of the weight while maintaining the required rigidity and safety [20]. Although the AM process offers design flexibility, it has process-specific limitations, including overhang, minimum size, and orientation. This study proposes a methodical approach to topology optimization that accounts for the limitations of the AM process, using ANSYS Workbench as described by [21]. Many researchers have studied and proposed various methods for optimizing structural components used in the food, automotive, and aerospace industries. They have found that topology optimization is a highly effective and powerful tool for structural optimization. The authors conducted experiments on the topology optimization of the automotive suspension bracket and the aero-engine bracket using CREO. Several efforts have been made to use topology optimization in the compressor mounting bracket using ANSYS [13], leaf spring mounting bracket using FEA [22], and a full car body using FEA [23], engine mounting bracket of a heavy commercial vehicle using FEA [24], commercial vehicles, and powertrain transmission housing [25], aircraft fuselage [26].

II. MATERIALS AND METHODOLOGY

a. Materials

Most of the parts of the hydraulic cylinder are susceptible to corrosion. Therefore, the choice of material is very important. The material should not only be corrosion-resistant but also relatively cheap. Stainless steel, which has strength, ductility, and toughness at high temperatures, is often used in mechanical parts.

The stress-strain curves are perhaps the most indicative of the differences in the mechanical properties of different stainless-steel types. The low yield strength and high ductility of the austenitic types, as well as the high yield and tensile strengths and low ductility of the martensitic types, are clearly apparent. Between these two types lie the ferritic and ferritic-austenitic types. Although ferritic-austenitic types have much higher yield strength than ferritic and austenitic types, ferritic types have slightly higher yield strength than austenitic types. Although ferritic and ferritic-austenitic types are slightly better in this respect, their ductility is of similar magnitude to that of [27]. A variation of Alloy 410, Alloy 420 is a martensitic, hardenable stainless steel. Like 410, Alloy 420 contains at least 12% chromium, which is sufficient only to resist corrosion. Compared to Alloy 410, Alloy 420 contains higher carbon content. Although it can be hardened to a minimum Rockwell hardness of 50 HRC, the maximum hardness of the 12% chromium grades, it also retains a good amount of ductility when annealed. Although it can be welded, alloy 420 is rarely welded due to its hardening properties. Because martensitic stainless steel is made very hard, other properties are sometimes slightly compromised. The usable working temperature range is limited by the loss of ductility at sub-freezing temperatures and the loss of strength from over-tempering at higher temperatures, and the corrosion resistance is lower than that of the common austenitic types. Surface grinding or polishing the metal after it has been hardened will provide the highest corrosion resistance.

Table1: Chemical composition of stainless steel martensitic 420.

Type	C	Mn	Cr	P	S	Cr
420	0.15max	1.25max	1.00max	0.04max	0.03max	max: 14.00-min: 12.00

Source: Authors, (2026).

The mechanical and physical properties of the martensitic 420 SS are also given in Table 2 and 3.

Table 2: Mechanical stainless steel martensitic 420.

Temperature (°C)	Ultimate Tensile Strength (MPa)	Yield Tensile Strength (MPa)	Elongation (% in 50mm)	Brinell Hardness (HB)
Annealed *	655	345	25	max
399°F (204°C)	1600	1360	12	444
600°F (316°C)	1580	1365	14	444
800°F (427°C)	1620	1420	10	461
1000°F (538°C)	1305	1095	15	375
1099°F (593°C)	1035	810	18	302
1202°F (650°C)	895	680	20	262

* Annealed tensile properties are typical for Condition A of ASTM A276; annealed hardness is the specified maximum.

Source: Authors, (2026).

Table 3: Physical properties of stainless steel martensitic 420.

Density	7750 Kg/ m3
Elastic Modulus	200 GPa
Mean Coefficient of Thermal Expansion	µm/m/°C
at 32 to 212°F (0 to 100°C)	10.3
at 32 to 599°F (0 to 315°C)	10.8
at 32 to 1000°F (0 to 538°C)	17.7
Thermal Conductivity	W/m.K
at 212°F (100°C)	24.9
at 932°F (500°C)	—
Specific Heat	J/kg.K
at 32 to 212°F (0 to 100°C)	460
Electrical Resistivity	550(nΩ.m)

Source: Authors, (2026).

b. Methodology

For experimentation, we use one or more analytical, numerical, or experimental methods in every scientific study. However, among the most widely used numerical methods, the Finite Element Method (FEM) is used in this study. The cylinder mounting bracket was created in SolidWorks 2022 based on the geometric requirements provided by Parker Miller Fluid Power, as the study requires three-dimensional modeling of the part for further analysis by [28]. To be compatible with Ansys software, the part is saved in the IGES format. Subsequently, Ansys 2020R1 reads the model in IGES format for static structural analysis. In this case, improper application of loads and boundary conditions can reveal the model's weaknesses. The commercial software PTC Creo Parametric applies topology optimization (TO). The same loads and boundary conditions from Ansys are applied in the TO process. The study's workflow is shown in Figure 1.

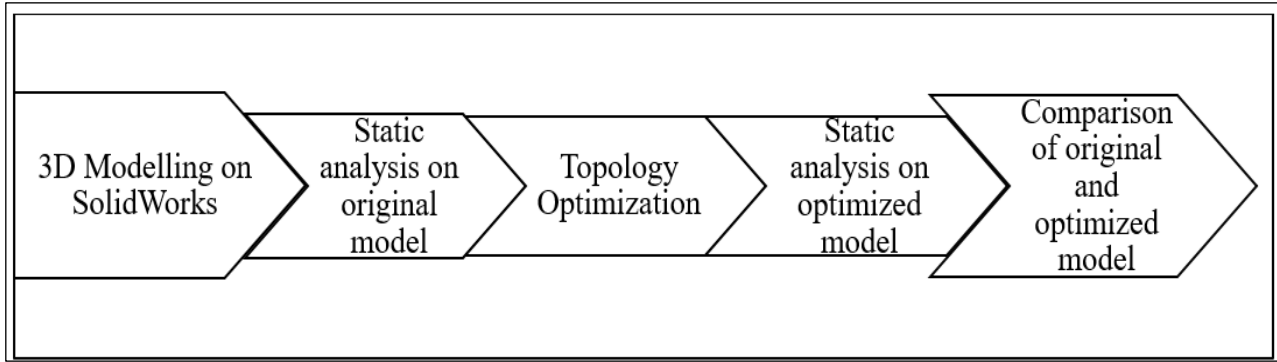


Figure 1: Workflow of the study.
Source: Authors, (2026).

III. STATIC STRUCTURAL ANALYSIS AND TOPOLOGY OPTIMIZATION OF THE COMPONENTS

a. Boundary conditions

A static structural analysis using the finite element method was performed to assess the structural behavior of the cylinder mounting bracket, as shown in Figure 2(a). The actual working conditions for which the mounting bracket is required to perform have been simulated by defining suitable loading and boundary conditions. The inner surface of the cylindrical pinhole on the top surface of the mounting bracket is subjected to a force of 1250.2 N. This force represents the operational load transmitted by the hydraulic cylinder to the construction via the pin connection under working conditions. To accurately simulate the actual conditions for which this type of force transfer is required from the cylinder to the structure via the mounting bracket, the loading direction is defined along the axis of the pin connection. The four bolt holes on the base plate are defined with suitable support boundary conditions to accurately simulate the conditions under which this type of bracket is mounted on the machine structure. The stiff connection between the structure constructed by this type of bracket and the supporting structure using bolted connections is accurately simulated by assuming this type of boundary condition.

a. Meshing of the Model

A three-dimensional model of the cylinder mounting bracket was created in PTC Creo and imported into ANSYS for topology optimization and finite element analysis, as shown in Figure 2 (b). To numerically simulate the problem, it is imperative to divide the geometry into smaller finite elements using a meshing procedure. Due to the complex features of the cylinder mounting bracket, including curved edges, ribs, and numerous holes, an unstructured tetrahedral mesh was used. In some areas, such as the pin connection hole and bolt mounting holes, where high stress concentrations are anticipated, the mesh was refined. The quality of the newly formed elements must be checked to ensure it meets acceptable limits for simulation purposes. The acceptable limits were checked against established criteria for mesh quality, including skewness, aspect ratio, and quality. The load transfer mechanisms between the cylinder pin and the base mounting points were captured using an adequately refined mesh while maintaining an acceptable level of efficiency. The stress, deformation, and structural response of the structure under applied loading conditions can be precisely calculated using the newly formed mesh, which is a dense network of elements covering the entire surface area of the cylinder mounting bracket.

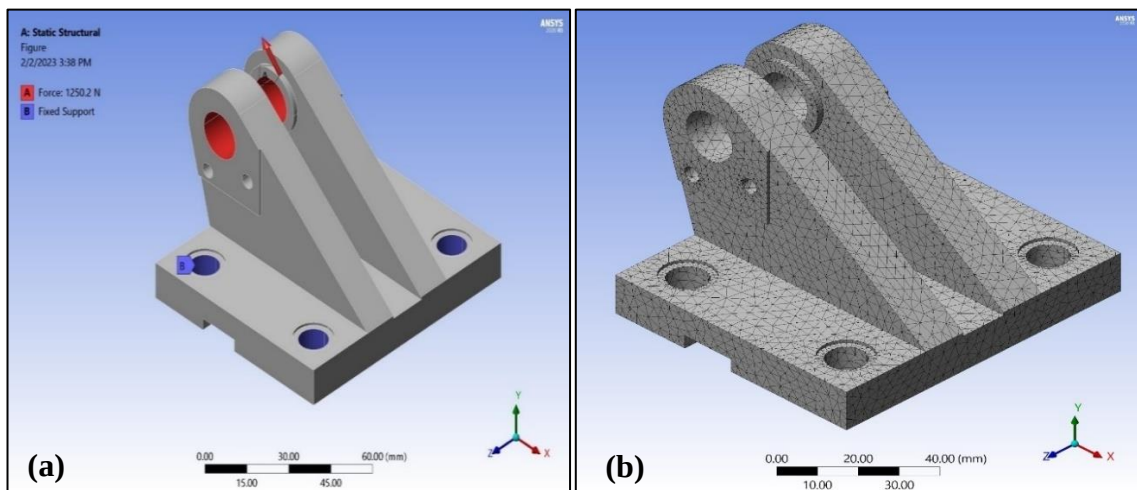


Figure 2: Loading and boundary condition (a) Boundary condition and loading (b) Meshing of the model.
Source: Authors, (2026).

Each finite element (i) created in the meshing phase of a general topology optimization problem statement is assigned a design variable (η_i), which is an internal, pseudo-density of the model. Material to be removed is denoted by $\eta_i = 0$, and material to be kept is denoted by $\eta_i = 1$. The pseudo-density ranges from 0 to 1. The above is another form of stating the topological optimization method:

Minimize/maximize $f(\eta_i)$

Subject to:

$$\begin{aligned} 0 \leq \eta_i \leq 1 & \quad (i = 1, 2, 3, \dots, n) \\ \bar{g}_j \leq g_j \leq \bar{\bar{g}}_j & \quad (j = 1, 2, 3, \dots, m) \end{aligned}$$

Where:

n is number of finite elements

m is number of constraints

$\bar{g}_j$ is computed constraint value

$\bar{g}_j$ and $\bar{\bar{g}}_j$ is lower and upper constraint bounds

PTC and Vanderplaats Research & Development, Inc. (VR&D) jointly developed Creo Topology Optimization. The software combines VR&D's optimization expertise with PTC's strength in bridging the physical and virtual realms. A rebuilt optimized part, either a tessellated (faceted) or B-rep (boundary representation), is the output of the Creo topology optimization procedure [29]. The applications tab, generative design, provides access to the Creo topology optimization interface (Fig. 3). Figure 4 shows the preprocess setup tree. Design spaces, structures, and design criteria are its three main constituents. The region where design and non-design areas are defined is called the design space. Although material, constraints, and objectives can be defined in the design criteria, loading and boundary conditions are defined in the structure.

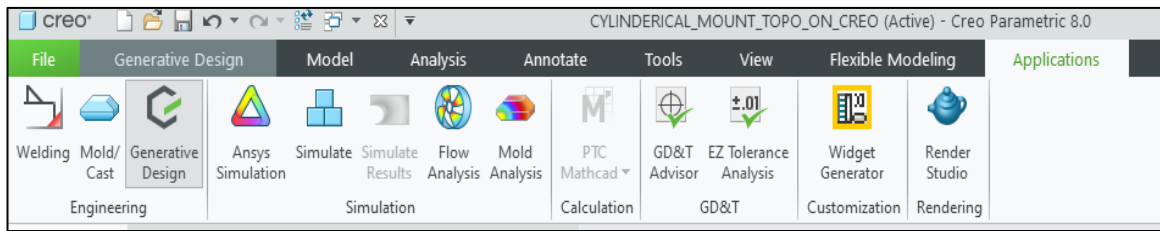


Figure 3: Topology optimization tool in Creo.
Source: Authors, (2026).

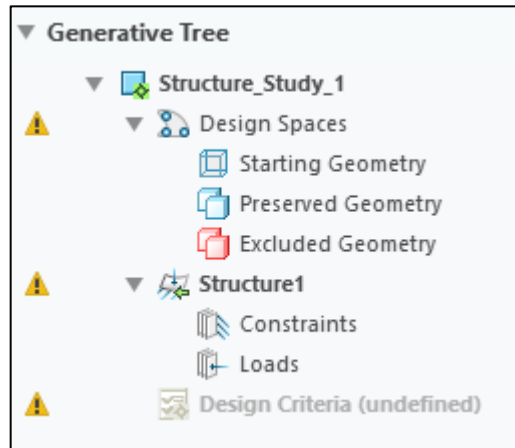


Figure 4: Topology optimization design tree.
Source: Authors, (2026).

IV. RESULTS AND DISCUSSIONS

The results and discussions of the findings of the study are presented in this section of the document. The Creo topology optimization solution for the six design points with weight-reduction values of 70%, 50%, 35%, 25%, 20%, and 17.5% is obtained first. A mesh size of 1.883 mm is used to merge each design-point solution with the static structural analysis solution. The static structural analysis of the original model is then performed in Ansys with the same mesh size as in Creo. Static structural analysis is employed in both Creo and Ansys to obtain the solutions for the original and optimized models at each design point.

a. Topology optimization result

The main aim of the topology optimization process conducted in this study is to minimize the weight of the bracket. The model's mass is gradually reduced during the weight-reduction process. The reason for the percentage difference, as shown in Figures 5-10, is the optimization constraints and the smoothing process. Following a weight reduction of nearly 70%, Figure 5 (a) illustrates the Von Mises stress results obtained from the parametric topology optimization analysis of the cylinder mounting bracket. To minimize the weight of the cylinder mounting bracket while retaining its functional properties, a topology optimization analysis was performed after specifying the material properties, boundary conditions, and mesh. Without affecting the load paths between the pin connection and the base mounting holes, the optimization algorithm removed unnecessary material from low-stress regions.

The solver evaluated the bracket's response to the applied load of 1250.2 N. The regions of maximum stress are near the pin joint area and the base area of the bracket, as indicated by the stress distribution plot after optimization. To maintain the integrity of the structure, these regions are retained in the optimal design. Notwithstanding the significant material reduction, the optimized bracket still maintains an efficient load path, as indicated by the stress contour plot. The results indicate that topology optimization is an effective technique for developing a light structure that maintains mechanical stability and satisfactory stress levels.

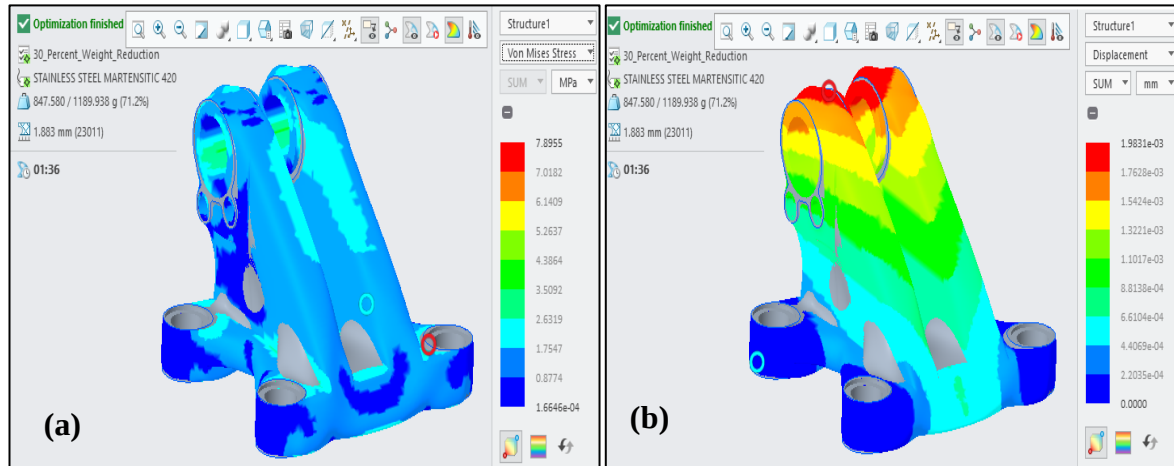


Figure 5: 70% weight reduction (a) Von-Mises's Stress (b) Deformation.
Source: Authors, (2026).

The deformation distribution of the cylinder mounting bracket, derived from a parametric topology-optimization study following a weight reduction of approximately 70%, is shown in Figure 5(b). The object's displacement is depicted in a color contour plot, where red denotes areas of maximum deformation and blue denotes areas of minimum deformation. The investigation shows that the maximum movement of the object is around 1.883 mm. This region experiences the maximum stress due to its direct interaction with the applied loads and constraint points. The base construction is stiff even after significant material reduction, as indicated by the very low deformation of the mounting holes in the lower base. The final mass of the optimized design is about 487.58 grams, which is 71.2% of the initial mass, indicating a significant weight reduction while maintaining performance efficiency in functional use. The enhanced bracket meets acceptable displacement levels in mechanical systems, as indicated by the deformation pattern.

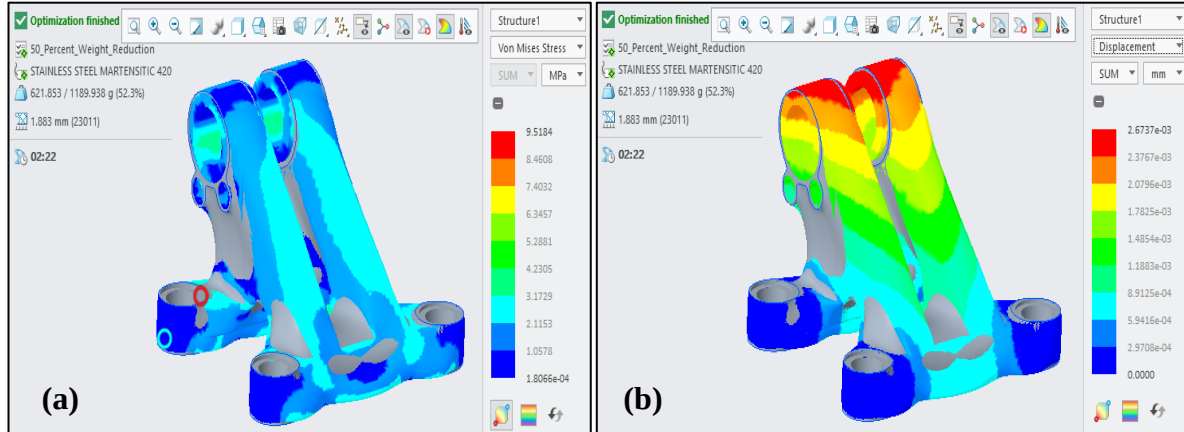


Figure 6: 50% weight reduction (a) Von-Mises's Stress (b) Deformation.
Source: Authors, (2026).

The Von Mises stress distribution of the cylinder mounting bracket after the parametric topology optimization process, with a 50% weight reduction, is shown in Figure 6(a). The Von Mises stresses at various locations of the design are shown in the contour map. The scale of the stresses on the right-hand side of the figure ranges approximately from 1.806×10^{-2} MPa to 9.518 MPa. The location of the maximum Von Mises stresses at the joint of the internal rib and the lower support region, as shown within the circle of red color, has been identified as per the Von Mises stress distribution. The stresses are concentrated at the location where loads are transferred to the upper cylindrical mounting section of the supports at the base. As the load passes through this bracket structure, it is affected by bending and shearing effects in this region. The cyan and green colors indicate low stress in the inclined support arms and the top cylindrical mounting holes, suggesting efficient load distribution along optimal structural paths. The outer edges and base mounting holes of the bracket, shown in dark blue, exhibit very low stress, suggesting that this region remains stable even after significant material is removed. The final mass of 621.853 g is approximately 52.3% of the original mass of 1189.938 g after optimization. The deformation contour map shows the displacement, measured in millimeters, across the entire optimized structure. 2.637×10^{-3} mm, as can be observed in Figure 6 (b), where the blue areas represent areas of low structural movement, and the red areas represent the areas of maximum movement.

It can be observed that the maximum deformation occurs at the upper cylindrical mounting part of the structure, at the pin connection point where the load is applied. The highest level of displacement due to direct load transfer via the bracket arms is represented by the red and orange bands in this section. The transition from the yellow to the green sections demonstrates how the distortion diminishes as the load continues to move downhill via the slanted support ribs. For the initial mass of 1189.938 g, the enhanced bracket has an ultimate mass of approximately 621.853 g, corresponding to 52.3% of the initial mass.

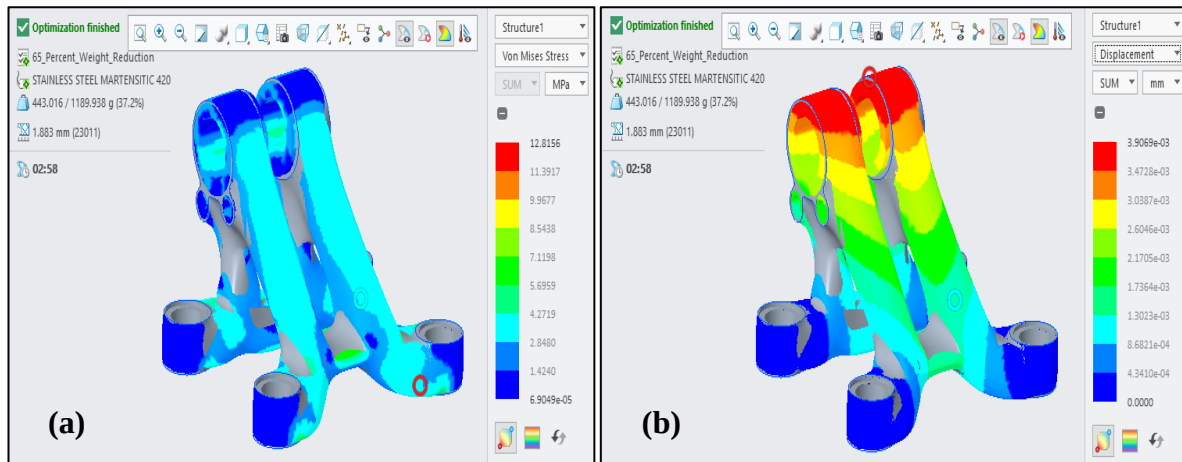


Figure 7: 35% weight reduction (a) Von-Mises's Stress (b) Deformation.
Source: Authors, (2026).

This is shown in the stress contour map in Figure 7 (a), whose color scale ranges from 8.0949×10^{-1} MPa to 12.8156 MPa. In this representation, red spots indicate the highest concentration of tension stress, whereas blue indicates the lowest. From the results, it is clear that the lower base support region, indicated by the red circle and the mounting hole location, has the maximum stress. This is due to the stress resulting from the inclined support arms. The weight distribution from the cylinder connecting point to the base support is heavily dependent on these regions. The optimized design has effectively distributed the weight along the major load paths, as shown by the stress distribution in the structural components. The largest portion of the mounting bracket's outer surface and the base platform, shown in dark blue, has very low stress levels. The final weight achieved through the optimization process, at approximately 443.016 g, is about 37.2% of the original weight of 1189.938 g. This indicates a substantial reduction in material usage while maintaining acceptable stress values within the material's allowed limits. The complete deformation distribution of the optimized cylinder mounting bracket is shown in Figure 7 (b). The magnitude of displacement in the bracket structure, measured in millimeters, is represented in the deformation contour map. The color bar indicates the regions of maximum (red) and minimum (blue) displacement in the structure, ranging from 0 mm (dark blue) to approximately 3.9069×10^{-3} mm (red). This region has the highest displacement as it is directly exposed to the applied load. Effective load transfer in the structure is indicated by the gradual reduction in distortion in the inclined supporting arms, as shown by the transition from the red and orange regions to the yellow and green regions. While allowing controlled deformation in the load-bearing top part, the improved design successfully maintains structural stiffness in the base region. While allowing controlled deformation in the load-bearing top part, the improved design successfully maintains structural stiffness in the base region. Compared to the initial mass of 1189.938 g, the topology optimization resulted in a final mass of approximately 443.016 g, which is 37.2% of the initial mass. Based on the deformation analysis, the parametric topology optimization approach successfully reduces the structure's mass while maintaining structural stability.

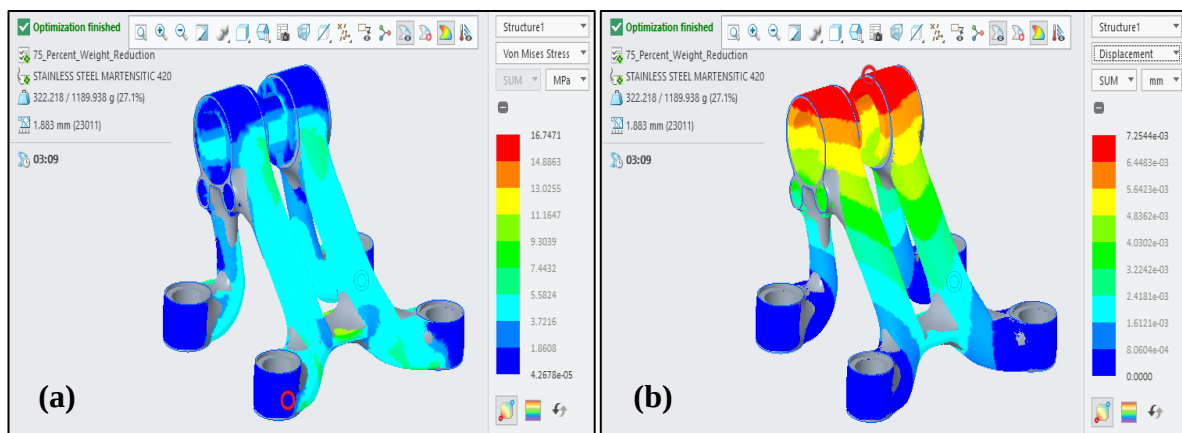


Figure 8: 25% weight reduction (a) Von-Mises's Stress (b) Deformation.
Source: Authors, (2026).

The Von Mises stress distribution of the cylinder mounting bracket, after parametric topology optimization with 25% weight reduction, is shown in Figure 8 (a). From the results, it can be observed that the lower base of the mounting bracket, especially the vicinity of the support hole, has the highest stress levels.

The stress contour map also shows the Von Mises stress levels at different points on the optimized shape. The dark blue areas represent low stress levels, and the red areas indicate high stress levels, indicating maximum stress concentration. The color scale ranges from 4.2678×10^{-1} MPa to 16.7471 MPa. Localized stress accumulation results from the interaction of shear and bending stresses in this region. It can be seen that the applied force is effectively transmitted from the interface of the cylinder mounting to the base mounting points via these structural components. Dark-blue color contours indicate that the majority of the base platform and the structural surfaces remain in low-stress regions. This means that despite the reduced amount of material, the bracket still has adequate structural stability. It can also be shown that the optimization process reduced the mass from 1189.938 g to approximately 322.218 g, a 27.1% reduction. The displacement across the optimum bracket, expressed in millimeters, is shown in the deformation contour map. According to the model, Figure 8(b) indicates that the area of maximum deformation is located in the top cylindrical mounting region, particularly at the pin connection. The displacement across the optimum bracket,

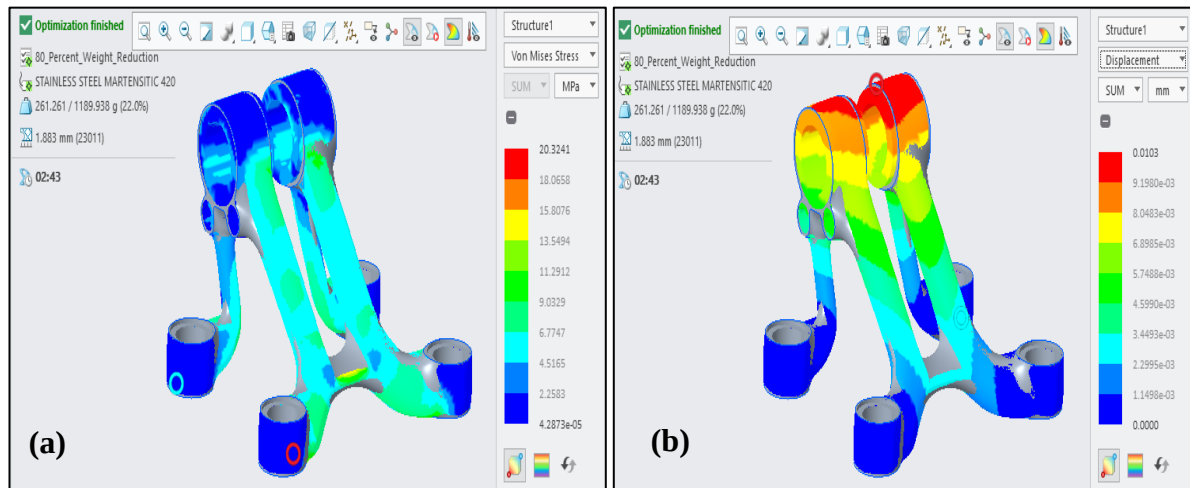


Figure 9: 20% weight reduction (a) Von-Mises's Stress (b) Deformation.
Source: Authors, (2026).

Expressed in millimeters, is shown in the deformation contour map. According to the color legend, the area of maximum displacement (red) and minimum deformation (blue) ranges from 0 mm (dark blue) to approximately 7.2544×10^{-3} mm (red). This means that constant support and adherence to the surrounding structure are guaranteed through the structural rigidity and good constraint of the base structure. Compared to the initial mass of 1189.938 g, the final mass obtained through optimization is about 322.218 g, or 27.1% of the initial mass. The viability of the lightweight structure for practical application is ensured by the optimization process, which guarantees the transmission of load from the upper supporting structure to the base supports. The Von Mises stress distribution obtained from the parametric topology optimization results, with a 20% weight reduction, is shown in Figure 9(a). From the figure, the stress contour map, with a color code ranging from approximately 4.2873×10^{-1} MPa to 20.3241 MPa, shows the Von Mises stress in the optimized bracket shape. From the results, it can be concluded that the lowest base mounting area, i.e., the region around the support hole as shown in the figure, corresponds to the highest stress. This is due to the transfer of load from the upper cylindrical mounting section to the base mounting points, where the structure is restricted, through the slanted support arms. The increase in stress in this region is due to a combination of bending, shear, and load-transfer effects. The applied force is effectively distributed from the cylinder mounting interface to the base supports through these areas, which are the major load-carrying parts. Compared with the initial mass of 1189.938 g, the optimization process yields a final mass of about 261.261 g, corresponding to 22.0% of the initial mass. Figure 9(b) shows the overall deformation pattern of the cylinder mounting bracket. The initial mass of the structure was 1189.938 g, and the final mass after the optimization process was about 261.261 g, corresponding to only 22.0% of the initial mass. The maximum displacement occurs at the top cylindrical mounting area, particularly near the pin connection point where the external load is applied, as indicated by the deformation values. As shown, due to direct load transmission through the cylinder connection to the bracket structure, the area marked in red and orange has the greatest deformation. Compared with the initial mass of 1189.938 g, the final mass, achieved through topology optimization, is approximately 261.261 g, or 22.0% of the initial weight.

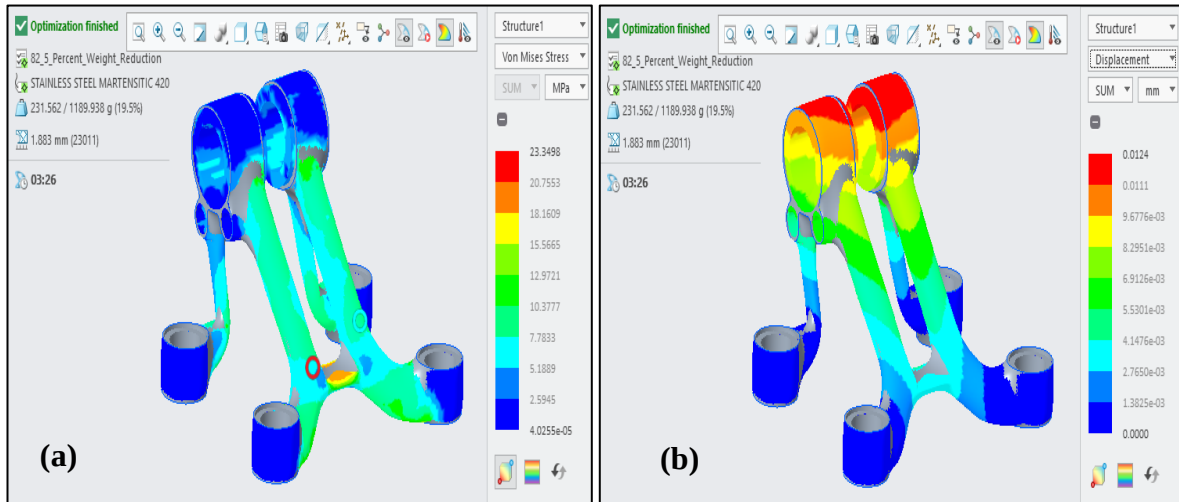


Figure 10: 17.5% weight reduction (a) Von-Mises's Stress (b) Deformation.

Source: Authors, (2026).

As shown in Figure 10(a), the von Mises stress contour map shows equal von Mises stress values across the bracket shape. The color scale ranges from about 4.0255×10^{-1} MPa to 23.3498 MPa. The results show that the maximum stress is near the junction point at the center base, where the structural intersection of the lower base platform and the inclined support arms occurs, as shown in the figure. The combined effects of bending, shear, and load transfer from the upper cylindrical mounting part to the base supports cause stress in this area. The initial mass was 1189.938 g, while the final mass, after optimization, was about 231.562 g, or 19.5 percent of the initial mass. The deformation distribution of the cylinder mounting bracket resulting from the parametric topology optimization process, with a 17.5% weight reduction, is shown in Figure 10(b). The red color shades in the color legend indicate displacement levels ranging from 0 mm to the maximum deformation in the mounting area at the top. The structural displacement of the bracket as the applied load is transmitted through it is shown through these colors. The maximum displacement occurs at the area around the top cylindrical mounting section of the bracket, especially near the pin connection where the applied load is transmitted, as shown by the deformation. As this area is directly exposed to the applied load and serves as the primary load-transfer interface between the cylinder and the bracket, it undergoes the greatest deformation. The change in color from red and orange to yellow, green, and cyan indicates the gradual reduction in deformation as the weight moves downhill along the slanted support arms. For the initial mass of 1189.938 g, the final mass calculated by the topology optimization technique is approximately 231.562 g, or 19.5% of the initial mass. Although the weight decreased compared to Von-Mises stress and the weight reduced compared to deformation are illustrated in "In addition to the above stated result evaluation criteria, strength per unit mass is also one of the evaluation mechanisms," the conclusion of the stress and deformation results is presented in "Table 4" and shown in Figure 11.

Table 4: Design points result summary.

No	Weight reduced (%)	Von-Mises's stress (MPa)	Deformation (mm)
1	70%	7.895	0.001983
2	50%	9.518	0.002673
3	35%	12.815	0.003906
4	25%	16.747	0.007254
5	20%	20.324	0.0103
6	17.5%	23.349	0.0124

Source: Authors, (2026).

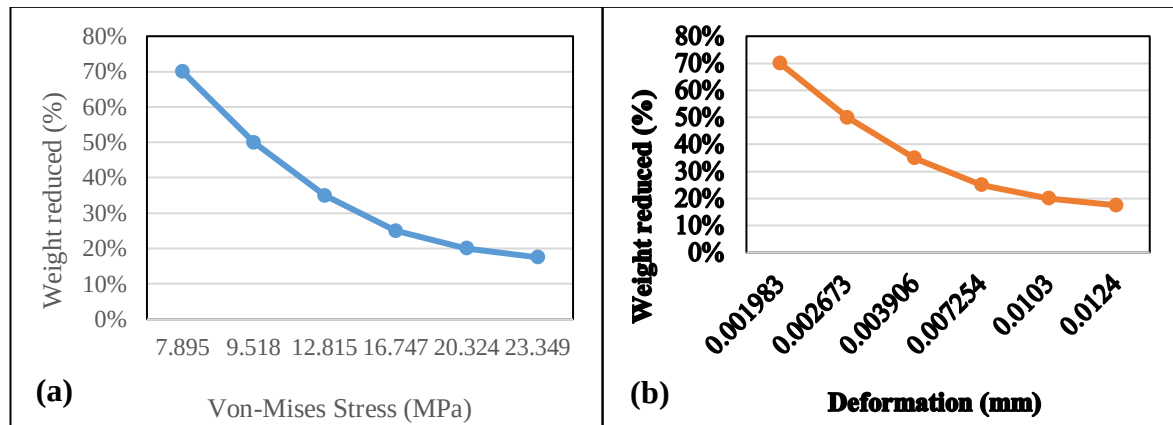


Figure 11: Graph of each design point with the respective result (a) Weight reduced vs Von-Mises's Stress (b) Weight reduced Vs Deformation.

Source: Authors, (2026).

In addition to the above-stated result evaluation criteria, the strength per unit mass is also one of the evaluation mechanisms given in Table 5.

Table 5: Strength per unit mass of the design points.

No	Weight reduction	Von-Mises stress (MPa)	Weight (Kg)	Strength per unit mass (MPa/Kg)
1	70%	7.895	0.848	9.315
2	50%	9.518	0.622	15.306
3	35%	12.815	0.443	28.927
4	25%	16.747	0.322	51.974
5	20%	20.324	0.261	77.792
6	17.5%	23.349	0.232	100.833

Source: Authors, (2026).

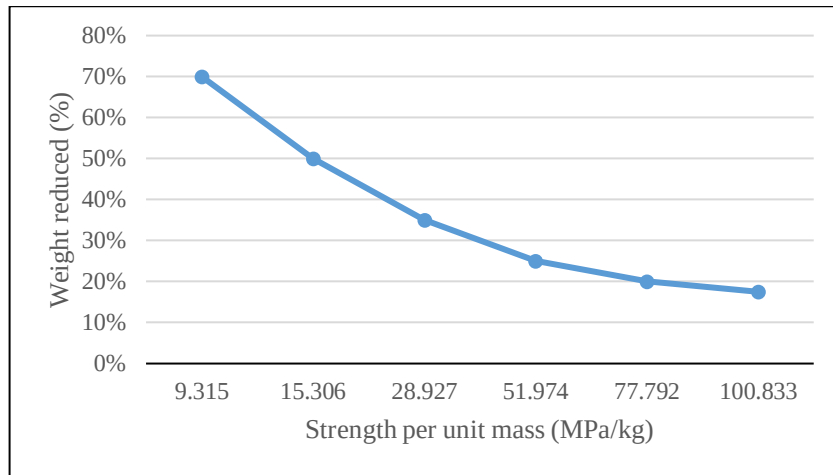


Figure 12: Design point vs strength per unit mass.

Source: Authors, (2026).

How6RTYever, the model was not selected as the best because the study was limited to static analysis and other loading cases. Moreover, the front part of the model's optimization results is very thin, and there are concerns about the model's behavior under bending stress. To explain the stress and deformation in "Figure 10," the fifth design point (20% weight-reduction) is selected as the final model. "Figure 13" shows the topologically optimized and original models. The final product weighs 0.261 kg, 22% lighter than the original model at 1.189 kg.

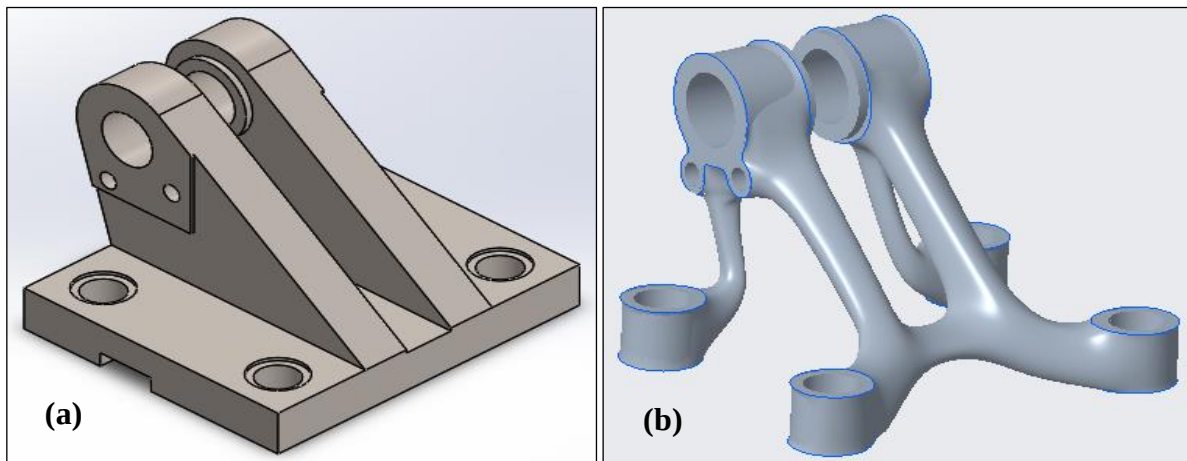


Figure 13: Original and topology optimized model (a) Original model (b) Optimized model.

Source: Authors, (2026).

b. Comparative study of the two models

Figures 14 and 15 show that the Von-Mises stress of the optimized model is 20.32 MPa, whereas it was 10.33 MPa for the original model. Also, the distortion value for the optimized model is 0.0103mm, whereas it was 0.00184mm for the original model. The resulting stress value is well below the material's tensile yield strength of 1360 MPa. This confirms that even if the optimized model has higher stress values, its factor of safety (FOS) will be greater.

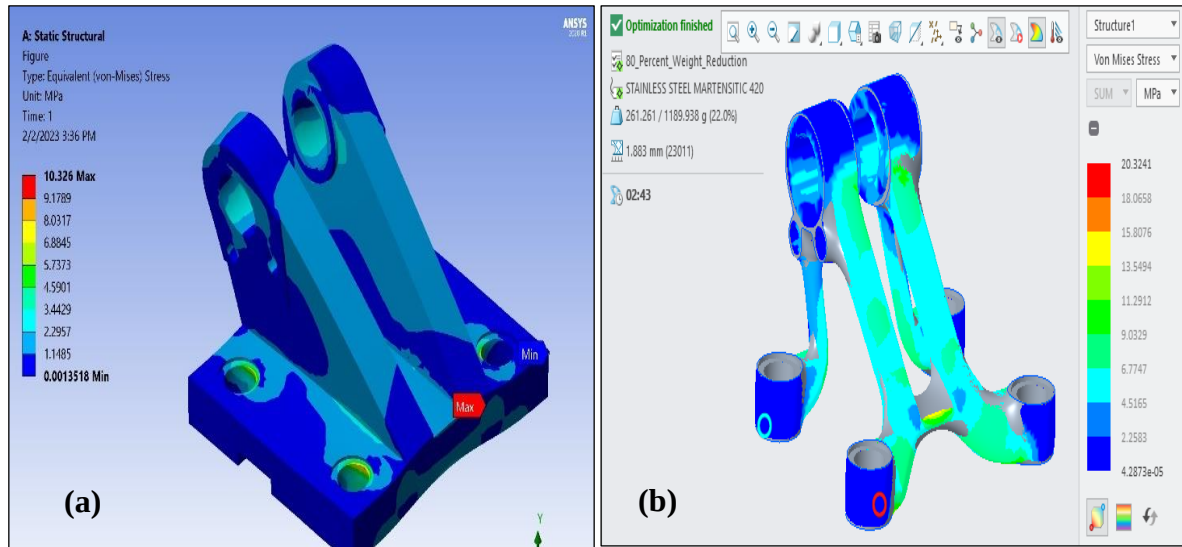


Figure 14: Von-Mises stress Result (a) Original model (b) Optimized model.
Source: Authors, (2026).

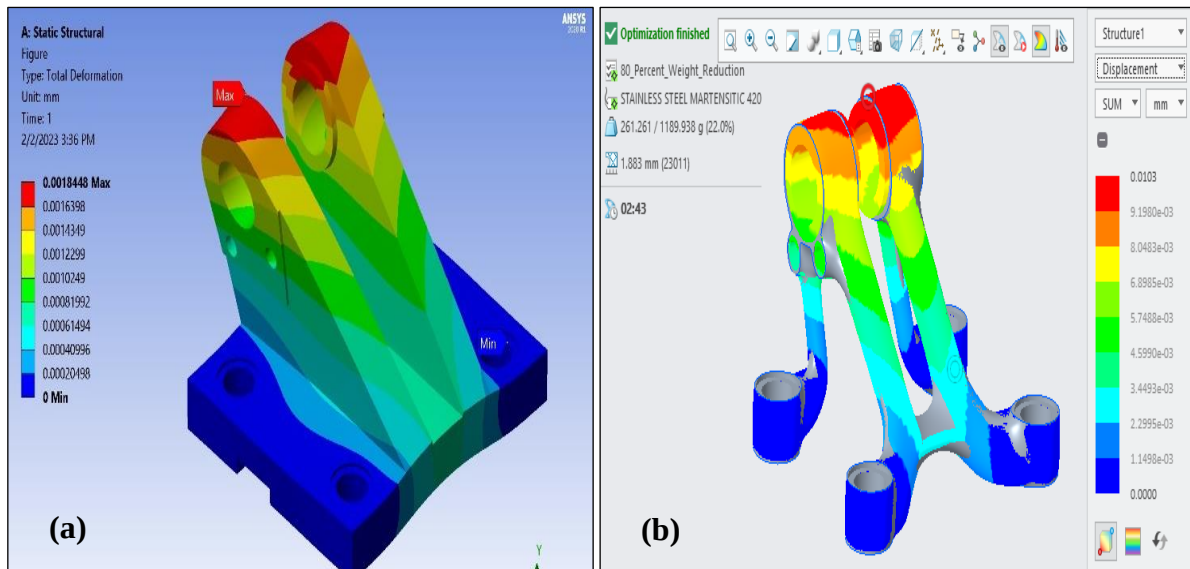


Figure 15: Deformation result (a) Original model (b) Optimized model.
Source: Authors, (2026).

V. CONCLUSION

This paper aims to optimize the topology of mounting brackets for cylinders used in fluid power systems. The aim is to check the original model at six design points: 30%, 50%, 65%, 80%, and 87.5% weight reduction, and 70%, 50%, 35%, 25%, 20%, and 17.5% weight retention. The original model weighed 1.189 kg and was created in the commercial SolidWorks software using martensitic stainless steel 420. The Parker Miller Fluid Power specification serves as the source for all geometric parameters, loading conditions, and constraints. The generative design module in PTC Creo Parametric software was used for topology optimization. The original model and the optimized model with an element mesh size of 1.883 mm were analyzed using static structural analysis. The original weight of the mounting bracket design was 1.189 kg. In the optimization process, six design points with weight retention of 70%, 50%, 35%, 25%, 20%, and 17.5% were chosen for evaluation. Even with the constraints, the optimization technique can still minimize the weight if required. Therefore, for 70%, 50%, 35%, 25%, and 17.5% design points, respectively, the actual weight retention values are 0.847 kg (71.2%), 0.622 kg (52.3%), 0.443 kg (37.2%), 0.322 kg (27.1%), 0.261 kg (22%), and 0.231 kg (19.5%). As the percentage of weight reduction increases, the weight per unit mass also increases. 9.315 MPa/kg, 15.306 MPa/kg, 28.927 MPa/kg, 51.974 MPa/kg, 77.792 MPa/kg, and 100.833 MPa/kg are the weights per unit mass. The choice of the model is not based on being the best, as it is constrained by the study's limitations related to static analysis and other loading conditions. Additionally, in the case of bending strain, the model appears suspiciously thin at the front. Therefore, the final model chosen to show stress and deformation is the fifth design point (20% weight-reduction design point). The completed model weighs 0.261 kg, which is 22% of its original weight of 1.189 kg. ANSYS performs a static analysis with the same loading and analysis conditions. The Von Mises stress of the improved model is 20.32 MPa, compared to 10.33 MPa of the previous model. Additionally, the distortion of the optimized model is 0.0103mm, compared to 0.00184mm of the original model. The generated stress is much lower than the material's tensile yield strength of 1360 MPa.

VI. AUTHOR'S CONTRIBUTION

Conceptualization: Edosa Kefyalew, Adugna Fikadu, Mahesh Gopal.

Methodology: Edosa Kefyalew, Adugna Fikadu.

Investigation: Edosa Kefyalew, Adugna Fikadu, Mahesh Gopal

Discussion of results: Edosa Kefyalew, Adugna Fikadu.

Writing – Original Draft: Edosa Kefyalew, Adugna Fikadu.

Writing – Review and Editing: Edosa Kefyalew, Adugna Fikadu.

Resources – Adugna Fikadu, Mahesh Gopal.

Supervision: Edosa Kefyalew, Mahesh Gopal.

Approval of the final text: Adugna Fikadu. Mahesh Gopal.

VII. ACKNOWLEDGMENTS

We are also grateful to the anonymous reviewers for their constructive comments, which helped us improve the quality of this manuscript.

VIII. REFERENCE

- [1] J. P. Wu, J., Sigmund, O., & Groen, "Topology optimization of multi-scale structures: a review," *Struct. Multidiscip. Optim.*, vol. 63, no. 3, pp. 1455-1480., 2021.
- [2] J. H. Zhu, W. H. Zhang, and L. Xia, "Topology Optimization in Aircraft and Aerospace Structures Design," *Arch. Comput. Methods Eng.*, vol. 23, no. 4, pp. 595–622, 2016, doi: 10.1007/s11831-015-9151-2.
- [3] A. Dubrovskaya, K. Dongauzer, and R. Faskhutdinov, "The design of lightweight gas turbine engine parts using topology optimization," *MATEC Web Conf.*, vol. 129, 2017, doi: 10.1051/mateconf/201712901067.
- [4] L. Krog, A. Tucker, and G. Rollema, "Application of topology, sizing and shape optimization methods to optimal design of aircraft components," 3rd Altair UK HyperWorks Users Conf., pp. 1–12, 2002, [Online]. Available: [http://www.soton.ac.uk/~jps7/Aircraft Design Resources/manufacturing/airbus wing rib design.pdf](http://www.soton.ac.uk/~jps7/Aircraft%20Design%20Resources/manufacturing/airbus%20wing%20rib%20design.pdf)5Cnftp://emci.udg.edu/mme/ODM/treballs/shape_optimization/airbus wing rib design.pdf
- [5] P. S. Tang and K. H. Chang, "Integration of topology and shape optimization for design of structural components," *Struct. Multidiscip. Optim.*, vol. 22, no. 1, pp. 65–82, 2001, doi: 10.1007/PL00013282.
- [6] J. Liu et al., "Current and future trends in topology optimization for additive manufacturing," *Struct. Multidiscip. Optim.*, vol. 57, no. 6, pp. 2457–2483, 2018, doi: 10.1007/s00158-018-1994-3.
- [7] S. Ramesh et al., "Topology optimization and finite element analysis of a jet dragster engine mount Topology optimization and finite element analysis of a jet dragster engine mount," *Cogent Eng.*, vol. 7, no. 1, 2020, doi: 10.1080/23311916.2020.1723821.
- [8] N. A. Faskhutdinov, R. N., Dubrovskaya, A. S., Dongauzer, K. A., Maksimov, P. V., & Trufanov, "Topology optimization of a gas-turbine engine part," in *In IOP conference series: materials science and engineering*, 2017, pp. 177, (1), 012077.
- [9] E. Armentani, V. Giannella, A. Parente, and M. Pirelli, "Design for NVH: Topology optimization of an engine bracket support," *Procedia Struct. Integr.*, vol. 26, no. 2019, pp. 211–218, 2020, doi: 10.1016/j.prostr.2020.06.024.
- [10] A. S. Shanmugasundar, G., Dharanidharan, M., Vishwa, D., & Kumar, "Design, analysis and topology optimization of connecting rod," *Mater. Today Proceedings*, vol. 46, pp. 3430-3438., 2021.
- [11] D. J. Munk, D. J. Auld, G. P. Steven, and G. A. Vio, "On the benefits of applying topology optimization to structural design of aircraft components," *Struct. Multidiscip. Optim.*, vol. 60, no. 3, pp. 1245–1266, 2019, doi: 10.1007/s00158-019-02250-6.
- [12] E. Talay, C. Özkan, and E. Gürtaş, "Designing lightweight diesel engine alternator support bracket with topology optimization methodology," *Struct. Multidiscip. Optim.*, vol. 63, no. 5, pp. 2509–2529, 2021, doi: 10.1007/s00158-020-02812-z.
- [13] S. Kalsi, D. Singh, and J. S. Saini, "Optimization of Compressor Mounting Bracket of a Passenger Car," *J. Inst. Eng. Ser. C*, vol. 100, no. 4, pp. 675–691, 2019, doi: 10.1007/s40032-018-0453-7.
- [14] S. Kirthana and M. Khaja Nizamuddin, "Finite element analysis and topology optimization of engine mounting bracket," *Mater. Today Proc.*, vol. 5, no. 9, pp. 19277–19283, 2018, doi: 10.1016/j.matpr.2018.06.286.
- [15] J. Fang, "Motion Simulation and Topology Optimization Design of a Mechanical Gripper Based on Creo Simulate 4.0," *Adv. Eng. Technol. Res.*, vol. 15, no. 1, p. 759, 2025, doi: 10.56028/aetr.15.1.759.2025.
- [16] A. Mortazavi and V. Toğan, "Simultaneous size, shape, and topology optimization of truss structures using integrated particle swarm optimizer," *Struct. Multidiscip. Optim.*, vol. 54, no. 4, pp. 715–736, 2016, doi: 10.1007/s00158-016-1449-7.
- [17] D. Yago, J. Cante, O. Lloberas-Valls, and J. Oliver, *Topology Optimization Methods for 3D Structural Problems: A Comparative Study*, vol. 29, no. 3. Springer Netherlands, 2022. doi: 10.1007/s11831-021-09626-2.
- [18] A. Kumar and D. Chhabra, "Parametric topology optimization approach for sustainable development of customized orthotic appliances using additive manufacturing," *Mech. Adv. Mater. Struct.*, vol. 31, no. 21, pp. 5276–5289, 2024, doi: 10.1080/15376494.2023.2214908.
- [19] G. Selvaraj et al., "Investigation on topology-optimized compressor piston by metal additive manufacturing technique: Analytical and numeric computational modeling using finite element analysis in ANSYS," *Open Phys.*, vol. 21, no. 1, 2023, doi: 10.1515/phys-2022-0259.
- [20] R. Ahmed, "Topology Optimization and Additive Manufacturing of Lightweight Cantilever Beams : Design , Fabrication , and Cost Analysis," *Res. Sq.*, 2025.

- [21] D. Jankovics, H. Gohari, M. Tayefeh, and A. Barari, “Developing Topology Optimization with Additive Manufacturing Constraints in ANSYS®,” IFAC-PapersOnLine, vol. 51, no. 11, pp. 1359–1364, 2018, doi: 10.1016/j.ifacol.2018.08.340.
- [22] A. Kalantre, V., Munde, K. H., & Pawar, “Topology Optimization of Front Leaf Spring Mounting Bracket,” Int. J. Sci. Dev. Res., vol. 3, no. 7, pp. 12–19, 2018.
- [23] B. Liu, J. Yang, X. Zhang, and X. Li, “Topology Optimization and Lightweight Platform Development of Pure Electric Vehicle Frame-Type Aluminum Body Considering Crash Performance,” J. Mater. Eng. Perform., vol. 34, no. 3, pp. 2424–2434, 2025, doi: 10.1007/s11665-024-09239-3.
- [24] M. N. Khan, A. K. A., Kadir, A. R. A., Nazri, Z., Razak, T. A. A., & Ahmad, “Topology optimization of an engine mounting bracket using finite elements,” 2022.
- [25] S. K. Kandreegula, N. Sukumar, S. Endugu, and U. Gupta, “Structural Non-linear Topology Optimization of Transmission Housing and Its Experimental Verification,” 2015. doi: 10.4271/2015-01-0098.
- [26] M. Kalanchiam and B. Mannai, “Topology Optimization of Aircraft Fuselage Structure,” World Acad. Sci. Eng. Technol., vol. 7, no. 5, pp. 133–136, 2013.
- [27] Outokumpu, Handbook of Stainless Steel. 2013. [Online]. Available: <http://www.outokumpu.com/sitecollectiondocuments/outokumpu-stainless-steel-handbook.pdf>
- [28] Parker, “Cylinder Mounting Accessories,” 2012.
- [29] Stefano Odorizzi, “Enginsoft Projects ’ insights,” 2022.