SafePro ceiling mounting mass reduction with ensured safety using ANSYS topology optimization

Abstract

This study explores the use of ANSYS Topology Optimization to improve the efficiency and safety of Droplet Ceiling Mounting. The focus is on minimizing mass while maintaining or enhancing the safety factor. By employing advanced computational techniques, the topology optimization process identifies underutilized material areas within the Mounting structure, allowing for substantial weight reduction without compromising structural integrity. The original Droplet Ceiling Mounting was 33.32g. Used as a baseline for comparison, five models were optimized and reanalyzed in ANSYS Topology Optimization, achieving weight reductions from 33.32g to 30.11g. The results demonstrate a significant decrease in mass alongside an optimized distribution of material, ensuring the Ceiling Mounting meets safety standards.

Key Words: Topology Optimization, Ceiling Mounting, Mass Reduction, Safety Factor, Finite Element Analysis, Ansys.

1. **Introduction**

IOT manufacturers are actively pursuing innovations to create lighter and more efficient devices, striving to enhance performance while reducing cost. To meet these objectives, they are employing a range of advanced techniques aimed at achieving optimal lightweight design. Among these methods, topology optimization has emerged as a popular approach among design engineers. This technique is integrated into the early stages of manufactured parts development to strategically minimize material usage and maintain structural integrity, contributing to the overall efficiency and sustainability of modern mounting.

1.1. Topology Optimization

Topology optimization is a mathematical approach used to optimize material distribution within a specified design space, considering loads, boundary conditions, and constraints to maximize system performance. Unlike shape and sizing optimization, it allows the design to take any form within the space, rather than sticking to predefined shapes.

This method is widely applicable in fields like aerospace, mechanical, biochemical, and civil engineering. Engineers typically use topology optimization at the conceptual stage of design. However, the resulting forms can be challenging to manufacture, which often necessitates further refinement for production feasibility. Research is ongoing to add constraints to enhance manufacturability. In some scenarios, direct manufacturing using additive techniques is possible, making topology optimization a crucial component in designing for additive manufacturing.

1.2. Ceiling Mounting Innovation in Design

The primary function of Mounting is to attach a plastic unit to the ceiling, which helps to hang any devices for a longer time with easy maintenance options to maintain. Although sheet metal is an ideal material to carry out a lightweight design, its design has seen little change, resulting in unnecessary material use and increased costs. This study focuses on using topology optimization for Ceiling Mounting, achieving a lightweight and aesthetically pleasing design.

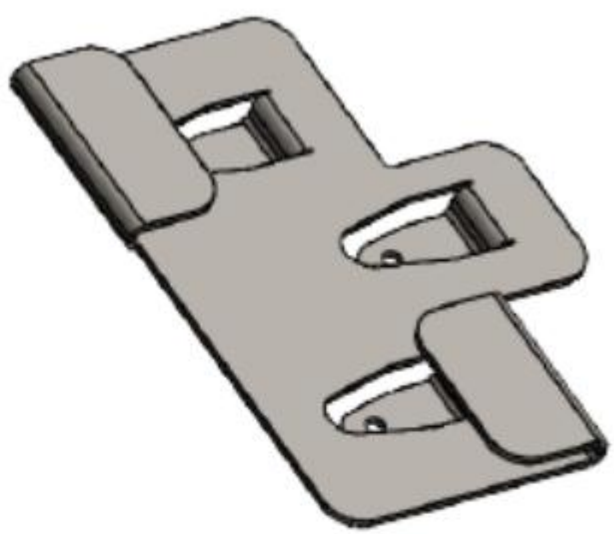

Fig. 1 .1 Ceiling Mounting

2. Objectives and Problem Definition

2.1. Objectives

The main objective of this study is to explore weight reduction opportunities for a Ceiling Mounting using topology optimization.

- This study addresses two aspects: static load stress analysis of the Ceiling Mounting and weight optimization.

- Finite element analysis of the Ceiling Mounting serves as a case study. The structural behavior of the Ceiling Mounting is analyzed using finite element techniques, and weight optimization is achieved through topology optimization tools.

2.2. Problem Definition

Ceiling Mounting presents an opportunity for weight optimization, as they are often designed with an unnecessarily high factor of safety, leading to excess material use and increased manufacturing costs. Topology optimization is an effective method to eliminate this surplus material, thereby lowering production costs and enhancing performance.

3. Methodology

The methodology that the study uses for the optimization of the Mounting, using the topology optimization approach, intends to improve the overall efficiency of the system and to reduce the related mass at the same time.

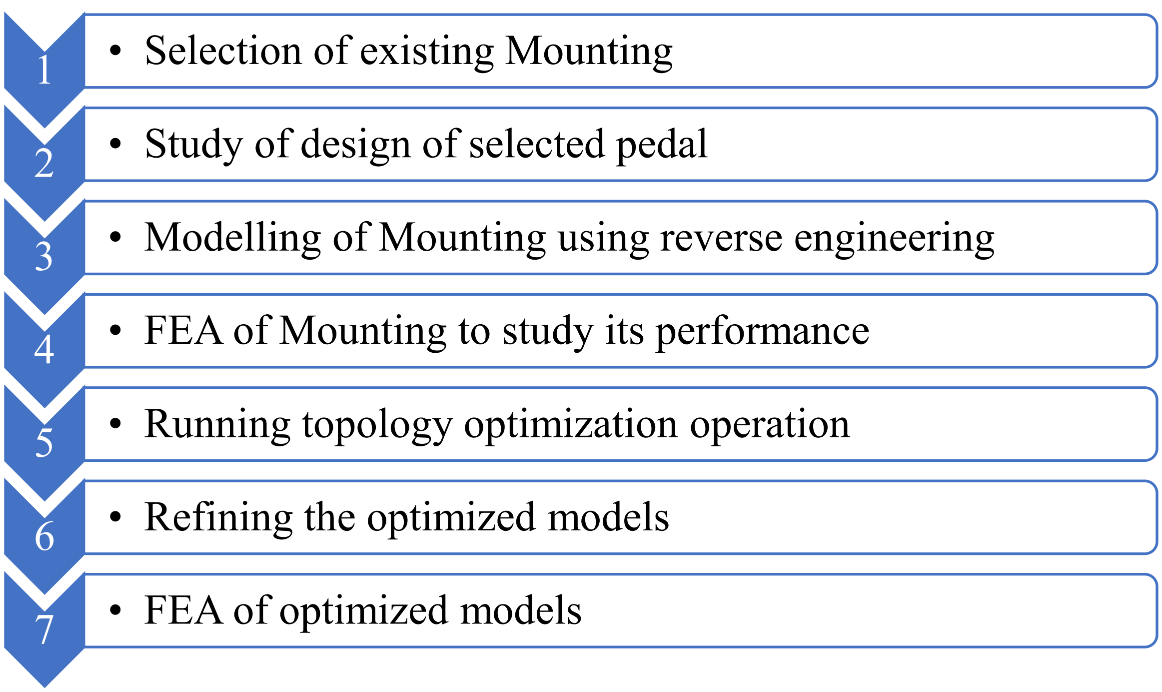

Fig. 3.1 Methodology to optimize Mounting

The data collected was used for 3D modeling and analysis in ANSYS Workbench 2024 R2 to assess stress and deformation through static structural analysis. ANSYS Topology Optimization was then applied to refine the design. The optimized design improves efficiency but requires adjustments for manufacturability.

4. Reverse Engineering and 3D Modelling using Ansys Space Claim

4.1. Reverse Engineering

Reverse engineering, also known as back engineering, is the process of uncovering the design or knowledge embedded in an existing product. This involves carefully disassembling the product to study its components and functionality in detail. The insights gained allow for replication or redesign based on the extracted information, enabling improvements or adaptations of the original design.

4.2. 3D Modelling using Ansys Space Claim

A 3D CAD model of the Mounting is created using ANSYS SpaceClaim. A detailed sketch of the component is developed in the CAD software. As the existing model was unavailable, dimensions were collected from existing research papers. The detailed sketch and 3D model of the pedal are shown in the figure.

5. Static Analysis of Pedal on ANSYS Workbench

5.1. Geometry Import into Ansys

Since geometry has already been created in ANSYS Space Claim, there's no need to import it. ANSYS will automatically read it in the model section. Open the geometry in the New Design Modeler to apply material.

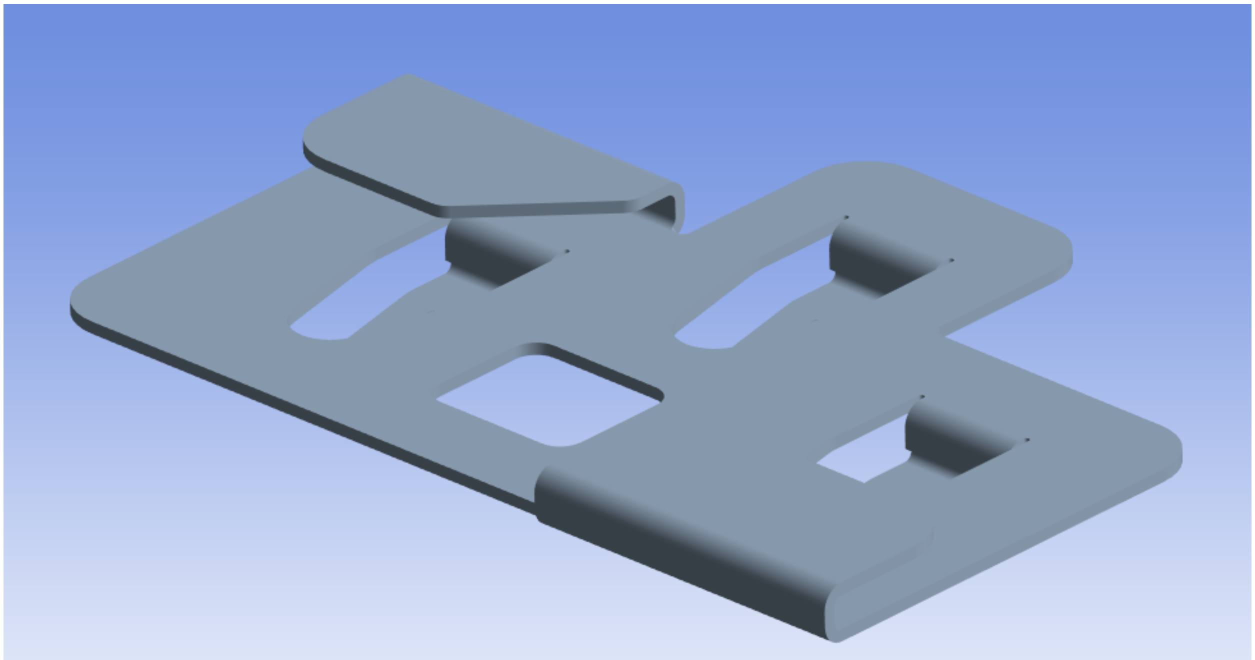

Fig. 5 .1 *Geometry opened in Ansys Design Modeler*

5.2. Material Selection

In this study, the FE analysis method begins with defining material properties, crucial for static structural analysis. Engineering data, including elasticity coefficients and Poisson's ratio, form the scientific basis, with A36 structural steel ensuring precise visualization.

Table-1. Material properties

Property	**Value**
Density	7.85e-06 kg/mm³
Young's Modulus	2e+05 MPa
Poisson's Ratio	0.3
Yield Strength	250 MPa
Tensile Strength	460 MPa

5.3. Meshing

ANSYS provides flexibility in meshing components, allowing users to either automatically or manually define the mesh. This flexibility includes choosing the shape and size of each mesh element, which is crucial for accurately analyzing complex geometries. In this specific case, the

Mounting was meshed using a combination of triangular (TRI) and hexahedral (HEX) elements. This hybrid approach optimizes the stiffness matrix, particularly in areas with pivots and curves, which are critical for maintaining structural integrity under stress. The mesh consisted of elements sized at 1.5 mm, resulting in a detailed model with 53,030 elements and 93,453 nodes. This high-resolution meshing ensures precise simulation results and enhances the overall accuracy of the finite element analysis.

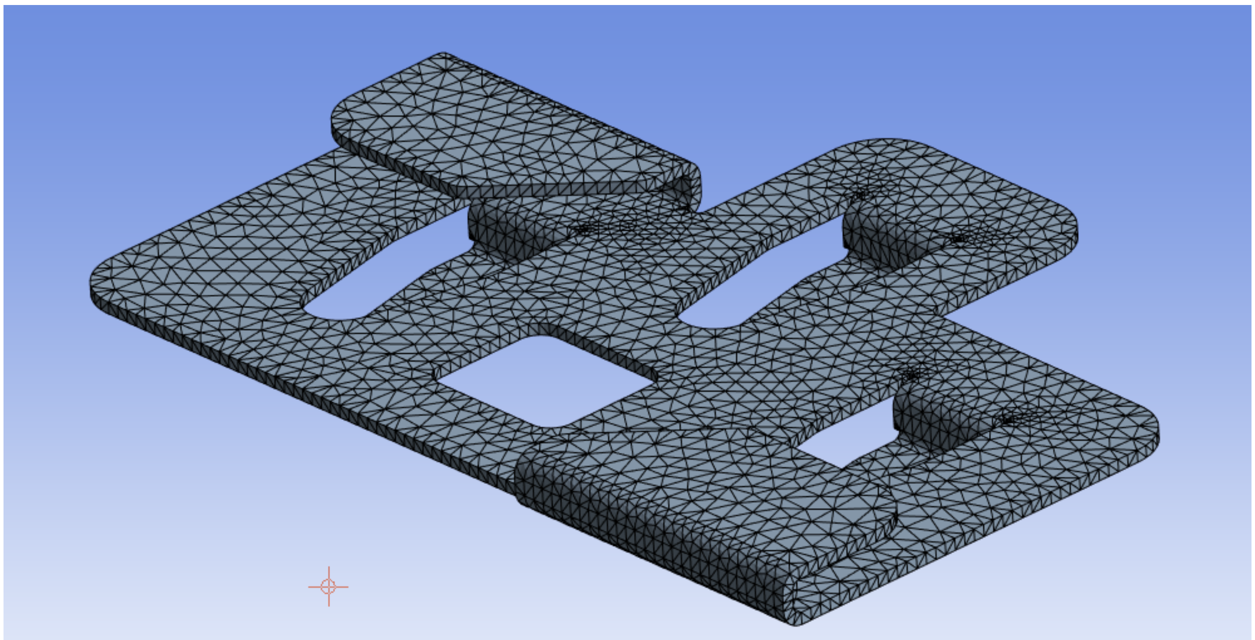

Fig. 5 .2 *Mounting model meshed*

5.4. Boundary Conditions for Analysis

In literature, the maximum force a person can apply to a Mounting is 40 kg. Thus, the total force on the Mounting is calculated as F = 40 * 9.81 = 392.4 N, approximately 400 N. The total pressure on the Mounting is determined by P = Force/surface area, resulting in P = 400 / 3249.1 = 0.12309 MPa. The pedal is pivoted at one end, with the master cylinder push rod attached at the center. The reactive forces from the push rod are neglected, as they are minimal in the pedal's design consideration. This setup ensures the stability of the braking system and enhances the overall performance.

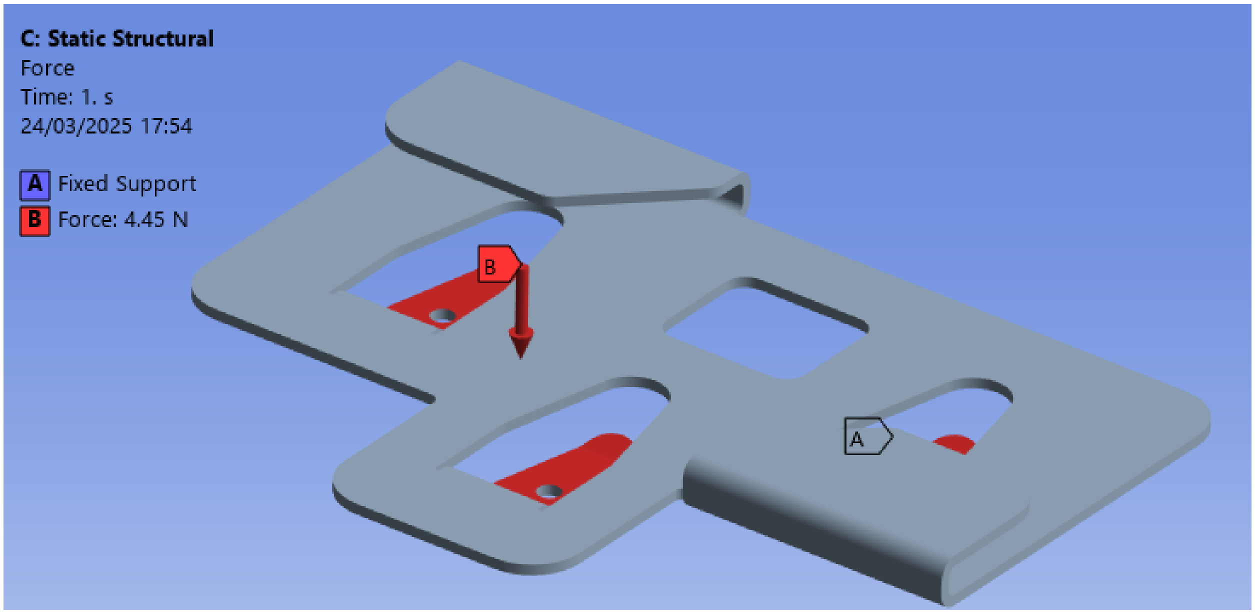

Fig. 5 .3 Boundary conditions established for analysis

5.5. FEA Results for Static Structural Behavior

Static structural analysis of a pedal uses Von Mises Stress and deflection to ensure safety under different loads. This process aids in optimizing design and performance, with clear graphics enhancing communication with technicians and manufacturers.

5.5.1. Equivalent Von Mises Stress

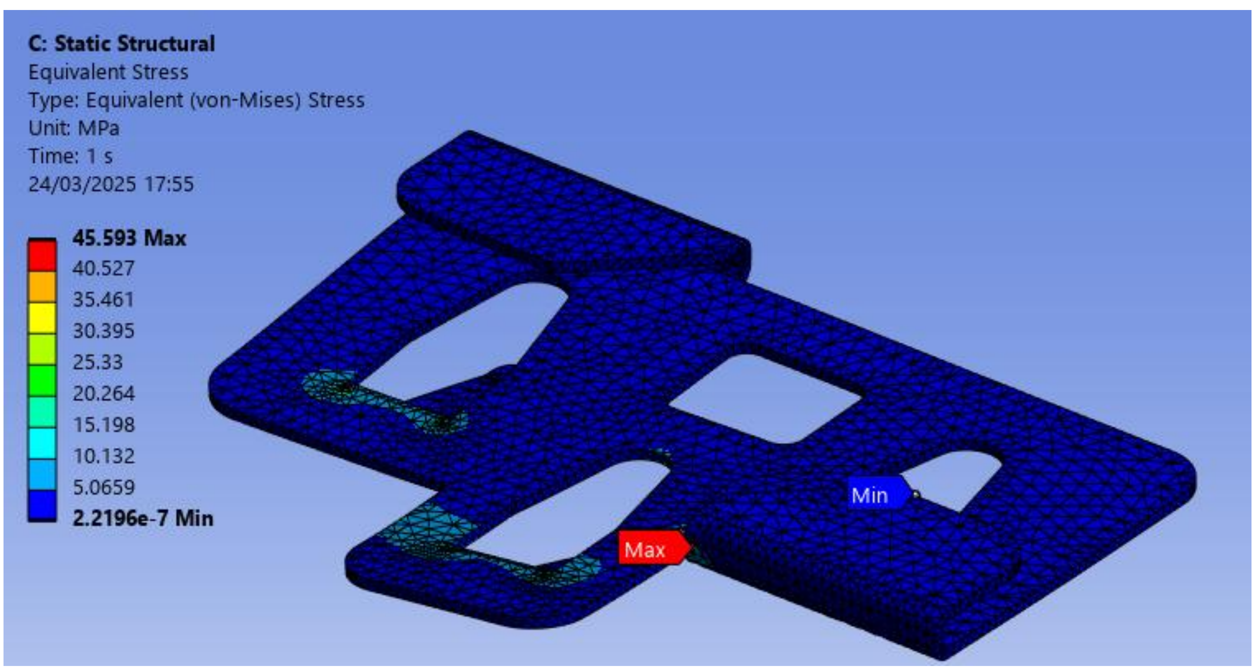

Fig. 5.4 *Equivalent Von-Mises Stress distribution*

The Von-Mises stress analysis showed maximum and minimum equivalent stresses of **70.74 MPa** and **0.0086 MPa**, respectively.

5.5.2. Total Deformation

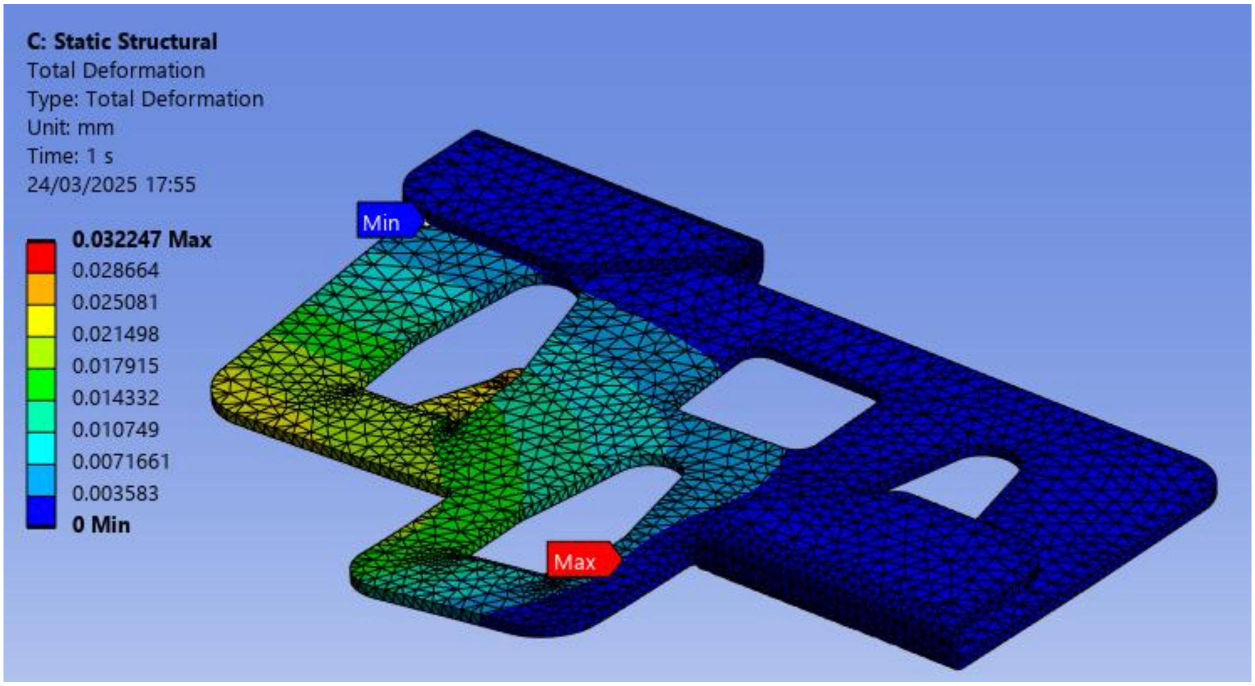

Fig. 5 .5 *Total deformation gradient*

The total deformation analysis showed maximum and minimum deformations of **0.76074 mm** and 0 mm, respectively.

5.5.3. Safety Factor

Static structural analysis displays safety factors using colors. Red indicates lower safety (higher stress), while blue shows higher safety (lower stress). This helps identify critical stress areas, with safety factors ranging from about 3.5341 to 15.

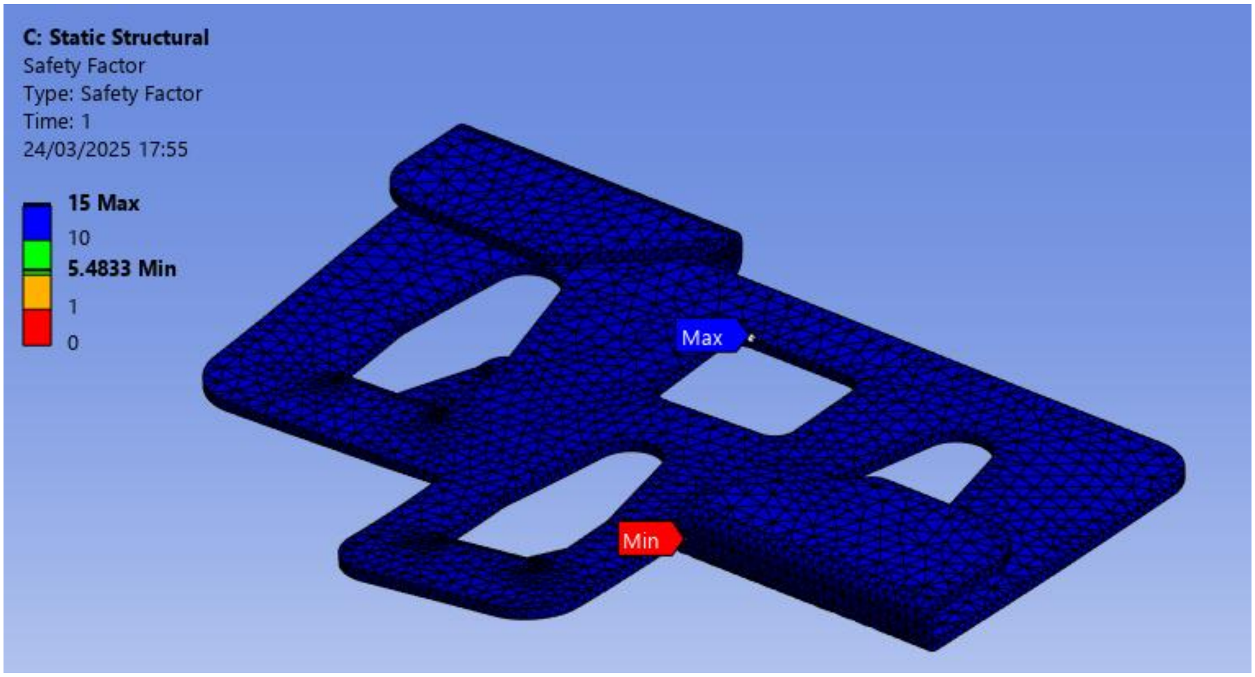

Fig. 5 .6 *Safety factor distribution*

Static structural analysis displays safety factors using colors. Red

Table-2. Results of the original Mounting	
Original Model Result	**Value**
Equivalent Von-Mises Stress (MPa)	70.74
Total Deformation (mm)	0.76074
Factor of safety	3.5341

The pedal is made from structural steel with a maximum tensile strength of 250 MPa. The current induced stress is much lower than this limit, indicating excess material that isn't necessary for safety. Topology optimization can be applied to reduce material use and lower the component's weight, leading to a more efficient design.

6. **Topology Optimization using Ansys**

ANSYS 2024 R2 Topology Optimization module features a user-friendly interface that simplifies running optimizations with constraints. It provides quick solutions to complex problems for users familiar with the process.

6.1. Allocation of Design Spaces

ANSYS 2024 R2 Topology Optimization focuses on material removal within the design space. It's crucial to define design and non-design regions, with non-design regions being areas where material cannot be removed, such as support and high-stress areas. The rest can be optimized for material removal without compromising the component's strength. The pedal design region is specified accordingly.

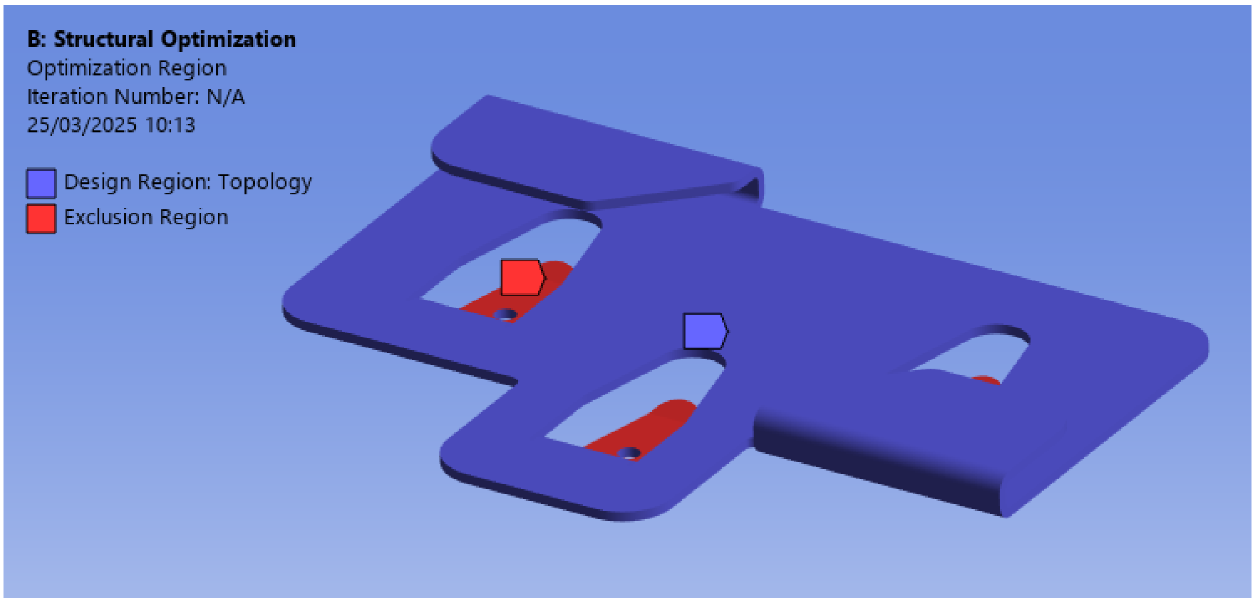

Fig. 6 .1 *Allocation of design spaces*

In the component, the blue region represents the design area eligible for optimization, while the red region indicates the non-design area, such as pivots and pads, which remain unchanged and do not require optimization.

6.2.Topology Procedure

After defining the design and non-design regions, the topology tool generates an optimized free-form model, resulting in a 50% weight reduction. The intricate design is ideal for additive manufacturing due to its complexity. Following this, the model will be redesigned to account for specific material removal areas and quantities, ensuring that the structural integrity is maintained while maximizing efficiency. This iterative process enables the creation of a lightweight, high-performance component tailored to meet engineering requirements.

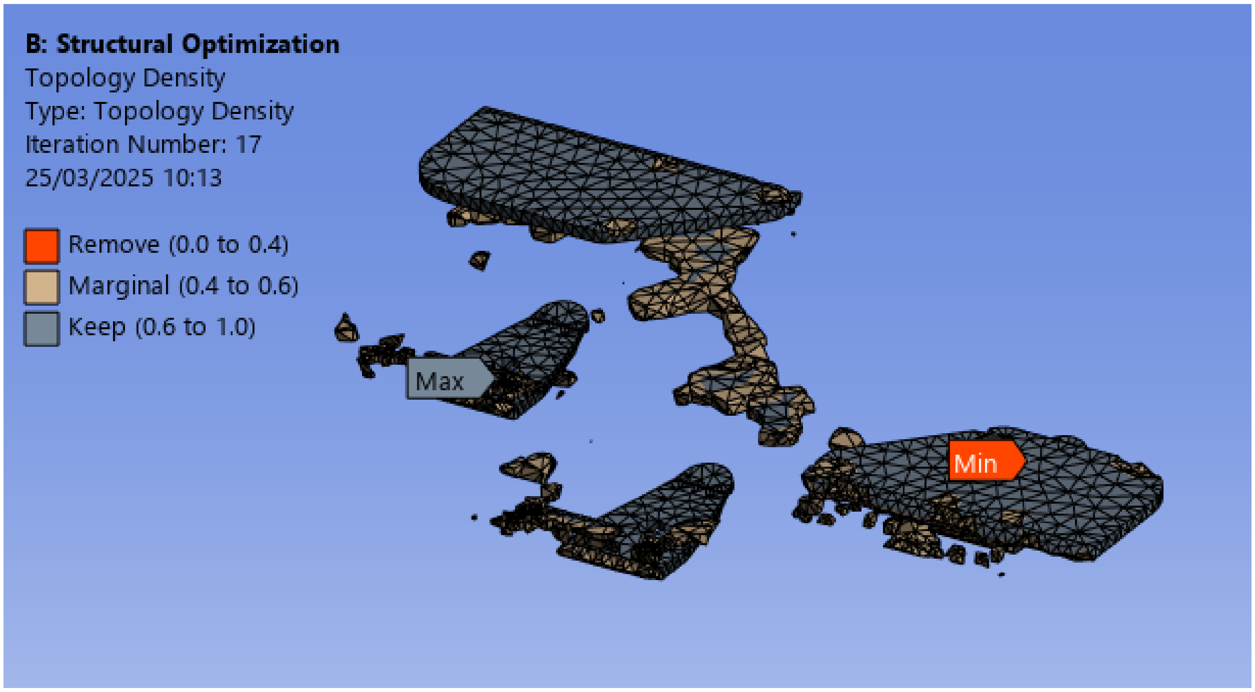

Fig-6.2 *Topology Optimized free from model*

6.3.Redesign and FEA Analysis of Optimized Models

The model from the Topology Optimization tool is further refined using Ansys Geometry SpaceClaim, a tool known for its precision in 3D modeling and design refinement. This process results in five distinct optimized models, each tailored to different design criteria and constraints. By leveraging SpaceClaim's capabilities, engineers can explore various configurations, enhancing the structural efficiency and functionality of the component while maintaining essential design parameters.

6.3.1. Static Structural Analysis of Optimized Model-1

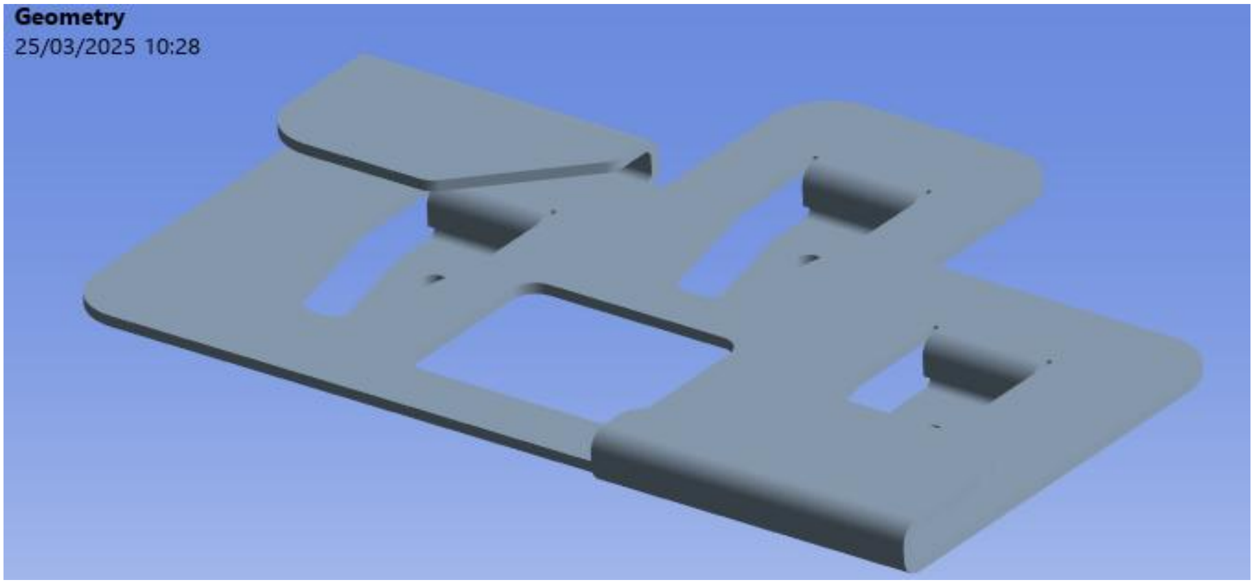

Fig. 6 .3 *Optimized model-1 (1.0524 kg)*

Fig. 6 .3 shows a 3D view of an optimized pedal model, weighing **1.0524 kg**. The design improves performance by minimizing weight while ensuring structural integrity.

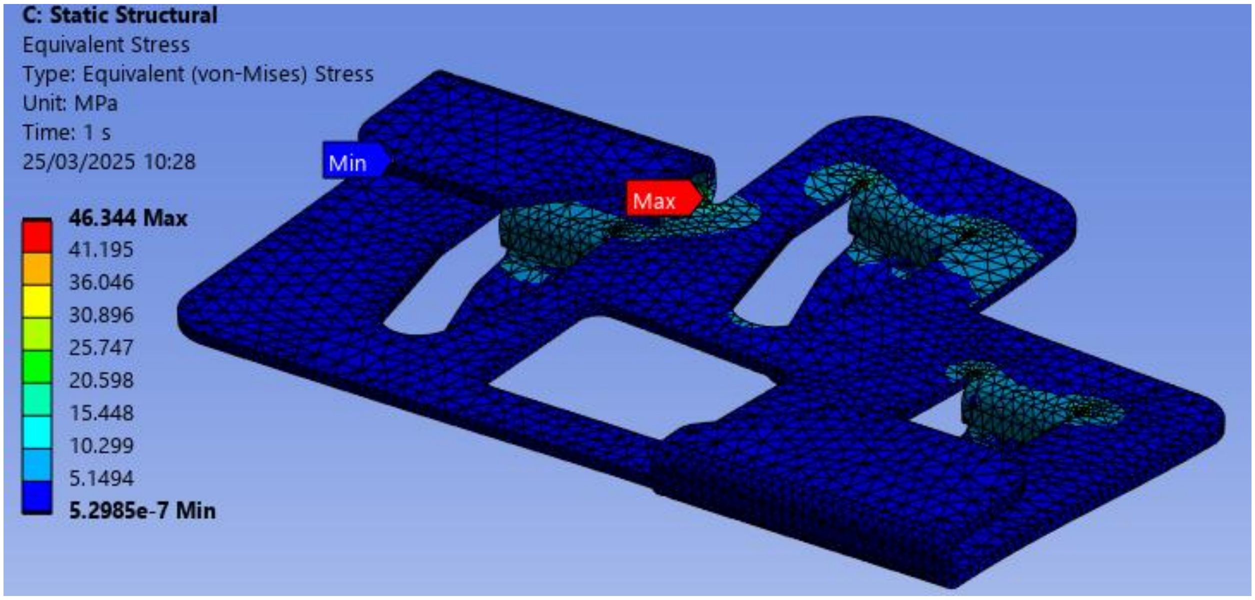

Fig. 6 .4 *Von-Mises Stress of optimized model-1*

Fig-6.4 depicts the color gradient from red to blue, which highlights areas of varying stress levels. Red indicates high stress, while blue shows low stress, with values ranging from approximately 0.027762 MPa to **79.591 MPa**.

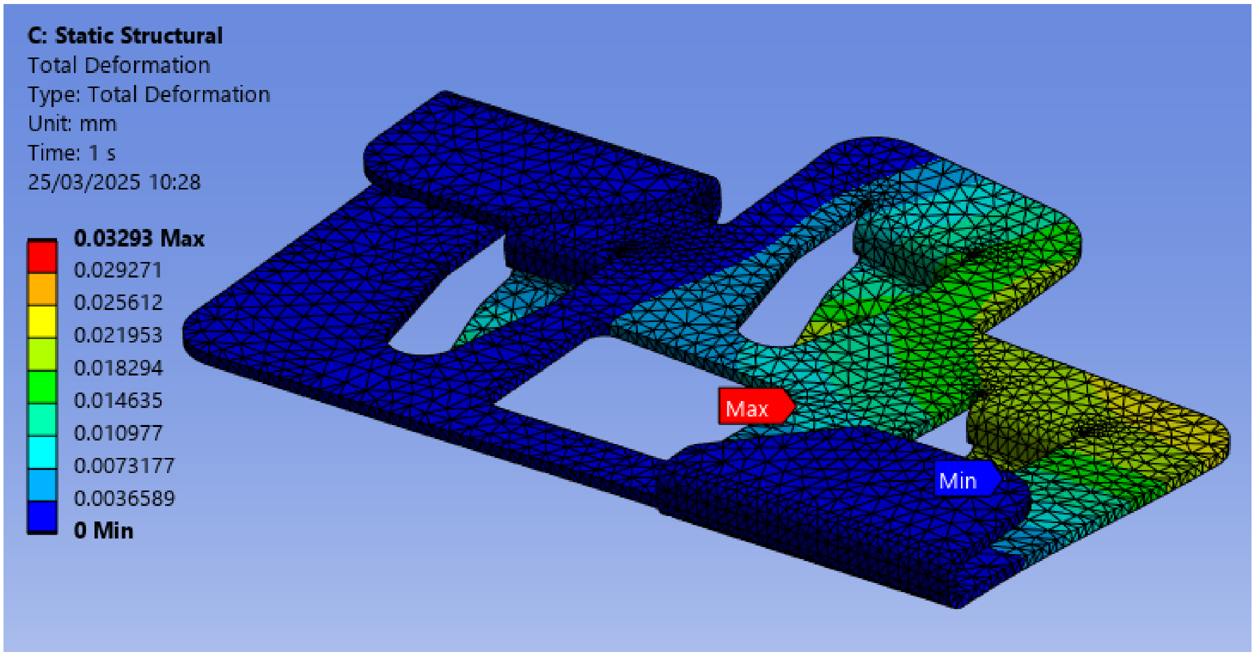

Fig. 6 .5 *Total Deformation of optimized model-1*

The maximum deformation value is **0.91604 mm**, and the minimum is 0 mm.

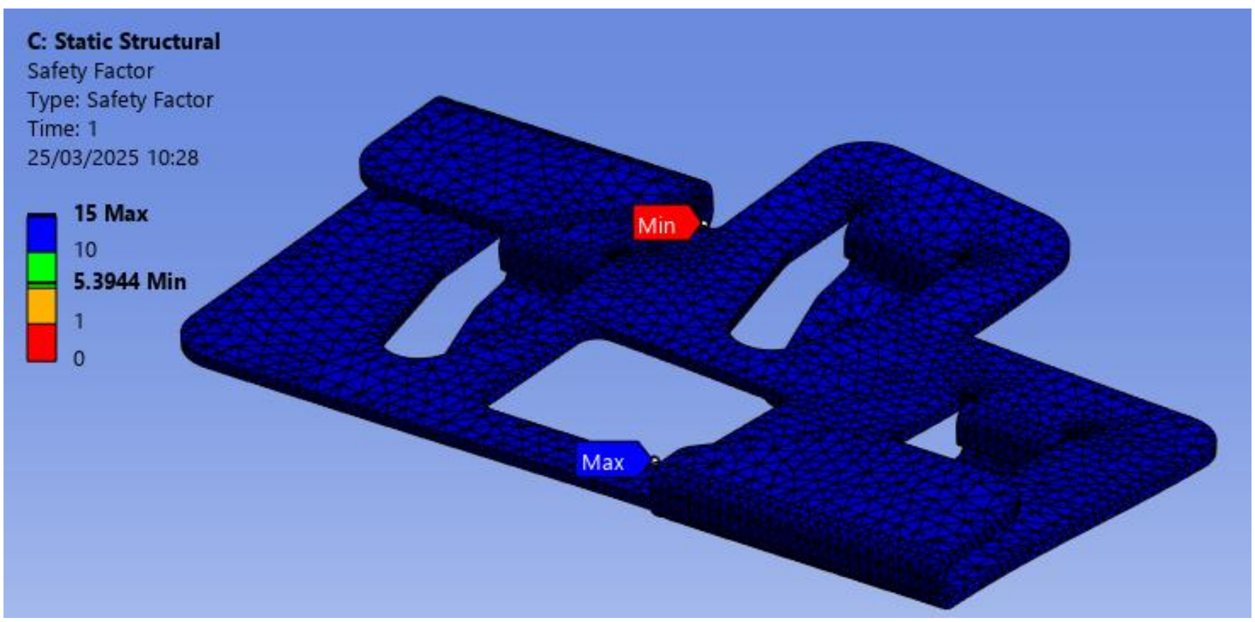

Fig. 6 .6 *Safety factor of optimized model-1*

The safety factor ranges from **3.1411** to 15.

7. Results and Discussion

Table-1. Von-Mises Stress (MPa)	
Model	**Equivalent Von-Mises Stress (MPa)**
Original	44.698
Optimized	46.344

Table-2. Total Deformation (mm)	
Model	**Total Deformation (mm)**
Original	0.031794
Optimized	0.03293

Table-3. Safety Factor	
Model	**Safety Factor**
Original	5.5931
Optimized	5.3944

Table-4. Masses of pedals (kg)	
Model	**Mass (gm)**
Original	33.53
Optimized	31.03

Based on the data presented:

- Von-Mises Stress (Table-1): The optimized model has a slightly higher stress (46.344 MPa), but this difference is minimal and likely acceptable.
- Total Deformation (Table-2): The optimized model has a slightly greater deformation (0.03293 mm), which is also negligible considering the small magnitude of the values.
- Safety Factor (Table-3): Both models have very similar safety factors, with the optimized model slightly lower (5.3944 vs. 5.5931), but still acceptable within typical designs.
- Mass of Pedal (Table-4): The optimized model achieves a mass reduction of 2.5 g (31.03 g vs. 33.53 g), which is beneficial for lightweighting efforts.

Conclusion:

The optimized model is the better choice due to its weight reduction while maintaining acceptable stress, deformation, and safety factor values. This is particularly advantageous in designs where weight saving is critical, such as in automotive or aerospace applications.Product