Methodology of Topology Optimization

Naziha Khandoker

Western Michigan University, nazihakhandoker12@gmail.com

Follow this and additional works at: https://scholarworks.wmich.edu/honors_theses

Part of the Aerodynamics and Fluid Mechanics Commons

Recommended Citation

https://scholarworks.wmich.edu/honors_theses/3217

This Honors Thesis-Open Access is brought to you for free and open access by the Lee Honors College at ScholarWorks at WMU. It has been accepted for inclusion in Honors Theses by an authorized administrator of ScholarWorks at WMU. For more information, please contact maira.bundza@wmich.edu.
Methodology of Topology Optimization

by

Mahnad Alawi
Naziha Khandoker
Ian New

Report: 12-19-08
Abstract

The need for lighter products is becoming increasingly essential as it cuts material cost and significant amount of weight, which is a key factor in the automotive and aerospace industries. Topology optimization allows us to achieve that by applying the concept to different structures with the goal of optimal distribution of material within finite volume design domain. Its algorithms selectively remove and relocate elements to achieve the optimum performance. This project performs a study on the effect of each design parameter over mechanical performance using Finite Element Analysis and applies it to a wing attachment brackets. The results are lighter and better performing structures.
Acknowledgements

The authors of this paper would like to thank the following people for the help in completing this project,

Dr. Jinseok Kim for his great support, guidance and encouragement throughout the project

Darren Promer for helping set up the testing equipment during the experimental stage of the project

Chris Rand for making sure all computers on Western Michigan University campus were equipped with Topology Optimization

Western Michigan University 3D print staff for making sure the parts were printed ahead of time for our presentation

Friends and family who showed their continuous love and support
Table of Contents

Abstract ........................................................................................................................................................................2
Acknowledgements ......................................................................................................................................................3
1.1 Background ..........................................................................................................................................................5
1.2 Scope ................................................................................................................................................................6
1.3 Limitations ........................................................................................................................................................6
2 Methodology .............................................................................................................................................................7
2.1 Simulation Process ...............................................................................................................................................7
2.1.2 Loading and Boundary Conditions ..............................................................................................................8
2.1.3 Topology Optimization ...................................................................................................................................8
2.1.4 Results ..........................................................................................................................................................9
2.2 Experimental Process .........................................................................................................................................12
2.2.1 Testing setup and Results ............................................................................................................................12
2.3 Results .............................................................................................................................................................14
3 Application to a Wing Attachment Bracket ...........................................................................................................15
3.1 Design, Material, and Boundary Conditions .....................................................................................................16
3.2 Z-Bracket Design ...............................................................................................................................................17
3.2.1 FE Analysis of Original .................................................................................................................................17
3.2.2 Design Space 1 ............................................................................................................................................19
3.2.3 Topology Optimization ................................................................................................................................19
3.2.4 FE Analysis ...............................................................................................................................................21
3.2.5 Design Space 2 ............................................................................................................................................22
3.2.6 Topology optimization ................................................................................................................................22
3.2.7 FE Analysis ...............................................................................................................................................23
3.2.8 Results comparison .....................................................................................................................................24
3.3 I-Bracket Design ...............................................................................................................................................25
3.3.4 FE Analysis ...............................................................................................................................................30
4 Conclusion and Recommendations .........................................................................................................................33
5 References .............................................................................................................................................................35
6 Appendices ..........................................................................................................................................................36
1 Introduction

This section introduces the reader of the concept of topology optimization; it starts with a background to structural optimization and its significance in the industry today. It is followed by the scope and limitations of the project.

1.1 Background

The design process of a selected component is an important step in the product development stage to produce sustainable and competitive products. To enhance strength and performance of a component, topology optimization provides the tool to obtain an optimal geometry in early phases of the design process. It improves the material layout of a component within a given design space, under a given set of loads, boundary conditions and constraints. Figure 1, shown below, was obtained from the topology optimization section of the ANSYS manual and shows a geometric redesign of a control arm used in a vehicle suspension system using topology optimization. The figure shows how the mass of a part can be reduced, leaving a new geometry with an optimal shape for the new specified maximum allowable mass.

![Figure 1: Topology Optimization Example](image)

By utilizing topology optimization in the development process, designers can study the structural behavior of the component. However, topology optimization can attain any shape within a given
design space which makes its application in industry rare in cases of complex geometries. Advancement in additive manufacturing is increasing the use of topology optimization in the industries as will be shown later in this report.

1.2 Scope

The purpose of this project is to establish a methodology for structural topology of existing components and apply it to two different models of wing attachment brackets. The methodology will include finite element analysis and experimental study to verify the methodology. The goal is to create a new and improved design of an existing part by reducing the mass of the part while complying with the same design requirements and standards associated with the original design. The overall mass of a system determines its performance parameters. Therefore, this project will make the use of finite element analysis software ANSYS Mechanical for the static structural simulation of the original and new designs, in order to assess the static load performance of the design. A comparison of maximum elastic strain, equivalent and shear stress, and total deformation values will be made throughout the project as these key parameters affect the performance of a structure.

1.3 Limitations

The above mentioned structural requirements (deformation, strain, and stress) which are considered in this project introduces finite element problems of both linear and non-linear structural behavior. However, only static linear FE problems are considered for the interest of this project. Furthermore, the wing attachment brackets will experience a dynamic loading due to structural vibrations. Due to constraint of time, dynamic load analysis of the structure was not considered to study the natural frequency of the parts. All engineering computers in WMU campus are equipped with educational level license for ANSYS, which only allows for 32000
cells/nodes for mechanical model. This prevented the use of finer mesh to get a more refined result with greater accuracy. However, the results obtained were sufficient for the purpose of the project, which is to see the changes in performance parameters within different geometries.

2 Methodology

The methodology development is divided into two parts. The first part is the simulation process, which focuses on topology optimization using Finite Element (FE) modeling on ANSYS. Multiple trials were run once the loading and boundary conditions were applied to the design space to find the effect of mass reduction. The second part is the experimental process during which the parts obtained from the topology optimization were 3D printed and tested under multiple force loadings. The results from each process were then compared to find to verify the method.

2.1 Simulation Process

The commercial software used in the simulation process FE modeling is ANSYS. A design space is created is first created in SpaceClaim and using ANSYS Mechanical, a static structural analysis is conducted for the geometry created. A topology optimization is then applied to the part. This is done several times for different optimization requirements. The results from the simulation process are then discussed.

2.1.1 Design Review

A simple beam, shown below in Figure 2, was chosen as the design space for the simulation process. The beam has a length of 100mm and a cross-sectional area of 30mmX30mm. A thicker beam was selected to keep shear stress dominant. This is because shear stress does not evenly distribute on structures. Therefore, it will show how the software optimizes the structures based on the stress distribution on the original part. A 3 mm element size mesh shown below in Figure
3, was used for this test since that is the smallest mesh size allowed for this structure in the student version of ANSYS.

2.1.2 Loading and Boundary Conditions

To perform a topology optimization, applying a certain set of load and boundary condition are required. The two lower short edges were pin supported which restricts displacement in all directions and allows for rotation only in y-direction. A concentrated force with a magnitude of 5000 N was applied along a line running across the top of the model. Figures 4 and 5 show the loading and boundary conditions applied to the beam.

2.1.3 Topology Optimization

Once the design space, meshing, and the loading and boundary conditions are determined, the topology optimization can be applied. The topology optimization was applied with a mass
constraint, which allows the user to specify the percentage of mass to retain from the original geometry. The entire design space was selected in the design area that assigns the software the regions to take out material from but excluded the regions where the boundary conditions (force line and supports) were applied. The software by default tries to keep the stress on the new structure as low as possible.

2.1.4 Results

The result of the topology optimization started by removing the corners of the structure and keeping a bridge like shape connecting between the force and the supports. In addition, that shows that the software starts by removing the parts that have low stress and keeps the parts with high stress as shown in Figure 6. The bridge goes in 45 degrees angle, which could be explained by saying that the shear stress is dominant on this structure. Figures 7 to 10 show examples of the optimized structures. Pictures of all the optimized models can be found in appendix A.
After obtaining nine different optimized parts, the same boundary conditions were applied to them with a force of magnitude 5000N. The obtained results of the stress, deformation, strain are shown in Table 1. Since the stress can be difficult to measure accurately in real life, the only focus will be on the deformation change.

<table>
<thead>
<tr>
<th>Structure</th>
<th>Mass (kg)</th>
<th>Volume (m$^3$)</th>
<th>Stress (Pa)</th>
<th>Deformation (m)</th>
<th>Strain (m/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>9.36E-02</td>
<td>9.00E-05</td>
<td>3.37E+07</td>
<td>-5.794E-04</td>
<td>1.43E-02</td>
</tr>
<tr>
<td>10% removed</td>
<td>8.50E-02</td>
<td>8.17E-05</td>
<td>2.59E+07</td>
<td>-5.5597E-04</td>
<td>1.10E-02</td>
</tr>
<tr>
<td>20% removed</td>
<td>7.75E-02</td>
<td>7.45E-05</td>
<td>2.28E+07</td>
<td>-5.35E-04</td>
<td>9.68E-03</td>
</tr>
<tr>
<td>30% removed</td>
<td>6.68E-02</td>
<td>6.42E-05</td>
<td>2.28E+07</td>
<td>-5.74E-04</td>
<td>9.62E-03</td>
</tr>
<tr>
<td>40% removed</td>
<td>5.72E-02</td>
<td>5.50E-05</td>
<td>2.56E+07</td>
<td>-6.34E-04</td>
<td>1.11E-02</td>
</tr>
<tr>
<td>50% removed</td>
<td>4.79E-02</td>
<td>4.61E-05</td>
<td>3.45E+07</td>
<td>-7.09E-04</td>
<td>1.47E-02</td>
</tr>
<tr>
<td>60% removed</td>
<td>3.87E-02</td>
<td>3.72E-05</td>
<td>3.93E+07</td>
<td>-7.93E-04</td>
<td>1.65E-02</td>
</tr>
<tr>
<td>70% removed</td>
<td>2.93E-02</td>
<td>2.82E-05</td>
<td>3.27E+07</td>
<td>-9.60E-04</td>
<td>1.37E-02</td>
</tr>
<tr>
<td>80% removed</td>
<td>1.54E-02</td>
<td>1.48E-05</td>
<td>6.44E+07</td>
<td>-1.88E-03</td>
<td>2.70E-02</td>
</tr>
<tr>
<td>90% removed</td>
<td>8.02E-03</td>
<td>7.71E-06</td>
<td>1.35E+08</td>
<td>-3.02E-03</td>
<td>5.68E-02</td>
</tr>
</tbody>
</table>

Figure 11 shows a plot of the deformation for each structure. It can be seen that the deformation stays constant until more than 20% of the mass removed. After removing 50% of the mass or more the deformation increases more than 20%, which could lead to structural failure in most cases. Therefore, the original structure and only the first four optimized structures will be tested.
The five structures were subjected to five different force magnitudes, and the displacement was observed at each force magnitude. The stiffness of each part can be found by getting the slope of the force versus displacement curve. Table 2 shows the forces and the corresponding displacement of each model and the plotted data can be seen in Figure 12. The stiffness results of the simulation presented on Table 3 and Figure 13.

![Figure 11: Change in Deformation vs Mass Removed at 5000N Load](image)

![Table 2: Displacement of Structures at Different Loads](image)

![Figure 12: Force vs Displacement Plot (Simulation)](image)
Table 3: Stiffness for Different Models (Simulation)

<table>
<thead>
<tr>
<th>Model</th>
<th>Stiffness, N/m</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>8.6984E+06</td>
</tr>
<tr>
<td>10% Removed</td>
<td>8.9171E+06</td>
</tr>
<tr>
<td>20% Removed</td>
<td>9.2175E+06</td>
</tr>
<tr>
<td>30% Removed</td>
<td>8.3419E+06</td>
</tr>
<tr>
<td>40% Removed</td>
<td>7.2475E+06</td>
</tr>
</tbody>
</table>

Figure 13: Plotted Simulation Stiffness Results

2.2 Experimental Process

To be able to apply this method in the real world applications, an experimental stage was required to verify the results obtained from the simulation process. This section describes the experiment set up and how the parts were tested. As the stress values cannot be recorded during this stage, the stiffness of the geometries was of interest.

2.2.1 Testing setup and Results

The parts were 3D printed using PLA plastic. 1K-16 Tension testing machine was used to perform the test. The machine can apply up to 1000 lb of compressive force. The model was
placed on a 3-point bending test stand and the load applied using a 0.25 in aluminum rod as shown in Figure 14. The test was performed using displacement control at rate of 1mm/min that allows recording more data points and plot curves that are more accurate. The machine stopped in the case of either the model fracturing or the machine reaching the load limit. All the parts were able to withstand the 4337 N (975 lb) load except the 40% mass removed part. The 40% model started fracturing at around a load of 2900 N. Figure 15 shows the fractured part.

The load and displacement data were recorded. The stiffness of each structure can be found by plotting the load vs displacement and finding the slope of the curve for each model. From Figure 16, it can be observed that the slope of the curves decreases when the percentage of the mass removed increase. Table 4 shows the stiffness values and Figure 17 shows the trend in stiffness.
Table 4: Stiffness for Different Models (Experiment)

<table>
<thead>
<tr>
<th>Model</th>
<th>Stiffness, N/m</th>
</tr>
</thead>
<tbody>
<tr>
<td>Original</td>
<td>2.1857E+06</td>
</tr>
<tr>
<td>10% Removed</td>
<td>2.1800E+06</td>
</tr>
<tr>
<td>20% Removed</td>
<td>2.1888E+06</td>
</tr>
<tr>
<td>30% Removed</td>
<td>1.7797E+06</td>
</tr>
<tr>
<td>40% Removed</td>
<td>9.2078E+05</td>
</tr>
</tbody>
</table>

Figure 16: Force vs Displacement Plot
Figure 17: Plotted Experimental Stiffness Results

2.3 Results

The results of the simulation and the experiment cannot be compared directly since it is hard to find the exact properties of the material to use in the simulation. Therefore, the normalized stiffness was used to compare the results, as summarized in Table 5. The structural stiffness depends on the material, boundary conditions, and the geometry. Since the material and the boundary conditions were the same for all simulation runs and they were same for all the experimental runs, the only factor that will affect the stiffness is the change in the geometry. Figure 17 shows that the patterns of the stiffness are close to each other. The variation between
the two curves caused by the fact that the material properties are different. In addition, the 3D printed parts are not homogeneous as the part in the simulation. Moreover, the small differences in the boundary conditions between the runs in the experimental process can affect the results.

Table 5: Normalized Stiffness Comparison

<table>
<thead>
<tr>
<th>Model</th>
<th>Normalized Stiffness</th>
<th>Error %</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>simulation</td>
<td>experiment</td>
</tr>
<tr>
<td>Original</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>10% Removed</td>
<td>1.025</td>
<td>0.999</td>
</tr>
<tr>
<td>20% Removed</td>
<td>1.0597</td>
<td>1.0004</td>
</tr>
<tr>
<td>30% Removed</td>
<td>0.959</td>
<td>0.953</td>
</tr>
<tr>
<td>40% Removed</td>
<td>0.833</td>
<td>0.855</td>
</tr>
</tbody>
</table>

Figure 18: Normalized Stiffness Curves Comparison

From all of that it can be said that the results of the topology optimizations using ANSYS can be translated in the real-life applications.

3 Application to a Wing Attachment Bracket

Once the method has been verified, it was possible to apply it to a part used in the industry today. For that purpose, two different models of wing attachment brackets were selected. Brackets are connector type elements that provide structural supports for hydraulic and electrical units in aircraft engine, wings and landing gears. A failure in the structure of a bracket can cause a catastrophe. The two models, Z-bracket and I-bracket, are shown in Figures 19 and 20. The loading conditions and specifications will be discussed in the following section. The parts will be
individually looked at and optimized. The FE Analysis will study the total deformation, equivalent elastic strain, equivalent and shear stresses. The overall results will be discussed at the end of the section.

![Z-bracket](image1)

![I-bracket](image2)

3.1 Design, Material, and Boundary Conditions

Based on the literature survey conducted [1], the two models are made out of Aluminum Alloy 7075-T6. This is a light weight and high strength material, usually used for highly stressed structures like airframe brackets.

The boundary and loading conditions were kept constant between the two models. There were only two conditions considered for this project, one surface from the bracket fixed and a pressure hole on the two holes. The loading conditions are based off the report on wing attachment brackets [2]. There, 100 N force was applied at the center of mass of the bolts in each direction. This gives a resultant force of 173.21 N. This resultant force was converted to a distrusted pressure load by dividing the force value by the area of the small holes, $0.000235 \text{ m}^2$.

\[
\text{Pressure} = \frac{\text{Force}}{\text{Area}} = \frac{173.21 \text{ N}}{\pi (0.005^2-0.0025^2)} \text{ m}^2
\]

\[
\text{Pressure} = 2.94 \text{ MPa}
\]
It is assumed that this pressure load is due to the pressure the bolt applies on the surface of the bracket upon contact. Figures 21 and 22 show the loading and boundary conditions on Z and I shaped brackets, respectively.

![Figure 21: Conditions on Z-bracket](image1)

![Figure 22: Conditions on I-bracket](image2)

3.2 Z-Bracket Design

Figure 23 shows how the original Z-bracket looks like with the mesh.

![Figure 23: Original Z-bracket, meshed](image3)

3.2.1 FE Analysis of Original

The following figures, Figures 24 to 27, shows the performance parameters of the original structure. These values will be later compared with the optimized geometries.
Table 6 shows a summary of maximum values of the above parameters from the original Z-bracket.

<table>
<thead>
<tr>
<th></th>
<th>Original Z-bracket</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass (g)</td>
<td>35.392</td>
</tr>
<tr>
<td>Total Deformation (mm)</td>
<td>0.193</td>
</tr>
<tr>
<td>Equivalent Elastic Strain (m/m)</td>
<td>0.00024879</td>
</tr>
<tr>
<td>Equivalent Stress (MPa)</td>
<td>17.837</td>
</tr>
<tr>
<td>Shear Stress (MPa)</td>
<td>1.786</td>
</tr>
</tbody>
</table>
3.2.2 Design Space 1

The first step in topology optimization is to create the design space. Figure 28 shows the first design space created for the z-shaped model with mesh. The targeted element size when constructing the mesh is 2 mm. The material is the same as the original, Aluminum alloy.

![Design Space-1 Z-bracket](image)

*Figure 28: Design Space-1 Z-bracket*

3.2.3 Topology Optimization

Figure 29 shows the optimization region. The areas shaded in blue are allowed to be taken out, and the areas in red are excluded from topology optimization. The areas in red were excluded for the purpose of giving the structure some support. The back sides of the holes are also excluded for even distribution during optimization process.

![Design region for optimization](image)

*Figure 29: Design region for optimization*
Under the optimization criteria, 75% mass reduction of the design space was requested. The following Figure 30 shows the result.

![Figure 30: Optimization Result on ANSYS](image)

However, the model obtained from ANSYS is not usable because the structure is unsymmetrical and has uneven surfaces. Therefore, this is only a template which needs to be refined and the final product is shown in Figure 31.

![Figure 31: Final Design - 1](image)
3.2.4 FE Analysis

After obtaining the optimized geometry, the design performance of the new geometry has to be evaluated. A static structural is applied to this part as was done with the original structure. Figures 32 to 35 show the results.

![Total Deformation on Original Z-bracket](image1)

![Equivalent Elastic Strain on Original Z-bracket](image2)

![Equivalent Stress on Original Z-bracket](image3)

![Shear Stress on Original Z-bracket](image4)

### Table 7: Summary of Optimized Design Space-1

<table>
<thead>
<tr>
<th>Optimized Design-1</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass (g)</td>
<td>22.733</td>
</tr>
<tr>
<td>Total Deformation (mm)</td>
<td>0.0818</td>
</tr>
<tr>
<td>Equivalent Elastic Strain (m/m)</td>
<td>0.00018212</td>
</tr>
<tr>
<td>Equivalent Stress (MPa)</td>
<td>13.055</td>
</tr>
<tr>
<td>Shear Stress (MPa)</td>
<td>1.452</td>
</tr>
</tbody>
</table>
3.2.5 Design Space 2

A parametric study was conducted to see if the holes on geometry were to be brought closer to each other, would that make a difference in the optimized geometry. The design space-2 is shown below in Figure 36 with the bottom two holes closer to each other. The material is also kept constant, Aluminum alloy.

![Figure 36: Design Space-2](image)

3.2.6 Topology optimization

The same topology optimization was applied to the new design space, to retain 25% of the mass. The regions of optimization were same as previous design space, shown in Figure 29. The images, figures 37 and 38, below show the unrefined and refined models obtained after applying topology optimization to design space-2.

![Figure 37: Optimization Result on ANSYS](image)  ![Figure 38: Final Design - 1](image)
3.2.7 FE Analysis

To compare the performance of this new geometry, FE Analysis was done again to see how the new optimized shape behaves under the same loading and boundary conditions. Figures 39 to 42 show the results. Table 8 tabularizes the maximum value of each parameter.

![Figure 39: Total Deformation on Original Z-bracket](image1)
![Figure 40: Equivalent Elastic Strain on Original Z-bracket](image2)
![Figure 41: Equivalent Stress on Original Z-bracket](image3)
![Figure 42: Shear Stress on Original Z-bracket](image4)

<table>
<thead>
<tr>
<th>Optimized Design-2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass (g)</td>
</tr>
<tr>
<td>Total Deformation (mm)</td>
</tr>
<tr>
<td>Equivalent Elastic Strain (m/m)</td>
</tr>
<tr>
<td>Equivalent Stress (MPa)</td>
</tr>
<tr>
<td>Shear Stress (MPa)</td>
</tr>
</tbody>
</table>
3.2.8 Results comparison

Table 9 below summarizes the results from the z-bracket. The results obtained after applying topology optimization to the z-bracket meets the goal of the project. There was a significant reduction in mass of over 35% for both the parts. The performance of the geometry also improved in most areas. The total deformation decreased by close to 60% for both the newer geometries. This could be due to the increase in stiffness of the geometry. The maximum stress on both the new geometries were more than 26% lower than the original bracket which was close to 18 MPa. The shear stress in Design 2 increased by 46% and this was due to design-2 having greater curvature in its geometry.

<table>
<thead>
<tr>
<th>Original Geometry</th>
<th>Design 1</th>
<th>Design 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass (g)</td>
<td>22.733</td>
<td>22.074</td>
</tr>
<tr>
<td>Mass (g)</td>
<td>22.733</td>
<td>22.074</td>
</tr>
<tr>
<td>Total Deformation (mm)</td>
<td>0.0818</td>
<td>0.0853</td>
</tr>
<tr>
<td>Total Deformation (mm)</td>
<td>0.0818</td>
<td>0.0853</td>
</tr>
<tr>
<td>Equivalent Elastic Strain</td>
<td>0.00018212</td>
<td>0.0017805</td>
</tr>
<tr>
<td>Equivalent Elastic Strain</td>
<td>0.00018212</td>
<td>0.0017805</td>
</tr>
<tr>
<td>Equivalent Stress (MPa)</td>
<td>13.055</td>
<td>12.765</td>
</tr>
<tr>
<td>Equivalent Stress (MPa)</td>
<td>13.055</td>
<td>12.765</td>
</tr>
<tr>
<td>Shear Stress (MPa)</td>
<td>1.452</td>
<td>2.613</td>
</tr>
<tr>
<td>Shear Stress (MPa)</td>
<td>1.452</td>
<td>2.613</td>
</tr>
<tr>
<td>Shear Stress (MPa)</td>
<td>1.452</td>
<td>2.613</td>
</tr>
<tr>
<td>Shear Stress (MPa)</td>
<td>1.452</td>
<td>2.613</td>
</tr>
</tbody>
</table>

To conclude, design-1 had the best result. It weighs 36% less than the original, deforms less compared to the original and experiences significantly lower stress levels.
3.3  **I-Bracket Design**

Figure 43 shows how the original I-bracket looks like with the mesh.

![Figure 43: Original I-bracket, meshed](image)

### 3.3.1 FE Analysis of Original

The following figures, Figures 44 to 47, shows the performance parameters of the original structure. These values will be later compared with the optimized geometries.

![Figure 44: Total Deformation on Original I-bracket](image)
Figure 45: Equivalent Elastic Strain on Original I-bracket

Figure 46: Equivalent Stress on Original I-bracket
Table 10 shows a summary of maximum values of the above parameters from the original I-bracket.

**Table 10: Summary of I-bracket**

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass (g)</td>
<td>39.46</td>
</tr>
<tr>
<td>Total Deformation (mm)</td>
<td>0.031818</td>
</tr>
<tr>
<td>Equivalent Elastic Strain (m/m)</td>
<td>0.00014967</td>
</tr>
<tr>
<td>Equivalent Stress (MPa)</td>
<td>11.042</td>
</tr>
<tr>
<td>Shear Stress (MPa)</td>
<td>4.0171</td>
</tr>
</tbody>
</table>
3.3.2 Design space

The first step in topology optimization is to create the design space. Figure 48 shows the first design space created for the I-shaped model with mesh. The targeted element size when constructing the mesh is 2 mm. The material is the same as the original, Aluminum alloy.

![Image: Design Space-1 Z-bracket](image)

*Figure 48: Design Space-1 Z-bracket*

3.3.3 Topology optimization

Figure 49 shows the optimization region. The areas shaded in blue are allowed to be taken out, and the areas in red are excluded from topology optimization. The areas in red were excluded for the purpose of giving the structure some support. The back side of the holes and surrounding material are also excluded for even distribution during optimization process. For the I-bracket the contact areas between the bracket, the wing structure, and surrounding components was kept constant and only the middle area of the part was chosen for the topology optimization.
Under the optimization criteria, 93% mass reduction of the design space was requested. The following Figure 49 shows the result.

However, the model obtained from ANSYS is not usable because the structure is unsymmetrical and has uneven surfaces. Therefore, this is only a template which needs to be refined and the final product is shown in Figure 51 with a mesh applied.
3.3.4 FE Analysis

After obtaining the optimized geometry, the design performance of the new geometry has to be evaluated. A static structural analysis is done with the same boundary conditions and loading as the original part. Figures 52 to 55 show the results.
Figure 53: Equivalent Elastic Strain on Optimized I-bracket

Figure 54: Equivalent Stress on Optimized I-bracket

Figure 55: Shear Stress on Optimized I-bracket
Table 11 shows a summary of maximum values of the above parameters from the optimized I-bracket.

<table>
<thead>
<tr>
<th>Optimized Design</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass (g)</td>
<td>35.746</td>
</tr>
<tr>
<td>Total Deformation (mm)</td>
<td>0.011514</td>
</tr>
<tr>
<td>Equivalent Elastic Strain (m/m)</td>
<td>0.00018816</td>
</tr>
<tr>
<td>Equivalent Stress (MPa)</td>
<td>12.639</td>
</tr>
<tr>
<td>Shear Stress (MPa)</td>
<td>3.7647</td>
</tr>
</tbody>
</table>

Table 12 below summarizes the results from the I-bracket. The results obtained after applying topology optimization to the I-bracket meets the goal of the project. There was a significant reduction in mass of close to 10%. The performance of the geometry also improved in most areas. The maximum total deformation decreased by over 60%. This could be due to the increase in stiffness of the geometry. The maximum shear stress decreased by about 6%. The maximum stress on the new geometry increased by almost 13% and the maximum strain also increased by almost 15%. The small increase in maximum stress and strain however do not prevent the new design from meeting the mechanical standards and safety factor imposed on the original design, because of this we can say the new design is an improvement upon the original design.
4 Conclusion and Recommendations

This project shows it is possible to improve upon designs of mechanical and structural components using topology optimization, to reduce the weight of the component by changing its geometry. Following the method described in this project, mechanical and structural components can be designed using an optimal shape that allows the minimum amount of material to be used to meet the mechanical standards and safety factor associated with the part. This reduces manufacturing cost of the parts and increases the performance of the overall system due to lighter parts being used. Two components being used in the aerospace industry were evaluated using ANSYS and compared to components that were redesigned using topology optimization. The redesigned components were made of the same material and were lighter and met the same mechanical standards and safety factors of the original parts. This was achieved by using topology optimization to generate the optimum shape.

A dynamic analysis will need to be performed on the redesigned parts in order to determine the natural frequencies associated with the part. The natural frequencies of systems are those frequencies at which the resonant response occurs under the right excitation conditions. Knowledge of these critical dynamic frequencies is an essential step in the design and evaluation
of a system subjected to dynamic loading. Different modes of frequency can be estimated by
modal analysis using Ansys software. Finally, the redesigned parts need to be manufactured
using tradition manufacturing methods or if it proves to be too difficult or expensive to
manufacture the complex geometries using tradition manufacturing methods, additive
manufacturing methods can be utilized to manufacture to parts. Once the parts are manufactured,
they can be tested to determine their actual performance.
5 References
Appendix A
(The Optimized Parts)

Original Beam

10% Mass Removed

20% Mass Removed

30% Mass Removed

40% Mass Removed

50% Mass Removed
60% Mass Removed

70% Mass Removed

80% Mass Removed

90% Mass Removed
Appendix B

**ABET student outcome 4:**

An ability to recognize ethical and professional responsibilities in engineering situations and make informed judgments, which must consider the impact of engineering solutions in global, economic, environmental and societal contexts.

Performance Indicator #2: student is able to make informed judgments based on the impact of engineering solutions in global, economic, environmental and societal context.

Did you adapt your project to make it useful in many countries? Y / N / NA If yes, explain:

Yes.
Our project can be useful in many countries as it describes a method to follow for the redesign of mechanical and structural components using topology optimization.

Did you consider standards and regulations, either U.S. or international? Y / N / NA If yes, explain how they affected your project:

Yes.
We considered the mechanical standards that were applied to the parts we redesigned using topology optimization. We had to make sure our redesigned part met the same standards as the original.

Did you consider the effects of manufacturing in various locations? Y / N / NA If yes, where in the report did you address this issue?

Yes.
We considered the manufacturability of the final components in the conclusion and recommendation section of the report.

Did you have to balance effects of costs and performance? Y / N / NA If yes, explain and refer to the report as appropriate.

No.
The goal of our project was to reduce material cost while improving performance or maintaining the performance of the original part.
Did you consider effects of maintenance, failure and repair on cost, safety, etc.? Y / N / NA If yes, where in the report did you address them?

No.

We did not consider the cost of maintenance failure of repair. However, we did considered safety as the components we redesigned had to meet a certain mechanical standard.

What were your considerations (e.g., cost, weight, manufacturing, availability, safety, recycling, etc.) in the selection of materials? List, explain and refer to the text of the report as appropriate.

We kept the material of the original design, a high strength lightweight aluminum alloy, constant throughout our project. Our project focused on the geometric optimization of the component without changing the material used.

Does your project impact air quality, water quality, noise levels, and other environmental aspects? Y / N / NA If yes, explain how and show what were your actions.

No.

Does your project impact human health during manufacturing or normal use? Y / N / NA If yes, explain what you did to alleviate the risks.

No.

Are there any other safety issues typical to your project? Y / N / NA If yes, explain your decisions and actions. Refer to the report as appropriate.

Yes.

Failure of the mechanical component we redesigned could lead to a failure of the surrounding system, an aircraft. Failure of an aircraft could lead to loss of life.