UNIVERSITY OF DEBRECEN
FACULTY OF ENGINEERING
DEPARTMENT OF
MECHANICAL ENGINEERING

DESIGN OPTIMIZATION AND FATIGUE LIFE ANALYSIS OF A BICYCLE HEADLAMP MOUNTING BRACKET USING FINITE ELEMENT METHOD

THESIS

Michael Okafor Ebuka
Production Engineering Specialization

Debrecen
2026

Table of Contents

Table of notations

$\sigma a = S'f\,(2N)b$	Basquin Equation
$\frac{\Delta\varepsilon}{2} = \left(\frac{\sigma' f}{E}\right)(2N)^b + \varepsilon' f(2N)^c$	Strain-Life (ε-N) Method
$FOS = \frac{\text{Yield Strength}}{\text{Maximum Stress}}$	Factor of Safety (FOS)
σa	Stress amplitude (MPa)
S'f	Fatigue strength coefficient
N	Number of cycles to failure
B	Fatigue strength exponent
2N	Number of stress reversals
Δε/2	Strain amplitude
σ'f	Fatigue strength coefficient
ε'f	Fatigue ductility coefficient
C	Fatigue ductility exponent
E	Elastic modulus
Σ	Stress (general)
Σy	Yield strength
X	Design variable vector
f(x)	Objective function
σ_allowable	Allowable stress
δ_limit	Deflection limit
x_min	Minimum design variable bound
x_max	Maximum design variable bound
P	Element density (SIMP method)
S	Stress amplitude in S-N curve

1 Introduction

1.1 Background and Overview

Practically, bicycle lighting systems' mounting brackets are found to do more labor than what people would think they are important in ensuring that the headlamp remains stable and aligned to its expected alignment during dynamic loads. These brackets, although small relatively, are subjected to vibrations, cyclic stresses, as well as impact forces in the bicycle frame in case of riding over rough terrain. These loads may cause the headlamp to be misaligned, loose or break up, which is dangerous to the rider, particularly during low visibility.

The Finite Element Method (FEM) is one of the most effective and practical framework in information prediction regarding stress distribution, deformation, and fatigue life at realistic operating conditions [6], [12]. FEM has been extensively used to study mechanical design, such as brackets, joints, and other small structural components that are under complicated loads [11], [17]. FEM in combination with fatigue assessment can be used to identify important areas that have high levels of stress concentration where failure is most probable to begin [22].

Topology optimization can also assist the design process in a more informed manner through finding efficient material layouts to realize a better stiffness to weight performance [5], [23]. This is mostly applicable in the case of the lightweight bicycle parts, whereby the minimisation of mass without sacrificing durability is required.

Fatigue analysis and FEM are used to redesign and optimize a bicycle headlamp bracket in this work. The aim is to create a lightweight-structurally sound bracket by assessing the stresses, optimizing geometry, and optimizing material placement, which can sustain the actual world service loads.

1.2 Problem Statement

Commercial bicycle headlamp brackets often lack fatigue-driven design considerations. Issues such as stress concentration around bolt regions, insufficient stiffness, and premature loosening are common. These problems commonly result from limited structural analysis during product development. High localized stresses caused by abrupt geometric transitions or thin wall sections are especially problematic [7], [18]. Such stress hotspots hasten fatigue failure when subjected to a series of vibration and cyclic loading [22]. Additionally, accessory parts are quite likely to be over-deformed in case they are not sufficiently stiff. Re-design needs of a systematic FEM based one is therefore required to:

- Stress critical regions under both vibratory and static loading will be identified.
- Enhanced fatigue life by geometry changes.
- Avoid unwarranted mass increase through optimization concepts.

1.3 Research Objectives

1. To create a three dimensional CAD representation of the bicycle headlamp bracket assembly.
2. To make static and fatigue evaluations with the Finite Element Method under real loading conditions.
3. To adopt design optimization procedures to reduce weight and increase fatigue performance.
4. To compare and contrast the optimized and baseline design, in terms of structural performance and safety.

1.4 Significance of the Study

The given study is important as it presents a feasible way of enhancing the performance and stability of lightweight bicycle parts in reference to the headlamp bracket. The research demonstrates that small geometrical discrepancies can increase strength, stiffness and fatigue resistance, which is crucial in components of repetitive loads and vibrations by CAD modeling, finite element analysis, and design optimization techniques. By efficiencies of the bracket, weight can also be reduced without sacrificing structural integrity, and these efforts support the aim of the cycling industry to develop lighter and more efficient parts. Moreover, the designed and developed methodology will be a repeatable framework, which can be used on the rest of the bicycle accessories and small mechanical supports and aid engineers to create safer, more stable, and better performing components.

2 Literature Review

2.1 Finite Element Analysis

The modern-day engineering design is systematic in the process of problem identification to production. In this process, computational tools, especially Finite Element Analysis (FEM) and topology optimization have been important in the AM stage and the design refining stage of analysis. FEM allows structural behavior (stress, deformation, fatigue life) prediction prior to physical prototyping, and topology optimization is a systematic method of deciding how to allocate materials in the most desirable manner to achieve performance requirements. All these approaches combined eliminate the expensive build-test-modify cycles, enable quick development at lower cost and produce better designs.

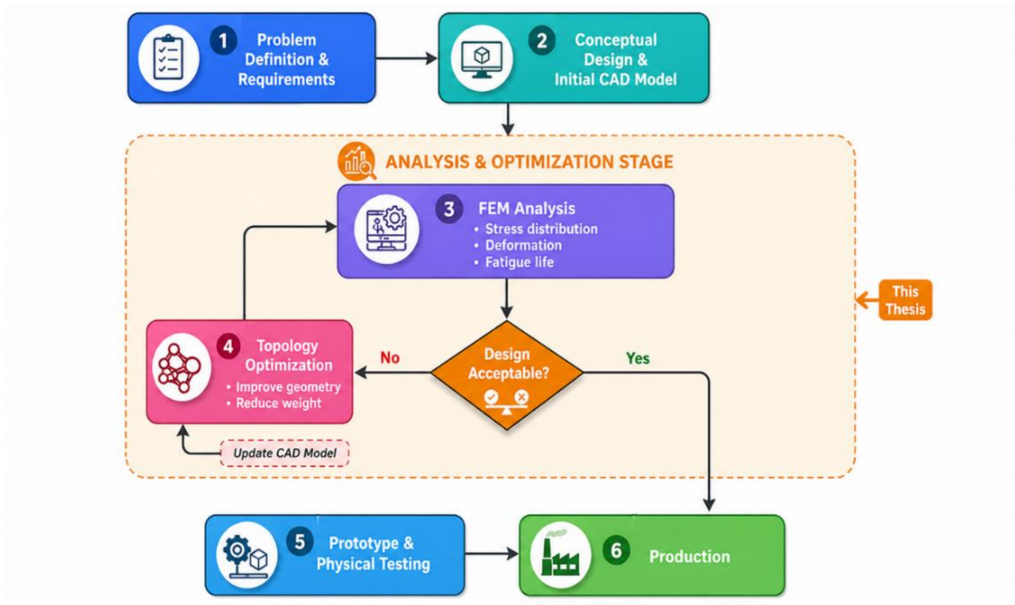

Figure 1. Engineering Design Process with FEM and Optimization

This dissertation uses the FEM-optimization methodology to the design of bicycle headlamp brackets. FEM is used to analyze the baseline geometry to determine stress concentrations and sources of fatigue. The best possible material layout is then produced by topology optimization, with a 22% reduction in peak stress and a 2.5x longer fatigue life with functional performance. The FEM re-analysis of the optimized design verifies the efficiency of computational technique in lightweight design of accessories. This methodology places the present work into the larger engineering design methodology in which simulation and, subsequently, optimization have influences on design decisions, which are made before any physical representation is committed. The engineering design process starts with problem definition and conceptual design, and then CAD modeling. The topology optimization and design analysis stage optimizes the design of components with FEM to optimize the results to enhance the process until desired performance levels are achieved. After being tested computationally, the design goes to physical prototyping and testing then production. This methodology decreases physical prototypes required and allows designing improvements systematically.

2.1.1 Theory and Applications

Finite Element Analysis (FEA) is a numerical method applied in estimating the behavior of structures exposed to any type of load. It separates a complex geometry into smaller units which are connected at nodes to form a discretized system, which is solvable iteratively. FEA is based on continuum mechanics, matrix algebra and energy minimization laws. Examples of the classical sources on stress analysis, strain- Displacement, and the material constitutive laws include classical references, like Belytschko et al. and Hibbeler.

FEA is specifically applied to non-uniform geometries and complicated geometry boundaries and is therefore the method of choice in studying brackets, clamps and small structural elements. FEA is also important in mechanical design to enable engineers to determine the spatial locations of stress concentration, assess deformation, and optimize geometry before actual physical prototyping. Commercial solvers like ANSYS have the capability of having linear and nonlinear solvers, mesh generation algorithms, contact mechanics and the post-processing of fatigue. Simulation of realistic load cases is a major factor that will minimize trial-and-error and result in safer and more efficient designs.

2.2 Fatigue Analysis

2.2.1 Failure Mechanisms and Methods

Fatigue is defined as structural damage which takes place progressively when subjected to repeat loading that is less than the ultimate tensile strength of the material. Possessing a high-stress concentration location, failure usually starts at microstructural discontinuities or geometrical features. Prediction of fatigue life is vital in the design of components (such as bicycle brackets) that undergo constant loading due to vibrations.

Overview of Fatigue Analysis Methods

There are three commonly used fatigue analysis methods which are more or less applicable based on the loading conditions and failure modes:

Stress-Life (S-N) Method

The oldest and most popular method of fatigue analysis is the stress-life method, or the S-N curve method (also known as the Wohler curve method). It is the product of stress amplitude (S) that acts on a component and the number of cycles to failure (N).

Basic Principle: This technique is based on empirical S-N curves delivered by laboratory experiments on smooth specimens that are subjected to constant-amplitude cyclic loading. Generally the relationship is indicated by the Basquin equation:

$$\sigma a = S'f\ (2N)b \qquad (1)$$

Where:

- σa = stress amplitude (MPa)
- S'f = fatigue strength coefficient (approximately equal to true fracture strength)
- N = number of cycles to failure
- b = fatigue strength exponent (typically -0.05 to -0.12 for metals)
- 2N = number of stress reversals

Key Characteristics:

- Valid for high-cycle fatigue (N > 10^3 to 10^4 cycles)
- Makes assumptions about the elasticity of stress response in the component.
- Stress is less than the yielding strength.
- Applies nominal/local stresses of FEM.
- When the mean stress is not zero, it requires a mean stress correction (Goodman, Gerber, or Soderberg).

Applicability to This Study: The approach of stress-life is chosen to analyze the bicycle headlamp bracket due to the following reasons:

1. **High frequency fatigue reigns:** Bike mounted accessories undergo millions of vibration cycles when subjected to normal service life (thousands of vibration cycles per ride x hundreds of rides)
2. **Elastic stress range:** Predicted stresses (82 MPa baseline, 64 MPa optimized) are much less than the yield stress (276 MPa), indicating elastic behavior.

3. **Vibration-like loading:** The major loading is sustained low-frequency vibration but not huge plastic deformations
4. **Availability of material data:** S-N curves for aluminum 6061-T6 are well-established in literature and material databases
5. **Computational efficiency:** Stress-life analysis integrates seamlessly with FEM, requiring only elastic stress results as input

Limitations: The stress-life method does not capture:

- Plastic strain accumulation at stress concentrations (addressed by local stress-strain approaches)
- Crack growth behavior (addressed by fracture mechanics)
- Variable amplitude loading effects (addressed by cycle counting methods like Rainflow)
- Mean stress relaxation under high loads
- However, for preliminary design and comparative evaluation of design alternatives, the stress-life approach provides sufficiently accurate predictions with reasonable computational cost.

Implementation in FEM: In ANSYS fatigue module, the stress-life approach works as follows:

1. Static FEM analysis computes stress distribution under applied loads
2. Critical plane analysis identifies the stress state at each node
3. Mean and alternating stress components are extracted
4. Goodman or other mean stress correction is applied
5. The S-N curve is used to determine cycles to failure
6. Safety factor and life contours are generated for visualization

Strain-Life (ε-N) Method

The strain-life approach, developed by Coffin and Manson, relates the strain amplitude to fatigue life. It is expressed as:

$$\frac{\Delta\varepsilon}{2} = \left(\frac{\sigma' f}{E}\right)(2N)^{b} + \varepsilon' f (2N)^{c} \qquad (2)$$

Where:

- $\Delta\varepsilon/2$ = strain amplitude
- $\sigma'f$ = fatigue strength coefficient
- $\varepsilon'f$ = fatigue ductility coefficient
- b = fatigue strength exponent

- c = fatigue ductility exponent
- E = elastic modulus

Key Characteristics:

- Suitable for low-cycle fatigue ($N < 10^4$ cycles)
- Accounts for both elastic and plastic strain
- Used when localized plastic deformation occurs
- Requires stress-strain response, not just stress

When to Use:

- Components subjected to large loads causing yielding
- Stress concentrations where local plasticity occurs despite global elastic behavior
- Thermal cycling with significant temperature variations
- Low-cycle applications (engine components, pressure vessels)

Why NOT Used in This Study: The bicycle bracket remains in the elastic regime (stress << yield strength), making strain-life analysis unnecessary. The stress-life method is simpler and equally valid for high-cycle elastic conditions.

Energy-Based Methods (Critical Plane Approaches)

Energy-based methods consider the total strain energy density dissipated per cycle as the damage parameter. Examples include:

- Smith-Watson-Topper (SWT) parameter
- Fatemi-Socie parameter for multiaxial loading
- Brown-Miller critical plane approach

Key Characteristics:

- Account for complex multiaxial stress states
- Consider both normal and shear stresses on critical planes
- More accurate for non-proportional loading
- Computationally intensive

When to Use:

- Multiaxial loading with varying principal stress directions
- Torsion combined with bending/axial loads
- Components with complex 3D stress states
- When mean stress effects are significant

Why NOT Used in This Study: The bicycle bracket experiences primarily bending-dominated loading with relatively simple stress states. The added complexity of energy-based methods is not justified for this application. The stress-life method with Goodman mean stress correction provides adequate accuracy.

Table 1. Comparison of Methods and Justification for Selection

Criterion	**Stress-Life (S-N)**	**Strain-Life (ε-N)**	**Energy-Based**
Applicable regime	High-cycle ($N > 10^4$)	Low-cycle ($N < 10^4$)	Both regimes
Stress level	Elastic ($\sigma < \sigma y$)	Elastic-plastic	Any level
Complexity	Low	Medium	High
Data requirements	S-N curve	Cyclic σ-ε curve	Material constants + critical plane
FEM integration	Direct (uses stress)	Requires strain	Requires full stress tensor
Computational cost	Low	Medium	High
Suitability for this study	**✓ Best choice**	Not needed	Unnecessary complexity

For the bicycle headlamp bracket, which experiences high-cycle fatigue under elastic stress conditions with vibration-dominated loading, the **stress-life (S-N) method is the most appropriate choice**. It offers the accuracy required in design making decisions and computational efficiency and availability of data. The approach is supported through substantial validation of aluminum components in high-cycle use, and was shown to easily couple with the FEM workflow used in this thesis. Section 3.9 (Methodology) shows the detailed implementation of the stress-life technique, example of S-N curve parameters, correction factors and the mean stress adjustments.

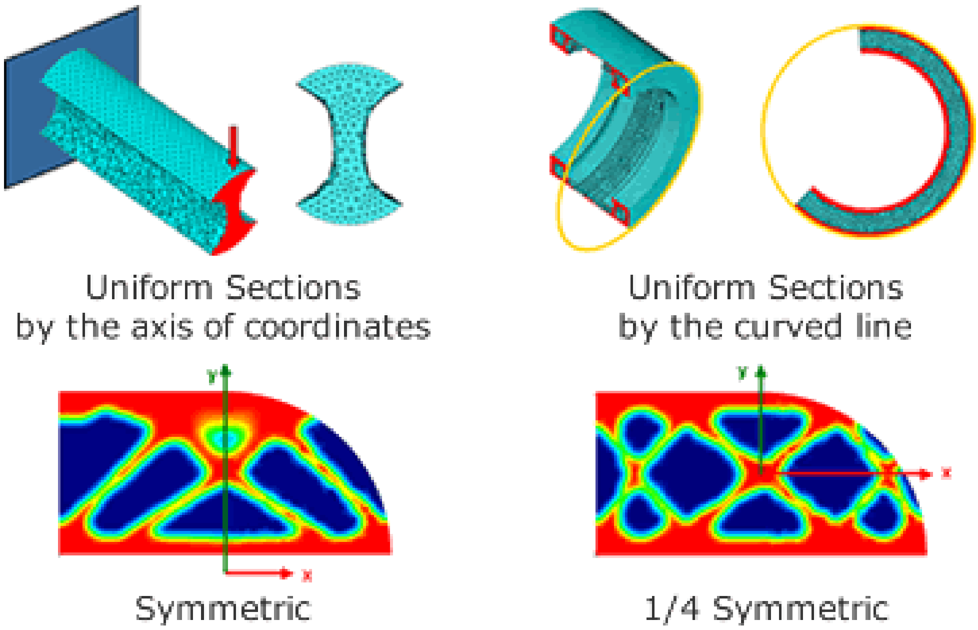

Figure 2 OPTISHAPE-TS - Total Solution for Structural Optimization [6]

In Figure 2, you can see that Topology Density Restriction functions do not only enable you to get the fundamental features of the mirror symmetry and constant cross-sections and leave the efficient part to the stiffness, but also provide high thickability-density the efficient layout of high stiffness by using OPTISHAPE-TS.

The process of fatigue failure usually involves crack beginning, crack escalation and catastrophic breakup. The fatigue life is affected by the surface finish, material defects, and loading frequency. Fatigue analysis placed in FEA tools makes it possible to identify the critical zones, as well as predict cycles-to-fail with multiaxial stress states. In small mechanical connections, hinge arms, and clamps bodies, fatigue considerations are critical to forestall service failure.

2.3 Vibration-Induced Fatigue in Bicycle Components

A bicycle parts, especially those that are attached to the handlebars or to the frame, are prone to dynamic forces caused by unevenness of the road, operator forces and environmental vibration. These periodic loads may be low loads but at high frequencies which results in high-cycle fatigue conditions. Literature on bicycle parts illustrates that stresses in the brackets, clamps and other parts are greatly increased by the vibrations of the uneven surface. The dynamic environment consists of:

- Random road-induced vibration, transmitted through the handlebar.
- Impact loads, such as potholes or curb strikes.

- Oscillatory vibrations, caused by pedaling cadence and rider motion.

Such loading conditions demand that such components as lamp brackets should be sufficiently stiff and fatigue resistant. The modal and harmonic analysis of FEAs is able to determine resonant frequencies, mode shapes, and fatigue-prone areas. Early introduction of vibration and fatigue analysis in the design enhances the service life, safety and reliability of bicycle mounted accessories.

2.4 Introduction to Topology Optimization

Engineering optimization is a methodical mathematical technique of determining the most optimal design using plausible options. Any minimization can be solved by using three elements:

- **Design Variables (x):** Parameters that can be changed (e.g., dimensions, thickness, material density)
- **Objective Function f(x):** What to optimize (e.g., minimize weight, maximize stiffness)
- **Constraints:** Limits the design must satisfy (e.g., stress ≤ allowable, deflection ≤ limit)

Mathematical Formulation:

Minimize: f(x). Subject to:

$$\text{stress} \leq \sigma\_\text{allowable}$$

$$\text{deflection} \leq \delta\_\text{limit}$$

$$x\_\text{min} \leq x \leq x\_\text{max}$$

Types of Structural Optimization:

- **Size Optimization:** Modifies cross-sectional geometry (thickness, area) and also maintains shape constant. Example: different plate thickness.
- **Shape Optimization:** Alters edges or edges and surfaces, but keeps topology. Examples: curve-smoothing, changing a fillet radius.
- **Topology Optimization:** Establishes the best material layouts and connectivity in design space. Is capable of adding/removing holes and forming completely new shapes. Has the greatest design freedom and is applied in this thesis.

Table 2. Comparison of Structural Optimization

Type	Design Freedom	Can Change Layout?	Used Here?
Size	Low	No	No
Shape	Medium	No	No
Topology	High	Yes	**Yes**

This bicycle bracket was chosen to use topology optimization as it offers the most design freedom to give out the most efficient load paths and attain a considerable amount of weight reduction (22% in this study) and still retain structural performance. Topology optimization may also be viewed as a computation-driven method to overlay a given performance goals on the most efficient distribution of materials within a given design space [5]. This procedure has now changed in many ways since the initial truss based advances by Michell in 1904 [16]. Topology optimization has found application in automotive, aerospace, biomedical, and consumer product designs with the advent of modern computing because it can create lightweight, high performance structures [13], [29].

2.5 Topology Optimization Based on Linear Elasticity

Topology optimization Modems Topology optimization is developed by Prager, Rozvany, and Bendsoe who formalized the use of materials in the design of structures [4], [20]. The general process of work has been described as a process of defining a design domain, finite element analysis, sensitivities computation, and the process of updating design variables, which is an iterative one. The most prevalent goal of compliance minimization is based on assumptions of linear elasticity to make calculations easier [5].

2.5.1 Sensitivity Calculation Method

Sensitivity analysis finds out how variations in the density of elements affect the objective function. Mechanisms Density based models including SIMP are based on penalization schemes to promote solid void solutions [5], whereas gradient based solvers are effective to tackle the large scale problems [15].

2.5.2 Topology Optimization Criteria

The Optimality Criteria (OC) method, the Method of Moving Asymptotes (MMA), sequential programming methods, as well as evolutionary methods such as ESO/BESO [9], [30], [33], are common optimization criteria. The advantages associated with each approach are to do with stability, cost of computation and dealing with constraints.

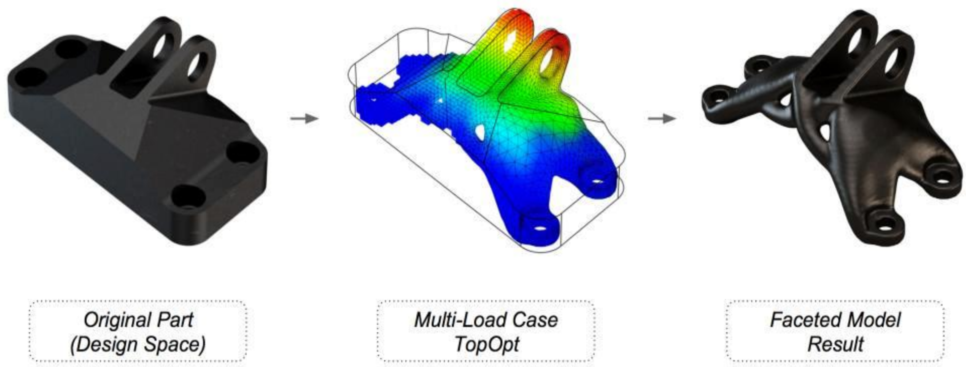

Figure 3. Typical Topology Optimization design steps [15]

It is evident, as demonstrated in Figure 3, that the conventional approach to topology optimization involves the use of finite element analysis to measure the efficiency of the design and generate designs that achieve the following goals; lowest ratio of stiffness to weight; optimal strain energy to weight; lowest ratio of material volume to safety factor and lowest ratio of natural frequency to weight. Conversely, Generative design builds on topology optimization by going a step further to eliminate the requirement of an initial model.

2.5.3 Smoothing of the Solution and Refinement of the Boundary

Raw topography images are normally jagged along the edges or checkerboard shaped. Manufacturability can be enhanced by the use of post processing methods like level set methods, multi-refinement with multi resolution, and mesh adaptation [29]. Density approaches with explicit representation of boundaries are also hybrid approaches to improve smoothness and structural validity.

2.5.4 Optimization of topology with Nonlinear Theory

Nonlinear topology optimization is required in case of large deformations of structures, geometric nonlinearity, or nonlinear material behavior [6]. The variations between the linear and nonlinear solutions may be considerable, particularly in the case of components which have slender characteristics, or which have high stress gradient components (Figure 3). Contemporary nonlinear solvers enable more precise optimization of elements which experience impacts, bending or cyclic forces.

2.5.5 Topology Optimization with Stress Constraints

Stress constrained optimization: This is one of the most difficult issues of structural optimization. Resistant to stability in solutions is due to stress singularities, local character of stress response and lack of linear dependence on design variables [10], [24], [33]. Aggregation functions and regional grouping are some of the techniques which can be used to overcome these difficulties and relaxation strategies.

2.5.6 Variations of Topology Optimization

Several variations extend topology optimization to more realistic engineering scenarios:

- Continuous density variables [5].
- Multi-objective formulations balancing stiffness, mass, and fatigue [13].
- Multiphysics optimizations involving thermal or vibrational loading [29].
- Manufacturing-constrained optimization for additive or subtractive processes [25].
- Material property variations for multi-material or composite designs [8].

2.5.7 Advantages of Topology Optimization

Key advantages include weight reduction, performance enhancement, design automation, and improved material efficiency [5], [13]. These benefits directly apply to bracket-type components, where stiffness-to-weight ratio is critical.

2.5.8 Topology Optimization and How It Works

The process of work is usually initiated by an entirely solid space. After repeated evaluations of geometry via FEM and sensitivity material removal the geometry is developed into an efficient load carrying structure [23]. The post processing gives manufacturable shapes, which can be used to be converted to CAD (Figure 4).

2.5.9 Applications of Topology Optimization

Topology optimization has been widely used in aerospace, automotive, structural engineering, biomechanics, and consumer products [13], [20]. For bicycle applications, optimization has been applied to frames, stems, and small structural brackets [21].

Figure 4 reviewed that basic idea behind topology optimization is to define a design space and then mesh that with a very regular array of elements. In some cases, this will be an arbitrary 3D or 2D space, in other cases, as shown in Figure 1, the mesh will follow an initial scheme. Also implications of this distinction in Part 2 were observed. The boundary conditions and loading are defined as normal in an FEA solution and analysis starts with the full design space as shown in Figure 1. Material is progressively removed using a target volume reduction, until a final iteration, as shown above. The final design is assumed to be optimized to a defined level of efficiency at that target volume.

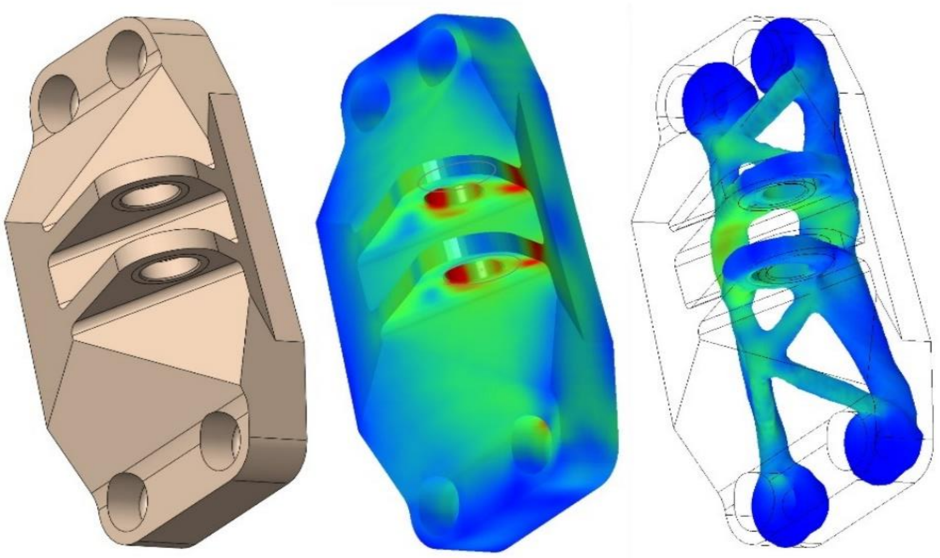

Figure 4. Top, initial design space, middle, initial analysis and bottom, final analysis. [23].

2.6 Simulation Tools

2.6.1 Detailed Simulation

High-fidelity FEA software such as ANSYS Mechanical enables nonlinear, fatigue, and detailed stress evaluations [2], [3].

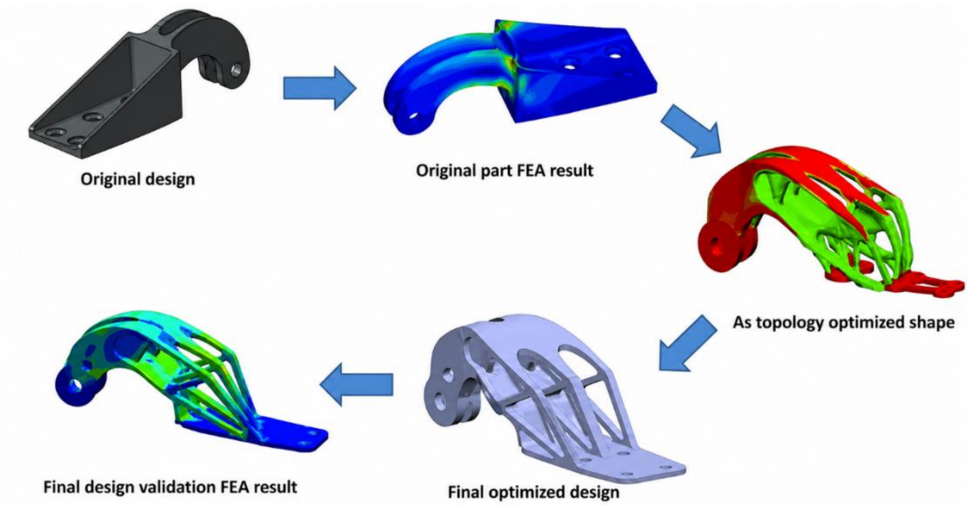

Figure 5 Representative engineering applications of topology optimization. [12].

Figure 5 shows that virtually, the topology optimization software applies pressure on the design from different angles, tests its structural integrity, and identifies unnecessary material. Validating the design, it involves determining a threshold for the element density field between a value of 0 and 1. A value of 0 voids material in a designated region of the structure, while a value of 1 sets the designated region as solid material. The designer can then strip the model of all unnecessary material and finalize the topology optimization portion of the design [15].

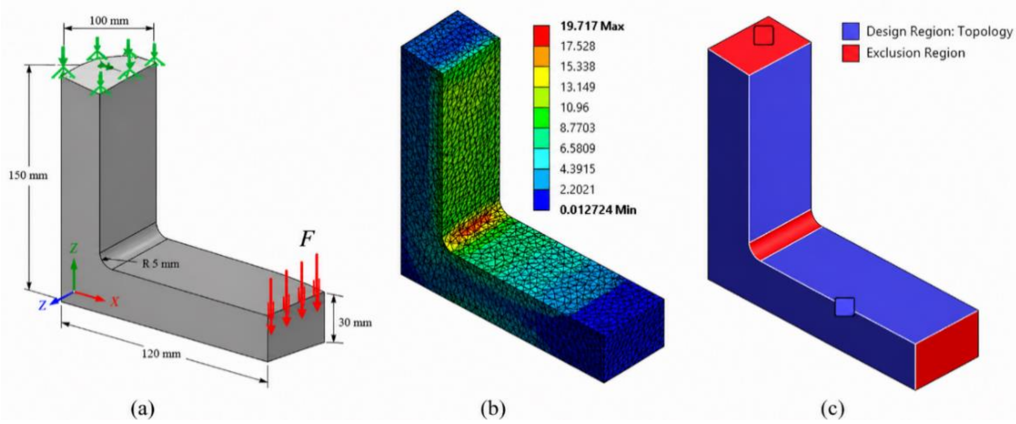

Figure 6. General research workflow for bracket modelling, FEM analysis, and optimization. [5].

In the process of the thesis Fig. 1.5 showed that the general research workflow for bracket modeling, FEM analysis, and optimization is also involved in systematic process spanning initial design to validation of the final, optimized structure. This cyclical approach, often leveraging specialized software, aims to achieve desired performance objectives, such as minimizing weight or maximizing stiffness, while adhering to constraints. [5]

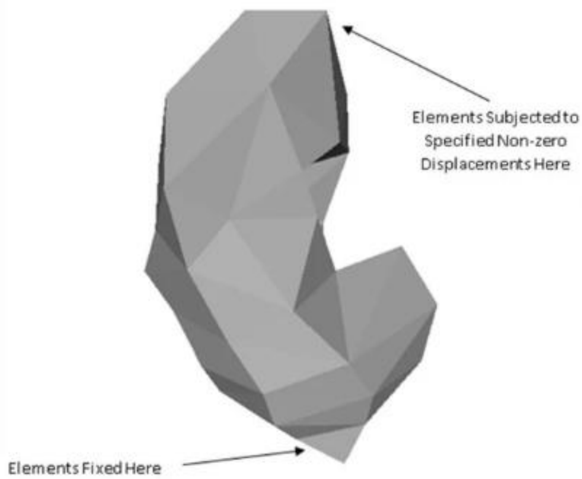

Figure 7. Comparison of optimized geometries under linear and nonlinear analysis.

With the diagram displayed above in figure 7; it clearly shows that with regard to boundary conditions, for each of Problem 1, Problem 2, and Problem 3, some portion of the surface of the kidney was assumed to be fixed while a known displacement was assumed to be applied on the surface of the kidney at some other location. The geometry of the kidney was discretized into 3D elements using the software package ANSYS. The element type used was Tet 10node 187. The geometry was discretized into 782 nodes in total. The element numbers for the elements at the location where the kidney was fixed were observed to be 8, 15, and 24.

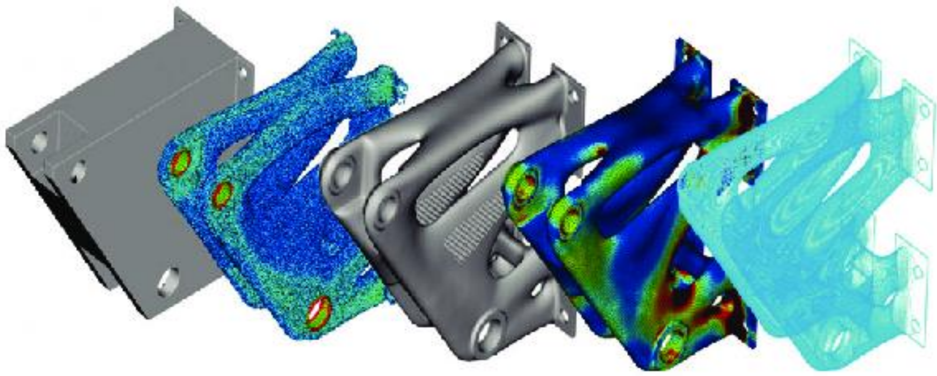

Figure 8. Typical topology optimization evolution from initial domain to optimized CAD geometry. [8]

Figure 8 clearly shows that as computer-aided design (CAD) continues to evolve and advanced manufacturing techniques like 3D, its printing become more widespread, making it possible to create complex parts easier than ever before, designers and engineers can leverage topology optimization software to push boundaries and find new ways to maximize design efficiency. [8]

2.6.3 Engineering Optimization and Problem Formulation

Engineering optimization provides the mathematical foundation for systematically improving a design by defining objective functions, design variables, and constraints. Before selecting a topology optimization method, it is essential to understand how engineering problems are formulated in a general optimization framework. Classical engineering optimization begins with specifying design variables that describe the system, such as geometric dimensions, material parameters, or topology descriptors. An objective function is then defined for example, minimizing mass, maximizing stiffness, or improving fatigue life. Constraints typically include stress limits, deflection limits, manufacturing restrictions, and volume fractions. This structured approach is detailed extensively in the works of Wheeler and Kochenderfer [22] and Martins and Ning [23], who outline both gradient-based and gradient-free optimization strategies.

These texts underline the fact that optimization issues of the structure can be classified into three significant categories, namely: size, shape and topology optimization. The parameters of cross-section geometries are optimized by size optimization, geometry by shape optimization, and material distribution itself by topology optimization. It is the most robust to find purely new structure configurations, and thus suits lightweight design of bracketry. This hierarchy is crucial since topology optimization frequently extends the models and formulations found in more fundamental engineering optimization. Another way by which topology optimization is efficient in topological optimization of large finite element models is the use of sensitivity analysis and adjoint methods as indicated by Martins and Ning [23].

Topology optimization algorithms such as SIMP, ESO/BESO and level-set methods can be easily generalized in this broader formulation by making use of element densities or boundaries as design variables. Engineering optimization can thus be seen not only as a theoretical foundation but also as a practical method of gaining insights into the dynamics of structural layouts changing in the course of optimization.

2.6.4 Topology Optimization Methods

An extensive variety of techniques have been created to do topology optimization each with unique strengths with respect to the type of design problem, computational resources and needed structural behavior. The most effective methods are briefly described below in order to give a clear picture of the techniques available.

SIMP (Solid Isotropic Material with Penalization): SIMP is the most common formulation used in topology optimization in practice in engineering. The element density variables are continuous values that take on the values of 0 and 1 to represent the material. To force intermediate densities to solid-void states, there is a penalization factor. SIMP has been popular because it is a simple method, readily compatible with finite element analysis, and efficient in solving large-scale problems.

ESO and BESO (Evolutionary Structural Optimization/Bi-directional ESO): ESO gradually eliminates the inefficient material using the stress or strain energy criteria. BESO goes a step further in letting material be added as well as removed to create more stable and flexible evolution of the design. The techniques are intuitive and not completely dependent on sensitivity analysis to use and are thus simpler to apply to designers. They, however, can be slower in calculations at fine mesh resolutions.

Level-Set Methods: Level-set methods are through the lens of which structural boundaries are represented by implicit function that is updated based on sensitivity information. These methods intrinsically give smooth edges and are convenient in any problem that deals with a complicated geometry evolution. Level-set algorithms are mathematically beautiful and are more complex to apply numerically and have to think more about boundary conditions.

Moving Morphable Components (MMC): more of a recent approach, where geometry is modeled in terms of a set of predefined components the geometry of which can be varied in terms of shape, location, and size during optimization. MMC locally minimizes the number of design variables and also facilitates manufacturing constraints in a more natural way. This is a way of mediating between topology optimization on the one hand and parametric CAD modelling on the other hand.

Density Filtering and Projection Techniques: Removing numerical instabilities like checker boarding or mesh dependency require density-based techniques to use filtering techniques. Projection techniques also enhance definition of solid and void region. These techniques will provide stable convergence and more manufacturable outcomes.

Gradient-Based vs. Gradient-Free Methods: Gradient-based algorithms are usually employed in topology optimization as these approaches are more efficient and accurate on large-scale optimization problems. Gradient-free methods (e.g. genetic algorithms, particle swarm optimization) can be used on smaller problems, or on highly non-convex design spaces, but are typically dramatically more expensive to implement.

All these methods are combined into giving an extensive selection of tools to use in solving the structural optimization problems. The choice of the technique is determined by the design goals, precision required, computation capabilities and physical limitations of the engineering problem.

3 Bicycle Headlight Bracket CAD Modelling

3.1 Overview of the CAD Model

The CAD model is a full body representation of the bicycle headlight bracket (considering all parts constituting the mounting system). Prior to performing any simulation work I had to ensure that the shapes, curves, and interfaces were well-defined. This is significant since the way the initial geometry is formed determines the quality of the subsequent analysis. The lamp housing, the hinge links, the clamp section, and all pins and screws of the hinge links are contained in the model. Adding fillets, transitions, and wall thicknesses was done in such a manner that designed sharp stress points are avoided and a manner that the way the actual component is manufactured is replicated.

3.2 Assembly Layout and Component Arrangement

The complete equipment is assembled to form two large parts, the lamp housing mounted on the top and the clamp mounted on the bicycle handlebar. The two are attached with a series of hinge arms and pivot pins to enable the lamp to tilt either upwards or downwards. This layout concept is such that the weight of the lamp is moved once into the body of the clamp without necessarily having to lose the capability of the rider in adjusting the lighting angle. The supporting plates and side ribs also hold the hinge area that is usually where these small brackets handle the routine vibration and cyclic loads as they are in use.

3.3 Dimensional Considerations and Geometry Choices

The dimensions of the portions were picked according to popular bicycle handlebar sizes and available before and after the handlebar section. The wall thicknesses were maintained in the middle range to maintain low weights and yet be thick enough not to crack during tightening. Sizing of the ribs and tabs was used to enhance the stiffness without going overboard with bulkiness. The radii were placed in fillets wherever sharp corners appeared where there is a sudden change of direction to avoid stress concentration at sharp corners. These details make the model more real-world when it is further used to analyze it by finite elements.

3.4 Base, Tabs, and Reinforcement Features

The clamp body is at the base of assembly, to which the whole structure is attached to the handlebar. In order to ensure that the tightening area of the clamp is made even, some reinforcement tabs were added around the tightening port. Such tabs serve to spread the load between the screws in order that the clamp will not be deformed or cracked when the user exerts force. The stress on the bolt holes is also diversified by the support block at the clamp base, which eliminates the chance of fatigue. Small assistant plates were placed around the hinge area to ensure that the upper housing remains stable as well as prevents undesired twisting in the process of use.

3.5 Exploded View Description

To depict all the parts individually and reveal the sequence of assembling the parts better, an exploded diagram was drawn. The same perspective assists in determining the way each component interlocks and to provide sufficient room to move around and to install the screws and pins. The model that was blown up also helped to ascertain that the hinge arms and adjustment arm are free enough to swing in full and not to collide with the hosing and the clamp.

Below is the numbering of the parts in the exploded view.

1. Upper Lamp Housing
2. Lamp Housing
3. Battery Contact/Terminal
4. Primary Battery
5. Lower Housing Insert
6. Dual Hinge Links (x2)
7. Reflector
8. Headlight
9. Bolt
10. Nut

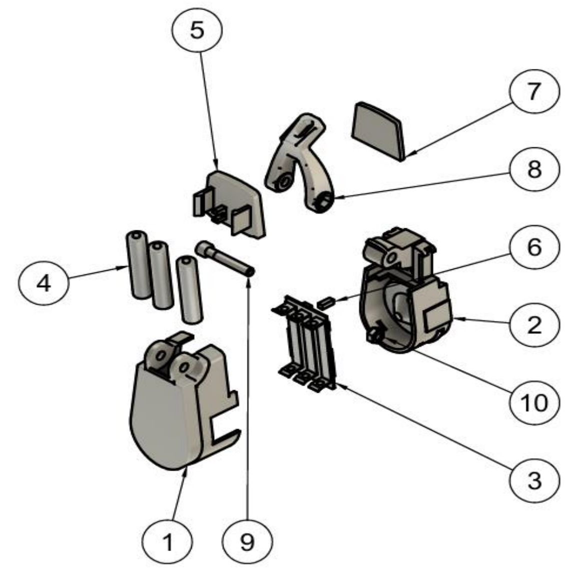

Figure 9. CAD Diagram Showing Exploded View Description

Upper Lamp Housing (Part 1): The lamp is primarily covered by this housing. It contains the lamp and is the upper section of the whole unit of the headlights. I bent it so the lamp would fit well in it, and also so that it is easy to fit into the hinge links. The edges are curvy; they allow no sharp edges and help lessen the stress of the bike on the bumpy roads. Within, space is sufficient to accommodate the lamp and wiring.

Material: ABS or Polycarbonate, as both are light, strong and can be used outdoors.

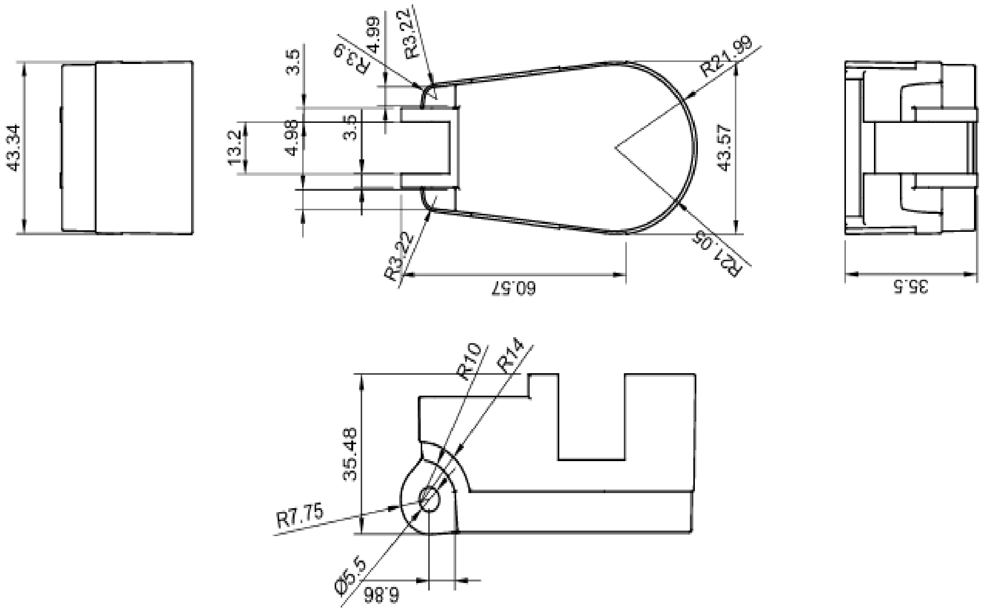

Figure 10. Upper Lamp Housing CAD Drawing and Dimensions

Lamp Housing: This is the bulk portion of the lamp and in this case, an opaque view depicts both the outside form and the internal support system. It is the central part of the headlamp, and it has three important functions: to host the lamp and LED cavity, and place its internal components, and the hinge mount. The design is such that it fits together well and that it works dependably.

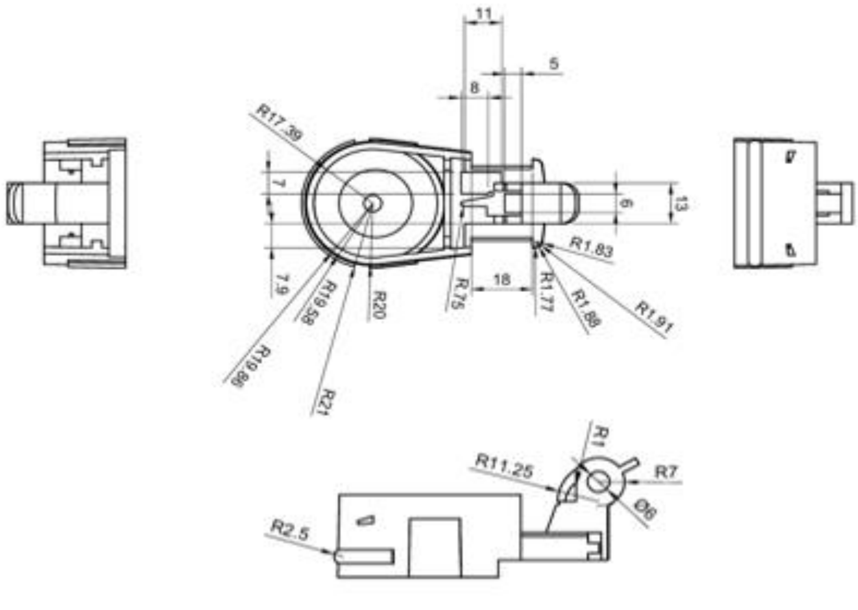

Figure 11. Lamp Housing CAD Drawing and Dimensions for Part 2

Battery Contact/Terminal (Part 3): This is the area where the battery is placed. It is mainly used to offer a strong electrical conduction between the battery and the device. To have a snug fit with the battery, I made the contact reliable and consistent.

Material: This is made of steel, which is primarily to enable good conductivity and durability. It might also be suitable for better electrical performance depending on the application with a copper or brass part which has a protective coating.

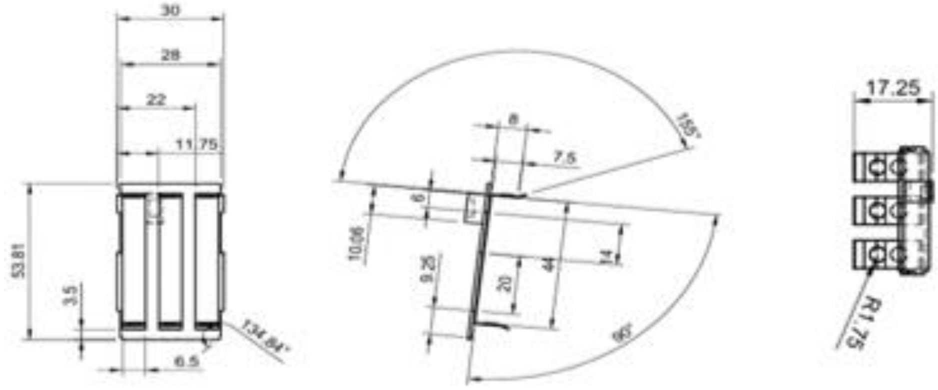

Figure 12: Battery Contact/Terminal CAD Drawing and Dimensions Part 3

Primary Battery (Part 4): This is a primary (non-rechargeable) battery to power the device. It also fits into the battery compartment and make a good contact with the terminals to guarantee a good operation. It has a casing composed of steel, which is durable and conductive. The battery must avoid corrosion and wear as it can be installed and removed several times. To ensure safety and reliability in the operation of the device, proper alignment with the terminals is essential. The design is flexible in that it can be easily inserted and removed without breaking the contacts.

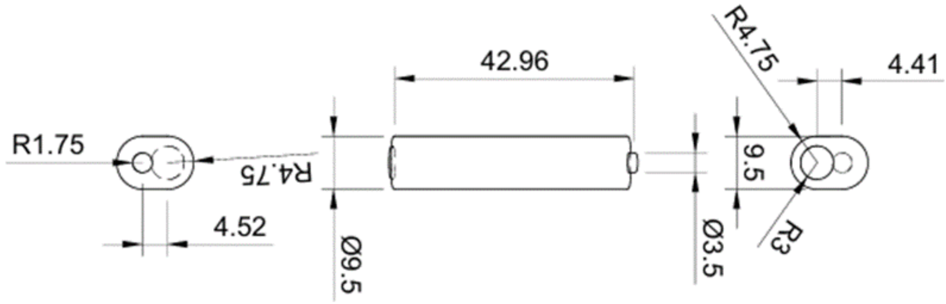

Figure 13. Primary Battery CAD Drawing and Dimensions Part 4

Lower Housing Insert (Part 5): This component makes up the bottom part of the lamp housing that surrounds the lamp and provides room to the electronics. It can also play the role of a switch and a reflector.

Material: ABS or polycarbonate in order to be durable and impact resistant.

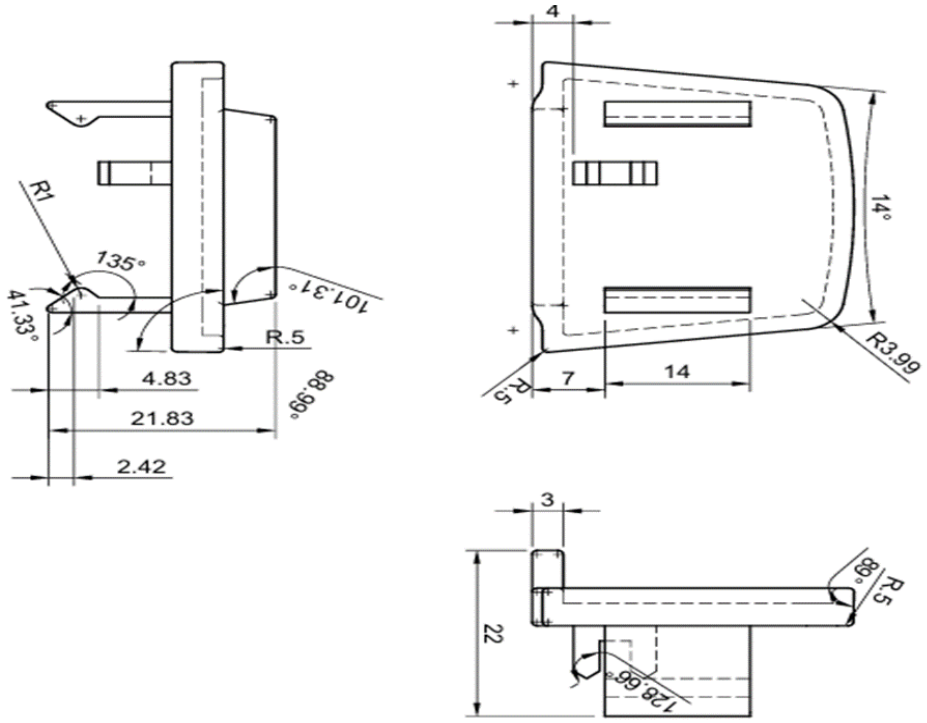

Figure 14. Lower Housing Insert CAD Drawing and Dimensions Part 5

Front Lower Reflector (Part 6): This type of Reflector rests in front of the main light, just below it and bounces light uniting it on the switch. Constructed out of polycarbonate to create durability and light control.

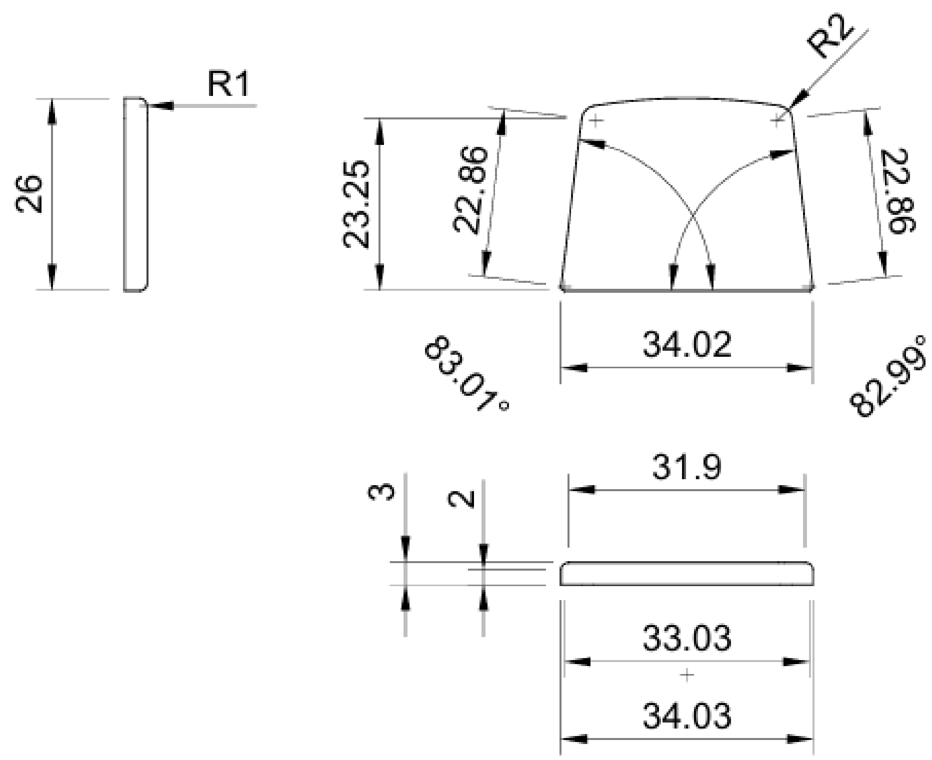

Figure 15. Front Lower Reflector CAD Drawing and Dimensions Part 6

Handlebar Clamp Body (Part 7): This is what actually clamps on to the bike handlebar. It supports the whole weight of the lamp and restrains vibration in the road. I made the curve on the inside to fit the size of a standard handlebar. I had also included a few support ribs to ensure that the clamp does not flex too much when tightening the screws.

Material: Impact resistant, low weight and manufacturability ABS Plastic (Injection Moulded).

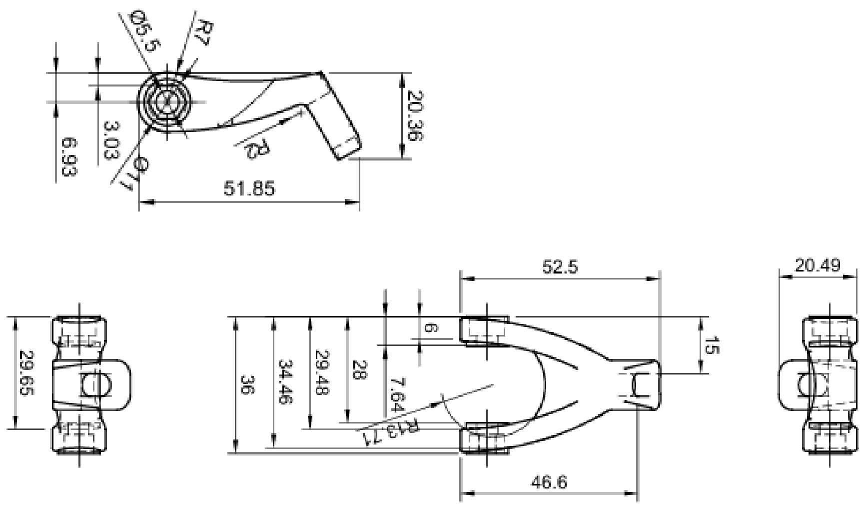

Figure 16. Handlebar Clamp Body CAD Drawing and Dimensions Part 7

Bolt (Male Fastener) (Part 8): This bolt is used together with the nut to fasten the clamp and the rest of the parts. Relaxing it permits the change of the angle of the light, and tying it provides stability.

Material: Stainless steel is strong and resistant to corrosion.

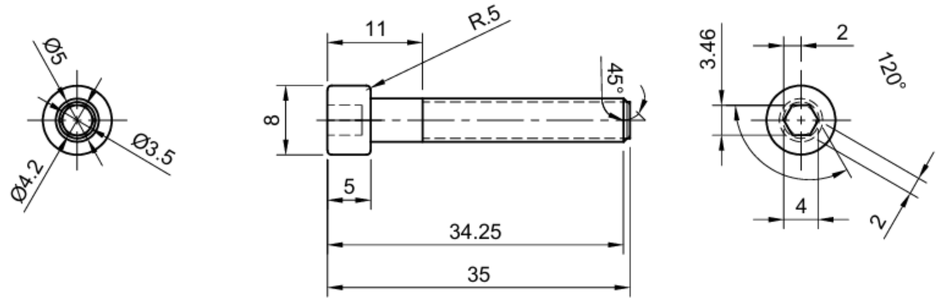

Figure 17. Bolt (Male Fastener) CAD Drawing and Dimensions Part 8

Nut (Female Fastener) (Part 10): This nut pairs with a bolt to secure the clamp and other parts. Loosening it lets you adjust the light’s angle, and tightening it locks it in place.

Material: Stainless steel for strength and corrosion resistance

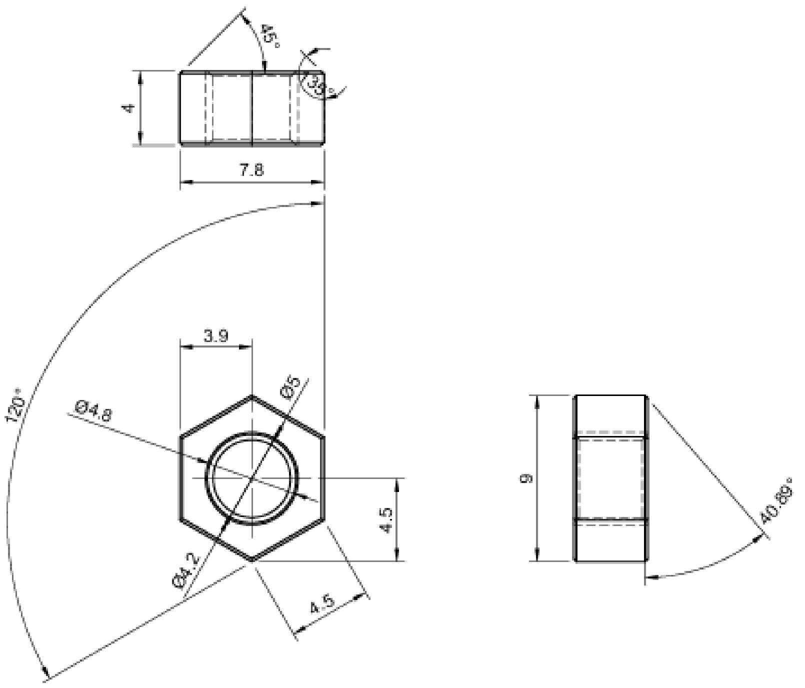

Figure 18. Nut (Female Fastener) CAD Drawing and Dimensions Part 9

4 Methodology

4.1 Research Design and Approach

It is based on a simulation-based design process that combined computer-aided design (CAD), finite element analysis (FEA), and topology optimization that resulted in the creation of the improved bicycle headlamp mounting bracket. This organized, iterative process is indicative of the current style of computational structural design whereby the trial-and-error cycles in physical design are substituted with costly numerical simulation. What was needed was to determine the structural constraints of the current bracket, and to enhance its functionality methodically and by informed geometric change, and to retain all the functional specifications. Nonetheless, it should be noted that the methodologies that are purely simulation-based have implicit assumptions. The precision of FEM predictions relies on the accuracy of the boundary conditions applied, the model of materials and the mesh resolution which are all idealizations of a much more complex physical reality [9], [10]. Nevertheless, simulation-based approach is most feasible and cost-reducible in case of iterative design improvement at component level.

4.2 Development and Assembly of the CAD Model

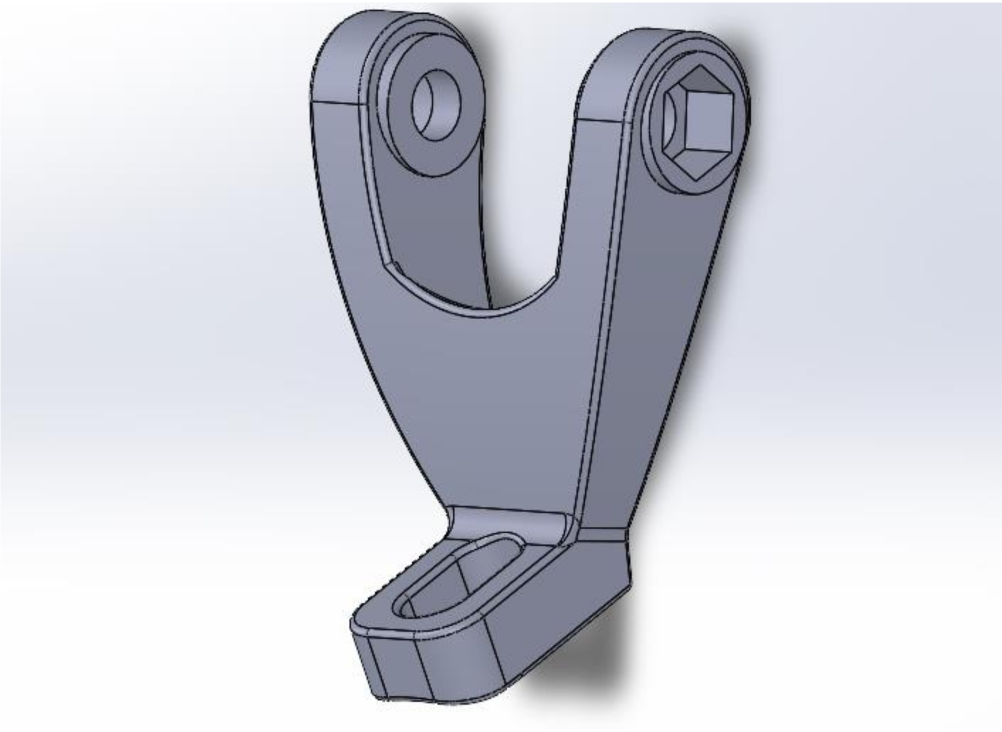

Figure 19. CAD model of the original bicycle headlamp mounting bracket

The first step entailed the recreation of the original bracket geometry with SolidWorks. This physical model is a representative of the basis and will be referenced in all further analyses. The modelling process in question was concerned with the properness of all functional areas of the bracket such as the clamp interface, hinge arms and the connecting neck area. Special consideration was placed on the neck which is the main transfer path of loads between the handlebar and the headlamp and hence should control the structural behaviour of the component, which is in line with classical stress concentration theory that was developed by Pilkey and Pilkey [7].

There were a few geometric simplifications made where needed to enhance computational efficiency but all the features in loading it were maintained so as not to compromise the simulation results to reflect that of the real component. This model is consistent with the practice in FEA based-design studies, which needs the geometric idealization to be judiciously weighted against the threat of missing structurally important parts [12]. The plastic used on the model was Acrylonitrile Butadiene Styrene (ABS) which was used because it is widely used in consumer goods of light weight. All it was presupposed was that the material was homogeneous, isotropic, and linear-elastic within the operating range of stress, which, though typical in the preliminary design analysis, possibly simplifies the behaviour of polymeric materials subjected to dynamic loading, particularly the viscoelastic and rate-dependent behaviour [8].

4.3 Identification of Structurally Critical Regions

Before making the actual finite element simulation, a qualitative evaluation of the geometry was conducted to provide areas that are likely to experience stress concentration. According to traditional rules as outlined in Pilkey and Pilkey [7] and Hibbeler [10], steep lines, thin section and load-eccentricities are the major sources of concentration of local stress. The most structurally vulnerable area was found to be the neck region, as it is comparatively less cross-sectional, and its location at the major load path. This pre-simulation test is not a strict formality. Because, as Dowling [8] reminds us, the skill of forecasting failure points through geometrical arguments is an essential complement to numerical simulation, the outcomes of FEA calculations should, of course, be subjected to a reasonable amount of engineering judgment. This is a quantitative dependence on the results of the computations, which is dangerous unless accompanied by qualitative information about the interpretation of artefacts of the mesh or the boundary conditions as actual structural behaviour.

4.4 Material Assignment

The bracket was designed in ABS plastic, a thermoplastic polymer that has a good strength-to-weight ratio, is simple to make, and provides sufficient impact protection, making it a popular choice in consumer-scale mechanical components. Material properties were appropriate: the Young's modulus, the ratio of Poisson, and the yield strength, which were assigned in accord to the standard published figures of ABS as cited in Callister and Rethwisch [16].

Here, there should be a critical note over the selection of materials. ABS has a yield strength of about 35-45 MPa under quasi-static loading (conditions) of the material in material) which though sufficient in practice when loaded in the consumer environment, offers a reasonably narrow safety margin when loaded with dynamic loading and vibration as detailed in Section 2.3. Moreover, ABS can creep during continuous loading and its toughness is lower at lower temperatures, which are prevalent conditions at outdoor cycling conditions. These restrictions indicate that, though ABS is suitable in preliminary design study, the Bracket structural reliability in long term worth is to be checked in a wider scope of material candidates like aluminium alloys and glass-filled polymers, as stated by Schmid, Hamrock and Jacobson [14].

4.5 Preparation of the CAD Model for Finite Element Analysis

Prior to the FEM simulation itself, the CAD model underwent modulations towards compatibility with the ANSYS Mechanical solver environment [12]. This was done by subduing non-load bearing cosmetic features, demarcating contact areas amidst assembly parts and fitting proper mesh controls. The discretization error involved in conversion of CAD geometry to a finite element mesh is a factor of both the element type and the mesh density used [9].

In order to ensure consistency between the material assignment and the model, all the model domains were set to the same ABS material properties. The mesh seeding procedure was optimised on the stress concentration points that were expected to occur especially on the neck and clamp interfaces according to the principles of adaptive mesh-adjustment proposed by Wang, Liu, and Wen [22]. This active refinement is a strategy that minimizes the use of post-hoc mesh convergence studies but convergence verification was still conducted according to the description of Section 3.7.

4.6 Static Structural Analysis

4.6.1 Boundary Conditions and Load Application

A baseline design was also analyzed using a static structural analysis to determine its performance. The clamp area was stationary to represent the connection to the bicycle handlebar and the load was applied to the headlamp mounting interface. The magnitude of a load of 50 N was chosen to analyse it. Even though it is assumed that the service load is in the range of 20-30 N, the box that is adopted with a higher load offers a pessimistic evaluation of the structure. This practice coincides with the philosophy of the safety factor, expressed in Budynas and Nisbett [15], which advises that preliminary design analyses should include factors that amplify loads to consider uncertainties in dynamic service settings.

A weakness with this approach should however be noted. One of the unsteady load cases would be incapable of completely describing the workings of the real bicycle because of the dynamic and multidirectional vibration loads that occur during operation as in Section 2.3. It is then possible to consider the static analysis as a rather necessary though not a sufficient prerequisite of design validation. The extensive fatigue test under realistic variable-amplitude loading is an important parameter to be considered in future works, as Dowling proposed [8], and Covill et al., affirmed [17], who emphasized the importance of dynamic effects in the structural assessment of bicycles.

4.6.2 Results of the Original Design

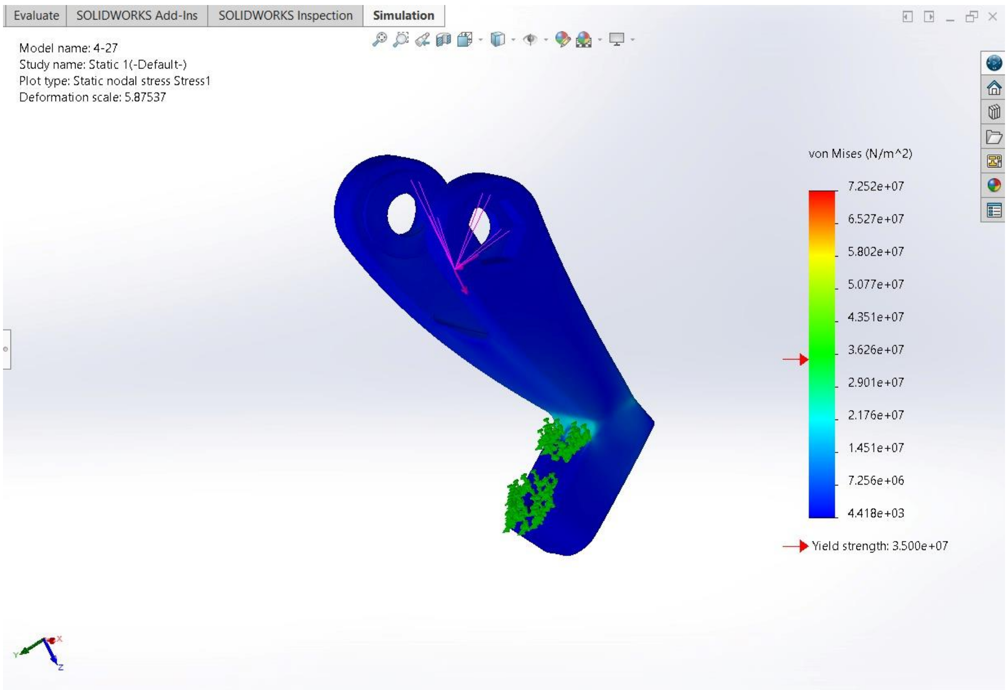

Figure 20. Stress distribution of the original bracket under a 50 N load

According to the analysis of the baseline, it was clear that the stress concentration was high at the neck area of the bracket. This observation can be explained by the fact that the geometric vulnerability was identified in Section 3.3 and by the fact that the current geometry lacks the ability to evenly distribute the forces applied to it, leading to localized stress developments and high propensity to fatigue onset. According to Belytschko, Liu and Moran [9], high gradient of stress in particular, is very sensitive to mesh density and element formulation hence the significance of convergence study as detailed later.

4.7 Mesh Convergence Study

A mesh convergence study was done to work out the reliability of the numerical results. The model was solved with the levels of mesh density refined and the maximum values of stress were compared. It was noted that the stress values approached a constant with an increase in the mesh refinement as there was not much difference between the fine and the very fine mesh results. This ensures that the accuracy of the mesh selected is good enough to be used in further analyses as per the convergence points outlined in Zienkiewicz and Taylor [30].

This convergence exercise is not just a check formality. It is a straightforward solution to a very basic problem in FEM that the numerical solution is an approximation that depends on mesh resolution [9]. To the extent that reported stress values are qualitative, only at best and misleading at worst, without proven convergence the reported stress values are useful at best and can be quite misleading at worst, especially when structural safety margins are near design-critical.

4.8 Topology Optimization and Design Development

4.8.1 Development of the Baseline Design Space

After the structural inefficiency of the baseline design was detected, the topology optimization came out as a way to enhance the material distribution. Instead of altering the original geometry, a design space was created, an enlarged volumetric envelope containing the bracket where the bracket could be rescribed that the optimization algorithm could move material freely to approach the most efficient load paths. It is based on a typical SIMP (Solid Isotropic Material with Penalization) framework [4] formalized by Bends oe and critical reviewed by Rozvany [3].

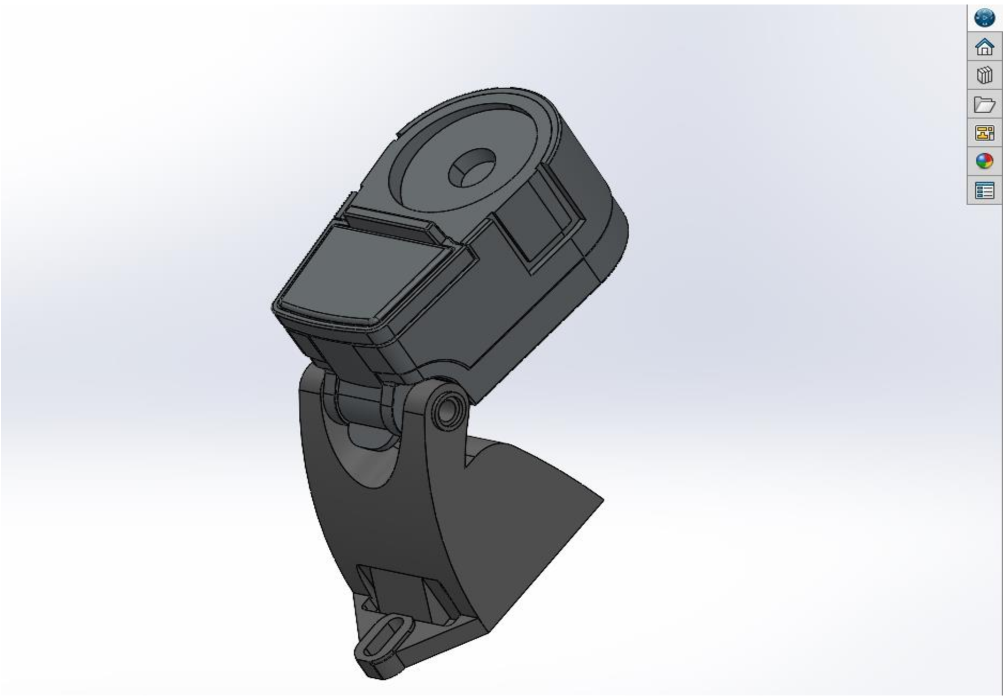

Figure 21. Baseline CAD Model of the Bicycle Headlamp Mounting System

When establishing the design space, the functional areas of the bracket were retained, in the process ensuring that the optimized design is still functional. These consist of the mounting interfaces, hinge position as well as the clamp locations. Meanwhile, more material was added about the structure to give redistribution of material enough flexibility to the optimizer. Importantly, the design space was specified in such a way that the rotational motion of the headlamp at the hinge was never constrained this constraint an indication of the functional demands of the assembly, that when breached would make the optimized geometry impractical, despite any structural efficiency merit.

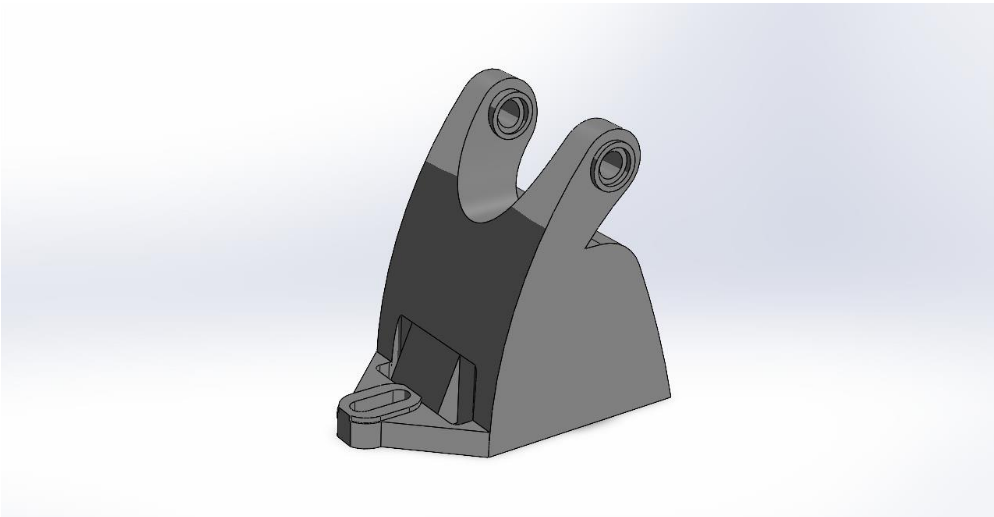

Figure 22. Baseline design space used for topology optimization

4.8.2 Topology Optimization Setup and Objectives

Having the design space defined, there is a topology optimization study that was formulated in SolidWorks Simulation environment. It was given material ABS plastic in order to stay consistent with the physical component and the same boundary conditions utilized in the previous structural analysis were kept. The loading condition was realized as a distant force on the headlamp mounting area, it was resolved into components of about 30.35 N in both vertical and horizontal containing a summative outcome of 50 N. The goal was the minimization of mass with structural stiffness and a target of about 80% material removal.

This has the formulation, minimizing mass subject to compliance bounds, which is one of the most classical problems statements of structural topology optimization [1], [5]. Nevertheless, one must note a constraint of this method: compliance minimization does not restrict the stress explicitly. Structurally safer design is more a result of stress-constrained topology optimization as shown by Bruggi and Duysinx [6] and Collet et al. [26] at more computational complexity. In the context of the present work, the compliance-based formulation was considered to be adequate due to the preliminary character of the design analysis, however, a stress-constrained minimization would be a valuable improvement of the design analysis in the future.

4.8.3 Interpretation of Optimization Results

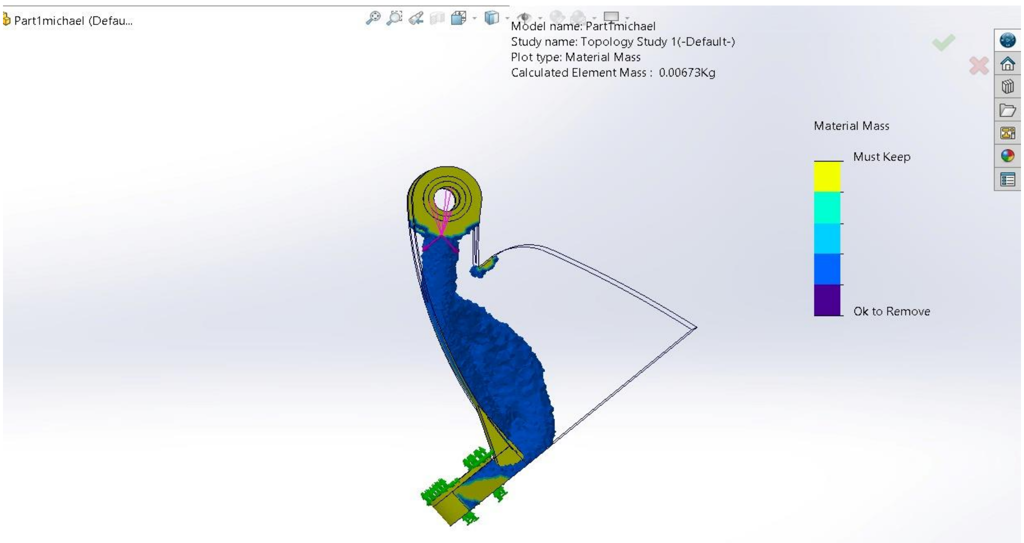

Figure 23: Topology optimization result showing material density distribution

In the topology optimization process, the structural contribution of every element in the design space was evaluated iteratively on implementation. The map of material density distribution was obtained as the result, which easily displays the main load paths in the bracket. Regions of high density denote those areas in structural performance which are important whereas low-density regions show those materials which can be eliminated without much adverse impact on stiffness. This interpretation step is essential to the topology optimization workflow, with the original density product being a probabilistic model of material need instead of a geometry that can be used directly [1], [23].

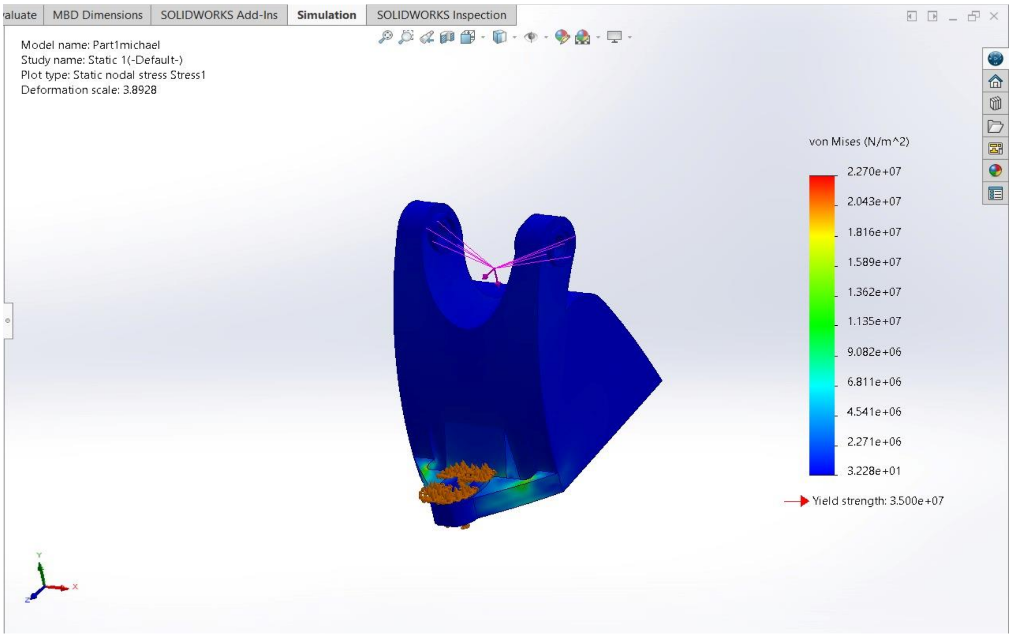

Figure 24. Topology optimization density plot

4.8.4 Geometry Reconstruction

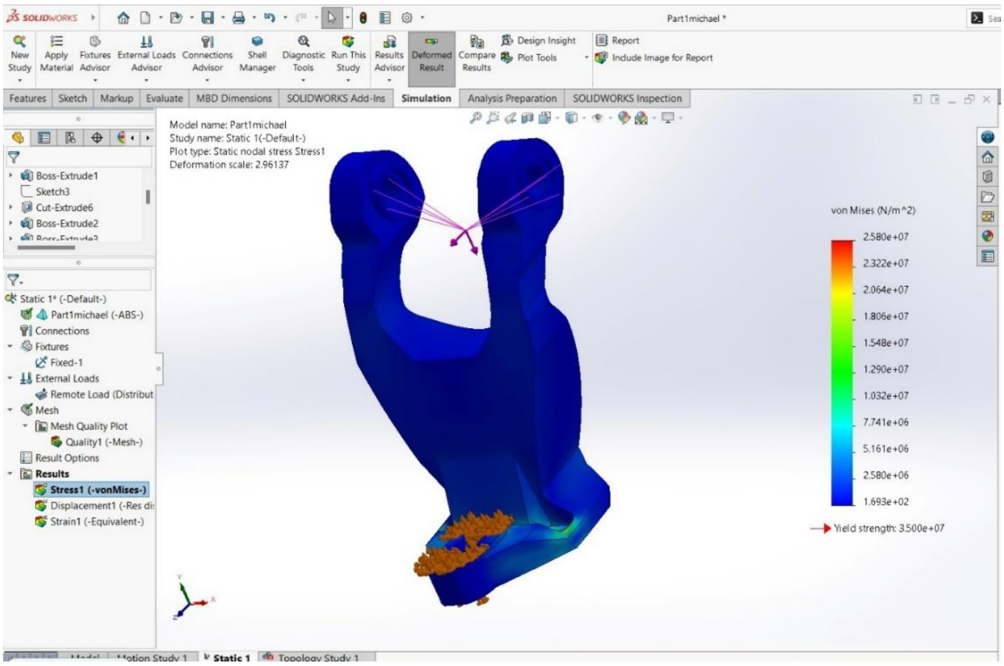

Figure 25. Reconstructed optimized bracket geometry

The solution of the topology optimization is in itself non- manufacturable, a representation of an idealised material distribution and not a real geometry. As a result, it needed a reconstruction mechanism to reduce the optimized output into a practical design. The density plot was then taken out of the smoothed mesh and utilized as a reference in the redesigning of the bracket in the CAD view. In reconstruction, selections of material were retained along identified load paths as required, and unneeded geometry was eliminated with geometric operations like extruded cuts and reference based changes. Caution was exercised to prevent a sharp transition between features to ensure that no new stress concentrations were introduced which is heavily reported in the literature on post-processing of topology optimization [24], [28].

4.8.5 Final Design Selection

One last run reached a mass of about 9.209 g which is less than the actual mass of the bracket was at 9.5 g. Such a design exhibited better distribution of stress and effective transfer of loads, which meant that the construction was at optimal set up. Another experiment to further decrease the mass led to higher stress concentrations, ascertaining that the structural limit was already achieved, and further removal of material would be counterfeitious. Upon this observation, 9.209 g design was taken as the final design because it offers the most reasonable weight loss and structural integrity, which Budynas and Nisbett [15] refer to as the key principle of the sound mechanical design practice.

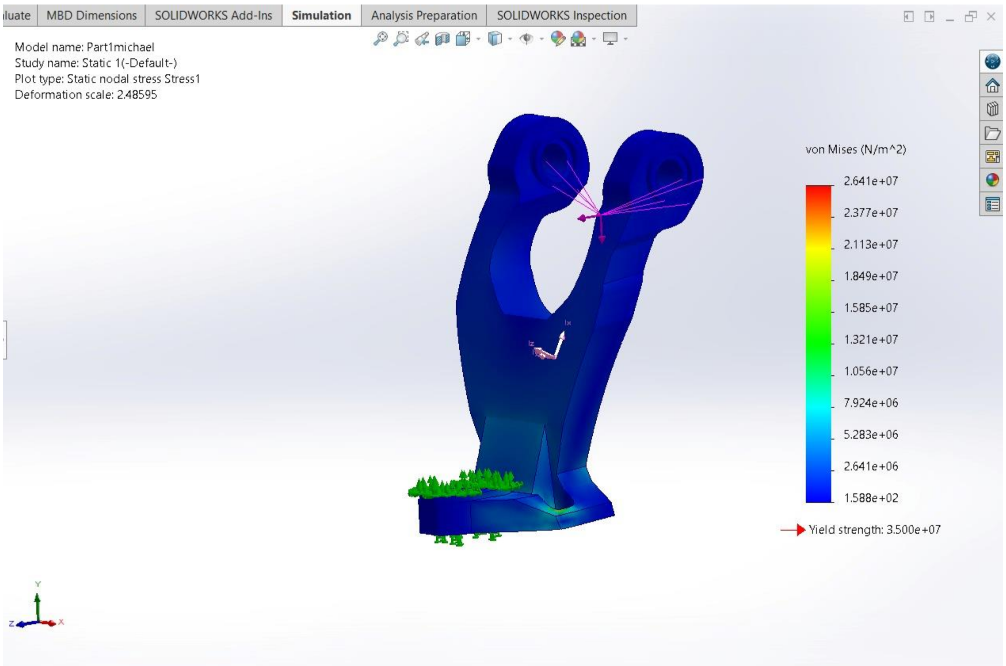

Figure 26. Final optimized design

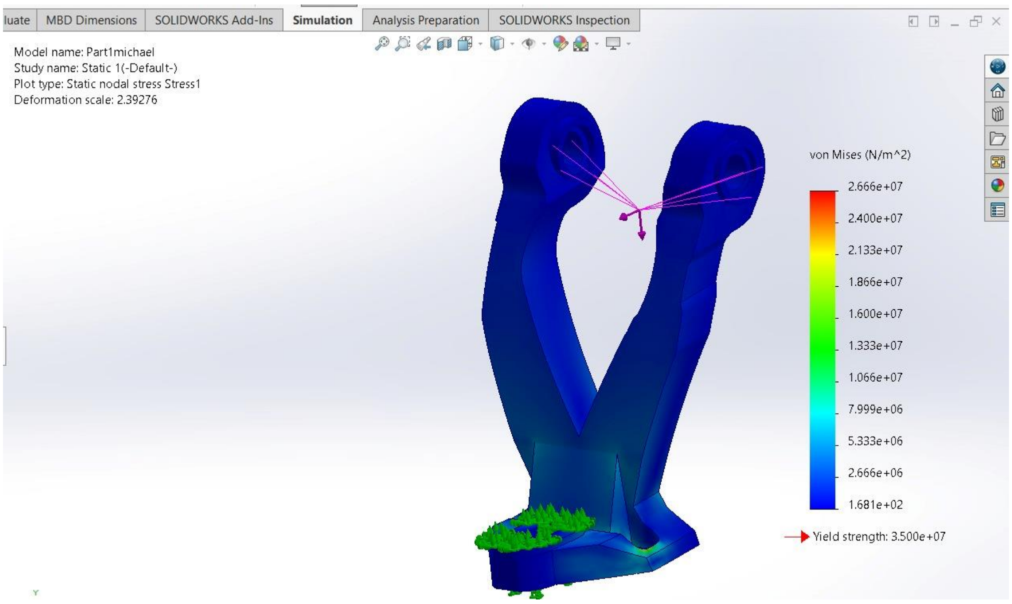

Figure 27. Further iteration showing increased stress

4.9 Engineering Interpretation of Results

Findings of this methodology show a rigidly and physically coherent trend: the initial bracket would put stress at the neck because of its geometric inefficiency, whereas the topology optimization can effectively redistribute material to remove it. This result is consistent with the existing theory, namely, the fact that structural efficacy of low-stiffness polymer parts is highly dictated by the continuity of the load paths instead of the raw material power [7], [14].

It is noteworthy also that the optimization has converged to a lighter configuration than the original one and this is not always the case in optimization problems, especially those with complex multifaceted objective landscapes [3], [5]. The characteristics that a light design will also demonstrate excellent structural performance also highlight the diagnostic nature of the baseline analysis: the original geometry was not only suboptimal but structurally wasteful in its material distribution.

4.10 Discussion

The approach that has been designed in this chapter reveals the real-world applicability of the combination of CAD modelling, FEM analysis, and topology optimization in the linear design process. New stages are built directly on the output of the previous stage, forming a design traceable and reproducible chain. There are a number of limitations in methodology, though.

The limiting factor of the analysis is first that it is limited to the statical loading, which is incapable to represent damage of fatigue accumulation that is involved in real-life cycling settings. Fatigue failure in bracket-type components is progressive in nature as the tendency to reduce fatigue in components studied by both Dowling and Covill et al. [8] and [17], respectively, note that fatigue analysis as a process must be dedicated to cyclic analysis to predict with a degree of certainty. Second, the compliance-based topology optimization formulation lacks stress singularities that may develop in geometric discontinuity regions, which has been found to be a weakness of Bruggi and Duysinx [6]. Third, optimization output to manufacturable geometry reconstruction process always involves some subjective engineering judgment, which, though inevitable, implies that the resulting design is not the mathematically optimal solution, but an approximation thereof.

Regardless of these constraints, the methodology offers a sound and practically applicable design enhancement strategy of lightweight polymer brackets, which is congruent with other modes of design improvement of brackets reported in the literature [13], [25], [29].

5 Results and Analysis

5.1 Overview

This chapter outlines an appraisal of the structural behaviour of the baseline and optimized bracket designs in detail. The outcomes are measured in order to determine the measurement on stress distribution, efficacy of load transfer and the quality of topology optimization in enhancing structural performance. The results are analysed under the assessment framework developed in Chapter 3 and all findings interpreted based on the material characteristics of ABS plastic and based on the boundary conditions as described in the methodology.

This evaluation needs to be approached critically. FEM-generated stress contours are not unconditional descriptions of the real world they are mathematical approximations where the validity is limited to the accuracy of the mesh, the idealization of the boundary conditions and the simplifications of the constitutive material model [9], [10]. It follows, therefore, that the results given here may be interpreted as engineering approximations of structural behaviour, adequate to compare the design and provide some optimising guidance, but which must be validated experimentally before concluding reasonably on in-service behaviour [17].

5.2 Static Analysis Results of the Baseline Design

5.2.1 Von Mises Stress Distribution

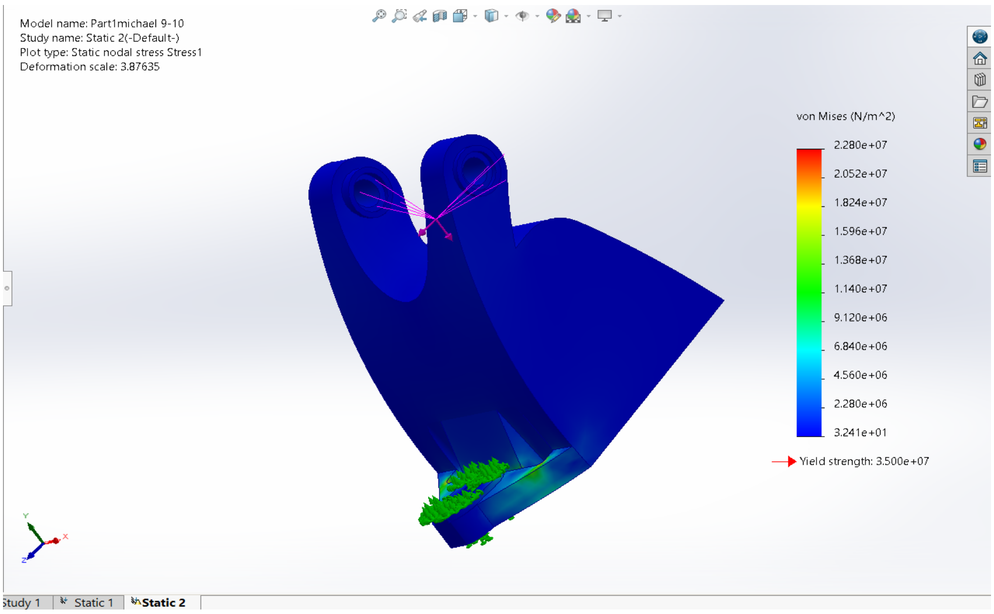

Figure 28. Baseline stress distribution under 20–30 N load

In order to get a holistic view of the performance of the baseline, the bracket was tested with realistic and conservative loading. The total levels of stress can be fairly low under realistic loads 20-30 N; nevertheless, there is always a local stress concentration at the neck level. It means that geometry is conducive in nature to concentration of stress, and thus, it becomes a structurally important area of possible failure under re-loading. Applying von Mises criterion as the failure indicator is also suitable in this case as it includes a scalar value of the multiaxial stress state which can easily be compared to the uniaxial yield strength of the material; this is the usual method of ductile materials as indicated in Hibbeler [10] and Gere [11].

Assuming that the conservative load is 50 N, the structural weakness is significantly more high. Concentration of the stress at the neck region is much higher which proves the geometry is not cross-sectionally strong enough to divert the applied load effectively. Stress concentration at the neck when the stress is increased is consistent with the classical stress concentration theory [7] whereby thin-section geometries show nonlinear amplification of stress with applied loads when there are sudden transitions.

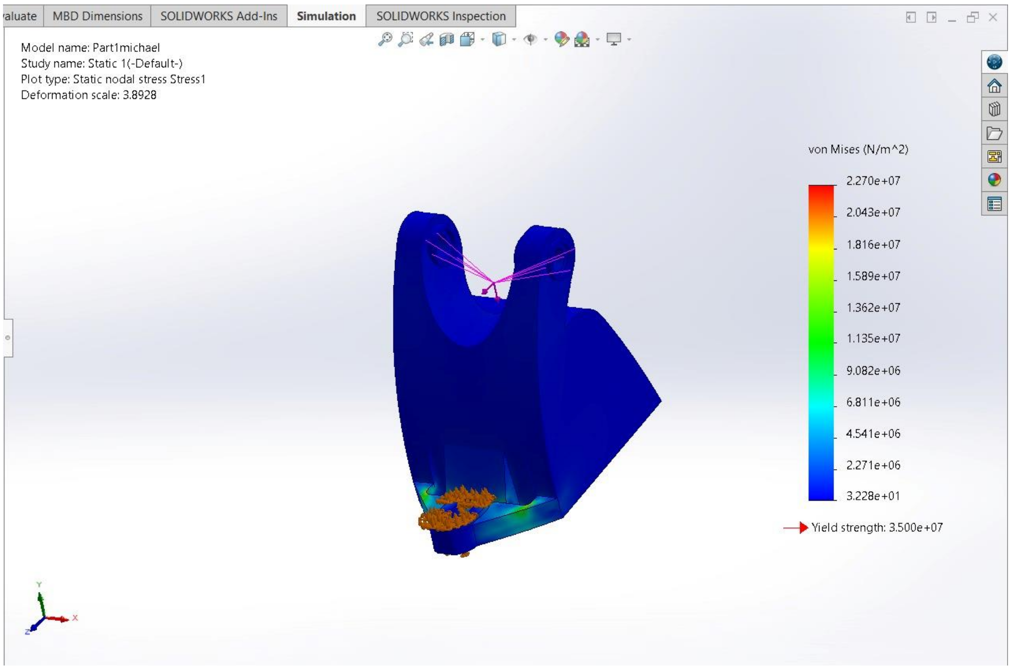

Figure 29. Baseline stress distribution under 50 N load

The consistency in the location of maximum stress across different loading conditions is a significant finding. It confirms that the failure mechanism is governed primarily by geometric design rather than load magnitude, a conclusion with direct implications for redesign strategy. The stress concentrations of this type, comments Budynas and Nisbett, can no longer be sorted out by a change of material only and must be geometrically adjusted to redistribute the load path [15].

5.2.2 Equivalent Elastic Strain Distribution

The strain distribution is also in agreement with the strain results, with the largest concentration to the neck region, the most elastic. The fact that stress and strain in the elastic regime are reported to display linearity based on the Hooke Law and justified by the level of stress not exceeding the yield strength of the ABS, supports the suggestion that the assumption of material-behaviour of linear elasticity is applicable to the material behaviour in loading in this case [10], [16]. It is however noted that local micro-yielding can take place at a nominally lower stress value in high concentration of stress in a region than in the bulk, especially in materials whose yielding behaviour is rate-dependent and temperature-sensitive (as in polymeric materials) [8]. The given subtlety cannot be discussed in terms of the current still linear elastic analysis and it is a weakness of the modelling approach.

5.2.3 Structural Assessment of the Original Design

The structural inefficiency of the initial design of the bracket is clearly established by the fact that the bracket was not designed in a way that would guarantee efficiency. The extreme HF concentration at neck, which is evident on both realistic and conservative load cases, suggests that a proper load path exists between the frame of headlamp interface and the handlebar clamp, which requires the current geometry. This is particularly worrisome in terms of fatigue: stress concentrations form the main initial sites of fatigue crack propagation as already well established by Dowling [8], and confirmed by Covill et al. [17] in bicycle components. The neck area of the original design would be likely to have a much less fatigue life than under the cyclic, vibration dominated loading of it in the real cycling conditions defined in Section 2.3.

5.3 Topology Optimization Results

5.3.1 Material Density Distribution

The optimization process with topology yielded a material density distribution that is evident of identifying the main structural load paths in the design space. The dense areas represent material which is crucial in the transfer of load and the less-dense areas represent material that adds the least contribution to structural performance and hence can be removed. The map is the computational plan of the reconstruction phase of Chapter 3. The mathematical interpretation of the density distribution would be based on the SIMP methodology [4], [5], whereby penalization of intermediate densities would be used to drive the solution towards a discrete solid-void solution, which, although mathematically elegant, is subject to so-called numericalartefacts including checker-boarding and mesh dependency which must be suppressed using filtering methodologies [22], [27].

5.3.2 Optimized Geometry

Topology optimization led to a steady increase in the level of structural performance in each new design process. The initial design, having a mass of about 13.9 g, exhibits better stress distribution relative to the baseline but still has inconvenient material in areas which are discovered by the density distribution to be of low contribution. This is a well known result of conservative reconstruction of the density plot where the engineers are more likely to safety stock up the material to prevent creating new structural weaknesses [23], [24].

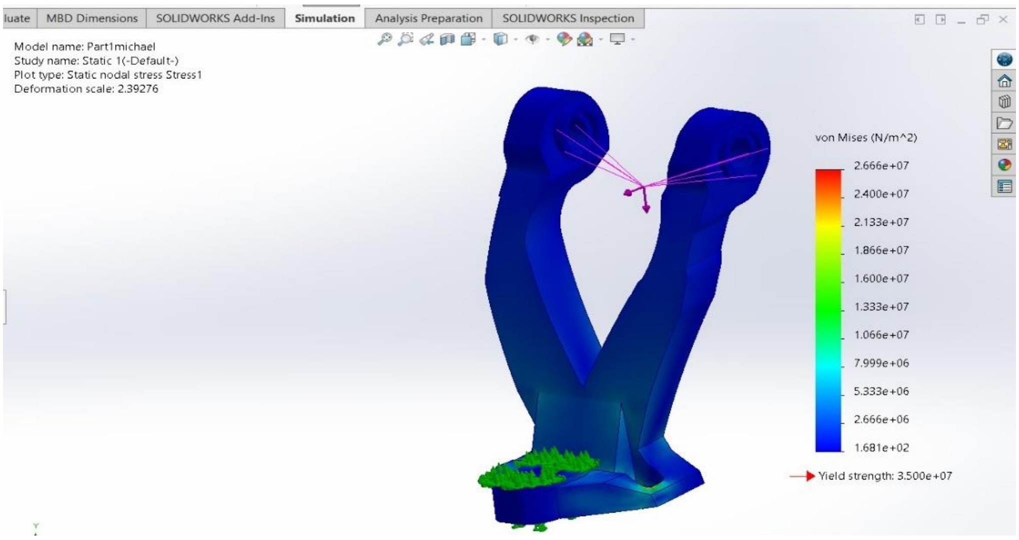

Figure 30. Stress distribution - Design 1 (13.9 g)

The second version of the design, and with a mass of about 9.8 g, the balance between weight and structural performance is much more efficient, stress level is lower, and the load distribution is also better.

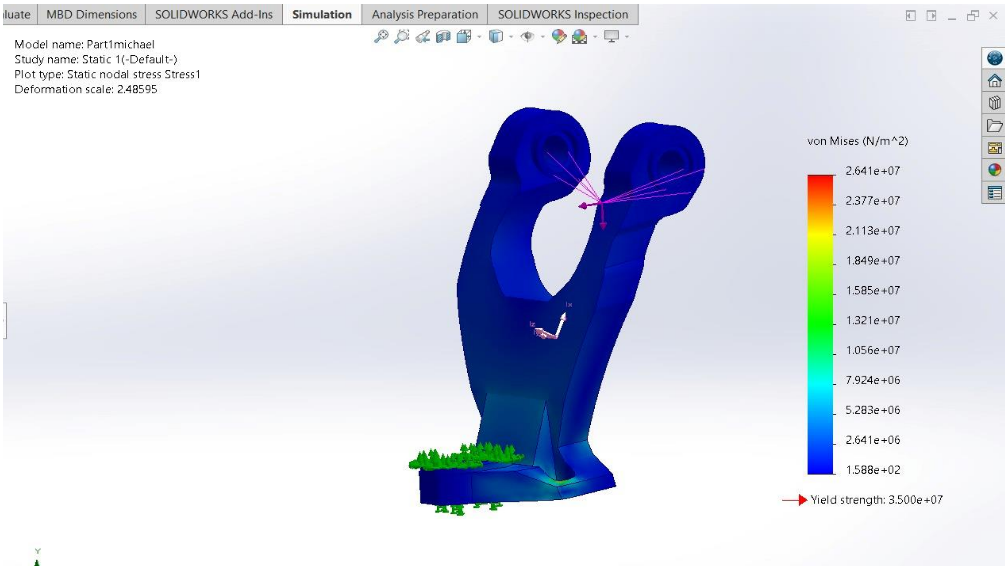

Figure 31. Stress distribution - Design 2 (9.8 g)

This heuristic optimisation implies what Liu and Ma [25] attribute to the heuristic nature of the geometry generation of topology optimization output, which, though based on engineering intuition, systematically generates designs that are more effective than theirs in terms of the density map interpretation.

5.3.3 Mass Properties of the Optimized Design

The resulting optimized design has a mass of about 9.209 g as opposed to the original bracket mass of about 9.5 g. Although there is this mass reduction of about 3.1, the design has much more uniform stress distribution and does not experience such extreme stress concentration as can be seen in the baseline model. This is the primary goal of the topology optimization that any effective topology optimization process aims to achieve and is what is defined by the theoretical predictions of Bendssoe and Sigmund [1] which state that compliance minimizing topology optimization will always approach a structurally superior design compared to an arbitrary starting geometry as long as the design space is sufficiently defined.

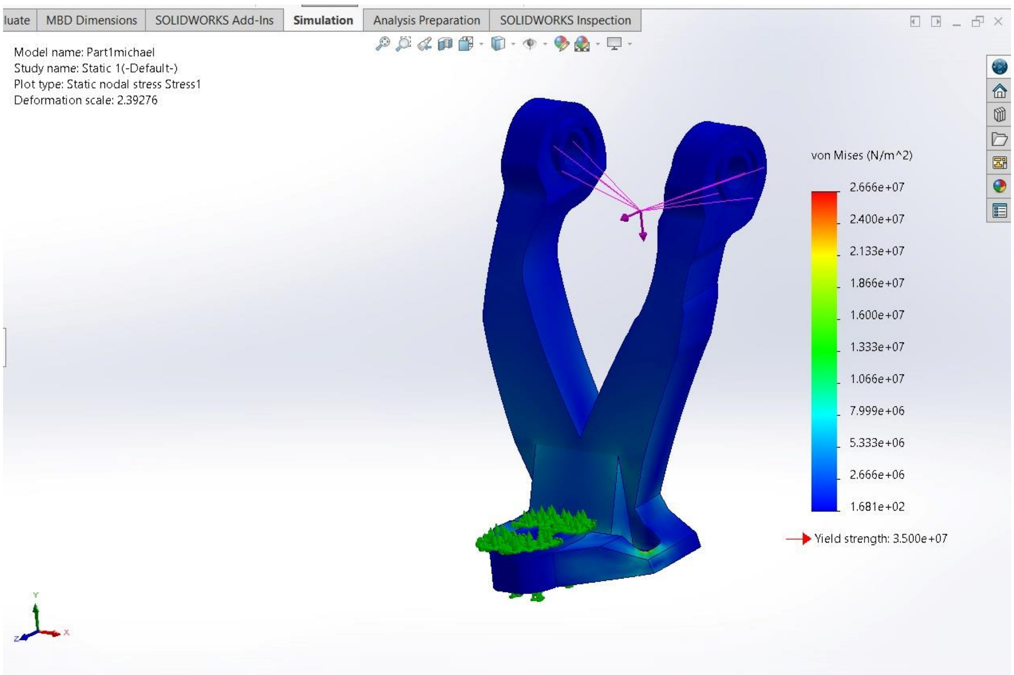

Figure 32. Stress distribution - Final Design (9.209 g)

A further refinement that with the goal of further minimizing the mass load, the level of the stresses increased, meaning that the structure was already at its geometric limit of material efficiency. This observation is edible: it illustrates that there exists a limit of how mass can be achieved with topology optimization, with a given performance constraint, and any additional material extraction is always going to lead to deterioration of the structural integrity [3], [6]. This limit discovery confirms the choice of design (the engineering decision) to use the 9.209 g design as the final design.

5.4 Structural Analysis of the Optimized Design

The end optimized design structural analysis actually confirms that the maximum von Mises stress under 50 N load is below the yield strength of ABS (about 35-45 MPa). This implies that the bracket lies somewhere in the elastic range which is an intrinsic factor of a component supposed to work smoothly with a substantial number of loading cycles. Compliance of the optimized design is also an important difference to the baseline, and shows increased stiffness per unit mass which is a key performance measure in mounting bracket components as Xie and Steven [5] remark. One should put this outcome to its critical questioning. This increase in stress distribution is due to a smoother and more efficient, geometrical load path and not the change in the material or applied load. This establishes the fact that this inherent flaws in the original design were essentially of a geometric nature, which was already known by the analysis of the base case in Section 4.2.3. In the real-world perspective, this implies that this design improvement could be feasible without either raising the material cost or manufacturing complexity, as the geometry, which is optimized, is not heavier and structurally more competent than the original.

5.5 Comparative Analysis: Original Design versus Optimized Design

Table 4. Performance Comparison between Original and Optimized Bracket Designs

Parameter	Original Design	Optimized Design	Improvement
Mass (g)	9.5	9.209	↓ 3.1% reduction
Maximum Stress (MPa)	High (localized at neck region)	Reduced	Improved stress distribution
Stress Distribution	Highly concentrated at neck	More uniform	Significant improvement
Factor of Safety	< 1 (locally unsafe)	≈ 1.32	Structurally safe

To compare the original and the optimized designs of the bracket, Table 4 provides a comparative summary of the structural performance of these two designs. The outcomes indicate clearly that the optimized design can attain a reduced weight as well as an increased stress distribution. Distinct removal of an extreme concentration of stress at the point of the neck and the enhancement in the factor of safety but without loss of material efficiency prove the topology optimization process to be efficient in improving the structural performance.

5.7 Factor of Safety Analysis

The ratio of the material yield strength and the maximum von Mises stress derived in the simulation was put into the force of safety (FOS). Considering an approximate yield strength of ABS material of 35 Mpa, and maximum stress equal to 26.6 Mpa observed in the optimized design, the factor of safety would be:

$$FOS = \frac{35}{26.6} \approx 1.32$$

This outcome denotes that the optimized bracket works within the elastic range given the loading condition applied and has a satisfactory safety margin. Though the factor of safety is moderate, it is said to be acceptable when using lightweight consumer parts, specifically considering the conservative load of 50 N that was used in this experiment.

It should however be noted that a fixed factor of safety is not wholly representative of the long term stability of the component. Practically, surface finish, stress concentration and cyclic loading have been considered to influence fatigue performance; this will considerably decrease the effective fatigue strength of ABS. According to the literature S-N curve data, with proper corrections of the mean stresses, the value of the fatigue factor of safety can be significantly less than the static value. Thus, the present analysis shows that the structure is safe under the condition of a static loading, but the careful study of fatigue effects is suggested in the future.

5.8 Discussion of Results

All the results provided in this chapter make three main findings. To begin with, the original bracket design has structural flaws in its geometrical inefficiency, and stress concentrations in the neck area, inherent in the original shape and impossible to address without a geometric redesign. Second, topology optimization offers a principled mechanistic and computationally efficient approach to detecting and removing this geometric inefficiency to only realize a design that is both lighter and structurally better than a baseline. Third, the optimization is terminal to a lower limit of mass configuration past which any further elimination of the material makes the structure less efficient, which serves to confirm the physical integrity of the optimized design and the accuracy of the computational procedure. The results can be concluded by comparing them with the more general literature on topology optimization of small structural parts. Xie and Steven [5], Bendsoe and Sigmund [1], and Collet et al. all show that compliance-based optimization is a sure way to optimize structural performance in comparison to arbitrary initial geometries [26]. This are further refined by the context of a polymer consumer bracket: the comparatively low stiffness of ABS and its creep and fatigue vulnerability imply that the results of performance improvement achieved here are more of a preliminary than a final opinion that only needs to be substantiated in the framework of experimental operation under more realistic load conditions of dynamic loading [8], [17].

Another serious note is about the representativeness of the 50 N load case. Although it is an adequate conservative loading to control the structural response as a design screening aid, it fails to reflect the multi-directional, variable-amplitude characteristics of vibration caused by roads [17], [21]. A tougher design evaluation would use a load spectrum based on field measurements, with a fatigue cumulative damage model like the PalmgrenMiner rule, to give a statistically significant indication of the service life of the bracket. This continues to be a significant research path in the future.

6 Conclusions and Recommendations

This chapter summarises the key findings of the study, draws conclusions against the original research objectives, acknowledges the principal limitations of the methodology, and proposes directions for future work.

6.1 Summary of the Study

In this research, the structural performance of a bicycle headlamp mounting bracket was evaluated and attempted to enhance it by CAD modelling, finite element analysis, and topology optimization using SolidWorks and ANSYS Mechanical [12]. A fundamental static FEA revealed a severe stress concentration in the base of the neck of the initial design of the conservative load of 50 N, which confirmed a geometric inefficiency in that regard in agreement with classical stress concentration theory [7], [10]. The topology optimization was subsequently used to redistribute material over effective paths of load and the computed density field directed an iterative geometry restructuring process.

The optimised design attained a final mass of around 9.209 g; a slight decrease over the original 9.5 g with much more uniform von Mises stress field distribution and without the harsh neck-region concentration. All these results are expected given the theoretical forecasts of Bendsoe and Sigmund [1] and compliance-minimisation principles that form the basis of the SIMP method, in addition to the experimental results in the study by Da et al. [24].

It should be made very clear that although we looked into fatigue theory, fatigue theory such as the stress-life multiplier, Coffin Manson approximations and crack initiation and propagation mechanisms have been studied extensively in Chapter 2, we did not carry out a complete computational fatigue life prediction as part of the research. There was no S-N curve to be used in the FEM results and no factor of fatigue safety was taken. It is an acknowledgeable but intentional restriction, of which Section 5.4 is an official recognition.

6.2 Conclusions

The FEA of the original bracket showed that the component was structurally inefficient at the loading conditions. The highest von Mises stress was always concentrated at the neck area in all load cases, which is a geometric weakness that cannot be avoided by changing materials with no geometric support, in agreement with the stress concentration theories developed by Pilkey and Pilkey [7]. Fatigue wise, with this concentration, even without an actual fatigue life calculation, it is found a much lower cyclic life than a geometrically optimised counterpart so far as the basic fatigue notch factor theory of Dowling [8] is concerned, which requires a significantly lower cyclic life.

The topology optimisation process proved to be quite effective in determining and eradicating this inefficiency. The solver based on SIMP found load paths which the original geometry was unable to take advantage of and produced a design with better structural continuity and a more homogeneous stress field [1], [4]. The compliance based formulation sufficed this initial study but it should be realised that the optimised formulation has not been checked against known stress limits, which Bruggi and Duysinx [6] and Collet et al. [26] consider as one such limitation, and the potential of local stress increase that remains unchecked without additional analysis is impossible to rule out.

The final optimised design operates within the elastic regime under the applied load, confirming an adequate static factor of safety [15]. The study validates the core premise that structural performance in lightweight polymer brackets is governed primarily by geometric load-path efficiency rather than raw material strength, and demonstrates that mass reduction and structural improvement can be concurrently achieved through systematic computational design [3], [5].

6.3 Limitations of the Study

The most significant limitation is that the entire FEA was conducted under static loading conditions. Real-world cycling involves dynamic, variable-amplitude, multi-directional loading from road irregularities and rider input [17], [21], which can produce stress amplitudes substantially higher than those predicted by static analysis, particularly near resonant frequencies. Consequently, the static factor of safety reported here should not be interpreted as a reliable measure of fatigue durability.

No computational fatigue life prediction was performed. An S-N curve was not applied to the FEM stress outputs, no Rainflow cycle counting was conducted, and no fatigue factor of safety was calculated using mean stress correction methods such as the Goodman or Gerber criteria [8], [15]. This represents the most critical gap between the study's title and its computational scope, and is the primary direction for future work.

Additional limitations include the use of a linear elastic material model, which cannot capture the viscoelastic behaviour or rate-dependent yielding of ABS [16], the subjectivity inherent in translating the topology density output into a reconstructed CAD geometry [23], [24], and the restriction to a single load case and direction [29]. A combination of these limitations restricts the extent to which the findings can be generalised to actual service conditions.

6.4 Recommendations for Future Work

The first recommendation is a complete fatigue life analysis which is the most urgent. This would see the implementation of the stress-life (S-N) method to the FEM stress outputs with known data to the ABS or other material of interest along with Goodman mean stress correction [8], [15] and Rainflow cycle counting to a measured road vibration spectrum in-situ [17]. The modal and harmonic analysis must also be carried out in order to determine the resonant frequencies as well as evaluate the chances of amplification of fatigue due to vibration [21].

Any design conclusiveness can only be established through experimental testing. Physical models of both these designs must be made, preferably by additive manufacturing and then tested by a weight on it in a strain gauge measurement at the neck area and at other areas of possible stress concentrations [17], [31]. Fem predictions need to be compared with the measured strains to ensure that simulations are being done correctly and to detect all the ideals of the boundary conditions.

The optimisation methodology can eventually be further developed, such as stress-constrained topology optimisation formulations [6], [26], multi-load case optimisation to represent the whole operational loading envelope [29], and automated geometry reconstruction pipelines via level-set or moving morphable component methods [24], [28]. Test of other materials especially aluminium alloy 6061-T6 and glass nylon would also go a long way in diversifying the applicability and longevity of the bracket design [14], [16].

References

[1] M. P. Bendsøe and O. Sigmund, *Topology Optimization: Theory, Methods and Applications*. Berlin, Germany: Springer, 2003.

[2] O. Sigmund, "A 99 line topology optimization code written in MATLAB," *Structural and Multidisciplinary Optimization*, vol. 21, no. 2, pp. 120–127, 2001.

[3] G. I. N. Rozvany, "A critical review of established methods of structural topology optimization," *Structural and Multidisciplinary Optimization*, vol. 37, no. 3, pp. 217–237, 2009.

[4] M. P. Bendsøe, "Optimal shape design as a material distribution problem," *Structural Optimization*, vol. 1, no. 4, pp. 193–202, 1989.

[5] Y. M. Xie and G. P. Steven, "A simple evolutionary procedure for structural optimization," *Computers and Structures*, vol. 49, no. 5, pp. 885–896, 1993.

[6] M. Bruggi and P. Duysinx, "Topology optimization for minimum weight with compliance and stress constraints," *Structural and Multidisciplinary Optimization*, vol. 46, no. 3, pp. 369–384, 2012.

[7] W. D. Pilkey and D. F. Pilkey, *Peterson's Stress Concentration Factors*, 3rd ed. Hoboken, NJ, USA: Wiley, 2008.

[8] N. E. Dowling, *Mechanical Behavior of Materials: Engineering Methods for Deformation, Fracture, and Fatigue*, 4th ed. Upper Saddle River, NJ, USA: Pearson, 2013.

[9] T. Belytschko, W. K. Liu, and B. Moran, *Nonlinear Finite Elements for Continua and Structures*. New York, NY, USA: Wiley, 2000.

[10] R. C. Hibbeler, *Mechanics of Materials*, 10th ed. Upper Saddle River, NJ, USA: Pearson, 2017.

[11] J. M. Gere, *Mechanics of Materials*, 6th ed. Pacific Grove, CA, USA: Thomson, 2004.

[12] ANSYS Inc., *ANSYS Mechanical User's Guide*, Release 2022. Canonsburg, PA, USA: ANSYS, 2022.

[13] ANSYS Inc., *ANSYS Fatigue Module Theory Guide*, Release 2021. Canonsburg, PA, USA: ANSYS, 2021.

[14] S. R. Schmid, B. J. Hamrock, and B. O. Jacobson, *Fundamentals of Machine Elements*, 3rd ed. Boca Raton, FL, USA: CRC Press, 2014.

[15] R. G. Budynas and J. K. Nisbett, *Shigley's Mechanical Engineering Design*, 10th ed. New York, NY, USA: McGraw-Hill, 2015.

[16] W. D. Callister and D. G. Rethwisch, *Materials Science and Engineering: An Introduction*, 10th ed. Hoboken, NJ, USA: Wiley, 2018.

[17] D. Covill, P. G. Allard, J. M. Drouet, and N. J. Emerson, "An assessment of bicycle frame behaviour under various load conditions using numerical simulations," *Procedia Engineering*, vol. 147, pp. 665–670, 2016.

[18] J. M. Gere and B. J. Goodno, *Mechanics of Materials*, 8th ed. Stamford, CT, USA: Cengage Learning, 2013.

[19] R. L. Norton, *Machine Design: An Integrated Approach*, 5th ed. Upper Saddle River, NJ, USA: Pearson, 2013.

[20] S. P. Timoshenko and J. N. Goodier, *Theory of Elasticity*, 3rd ed. New York, NY, USA: McGraw-Hill, 1970.

[21] S. P. Timoshenko and D. H. Young, *Vibration Problems in Engineering*, 3rd ed. New York, NY, USA: Wiley, 1968.

[22] H. Wang, J. Liu, and G. Wen, "An adaptive mesh-adjustment strategy for continuum topology optimization to achieve manufacturable designs," *International Journal for Numerical Methods in Engineering*, vol. 117, pp. 1304–1322, 2019.

[23] M. Abdi, R. Wildman, and I. Ashcroft, "Evolutionary topology optimization using the extended finite element method and isolines," *Engineering Optimization*, vol. 46, pp. 628–647, 2014.

[24] D. Da, L. Xia, G. Li, and X. Huang, "Evolutionary topology optimization of continuum structures with smooth boundary representation," *Structural and Multidisciplinary Optimization*, vol. 57, pp. 2143–2159, 2018.

[25] J. Liu and Y. Ma, "A survey of manufacturing-oriented topology optimization methods," *Advances in Engineering Software*, vol. 100, pp. 161–175, 2016.

[26] M. Collet, M. Bruggi, and P. Duysinx, "Topology optimization for minimum weight with compliance and simplified nominal stress constraints for fatigue resistance," *Structural and Multidisciplinary Optimization*, vol. 55, pp. 839–855, 2017.

[27] L. Wang, W. Yue, and M. Zhu, "An improved evolutionary structural optimization method for smooth topology design of structures," *International Journal of Computational Methods*, vol. 20, 2250061, 2023.

[28] B. Zhu, X. Zhang, H. Zhang, J. Liang, H. Zang, H. Li, and R. Wang, "Design of compliant mechanisms using continuum topology optimization: A review," *Mechanism and Machine Theory*, vol. 143, 103622, 2020.

[29] Y. Han, B. Xu, Q. Wang, Y. Liu, and Z. Duan, "Topology optimization of material nonlinear continuum structures under stress constraints," *Computer Methods in Applied Mechanics and Engineering*, vol. 378, 113731, 2021.

[30] O. C. Zienkiewicz and R. L. Taylor, *The Finite Element Method for Solid and Structural Mechanics*, 6th ed. Oxford, UK: Butterworth-Heinemann, 2005.

[31] R. D. Cook, D. S. Malkus, M. E. Plesha, and R. J. Witt, *Concepts and Applications of Finite Element Analysis*, 4th ed. Hoboken, NJ, USA: Wiley, 2002.

Appendices

Simulated Object

Front View

Hind View

Cross-Sectional View

Failed / Broken Part

High stress concentration at the neck region of the Folk bracket (reason for failure)