Lightweight Design Method for Bike Frames Using Topology Optimization under Multiple Loads

Dong Hyun Ko¹, Dong Seok Shin², Euy Sik Jeon³

¹ Master’s degree candidate, Department of Mechanical Engineering, The College of Kong-ju National University, South Korea, dhko@nentkorea.com
² Senior researcher, Industrial Technology Research Institute, Kong-ju National University, South Korea, believe@kongju.ac.kr
³ Professor, Department Mechanical Engineering, The College of Kong-ju National University, South Korea, osjun@kongju.ac.kr

Corresponding author: Euy Sik Jeon

Abstract: Mobility is a complex transportation consisting of not only frames but also seats, sensors, and steering systems. Among them, the frame is an important structure that determines the safety and performance of the device as a part that supports the load by combining all other components and receives external impact. It is necessary to study the skeletal design of the frame of mobility devices to achieve a safe and simple structure. This study proposed a conceptual design method for a skeletal mobility frame. An optimized frame design was investigated for multiple loads using a topology optimization technique in automotive parts design. Finite element modeling was conducted by measuring the actual model, and finite element analysis (FEA) was performed. Post-processing was performed along with FEA. Furthermore, finite factor analysis was performed on the post-processed model and was compared with the conceptual model. Results of the study revealed that compared with the conceptual model, the weight of the design model decreased by 93.7%. This process was carried out through computer modeling and computer simulation, and it was possible to avoid separate prototyping and evaluation. Furthermore, Through this research process, the optimal skeletal structure was extruded based on the optimal topology design results for multiple loads. During the optimization process, the maximum strain and stress satisfied the yield condition of the material, proving the safety of the structure. It is believed that this research process can be used for designing other mobility frames that require an engineering design method.

Keywords: Bike Frame, Finite Element Analysis, Personal Mobility Device, Topology Optimization

1. Introduction

Owing to climate change, environmental problems, and energy crisis, sustainable and eco-friendly transportation is required. Personal mobility(PM) devices provide lower mileage and exhibit poorer driving performance than cars; however, they have several advantages, such as being powered by electric energy, increasing traffic density, and transporting one person or individual cargo. PM devices serve as support for passengers and cargo at the bottom, and major accessories such as batteries and wheels are installed[1][2]. Furthermore, their simple structure and design make PM devices easier to design and manufacture than general cars. However, high accessibility results in an overweight, low-safety design[3-5]. An engineering method is needed to make the mobility frame safe. In line with this,
the study was proposed.

The number of PM-related accidents are on the rise every year with the increase in the use of PM devices. It is necessary to study a conceptual design method for PM products that considers their safety [6][7]. In the field of general automotive parts design, finite element analysis (FEA) is used for design-performance prediction–optimization[8][9]. Using FEA, structural performance can be predicted relatively easily and performance changes due to design changes can be observed with fewer resources, thereby achieving the desired structural performance. General structural analysis is based on the theory of solid mechanics, and it can be solved based on the rigidity method or potential energy formulation theory. ANSYS, which automates this theory, is used in this study.

ANSYS provides a topology optimization process based on the density method. Users can adjust the objective function, penalty coefficient, and constraints to observe the topology optimization results. Topology optimization results were observed with shape, weight, and maximum displacement and stress[10][11].

Most of the existing mobility frame research cases involve the structural analysis of existing frames. However, studies on the optimization results using phase optimization that are widely used in other industries for frame fabrication and the performance of post-processing and post-processing models are limited[12-16]. Existing conceptual design methods, including topology optimization, are common but not approached in three-wheel mobility devices. An engineering method is needed to make the mobility frame safe.

In this study, a plan to use the FEA and topology optimization for the conceptual design of mobility frames was proposed. A three-wheel mobility device was used in the study, and the layout design was presented using FEA. In addition, weight reduction was performed by removing unnecessary design areas using topology optimization techniques. The result of topology optimization was post-treated with a bone-forming frame, and the structural performance was compared after prototype production.

2. Finite Element Analysis

2.1 Research Methodology

![Fig. 1] Topology Optimization Process
The following process was performed to perform the optimal design using topology optimization as a mobility frame design method. Modeled using Creo Parametric to perform finite element modeling for the concept model. The concept model was actually measured on a commercially available 3-wheel mobility frame. ANSYS was used for finite element modeling, and the finite elements were configured in 3D (Hexahedron). Optimization was performed by comparing the finite element analysis results of the conceptual model with the phase optimization results. For the load, five load conditions were applied in consideration of the maximum loading weight, the weight of passengers and mobility parts, etc. It was confirmed that deformation and defects exceeding the yield point of the material did not occur. Post-processing was carried out with a skeletal structure using a pipe made of steel. All these processes were performed in the same way as [Fig. 1].

2.2 Layout Design of the Lower Frame of the Motorized Bicycle

CAD modeling was performed for finite element analysis and further phase optimization. The layout model used in this study had a size of 1180 × 400 × 30 [mm3], which is suitable for Japanese mobility device standards, as shown in [Fig. 2][17]. When designing the layout model, the non-domain areas considering the installation of the forward steering wheel and the installation of the rear driving wheel were also considered.

2.3 Finite Element Modeling

Finite element modeling is a procedure that must be performed in order to divide and analyze complex structures that are difficult to solve into solvable problems. The finite element model represents a complex form by continuously arranging several elements of a simple form. The finite element model for the lower model is shown in [Fig. 3]. The model consisted of 11,235 hexahedral
elements and 57,744 nodes. The height of the layout model had six equal parts, and the average element length was set to 5 mm. The element properties are summarized in [Table 2].

![Fig. 3] Finite Element Model

**[Table 1] Element Properties**

<table>
<thead>
<tr>
<th>Content</th>
<th>Unit</th>
<th>Values</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of Elements</td>
<td>EA</td>
<td>17,460</td>
</tr>
<tr>
<td>Number of Nodes</td>
<td>EA</td>
<td>61,126</td>
</tr>
<tr>
<td>Element size</td>
<td>mm</td>
<td>5.0</td>
</tr>
<tr>
<td>Element Type</td>
<td>-</td>
<td>hex</td>
</tr>
</tbody>
</table>

**[Table 2] Material Properties of Structural Steel**

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Unit</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yield Stress</td>
<td>MPa</td>
<td>109.0</td>
</tr>
<tr>
<td>Ultimate Stress</td>
<td>MPa</td>
<td>293.0</td>
</tr>
<tr>
<td>Young’s Modulus</td>
<td>MPa</td>
<td>120,000.0</td>
</tr>
<tr>
<td>Poisson’ Ratio</td>
<td>-</td>
<td>0.3</td>
</tr>
<tr>
<td>Density</td>
<td>kg/mm3</td>
<td>7.8 × 10^-6</td>
</tr>
</tbody>
</table>

The layout model considered the properties of structural steel, which is commonly used in these devices; these are summarized in [Table 2]. The boundary and load conditions are shown in [Fig 3]. We restricted the triaxial (X-, Y-, and Z- axes) movement-rotation freedom for all the elements of the non-domain region. It was assumed that the external force applied to the model caused a bending moment and twisting. A load of 980 N was applied to the mobility device.

![Fig. 4] Static Load Cases for Layout Model
The results of FEA for various load conditions of 980N are shown in [Fig. 5] and [Fig. 6]. [Fig. 5] shows the stress value for each analysis. Excluding the analysis result of [Fig. 5(f)], to which all loads were applied, the greatest stress value was found for the forward bending moment in [Fig. 5(b)]. [Fig. 6] shows the maximum displacement for each analysis. Excluding the analysis result of [Fig. 6(f)], the largest displacement was found for [Fig. 6 (d)]. [Table 3] presents the results of the maximum displacement and maximum equivalent stress. Under the given conditions, it was confirmed that the maximum value of the displacement was $1.8 \times 10^{-2}$ mm, the maximum value of the equivalent stress was 18.5 MPa, and the stress and displacement of the conceptual model did not meet the yield conditions of the material.
2.4 Topology Optimization

Topology optimization was used to present a topological structure by removing unnecessary elements using FEA. Density method approach – solid isotropic material with penalization (SIMP), which is a commonly used topology optimization – was used in this study.

The topological optimization problem is defined as minimizing the strain energy \( U \) of the structure and satisfying a set of constraints in the optimization process. In general, topological optimization requires the amount of material present in the design area to be constant. Therefore, the topological optimization problem is mathematically defined as follows.

Objectiv : 

\[
U(x) = u^T K u = \sum_{e=0}^{n} (x_e)^2 u_e^T K_e u_e
\]

Subject to \( \frac{V(x)}{V_0} = V_S, x_{min} < x \leq 1 \)

Here, \( U(x) \) is the strain energy, \( u \) is the total displacement vector, \( K \) is the total stiffness matrix, \( e \) is the element sequence number, \( u_e \) is the element displacement vector, and \( K_e \) is the element stiffness matrix. \( x \) represents the density of the material as the design variable vector. \( x_{min} \) is the smallest
density vector used to avoid instability in the numerical analysis. \( n^* \) is the number of finite elements discretized over the entire design domain (the maximum value of \( e \)), and \( \gamma \) is the exponential value of the discretized penalty point function over the entire design domain. In general, a \( \gamma \) value between 3 and 9 is used, and as the value increases, the derived phase becomes clearer. \( V(x) \) and \( V_0 \) are the volumes of the material and the entire design area, respectively. \( V_s \) is the volume constraint given as the initial condition.

![Fig. 7] Topology Optimization Region

<table>
<thead>
<tr>
<th>Restrictions</th>
<th>Condition</th>
</tr>
</thead>
<tbody>
<tr>
<td>Design variable</td>
<td>Domain</td>
</tr>
<tr>
<td>Objective Function</td>
<td>Min(w)</td>
</tr>
<tr>
<td>Constraint</td>
<td>Volume Fraction ( \geq 30% )</td>
</tr>
<tr>
<td>Option</td>
<td>YZ Plane Symmetry</td>
</tr>
</tbody>
</table>

![Table 4] Condition of Topology Optimization

The design area of the topology optimization is shown in [Fig. 7], and the conditions are summarized in [Table 4]. [Fig. 8] shows the shape obtained by the topology optimization.

Based on the result derived by the finite element topology optimization technique, a \( 30 \times 30 \times 1.2t \) pipe was used as, shown in [Fig. 9], to ensure that the optimal shape was as close as possible, and the skeletal structure can be constructed.

![Fig. 8] Topology Optimization Result

![Fig. 9] Topology Optimization Result
3. Results & Conclusion

3.1 Results

The ANSYS workbench generated six structural analysis modules, as shown in [Fig. 4], which includes the structural analysis.

[Fig. 10] and [Fig. 11] show the results of the equivalent stress and the total deformation calculated by ANSYS, respectively.

[Table 6] summarizes the maximum stress values and the deformations for each load. The maximum stress value was calculated to be 18.5 MPa in the conceptual model and 85.9 MPa in the post-processed model. The maximum deformation values in the conceptual and post-treatment models were calculated as 1.8×10⁻² and 3.8×10⁻¹ mm, respectively.

In the optimization process, the deformation amount and stress increased compared with the conceptual model, but it was confirmed that the maximum deformation amount and the maximum stress satisfy the material yield condition.

The masses of the conceptual and post-processing models are listed in [Table 5]. The conceptual and optimization-postprocessing models were designed to weigh 102.37 and 6.48 kg, respectively. Compared with the conceptual model, the weight of the post-processed model was reduced by 93.7%, and the maximum deformation and von Mises stress of the structural analysis module were collected as real data. The collected data are presented in [Table 5] and [Table 6].
Lightweight Design Method for Bike Frames using Topology Optimization under Multiple Loads

[Fig. 10] Redesigned Frame Analysis Equivalent Stress Results

![ Equivalent Stress Results ]

(a) Front Band  (b) Front Twist  
(c) Rear Band  (d) Rear Twist  
(e) Vertical Load  (f) Total

[Fig. 11] Redesigned Frame Total Deformation Analysis Results

[Table 5] Comparison Mass of Original and Redesigned Frames

<table>
<thead>
<tr>
<th>Load cases</th>
<th>Unit</th>
<th>Original frame</th>
<th>Optimized frame</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weight</td>
<td>kg</td>
<td>102.37</td>
<td>6.48</td>
</tr>
<tr>
<td>Front Bend</td>
<td>mm</td>
<td>7.2 × 10^{-4}</td>
<td>6.2 × 10^{-3}</td>
</tr>
<tr>
<td>Front twist</td>
<td>mm</td>
<td>1.8 × 10^{-2}</td>
<td>1.0 × 10^{-1}</td>
</tr>
<tr>
<td>Rear Bend</td>
<td>mm</td>
<td>6.0 × 10^{-3}</td>
<td>1.1 × 10^{-1}</td>
</tr>
<tr>
<td>Rear Twist</td>
<td>mm</td>
<td>1.0 × 10^{-2}</td>
<td>4.4 × 10^{-2}</td>
</tr>
<tr>
<td>Vertical Load</td>
<td>mm</td>
<td>1.0 × 10^{-2}</td>
<td>2.8 × 10^{-1}</td>
</tr>
<tr>
<td>Total</td>
<td>mm</td>
<td>1.8 × 10^{-2}</td>
<td>3.8 × 10^{-1}</td>
</tr>
</tbody>
</table>

[Table 6] Analysis Result Value of Redesigned Frames

<table>
<thead>
<tr>
<th>Load cases</th>
<th>Unit</th>
<th>Original frame</th>
<th>Optimized frame</th>
</tr>
</thead>
<tbody>
<tr>
<td>Weight</td>
<td>kg</td>
<td>102.37</td>
<td>6.48</td>
</tr>
<tr>
<td>Front Bend</td>
<td>Pa</td>
<td>1.8</td>
<td>11.0</td>
</tr>
</tbody>
</table>
3.2 Conclusion

Topology optimization was performed using the finite element method to obtain a safe and simple design model for a three-wheeled mobility frame. These results show no difference from research cases on the conceptual design of automobile parts using phase optimization[18]. Accordingly, the resulting model was post-treated with a skeletal structure that can be produced using each tube. No significant change in the performance was observed after post-processing the phase optimization result[19][20]. The following conclusions were obtained by comparing the conceptual model with the post-processed model.

1) The optimal skeletal frame structure was protruded based on the optimal topological design results for multiple loads.

2) In the optimization process, the maximum strain and stress satisfy the yield condition of the material, thereby proving the safety of the structure.

The results of the study are expected to be utilized in designing various mobility frames for safe and optimal structures. In this study, only the optimization related to the vertical load and torsion was conducted. In the future, research on various conditions such as vibration and shock can be conducted and used as basic data for designing mobility frames for various purposes that satisfy more diverse conditions.

4. Acknowledgment

This study was conducted with the support of Ministry of the Industrial Cluster Competitiveness Enhancement Project (PCC21006).

References

DOI: http://dx.doi.org/10.1016/j.matdes.2009.10.036

DOI: https://doi.org/10.14775/ksmpe.2020.19.12.113

DOI: http://dx.doi.org/10.14775/ksmpe.2015.14.3.074

DOI: http://dx.doi.org/10.7735/ksmte.2015.24.2.211
Lightweight Design Method for Bike Frames using Topology Optimization under Multiple Loads


DOI: https://doi.org/10.12815/kits.2021.20.1.100


DOI: http://dx.doi.org/10.5762/KAIS.2013.14.9.4171


DOI: https://doi.org/10.1007/s001580050176


DOI: http://dx.doi.org/10.7735/ksmte.2013.22.6.969

DOI: https://doi.org/10.5762/KAIS.2020.21.6.1


DOI: http://dx.doi.org/10.7467/KSAE.2012.20.6.146

DOI: https://doi.org/10.20910/JASE.2018.12.4.18


DOI: http://dx.doi.org/10.7467/KSAE.2015.23.3.336

DOI: http://dx.doi.org/10.3795/KSME-A.2012.36.12.1683

DOI: http://dx.doi.org/10.3795/KSME-A.2014.38.3.309