International Conference on Engineering Design and Environmental Sustainability (ICEDES) 2025
ISSN: 3031-7320
Website: https://ic.upstegal.ac.id/index.php/icedes

ANALYSYS DESIGN AND FINITE ELEMENT ANALYSIS FOOTSTEP FOR ATTACHMENT ELECTRIC BIKE

Rendy Fathurrohman [1 *]; Muhammad Agus Shidiq[2], Galuh Renggani Wilis[3], Ahmad Farid[4], Rusnoto[5], Mustaqim[6], Irfan Santosa[7]
Department of Mechanical Engineering, Universitas Pancasakti Tegal, Indonesia.
Corresponding author: ci_ulya@yahoo.co.id

ABSTRACT

The 3D modeling of footstep brackets for electric bicycles will be analyzed using finite element-based software. This research aims to optimize the initial design of the footstep bracket components through the topology optimization method. Topology optimization produced three designs topology design 1, 2, and topology design 3 by varying the holes in the footstep. The static force load chosen was 30 kg. Ductile iron and AISI 4340 materials are used in the footstep bracket. The simulation results show a 23% mass reduction from the initial design, with designs 1, 2, and 3 having respective masses of 0.21, 0.16, and 0.16 kg. For ductile iron material the maximum stress values for designs 1, 2, and 3 are 459.03 MPa, 684.09 MPa, and 643.50 MPa. The FOS values for designs 1, 2, and 3 were 1.2, 0.81, and 0.86. For AISI 4340 material, the maximum stress value from the initial design is 644.60 MPa, topology optimization design 2 686.20 MPa, topology optimization design 3 is 644.06 MPa. The factor of safety (FOS) value for the initial design was 1.1, topology optimization design 2 value 1, and topology optimization design 3 1.1.

***Keywords**: bracket footstep, finite element method, weight, maximal stress, safety factor.*

1. INTRODUCTION

Engineering involves research, development, design, analysis, and production (Teke et al., 2021). Efforts in the engineering sector to create lightweight designs in automotive components (Puji Prasetyono et al., 2023) and lifting hook design analysis (Hasan et al., 2022) have proven useful in reducing the overall weight of the components. Approaching the problem from this perspective reduces weight, but it hardly exploits the full potential for lightness (Albers et al., 2021). The aim of topology optimization itself is to reduce the mass of the component to be tested but still consider the strength of the component (Sakti et al., 2020).

3D bracket modeling in finite element-based software is an effort to replicate actual conditions, with the expectation that the research results will be accurate (Alamsyah et al., n.d.). The footstep bracket is a component of the electric bicycle located at the rear. This component functions as a support for the passenger's feet. When viewed in virtual reality (VR), the design of this component demonstrates a solid concept**.** Good design and aesthetics do not necessarily guarantee safety. Many of these component designs are bent or broken.

This research aims to optimize the initial component design using the topology optimization method with a static load of 30 kg**.** The expected outcome of this research is to produce a design with strong safety factors. Mass reduction was also taken into account in this study with effective mass reduction and a safety factor greater than 1, the design is considered safe and does not exceed the maximum yield strength limit.

2. METODE

2.1 Work of Principal

The research begins by validating the data obtained from the object to be used**.** The electric bicycle footstep bracket is the object of this research, which will be redesigned and simulated using SolidWorks 2024 software. The data generated in the simulation is used to compare results, particularly those from Von Mises stress. Valid data plays a crucial role in ensuring the sustainability of the resulting design. After validating the data, data collection is carried out using both the initial and optimized designs. The research continues by collecting static loading data, including deformation, Von Mises stress, mass, and strain, obtained from the initial design. The data obtained from the simulation results of the initial design are then used for topology optimization.

2.2 Specification of Electric Bike

The electric bicycle specifications in this research are based on the U Winfly D60 type, with a battery capacity of 0.58 kWh (48 V / 12 Ah). It has a maximum speed of 40 km/h and a range of up to 45 kilometers. This electric bicycle weighs 55 kg and has a maximum load capacity of 200 kg. The specifications of the electric bicycle are shown in Table 1.

Table 1. Specification of electric bike

Variables	Value
Empty weight (kg)	55
Maksimum load (kg)	200
Distance traveled (km)	45
Maksimum speed (km/h)	40
AC charging (hours)	4
Length (mm)	1480
Width (mm)	640
Height (mm)	1050

2.3 Design and Topology Optimization Setup

The proposed three-dimensional footstep bracket design uses SolidWorks software. Design optimization is also done using the same software. SolidWorks software is a tool with many features that make tasks easier for researchers. To achieve good design optimization, you must first go through the finite element method (FEM).

Figure 1 explains the dimensions used in the manufacturer's footstep bracket design. The initial footstep bracket design will be used as material for design optimization. The manufacturer's design will produce one design that will be optimized. Figure 2 shows the basic design used for topology optimization. Optimization design 1 is derived from the initial design by reducing the mass by up to 50%. Optimization design 2 is derived from the design proposed by the researcher, reducing the same amount of mass. Thus, optimization design 1 and optimization design 2 achieve the same mass reduction.

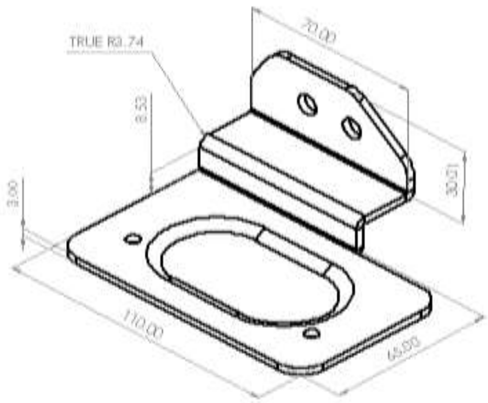
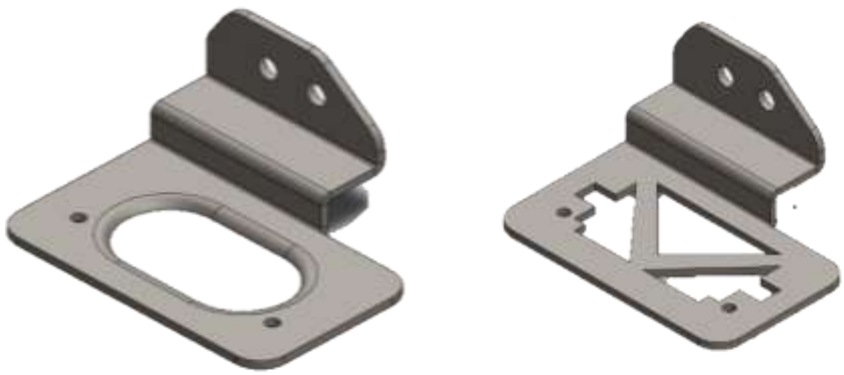


Figure 1. Option 1st, Initial design



(a) **(b)**

Figure 2. The base structure for topology optimization is optimized for (a) standard design 1 and (b) design option 2.

2.4 Materials

The materials used in this research are AISI 4340 steel and ductile iron. There are several reasons for using this material. Material AISI 4340 steel has a very high yield strength, reaching up to 710 MPa, while ductile iron has a lower yield strength of 551,485 MPa. However, ductile iron is easy to find, which is why this material is used in the research. Complete information regarding the differences between the two materials is provided in Table 2.

Table 2. Properties of material

Material	*Property*	*Value*
Steel AISI 4340	ρ, *Density* (kg m−3)	7850
	σy, *Yield strength* (MPa)	710
	v, *Poisson's ratio* (N/A)	0.32
	G, *Shear modulus* (MPa)	80.000
Ductile iron	ρ, *Density* (kg m−3)	7100
	σy, *Yield strength* (MPa)	551.485
	v, *Poisson's ratio* (N/A)	0.31
	G, *Shear modulus* (MPa)	77.000

2.5 Meshing

Figure 3 illustrates the details of the mesh used for the initial design. The curvature-based mesh type is an excellent choice for this research. By using a high-quality mesh, the total number of nodes obtained was 168,436, and the number of elements used was 83,498, with a maximum element size of 0.5 mm. Further specifications of the mesh values are provided in Table 3.

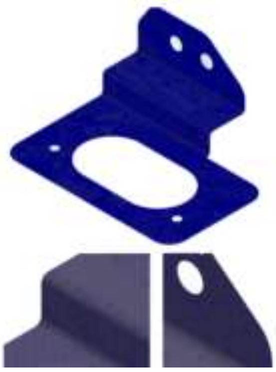


Figure 3. Model mesh of intial design

Table 3. Specification of mesh

Parameter	Value
Total node	168436
Total element	83498
Maximum element size (mm)	0.5
Minimum elemnt size (mm)	0.16
Aspect rasio	2.94
Jacobiant	1.23
Skewness	0.69

Figure 4 illustrates the flow diagram used in this research. Flow diagrams assist in planning research methods in a structured manner.

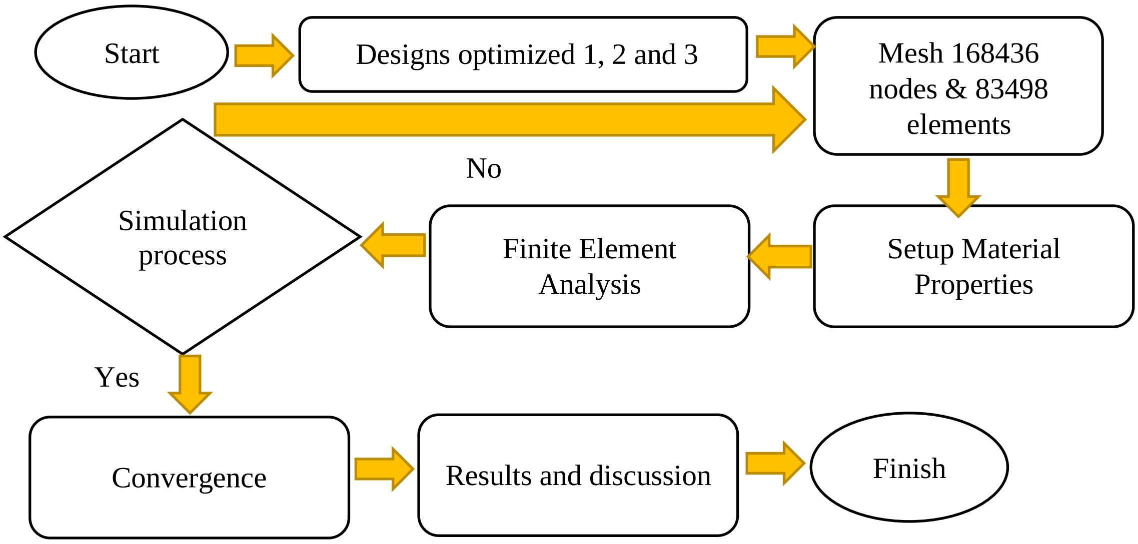


Figure 4. Research flowchart

2.8 Load and Fixed Geometry

The boundary condition in this research is defined by the position of the bolt that connects the footstep bracket to the electric bicycle. The two bolt holes at the top are considered fixed geometry, as shown in Figure 5.

Figure 5. Fixed Geometry

Figure 6 illustrates the load points used on the footstep bracket. Static loading is applied at the point where the passenger places their feet. The load applied is a static load. Researchers assume that the electric bicycle owner weighs 60 kg. However, under real conditions, electric bicycles have two footstep brackets left and right so the static load is divided, with 30 kg assigned to each footstep bracket. Next, the 30 kg load is converted to 294.199 N.

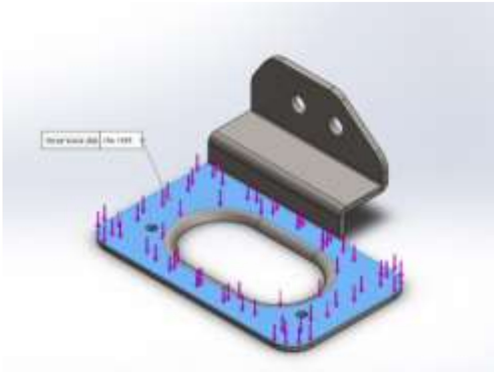


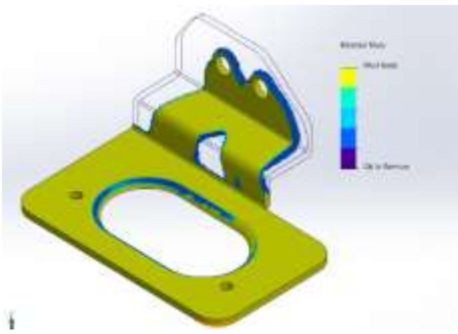
Figure 6. Load

3. Result and Discussion

The goal of this research is to reduce the mass of the initial design while preserving the quality and performance of the footstep bracket. The goal is to minimize material usage to enhance the acceleration of the electric bike while reducing costs. To provide an in-depth analysis of the bracket design for mass reduction, it is essential to discuss the research findings supported by several other studies (Effendi et al., 2023). This chapter presents images and graphs of the research data, showcasing the results of stress, deformation, and the factor of safety for each material used.

3.1 Topology Optimization

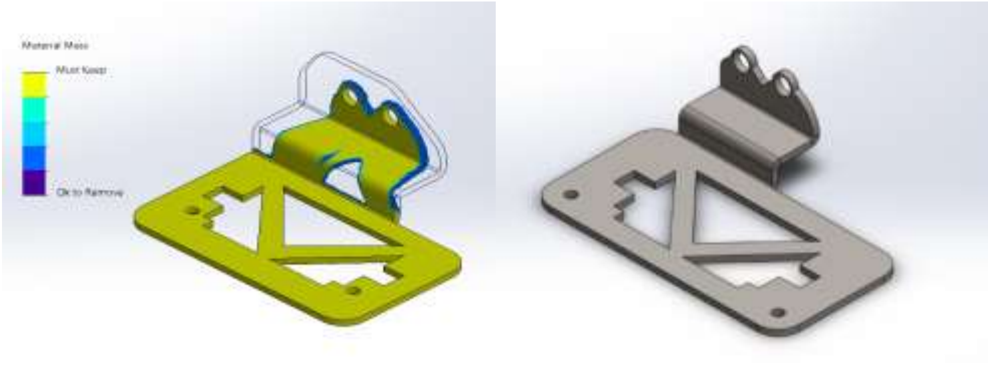
The mesh design resulting from the topology-optimized method does not form a solid structure but rather a design with only a surface (Tristanto et al., 2023). Next, the results of the topology optimization are refined to obtain a functional structure that can be used as intended. Figure 7. a and b show the optimization of topology design 2, which is the result of optimizing topology design 1. Figure 8. a and b show the optimization of topology design 3, which is also derived from topology design 1.



(a)

(b)

Figure 7. **(a)** Result topology optimization 2 **(b)** Design topology optimization



(a) (b)

Figure 8. (a) Result topology optimization 3 **(b)** Design topology optimization 3

3.1 Static Structural Analysis

Static loading is a type of load that remains constant in magnitude and direction (Y.M. Astomo D.S. et al., 2020). Static loading is also required in topology optimization. he load is used to find lightweight topology optimization designs. The topology optimization design results are then modified to ensure that each result has a different weight. Table 4 presents a comparison of the weight results for the initial design (Design 1), topology optimization design 2, and topology optimization design 3.

Table 4. Compare mass

Parameter	Initial design	Topology optimization 2	Topology optimization 3
Mass (kg)	0.21	0.16	0.16

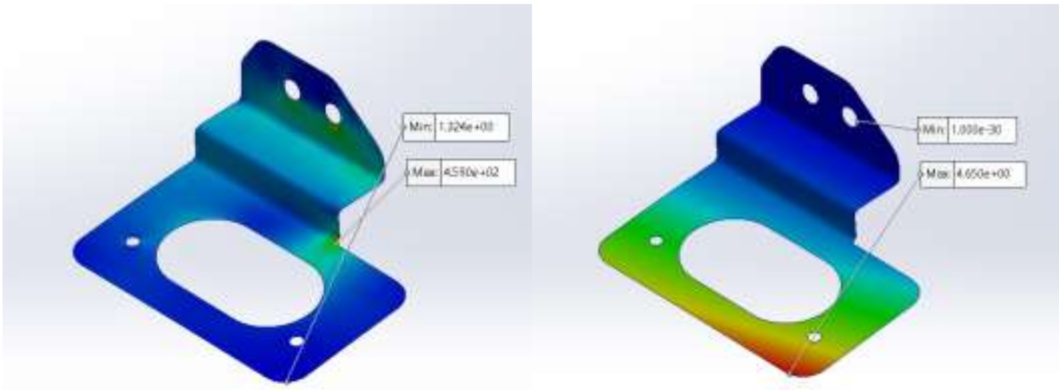


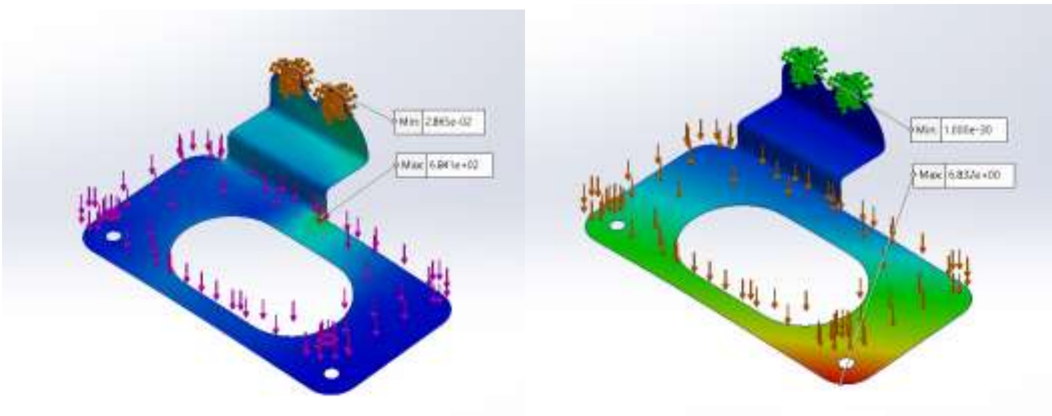
Figure 9. (a) *Stress* Initial design (ductile iron). **(b)** *Displacement* Initial design (iron).

The initial design exhibits a maximum stress of 459.03 MPa, with ductile iron as the material used, as shown in Figure 9.a. The stress is concentrated in the elbow area, which

.

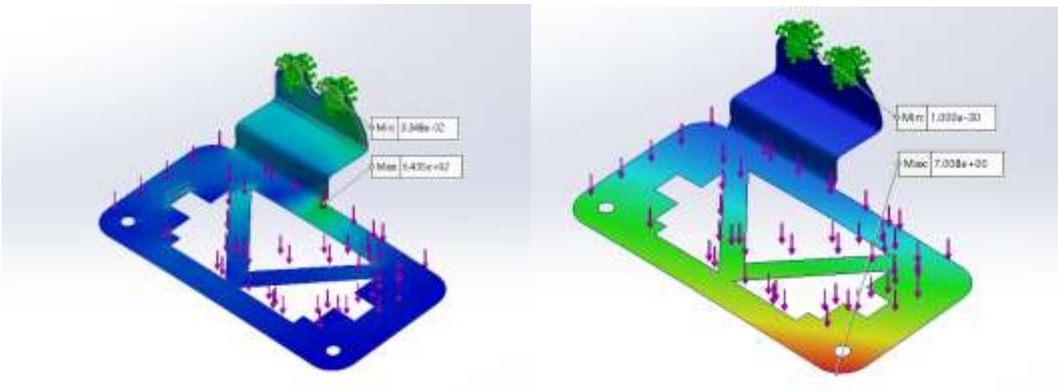
lacks a top design support. Figure 9.b illustrates the displacement position, showing a maximum value of 4.65 mm. Displacement occurs at the end of the component due to the significant distance from the bolt support.

Figure 10.a shows that the topology optimization of Design 2 has a maximum stress value of 684.09 MPa, representing a 32.89% increase from Design 1. The stress occurs in the same area as in Design 1. Figure 10.b shows that the deformation of the footstep bracket shape in Design 2 is 6.8 mm, a 32.35% increase from Design 1. Deformation occurs at the same point, specifically at the end of the passenger foot support.



(a) (b)

Figure 10. **(a)** Von misses/stress topology optimization 2 (ductile iron). **(b)** displacement topology optimization 2 (ductile iron).



(a) (b)

Figure 11. **(a)** Von misses stress topology optimization 3 (ductile iron). **(b)** displacement topology optimization 3 (ductile iron).

Figure 11.a explains the maximum stress in design topology optimization 3, namely

643.50 MPa. The maximum voltage value of topology optimization design 3 is 28.67% greater than the initial design and 5.9% smaller than that of topology optimization design 2. Figure 11.b explains that the deformation value in design topology optimization 3 is 7.0 mm, which occurs at the same end as in the initial design (design 1) and topology optimization design 2. The deformation value of Design 3 has increased by 34.28% compared to Design 1 and by 2.85% compared to Design 2.

Table 5 explains the safety factors obtained by using ductile iron as the material. The initial design had a high safety factor, reaching 1.2. Meanwhile, topology optimization design 2 has a safety factor of 0.81, which is lower than that of the initial design (design 1). The value of topology optimization design 3 is 0.86, which means it is higher than that of topology optimization design 2 and lower than that of the initial design (design 1). The decrease in FOS values in designs 2 and 3 is 32.5% and 28.3%, respectively, compared to design 1.

Table 5. Compare the stress, maximum stress, displacement, and factor of safety of the ductile iron material

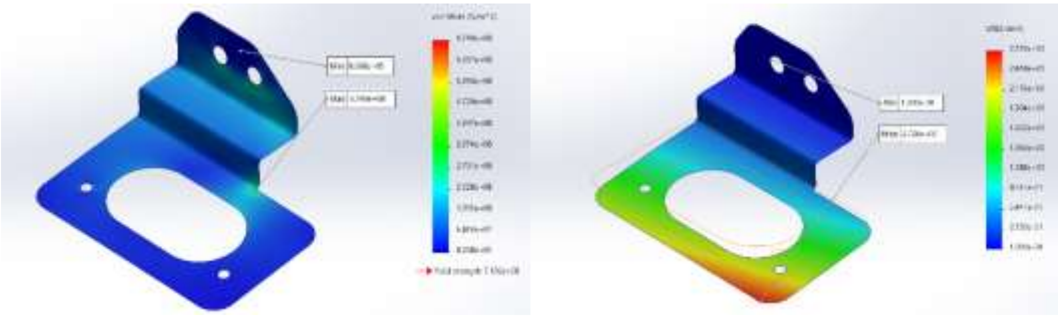
	Initial design	Topology optimization 2	Topology optimization 3
Maximum stress	459.03 MPa	684.09 MPa	643,50 MPa
Displacement	4.6 mm	6.8 mm	7.0 mm
Factor of safety	1.2	0.81	0.86

Figure 12. a shows the von Mises stress result from the initial design using AISI 4340 material. Maximum stress occurs in the elbow area, where the right corner is adjacent to the bracket. The maximum stress value using AISI 4340 material is 644.60 MPa. Meanwhile, Figure 12.b (see attachment) shows the deformation results of the initial design using AISI 4340 material. Maximum deformation occurs at the end of the bracket, which is farthest from the fixed support. The deformation value in the initial design using AISI 4340 material was 2.72 mm.

Figure 13. a shows the maximum stress results in topology optimization design 1, which uses AISI 4340 material. Similar to the initial design, the maximum stress point occurs at the same location. The maximum stress value is 686.20 MPa. This value is greater than the maximum stress value in the initial design, which uses the same material. Figure 13.b (seen in the attachment) shows the deformation results in topology optimization design 1 using AISI 4340 material. The maximum deformation occurs at the end of the bracket, with a value of 3.99 mm.

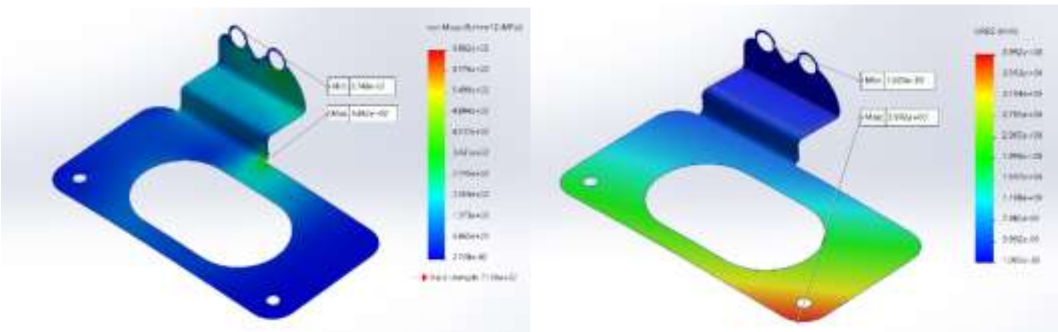
Figure 14. a shows the maximum stress results in topology optimization design 2 using AISI 4340 material. The maximum stress value is 644.06 MPa. Figure 14.b shows the deformation in topology optimization design 2 using AISI 4340 material. The maximum deformation value is 4.10 mm.

Figure 15. a shows the safety factor value for the initial design using AISI 4340 material. The safety factor in the initial design is 1. Meanwhile, Figure 15.b shows the safety factor value in topology optimization design 1, which is smaller than that of the initial design, with a value of 1. Figure 15.c (in the attachment) shows the safety factor value in topology optimization design 2, which is 1.1.



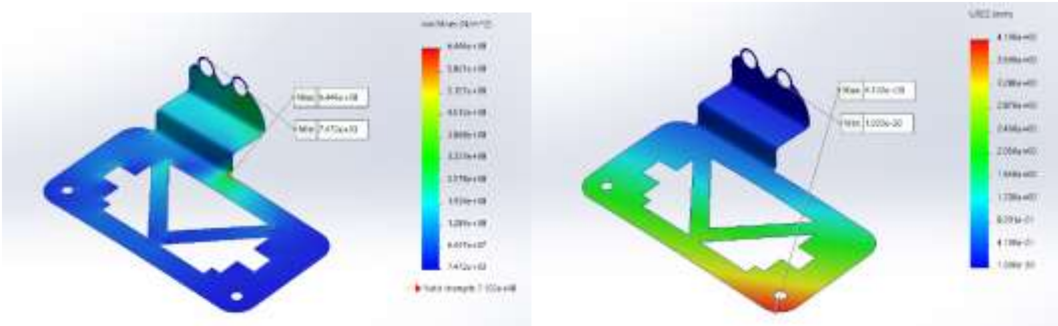
(a) (b)

Figure 12 : (a) Von misses/stress initial design (AISI 4340) **(b)** Displacement initial design (AISI 4340).



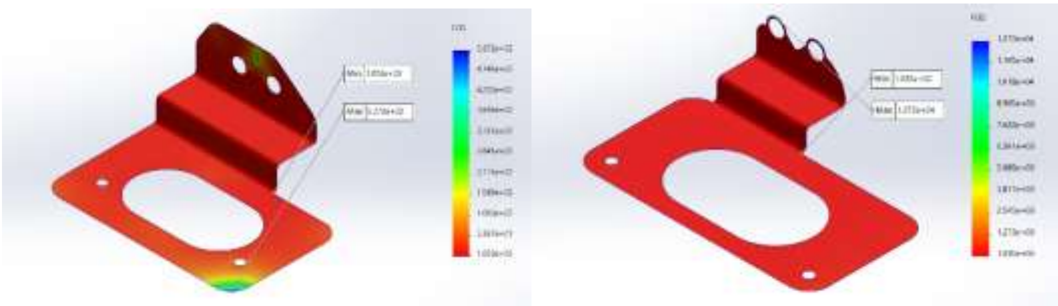
(a) (b)

Figure 13 : (a) Von misses/stress topology optimization 1 (AISI 4340) **(b)** Displacement topology optimization 1 (AISI 4340).

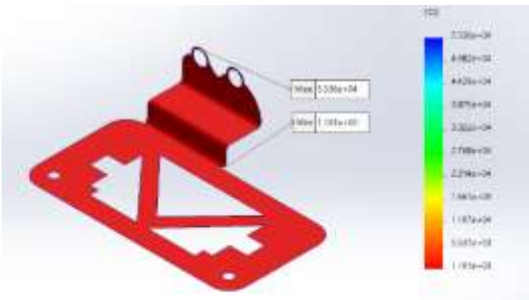


(a) (b)

Figure 14 : (a) Von misses/stress topology optimization 2 (AISI 4340) **(b)** Displacement topology optimization 2 (AISI 4340).



(a) (b)



(c)

Figure 15 : (a) Factor of safety initial design (AISI 4340) **(b)** Factor of safety topology optimization 1 (AISI 4340) **(c)** Factor of safety topology optimization 2 (AISI 4340).

Table 6 compares the maximum stress values, deformation, and safety factors in the initial design, topology optimization design 1, and topology optimization design 2. The material used is AISI 4340, with the smallest maximum stress value found in topology optimization design 2. The smallest deformation value occurred in the initial design, while the largest deformation occurred in topology optimization design 2. The safety factor value in the initial design and topology optimization design 2 is the same.

Table 6. Compare stress maximum stress, displacement, and factor of safety material AISI 4340

	Initial design	Topology optimization 2	Topology optimization 3
Maximum stress	644,60 MPa	686,20 MPa	644,06 MPa
Displacement	2,72 mm	3.99 mm	4.10 mm
Factor of safety	1.1	1	1.1

4. CONCLUSION

This research presents the optimization of the design topology to reduce the mass of the footstep bracket. The topology optimization results for design 2 and design 3 produce a design weight of 0.16 kg each, resulting in a 23% mass reduction compared to design 1. For ductile iron material, the resulting stress values for designs 1, 2, and 3 are 459.03 MPa, 684.09 MPa, and 643.50 MPa, respectively. The displacement values for designs 1, 2, and 3 are 4.6 mm, 6.8 mm, and 7.0 mm, respectively. The safety factor values for designs 1, 2, and 3 are 1.2, 0.81, and 0.86, respectively.

The simulation results using AISI 4340 material show that the stress values for designs 1, 2, and 3 are 458.76 MPa, 684.25 MPa, and 643.06 MPa, respectively. The displacement values for designs 1, 2, and 3 are 4.33 mm, 3.99 mm, and 4.09 mm, respectively. The safety factor value for design 1 with AISI 4340 material is 1.1. In design 2, using the same material, the safety factor value is 1, while in design 3, the safety factor value is 1.1.
Returning to the primary aim of this research reducing the mass of the footstep bracket without compromising the design quality. In this study, designs 1 and 3, both using AISI 4340 material, have the same safety factor value. However, the mass in design 3, using AISI 4340 material, is lighter. Design 3, using AISI 4340 material, is the most recommended in this research.

5. Acknowledgment

The research was founded by the Department of Mechanical Engineering, Universitas Pancasakti Tegal, Indonesia.

6. Authors' note

There is no conflict of interest and free of plagiarism.

7. Credit authorship contribution statement

Rendy Fathurrohman: Conceptualization, Methodology, and Writing. **M, Agus Shidiq & Rusnoto**: Visualization, Formal analysis. **Galuh Renggani Wilis & Ahmad Farid**: Supervision, and Head project. **Irfan Santosa & Mustaqim**: Writing review and editing.

References

[1]Alamsyah, A., Ika Wulandari, A., Harseno Ramadhan, B., Studi Teknik Perkapalan, P., Sains, J., Pangan dan Kemaritiman, T., & Teknologi Kalimantan, I. (2023). Kekuatan Bracket Pada Kapal Ro-Ro Menggunakan Aplikasi Finite Element. SPECTA Journal of Technology. *SPECTA Journal of Technology*, *4*. https://journal.itk.ac.id/index.php/sjt

[2]Albers, A., Holoch, J., Revfi, S., & Spadinger, M. (2021). *Lightweight design in product development: A conceptual framework for continuous support in the development process. Procedia CIRP*, *100*, 494–499. https://doi.org/10.1016/j.procir.2021.05.109

[3]Effendi, M. Y., Ubaidillah, U., Budiana, E. P., & Lenggana, B. W. (2023). Performance analysis on the structure of the bracket mounting for hybrid converter kit: Finite-element approach. *Curved and Layered Structures*, *10*(1). https://doi.org/10.1515/cls-2022-0206

[4]Puji Prasetyono, A., Yudianto, A., & Adiyasa, I. W. (2023). Lightweight Design and Finite Element Analysis of Brake Lever for Motorcycle Application. *ComTech: Computer, Mathematics and Engineering Applications*, *14*(1), 21–32. https://doi.org/10.21512/comtech.v14i1.8604

[5] Muhammad Hasan, Irfan Santosa, Agus Wibowo, Agus Shidik, Ahmad Farid, Hadi Wibowo. Topologi Optimalisasi dan Analisis Perancangan Lifting Hook. Maestro Jurnal Ilmiah. ISSN: 2657-1072. https://www.jurnal.publikasi-untagcirebon.ac.id/index.php/mestro/article/view/389/273

[6] Sakti, A., Kholis, N., Achmad, F., Yudianto, A., Adiyasa, W., & Solikin, M. (2020). *Seminar Nasional Hasil Riset Prefix-RTR OPTIMALISASI STRUKTURAL PADA HANDLE KOPLING SEPEDA MOTOR*.

[6]Teke, I. T., Akbulut, M., & Ertas, A. H. (2021). Topology optimization and fatigue analysis of a lifting hook. *Procedia Structural Integrity*, *33*(C), 75–83. https://doi.org/10.1016/j.prostr.2021.10.011

[7] Tristanto, D. A., Mulyadi, S., Kustanto, Muh. N., Triono, A., & Hardiatama, I. (2023). Analisis optimasi topologi desain support bracket pada steering main shaft mobil TITEN EV-2. *Turbo : Jurnal Program Studi Teknik Mesin*, *12*(1). https://doi.org/10.24127/trb.v12i1.2388