ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun363

Fatigue Analysis and Topology Optimization of a Brake Pedal Manufactured from AISI 1020 Material: A Performance Evaluation

Abakar Hamid Youssouf[1]; Dr. Yahya Işik[2]; Dr. Emre Isa Albak[3]

[1](Students); [2](Professor); [3](Associate Professor)
[1;2;3]Mechanical Engineering Department Bursa Uludad University, 16059 Bursa, Türkiye

Publication Date: 2025/06/13

Abstract: The brake pedal is a critical component for vehicle safety and driving performance. It is necessary to optimize weight and cost due to the high-strength steels or alloys typically used in conventional brake pedals. In this study, the performance of a brake pedal designed using AISI 1020 material is evaluated by topology optimization and fatigue analysis. In this case, a brake pedal designed using SolidWorks was analyzed using Hyperwork 2019 Optistruct and fatigue analysis was performed considering the mechanical properties of AISI 1020 material. The aim of this study is to propose an optimized brake pedal design in terms of both weight and fatigue resistance.

Keywords: *Brake Pedal, Static Structural, Topology Optimization, Fatıgue Analysis.*

How to Cite: Abakar Hamid Youssouf; Dr. Yahya Işik; Dr. Emre Isa Albak (2025) Fatigue Analysis and Topology Optimization of a Brake Pedal Manufactured from AISI 1020 Material: A Performance Evaluation. *International Journal of Innovative Science and Research Technology*, 10(6), 358-367.
https://doi.org/10.38124/ijisrt/25jun363

I. INTRODUCTION

The automotive industry is intensifying its efforts to reduce the weight of vehicle components in line with global goals such as reducing environmental impacts and increasing fuel efficiency. Lightening vehicle components not only reduces fuel consumption but also increases vehicle performance and driving safety. Basic components such as the brake pedal are critical to ergonomic design requirements and driving safety. The continuous loads to which these components are exposed require high fatigue resistance [1]. Mortimer (1974) conducted an important study that determined the ergonomic and safety design criteria of the brake pedal by examining the maximum forces that drivers can apply to the brake pedal. In this study, brake pedal optimization was performed based on the maximum pedal force data obtained by Mortimer.

Fatigue analysis is a critical element in the design of mechanical parts subjected to cyclic loads, especially brake pedals in automotive systems. The phenomenon of fatigue leading to material damage under repeated stresses below the ultimate tensile strength has been extensively studied since the pioneering work of Wöhler in the 19th century [2]. Modern fatigue analysis methods (stress-life [S-N curves] and strain-life approaches) are widely used to predict component durability. High Cycle Fatigue (HCF), characterized by low stress amplitudes and high cycle counts (>10^5 cycles), is particularly important for automotive components such as pedals that are subjected to frequent load cycles [3].

The Goodman method (mean stress correction theory) is often used to account for the combined effects of mean and variable stresses on fatigue life [4]. Finite Element Analysis (FEA) tools such as ANSYS provide detailed simulation of stress distributions and modal responses, allowing for accurate fatigue life predictions [5]. Validation of these models with experimental techniques such as hammer testing increases reliability, as demonstrated by studies on gas turbine engines [6]. This literature highlights the importance of integrating theoretical models, computational tools, and experimental validation to optimize the fatigue performance of mechanical components such as brake pedals.

Topology optimization is an advanced design methodology that aims to maximize structural efficiency with minimum material usage in engineering applications. In the automotive industry, especially the requirements for weight reduction bring this method to the forefront in order to comply with environmental regulations and increase vehicle performance. In the study conducted by Viqaruddin and Reddy (2017), optimum material distribution was achieved without compromising structural strength and rigidity criteria with the topology optimization process performed on the control arm component. In this process; material density as the design variable, mass reduction as the design objective and volume ratio as the restrictive parameter were taken into

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun363

account. The accuracy and engineering validity of topology optimization largely depend on correctly defined boundary conditions. In the relevant study, the connection regions were modeled using rigid elements (RBE2); thus, realistic boundary conditions were obtained in the load transfer regions. As a result of the optimization performed under multi-directional loading conditions applied to the control arm, a 41% reduction in the mass of the component was achieved; However, the strength, stiffness and vibration performance requirements were successfully met [7]. This clearly demonstrates the contribution of topology optimization in engineering design in terms of both efficiency and cost effectiveness.

Topology optimization is generally defined in the literature within the framework of three basic parameters: design variables (mostly material density), design objective (usually minimum weight or maximum stiffness) and design constraints (volume ratio, displacement limits, etc.) [8–9]. Optimization studies carried out in line with these parameters, integrated with solution approaches based on finite element analysis, enable important decisions to be made in the early stages of the engineering design process.

Finally, a fatigue life analysis was carried out to evaluate the durability of the pedal under cyclic loading conditions, in line with the methodologies presented by Quan and Ngoc (2023) in their work on control arm optimization.

II. MATERIAL AND METHOD

In this study, a brake pedal was designed and numerical analyzes were performed to evaluate the structural and fatigue strength of brake pedals used in the automotive industry. The study consists of three main stages: 3D modeling, finite element analysis and topology optimization.

First, the brake pedal model was designed in three dimensions using SolidWorks 2022 software. In the design process, typical usage conditions and geometric limitations for passenger cars were taken into account. The designed model was exported in “.IGS” format and imported into Altair HyperMesh 2019 software to prepare for the analysis process.

AISI 1020, which is in the low carbon steel class, was selected as the material. For this material, mechanical properties elastic modulus, poisson ratio, density, yield strength and fatigue limit were defined in the OptiStruct analysis environment based on literature data.

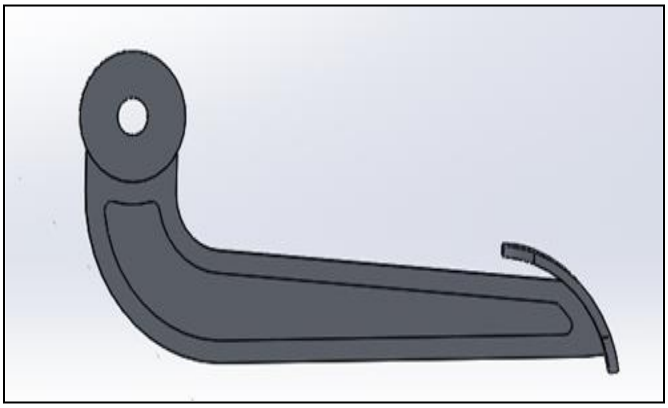


Fig 1 3D Model of Brake Pedal

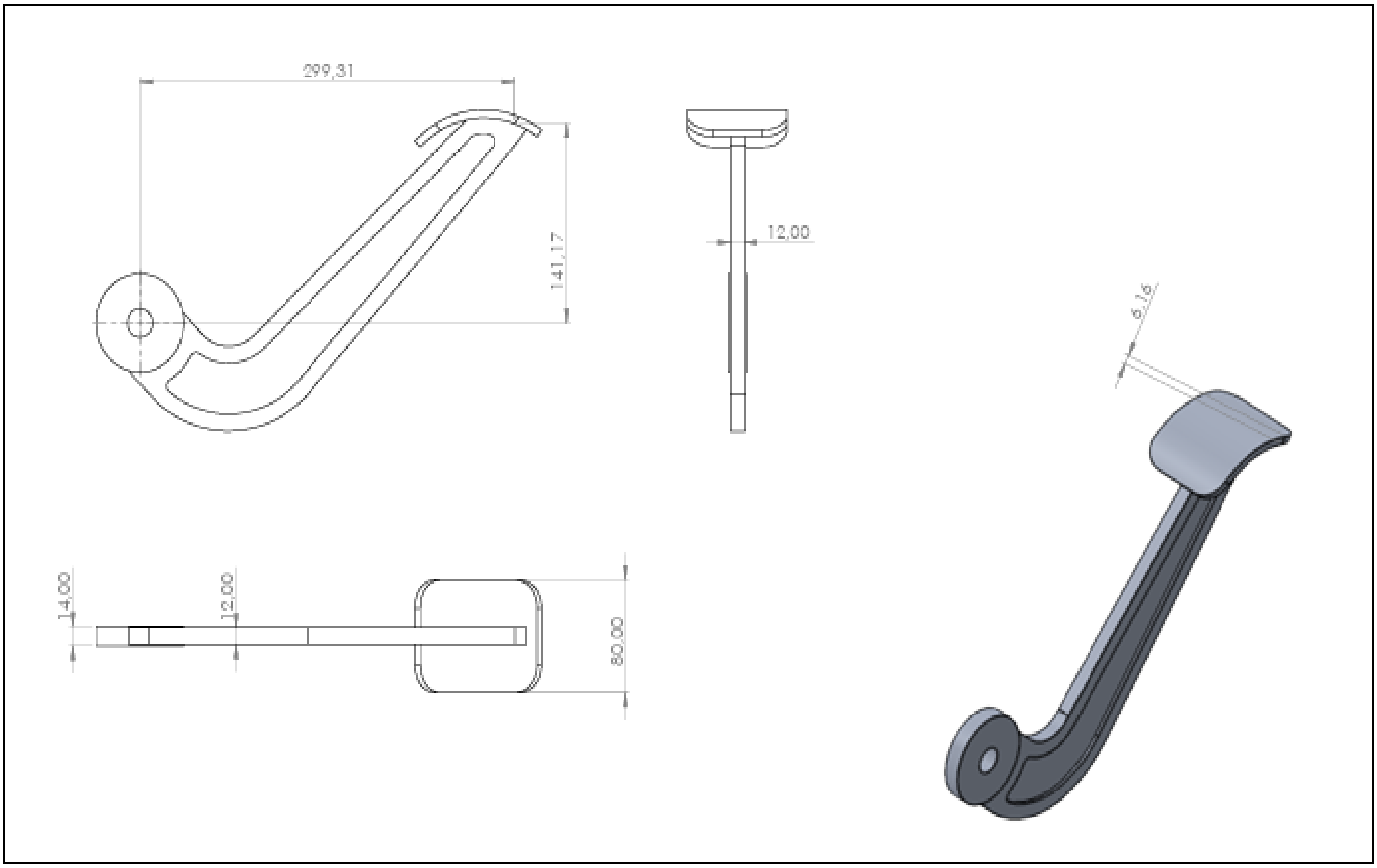


Fig 2 Drafting of Brake Pedal

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun363

➢ *Linear Static Analysis of Brake Pedal*

In static analyses, the force applied to the brake pedal was set to 400 N with reference to Mortimer's study [1]. This force was applied perpendicular to the pedal surface and the pedal pivot point was fixed. As a result of the static analysis, the maximum stress and displacement values on the pedal were obtained as shown in Figure 3. While creating the finite element mesh on the model, elements size of the mesh 2 were preferred. A total of 69657 elements were used to create the mesh and the mesh quality was optimized. Especially tet collapse, jacobian and aspect ratio values were taken into consideration in the quality of the mesh structure.

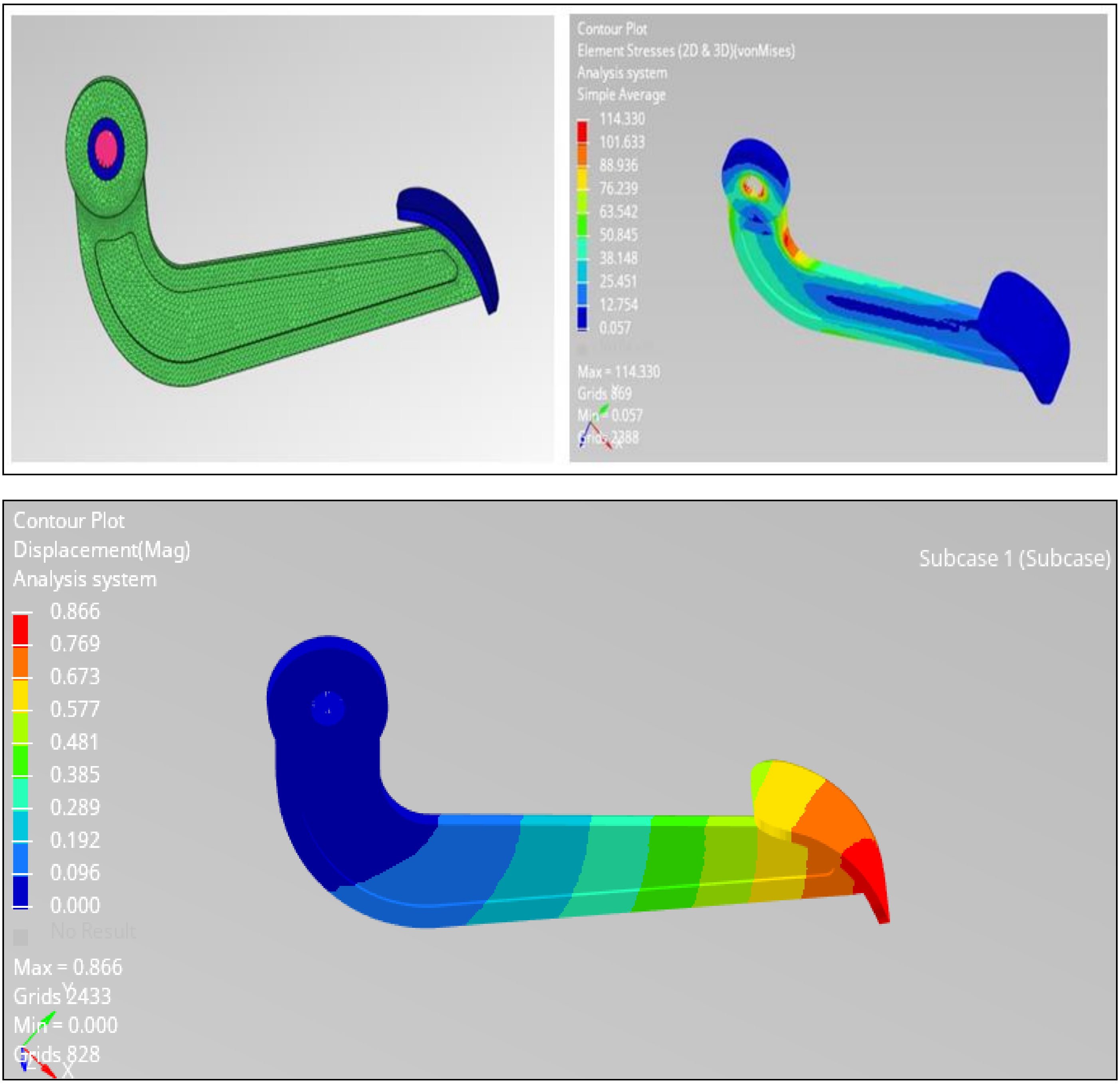


Fig 3 Meshing, Maximum Stress and Displacement

As a result of static analysis, maximum stress was obtained as 114 MPa and displacement as 0.866 mm. now that the static analysis is complete, we will perform topology optimization.

➢ *Topology Optimization*

In the topology optimization process, it was aimed to reduce the mass of the pedal while preserving its structural integrity. For this purpose, the target mass ratio was determined as 30% in the optimization analyses performed via OptiStruct and an optimized structure that maintains the load carrying capacity was obtained. While doing this, the most important Parameters, design variables (mostly material density), design purpose (usually minimum weight or maximum rigidity) and design constraints (volume ratio, displacement limit) were taken into consideration and the process was carried out, as determined in the introduction.

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun363

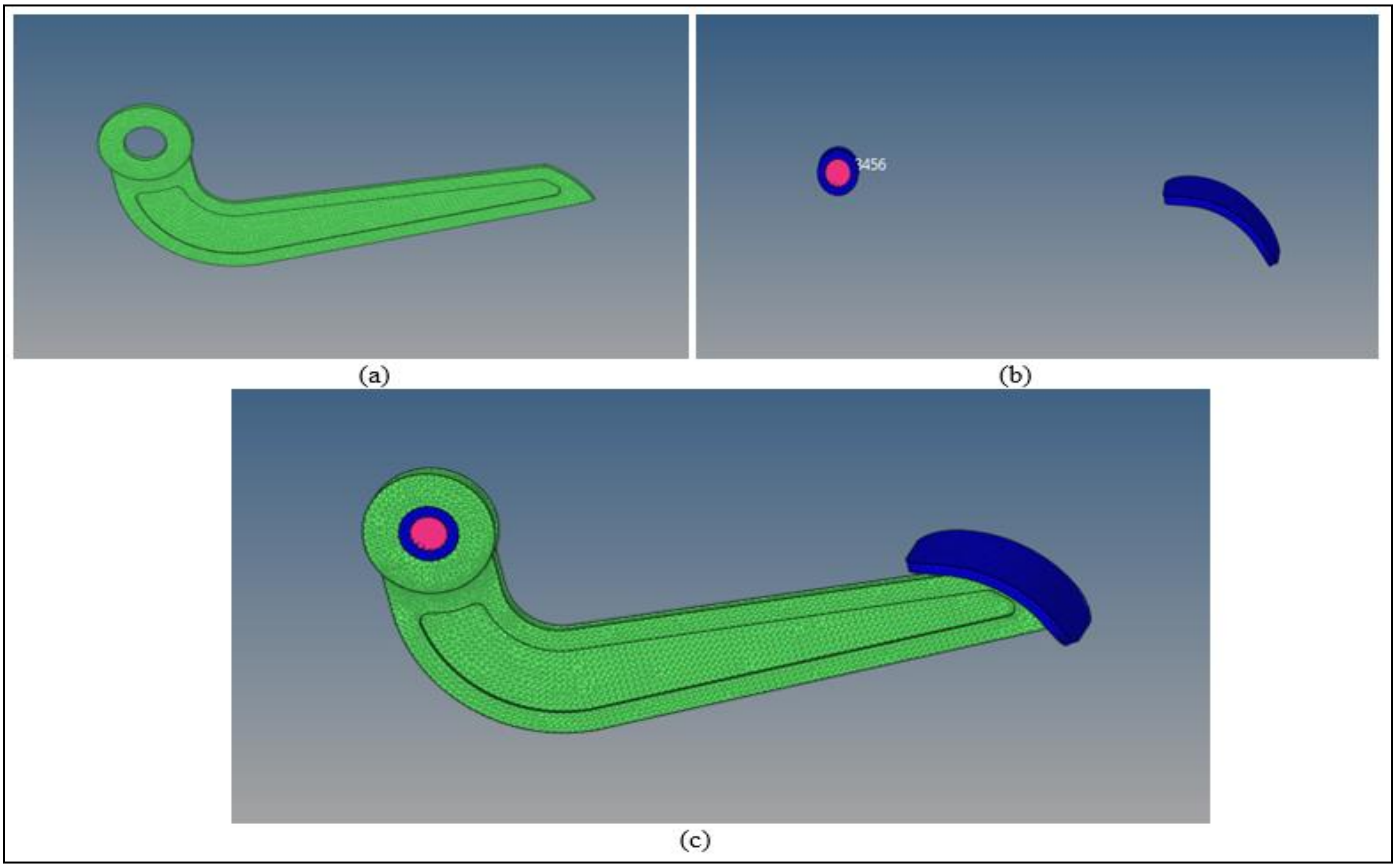
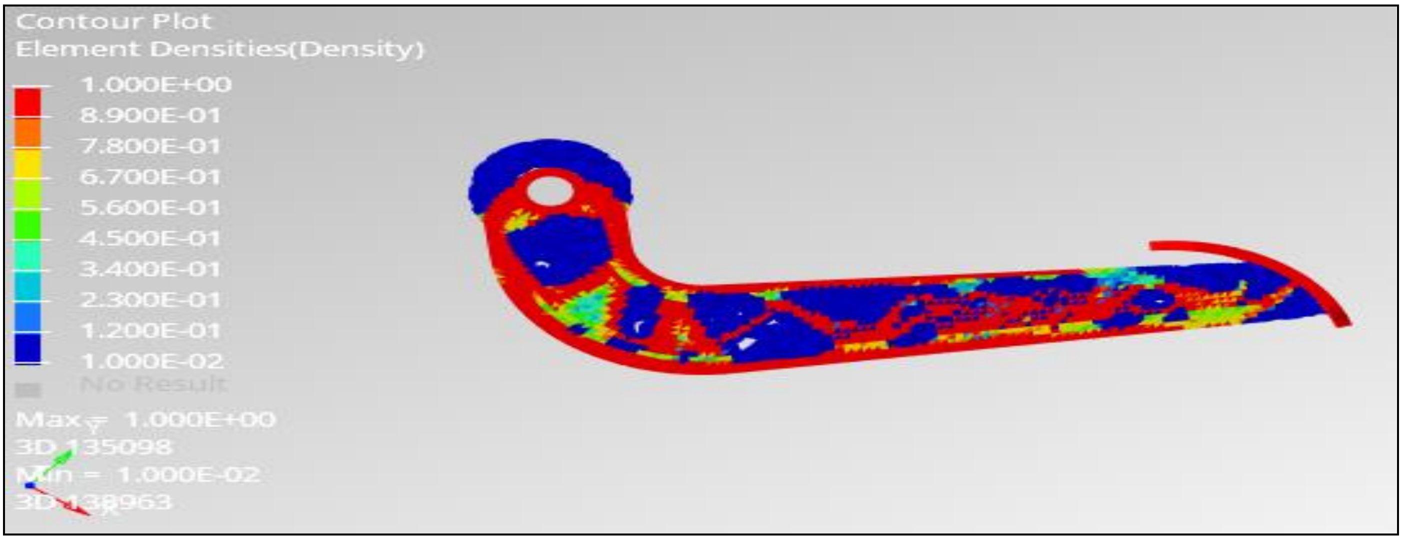


Fig 4 (a) Design Component, (b) Non-Design Component, (c) Full Component

In the topology optimization process, only certain structural regions are allowed to be modified, while the geometric integrity of some regions must be preserved. In this context, two main components are defined in the model: design region (design_component): This region is defined as open to optimization, and the material distribution can be freely changed by the algorithm. The design region represents areas that allow topological evolution in order to increase structural performance.

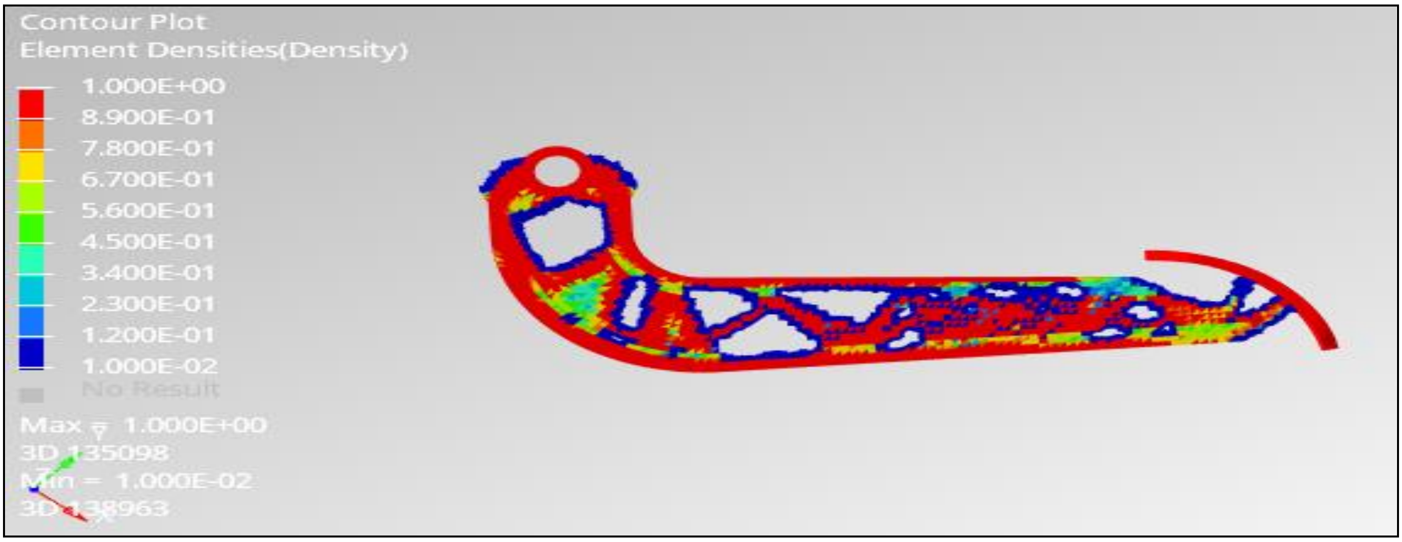
Non-design region (nondesign_component): This region is closed to optimization, and its geometry is kept constant throughout the analysis process. Connection surfaces, assembly points or load application areas that are generally not possible to change due to functional requirements are evaluated in this context.

Topology optimization definitions are made by determining the response, constraint and objective components, respectively. Following these definitions, the optimization process was performed and the results obtained are presented below.

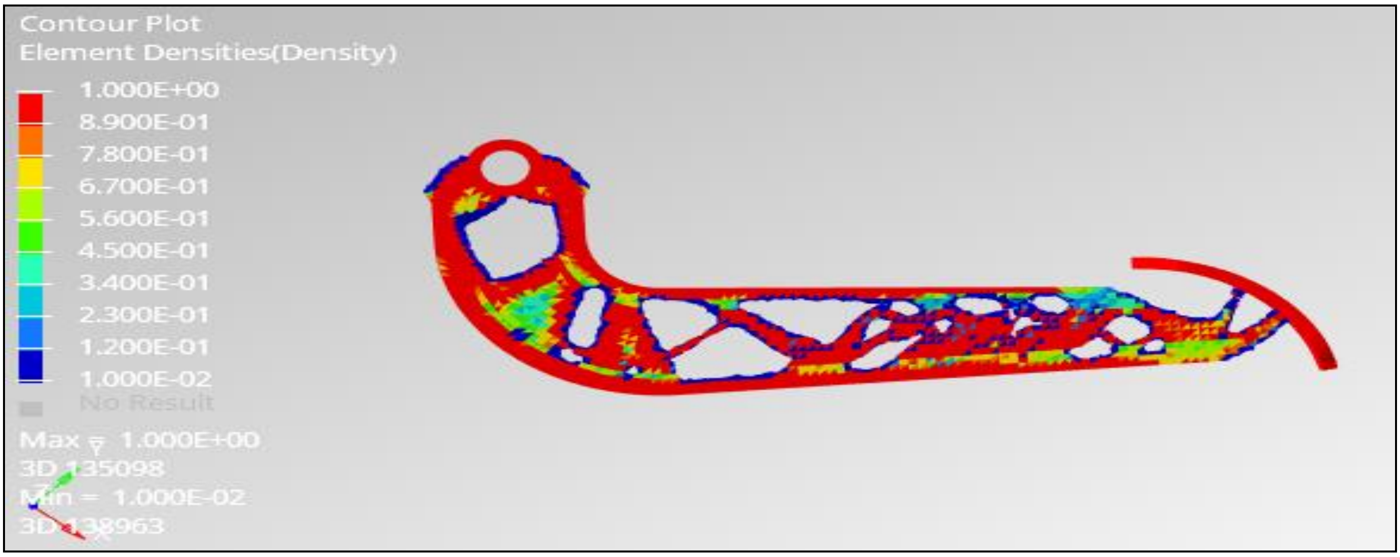


(a)

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun363



(b)



(c)

Fig 5 (a) Iteration41_0.010, (b)Iteration41_0.050, (c) Iteration41_0.219

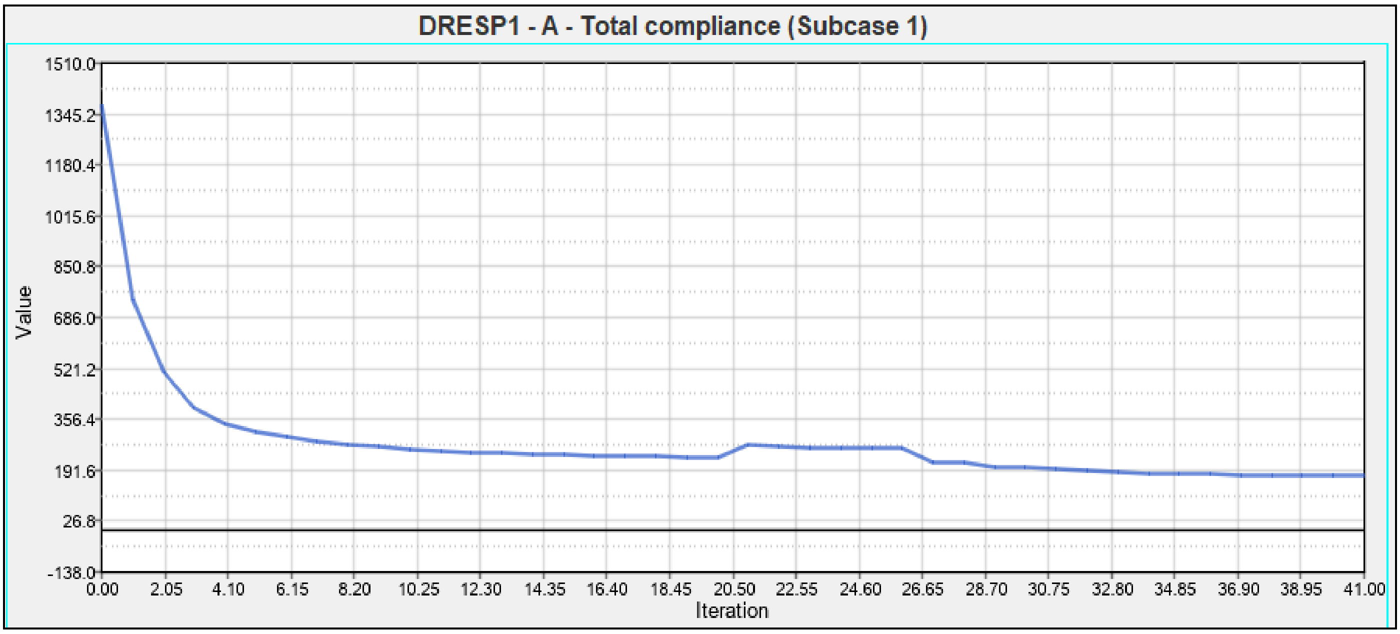


Fig 6 Iteration Graph

➢ *Fatigue Analysis*

After static analysis and topology optimization, fatigue analysis was performed on the brake pedal. Fatigue analysis in OptiStruct was performed using dedicated input cards to ensure accurate simulation of material behavior under cyclic loading. The key cards used are the following: FATPARM: Defines fatigue parameters including ultimate tensile strength and surface finish correction factors. FATSEQ:Specifies the sequence of load steps to be used for fatigue evaluation. FATLOAD: Assigns load cases to be considered in fatigue analysis. FATEVNT: Groups FATLOAD entries to represent complex load histories. FATDEF: Sets default fatigue settings such as the Goodman mean stress correction model. MATFAT: Material card that defines fatigue-specific properties such as S-N curve, strength limit and average stress sensitivity. Collectively, these cards enabled the solver to accurately predict damage accumulation and life prediction under variable amplitude loading conditions.

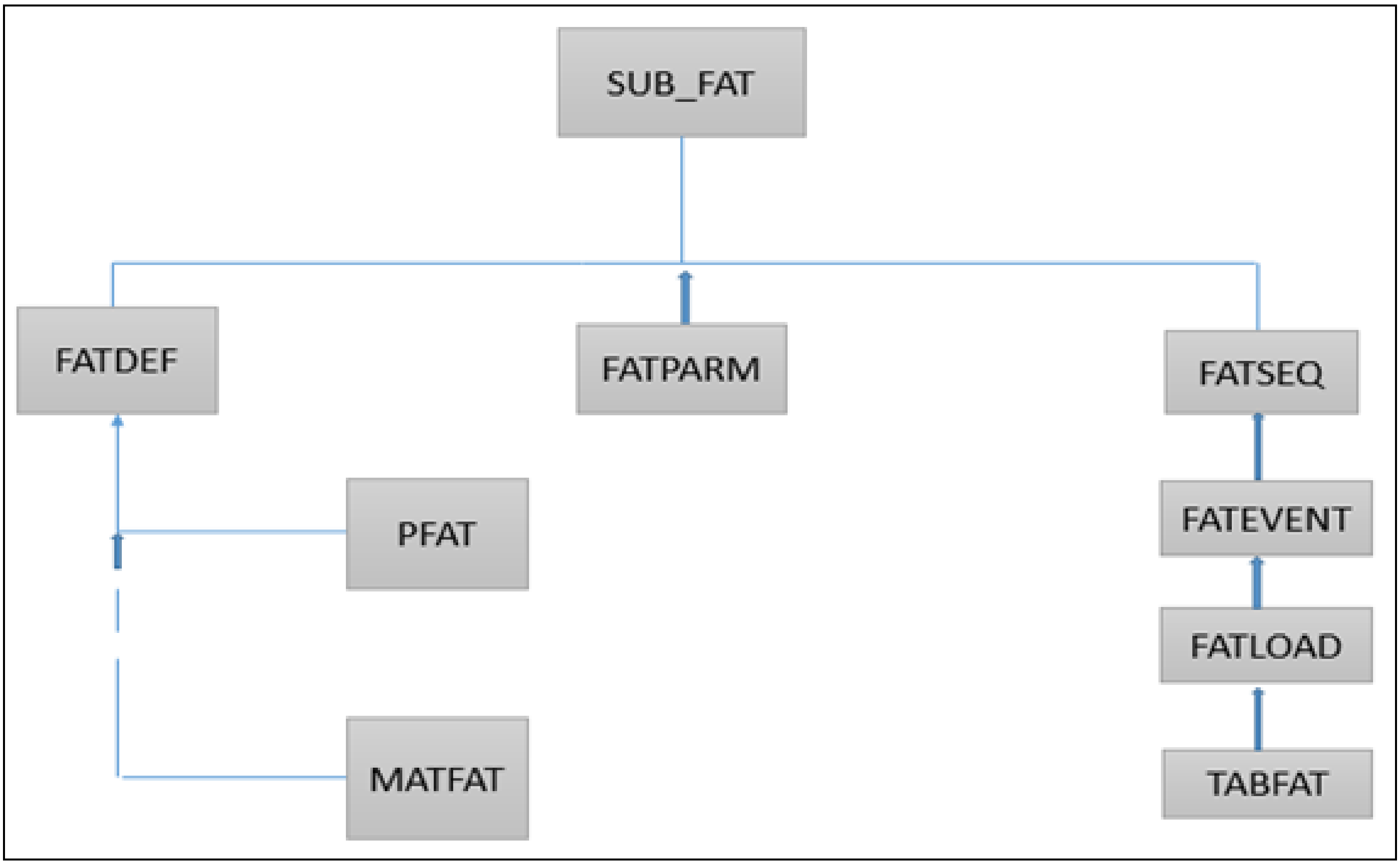


Fig 7 Card Structures that Need to be Defined in Hypermesh for Fatigue Analysis

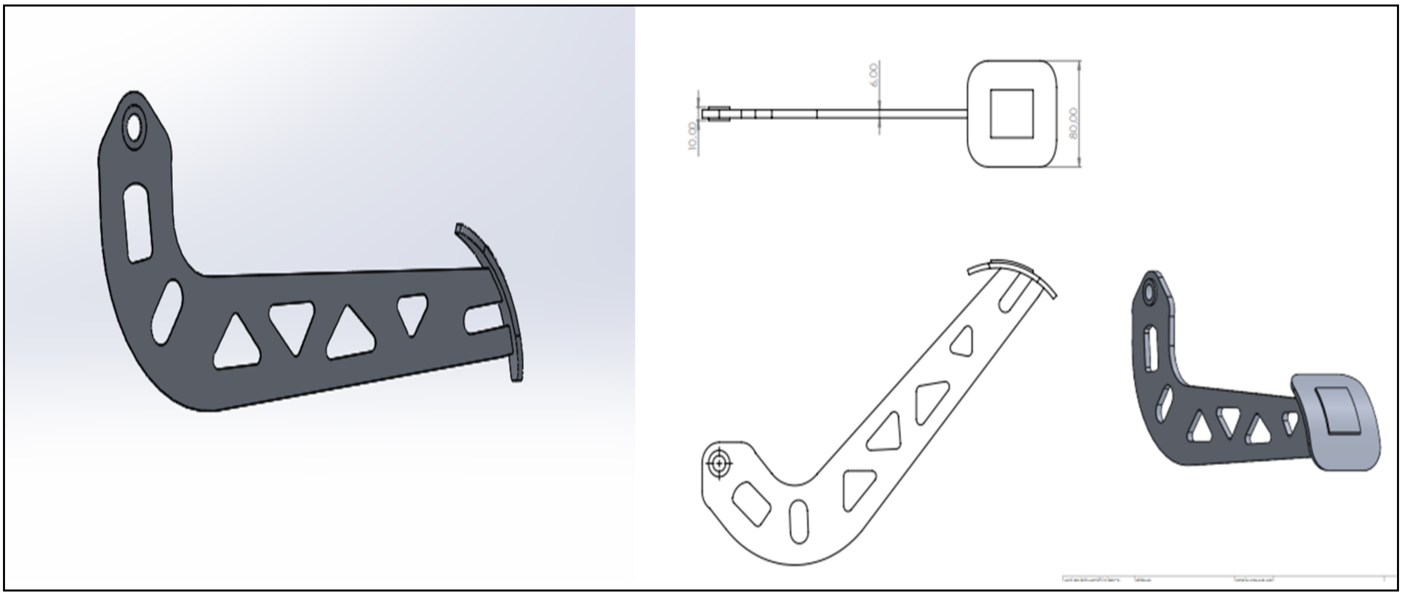


Fig 8 Model & Drafting of Brake Pedal after Topology Optimization

Volume 10, Issue 6, June – 2025 International Journal of Innovative Science and Research Technology
ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun363

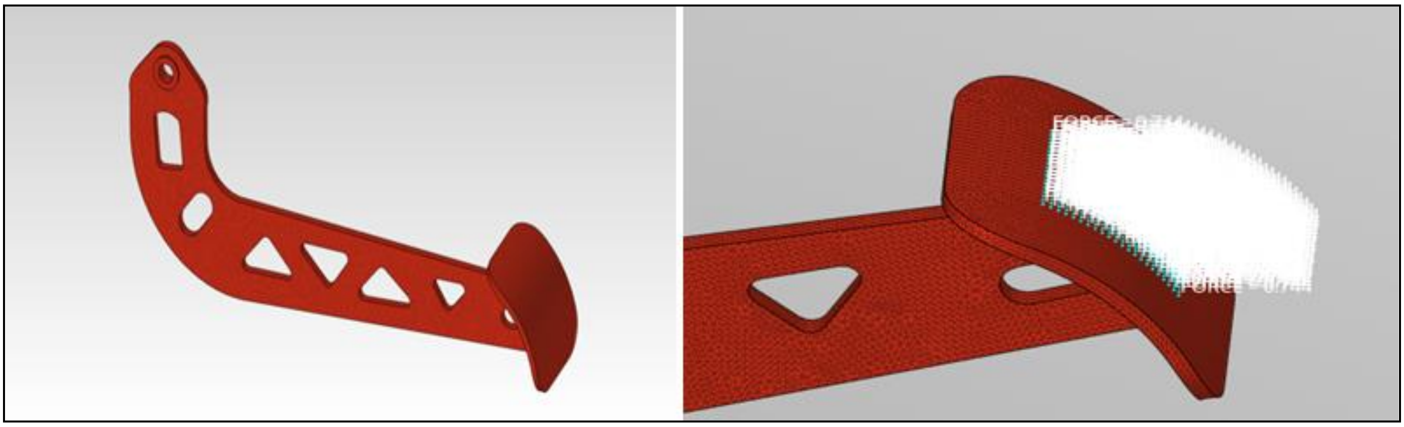


Fig 9 Mesh Generation & Load

Another issue to be considered before starting the fatigue analysis is the material properties, especially the elasticity modulus yield strength, ultimate tensile strength and poisson's ratio parameters are of great importance. They are shown in the table below.

Table 1 Properties of the Material

Properties	**Values**
Modulus of Elasticity	20 GPa
Tensile Yield Strength	350 MPa
Ultimate Tensile Strength	420 MPa
Poisson's Ratio	0.29
Mass Density	7900 kg/m^^{3}

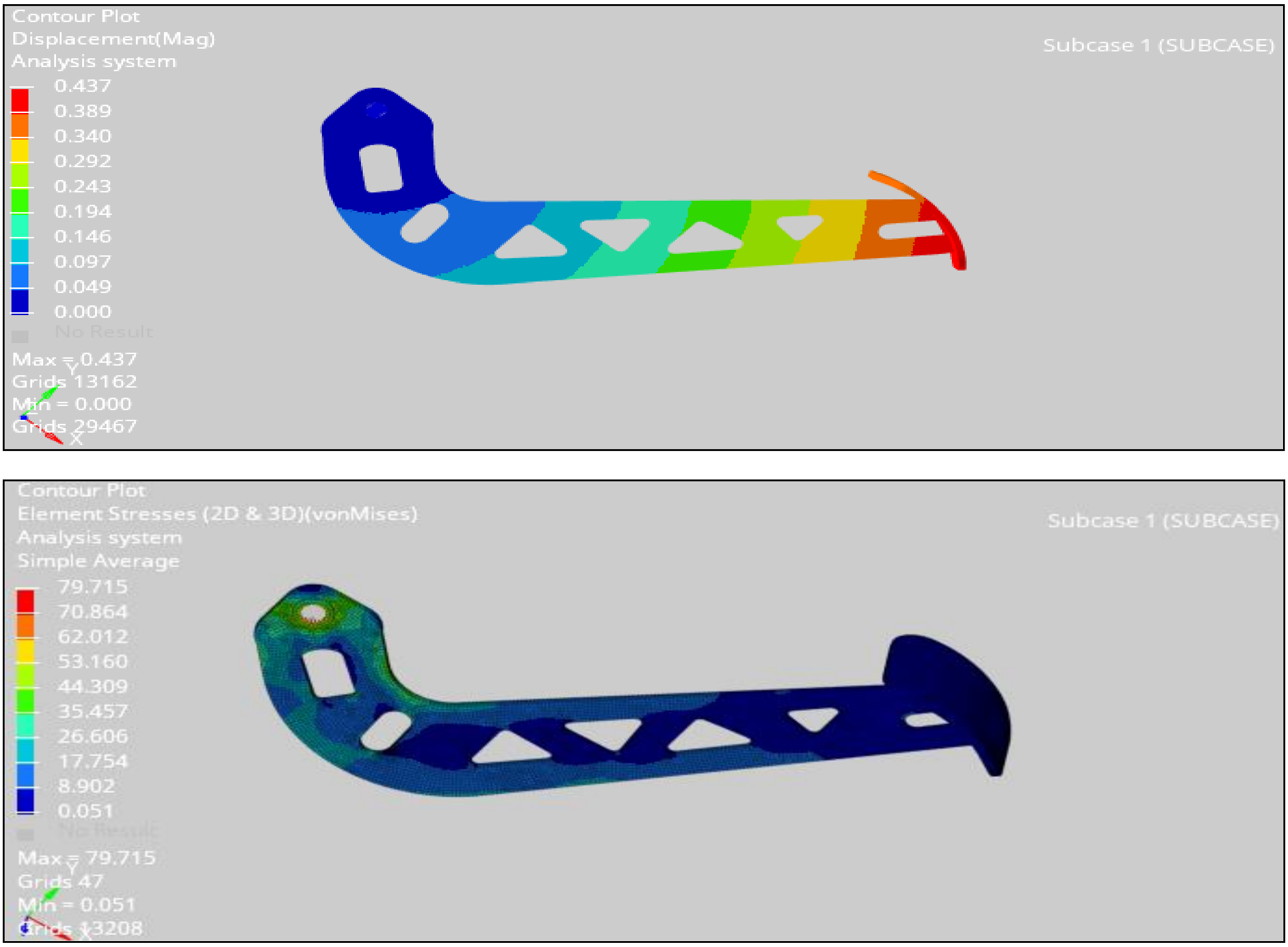
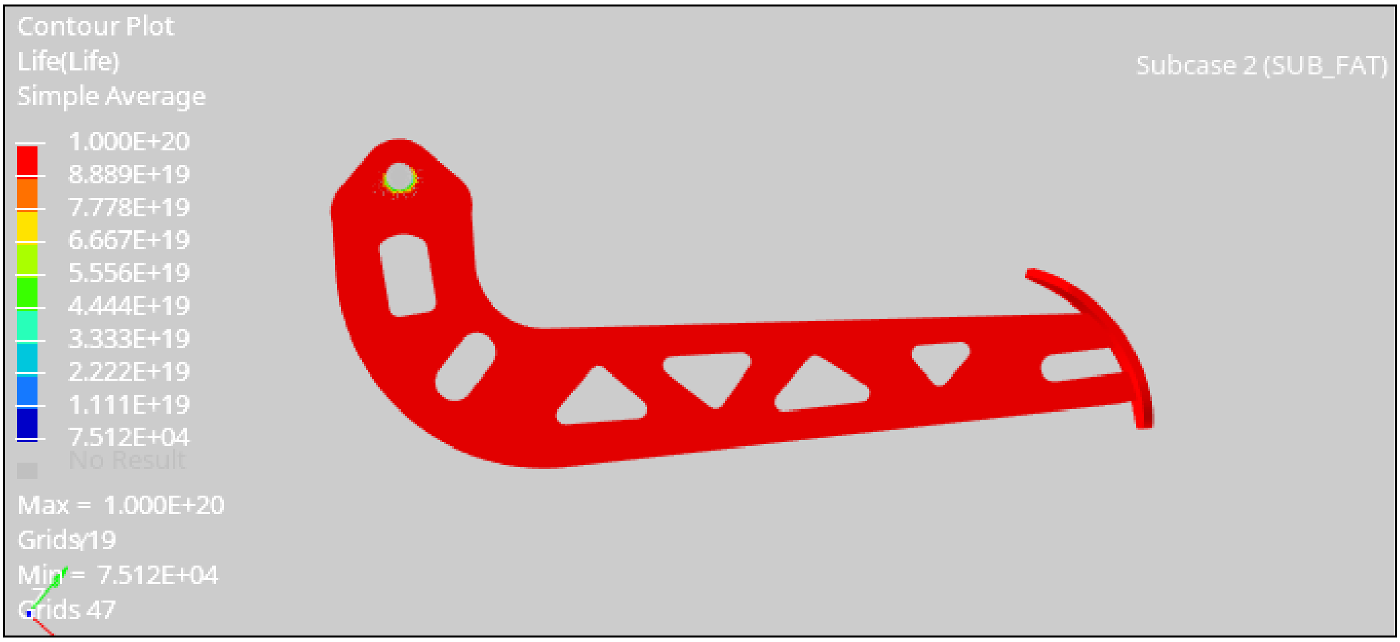


Fig 10 Displacement & Stress values of Fatigue Analysis

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun363



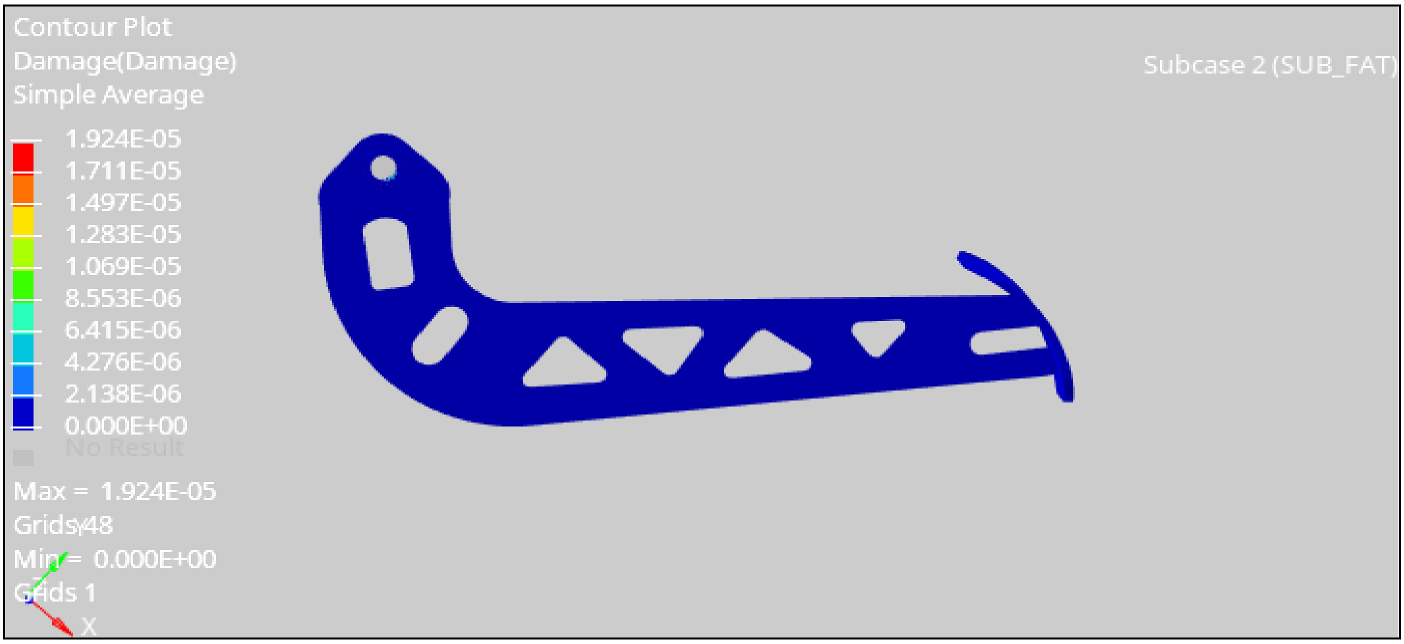


Fig 11 Fatigue Life & Damage Results

After carrying out the fatigue analysis by taking into account all boundary conditions, the results obtained are as follows: maximum displacement was found to be 0.437 mm, maximum stress 79.7 MPa and fatigue life was found to be 7.5x10^4 cycles.

Fatigue Subcase Summary :

Subcase ID	FATDEF ID	FATSEQ ID	MULTI AXIAL	TYPE	PSEUDO DAMAGE	UCORRECT	COMB. STS	SURF. STS	MCORRECT	RANDOM PDF
2	4	7	NO	SN	NO	GERBER	SGVM	NO		

Fig 12 Fatigue Subcase Summary

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun363

III. FINDINDS

➢ *Gerber Fatigue Criteria*

The Gerber criterion provides a margin of safety that models the strength of a material under mean stress (σ_m) and alternating stress (σ_a) with a parabolic relationship. The Gerber criterion determines the stress conditions under which the part can operate before it fractures. Fatigue analysis was done by defining the Gerber criterion. The results of the designed part according to the Gerber criterion are shown in Fig. 13 is given in Fig.

The result obtained after the calculation using the Gerber Curve Equation (Safe limit) is 0.19, which is less than 1, meaning the part is in the safe zone. The result obtained after the calculation using the Gerber Curve Equation (Safe limit) is 0.19, which is less than 1, meaning the part is in the safe zone. In addition, the S-N curve of the material used in the calculations is shown in Fig. 14 is given.

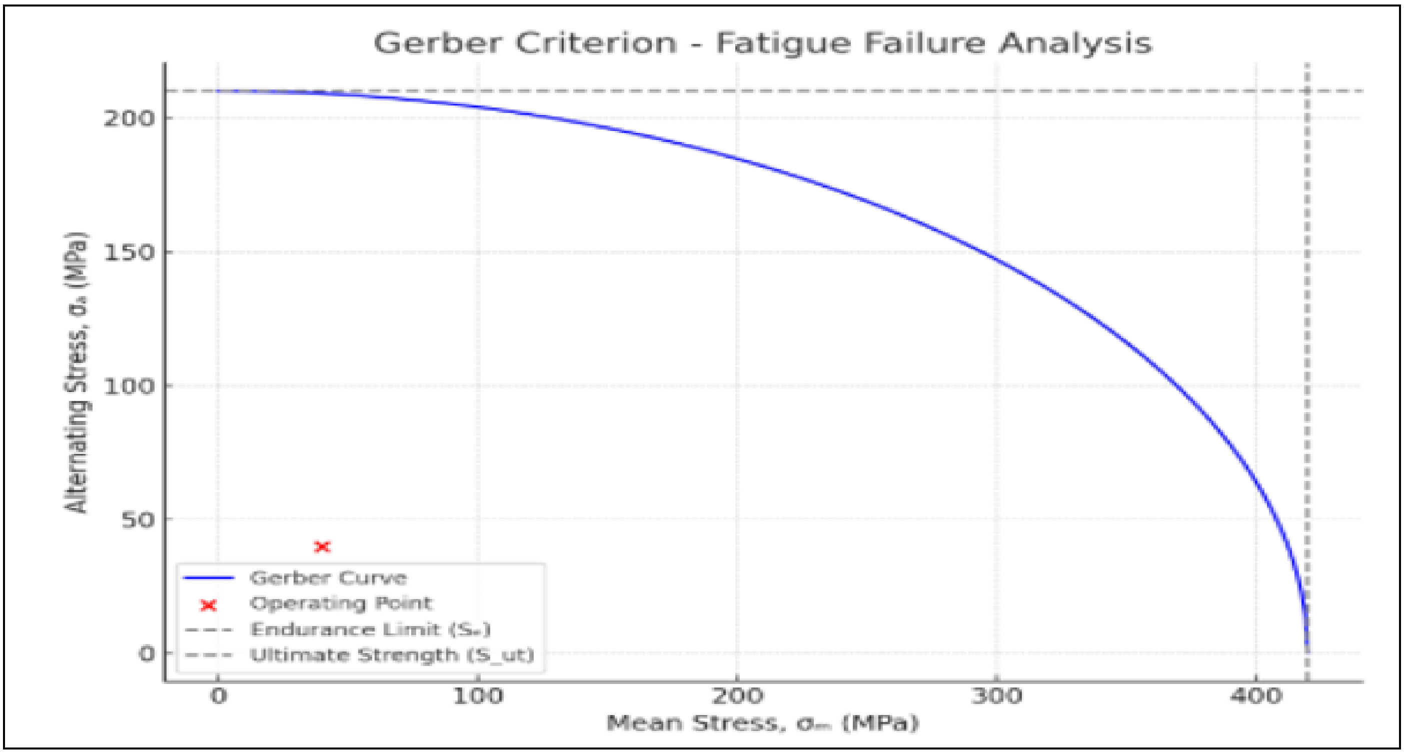


Fig 13 Gerber Criterion- Fatigue Failure Analysis

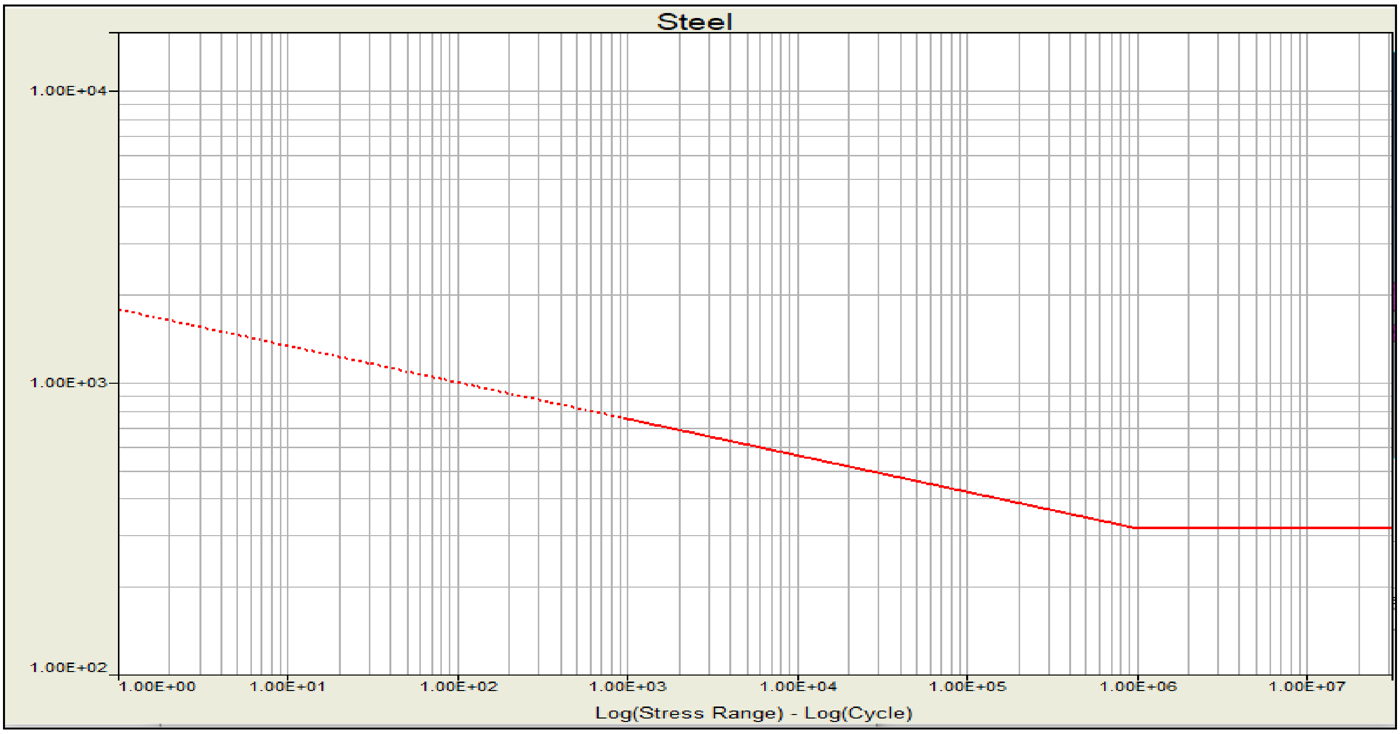


Fig 14 S-N Cuvre of the Material

ISSN No:-2456-2165 https://doi.org/10.38124/ijisrt/25jun363

AVERAGE FATIGUE ANALYSIS RESULTS

Subcase	Damage Top 0.1%	Life Top 0.1%	Damage Top 1.0%	Life Top 1.0%	Damage Top 5.0%	Life Top 5.0%
2	1.524E-05	6.561E+04	2.074E-06	4.822E+05	4.151E-07	2.409E+06

Fig 15 Average Fatigue Analysis Result

IV. CONCLUSION

In this study, a new brake pedal model was developed using SolidWorks and Hypermesh OptiStruct software in order to reduce the weight of the automotive brake pedal, using the fundamental studies conducted by Mortimer (1974) on brake pedal force capacity. Initially designed from AISI1020 steel material, the pedal weight was reduced by 51% from 2.087 kg to 1.014 kg with the applying topology optimization. As a result of static, topology and fatigue analyses, it was determined that the optimized pedal met the required mechanical strength criteria. The findings showed that the performance and reliability of the optimized brake pedal were preserved by reducing its weight and that it could contribute positively to vehicle efficiency. Also, the result obtained after the fatigue analysis according to the Gerber criterion also showed that the design was in the safety region.

REFERENCES

[1]. R. G. Mortimer. (1974). Foot Brake Pedal Force Capability of Drivers, Ergonomics, vol. 17, no. 4, pp. 509-513, 1974.

[2]. Schütz, W. (1996). A history of fatigue.

[3]. Stephens, R. I., et al. (2001). Metal Fatigue in Engineering.

[4]. Engin, Z. (2018). Evaluation of Multiaxial HCF Criteria.

[5]. ANSYS Inc. (2017). Ansys Online Help.

[6]. Sahin, A. O. (2024). High Cycle Fatigue Life Assessment.

[7]. Viqaruddin, M., & Reddy, D. R. (2017). Structural optimization of control arm for weight reduction and improved performance. Materials Today: Proceedings, 4, 9230–9236.

[8]. Bendsøe, M. P., & Sigmund, O. (2003). Topology Optimization: Theory, Methods, and Applications. Springer.

[9]. Darge, S., et al. (2014). Optimization of Lower Control Arm Using Finite Element Analysis. International Journal of Engineering Research and Applications, 4(7), 112–117.