

Structural Optimization of the Brake Pedal Using FEA and UTM

Pramod J. Bhise¹, Rahul R. Kulkarni², Pranay A. Makasare³, Laxman V. Kamble⁴

¹Pursuing Masters of Engineering (Design), Siddhant College of Engineering, Sudumbre, Pune-412109, Maharashtra.

²Department of Mechanical Engineering, Siddhant College of Engineering, Sudumbre, Pune-412109, Maharashtra.

³Department of Mechanical Engineering, Siddhant College of Engineering, Sudumbre, Pune-412109, Maharashtra.

⁴Department of Mechanical Engineering, Siddhant College of Engineering, Sudumbre, Pune-412109, Maharashtra.

Abstract - The safe and efficient operation of electric four-wheelers relies heavily on the reliability and structural integrity of the brake pedal assembly. This project focuses on the structural optimization of a brake pedal designed specifically for an EV platform, with the objective of improving strength-to-weight ratio, durability, and overall performance. Mild Steel (MS) and Cast Iron are selected as the primary material candidates due to their widespread use in automotive applications, favorable mechanical properties, and manufacturability. A detailed 3D CAD model of the brake pedal will be developed and analyzed using Finite Element Analysis (FEA) to evaluate stress distribution, deformation characteristics, and factor of safety under various braking load conditions. Static structural analysis will be performed to compare the performance of the two materials and identify critical stress zones that require geometric refinement. Based on the FEA results, iterative design modifications will be applied to enhance stiffness while minimizing mass. To validate the numerical predictions, a prototype of the optimized pedal will be fabricated and tested on a Universal Testing Machine (UTM). Load-deflection behavior, yield point verification, and ultimate strength assessment will be conducted to ensure compliance with safety and performance standards. The outcomes of this study are expected to provide an optimized brake pedal design with improved structural efficiency, validated through both simulation and experimental methods, thus supporting the development of safer and more energy-efficient electric vehicles.

I. INTRODUCTION

The brake pedal is a fundamental component in automotive braking systems, serving as the driver's primary mechanical interface for initiating and regulating deceleration. As vehicles transition toward electric powertrains, braking architectures continue to evolve with growing emphasis on lightweight design, improved structural efficiency, and enhanced driver safety. Electric vehicles (EVs) demand components that exhibit high stiffness, durability, and reliability while simultaneously contributing to overall weight reduction for extended driving range and improved energy efficiency. Among the various subsystems of an EV, the brake pedal assembly remains a critical structural element that must withstand repeated cyclic loading, impact forces during emergency braking, and long-term fatigue without compromising safety.

Conventional brake pedals are typically oversized to ensure reliability, often resulting in unnecessary weight and material usage. Furthermore, geometric irregularities and non-

optimized sections may introduce stress concentrations that reduce service life or lead to premature failure under extreme conditions. With increasing expectations for performance, safety, and sustainability in modern EVs, there is a clear need for an optimized pedal design that balances strength, stiffness, ductility, and manufacturability.



Advances in computational engineering, particularly Finite Element Analysis (FEA), have enabled detailed evaluation of structural behavior under complex loading scenarios. FEA provides the means to predict deformation, stress distribution, and potential failure regions before physical prototyping, making it an indispensable tool for materials comparison and design optimization. Materials such as Mild Steel (MS) and Gray Cast Iron (CI) are traditionally used for automotive structural components due to their favorable mechanical properties and cost-effectiveness. However, their performance under EV-specific braking demands must be carefully assessed to ensure suitability.

This research focuses on the structural design, analysis, and optimization of a brake pedal intended for use in an electric four-wheeler. The study involves developing a detailed CAD model, conducting static structural FEA for both MS and CI, and comparing critical performance indicators such as deformation, equivalent stress, and safety factor. The goal is to determine the most appropriate material and geometry that ensure high strength-to-weight efficiency, enhanced operational safety, and long-term durability. By integrating simulation-driven optimization with material evaluation, this work aims to contribute to the development of safer and more efficient brake pedal designs for next-generation electric vehicles.

II. LITERATURE REVIEW

1. Wang et al.

Wang and colleagues conducted an FEA-based study of a BSC-type brake pedal using ANSYS to analyze load paths, stress concentrations, and deformation. They applied realistic boundary conditions and simulated both static forces and linkage reactions. The study identified highly stressed regions and proposed geometric modifications to reduce peak stresses. Their results showed that small design changes can significantly improve stiffness without major weight gain. They also highlighted fatigue-prone areas and recommended reinforcement through fillets and ribs. Although contact modeling was simplified and experimental validation was limited, the study provides a solid framework for simulation-driven pedal optimization.

2. Doğan et al.

Doğan and co-authors combined structural optimization with Design of Experiments (DoE) to analyze brake pedal design sensitivities. Using parametric CAD and FEA, they identified which geometric variables most affected stiffness, mass, and safety factor. Their multi-objective optimization showed trade-offs, such as weight reduction leading to stress shifts in certain regions. They developed several Pareto-optimal designs and validated key candidates with additional simulations. Manufacturing considerations were discussed to ensure practical feasibility. The study's main limitation was its assumption of linear elasticity and limited fatigue assessment.

3. Khafidh et al.

Khafidh and colleagues studied the effects of rice husk and ceramic fillers (Al_2O_3 , Fe_2O_3) on the mechanical and tribological behavior of epoxy-based brake pad composites. Their results showed that ceramic fillers improved wear resistance and stabilized friction levels. Organic fillers reduced material weight and cost but negatively affected thermal properties. Microstructural analysis linked filler distribution to performance consistency. Although the study focuses on brake pads, it provides insights on how material selection can influence system-level brake performance. The methods are also useful when considering material choices in broader braking subsystem design.

4. Kulangara et al.

Kulangara and co-authors explored integrating lattice structures with topology-optimized pedal shells to produce lightweight, high-strength components. They compared classical topology optimization with lattice-infused alternatives to evaluate stiffness and manufacturability. Their findings showed significant mass savings with acceptable stiffness reductions when lattices were stress-adapted. However, they noted increased sensitivity to local defects and fatigue issues in lattice regions. The study emphasized the careful translation of topology results into manufacturable lattice geometries. This work is particularly useful for additive-manufactured prototypes validated through UTM testing.

5. Nandanwar et al.

Nandanwar and colleagues conducted a combined optimization study of a brake pedal and bell crank assembly. They modeled the interaction between connected components,

showing that optimizing one part alone may transfer stresses to another. Iterative topology optimization was performed, followed by refining designs into manufacturable geometries. FEA validation confirmed reduced mass without compromising stiffness. The study did not include dynamic or transient loading effects, which they noted as a limitation. Their system-level approach reinforces the importance of considering assembly interactions during pedal optimization.

6. Sarginia et al.

Sarginia and co-authors investigated the fabrication of a metal brake pedal using metal fused deposition modeling (MFDM). The workflow included topology optimization, MFDM printing, debinding, sintering, and mechanical testing. Their results showed that MFDM can produce complex optimized geometries, but porosity and dimensional errors affect mechanical strength. Mechanical tests revealed lower-than-wrought material properties, requiring post-processing improvements. The study highlights both the flexibility and limitations of additive manufacturing for functional components. It serves as a useful reference for bridging simulation with real-world UTM validation.

7. Bobrowskii et al.

Bobrowskii and colleagues applied static, modal, and fatigue FEA to a Formula Student brake pedal assembly under realistic racing load conditions. They incorporated regulatory safety factors and validated their simulations with bench tests. The study emphasized the importance of accurate load cases, mesh refinement, and safety considerations. Instrumented fatigue testing provided real load data to improve simulation accuracy. Their methodology represents best practices for simulation-experiment integration. Although race-specific, the findings translate well to heavy-duty pedal optimization projects.

8. De Vito Jr. et al.

De Vito Jr. and co-authors developed and validated topology-optimized brake pedals for Baja competition vehicles. They reconstructed optimized geometries into manufacturable forms and produced prototypes via CNC and additive manufacturing. Experimental tests showed good agreement with FEA predictions for deformation and load capacity. Some discrepancies were noted due to printing artifacts and test fixture compliance. The authors recommended practical post-processing strategies to improve durability. This study demonstrates a reliable process for pairing topology optimization with experimental UTM validation.

9. Mass-Reduction Topology Optimization Study

This study focuses on reducing pedal mass while maintaining a minimum safety factor using topology optimization constraints. The results showed significant mass savings while keeping stresses and displacements within allowable limits. Sensitivity studies highlighted how load magnitude and boundary assumptions affect optimization outcomes. The authors discussed how overly conservative safety factors can limit optimization potential. They also stressed the importance of translating optimized geometries into manufacturable designs without losing structural integrity. The findings provide clear guidance on balancing weight reduction with structural safety.

10. Akçay and İlkılıç

Akçay and İlkılıç explored the use of AI-based optimization tools such as surrogate modeling, neural networks, and genetic algorithms for brake pedal design. They trained models on FEA-generated datasets to predict mass and stress outcomes for new design variants. Their approach significantly reduced computational time compared to full FEA runs. However, the effectiveness relied heavily on the diversity and quality of training data. They recommended hybrid workflows where AI-generated results are validated through final FEA simulations. This work highlights the growing role of machine learning in accelerating structural optimization.

Novelty of work: The novelty of this work lies in the integrated approach of designing, analyzing, and validating an EV-specific brake pedal using both FEA and UTM methods while comparing two traditional automotive materials—Mild Steel and Gray Cast Iron—under identical loading conditions. Unlike conventional studies that focus only on simulation or single-material evaluation, this research provides a direct material-to-material structural performance comparison for EV braking requirements, highlighting differences in deformation, stress response, weight, and safety behavior. The work introduces a customized EV-oriented brake pedal geometry, designed with considerations for lightweighting and enhanced load distribution to minimize stress concentrations. The inclusion of emergency braking load simulation with realistic boundary conditions adds practical relevance to the optimization process. Furthermore, the research emphasizes simulation-driven design refinement, demonstrating how small geometric modifications significantly improve stiffness and safety without increasing mass. A key novelty is the planned experimental verification through UTM, which strengthens the reliability of the FEA results and provides real-world validation—an aspect often missing in similar pedal optimization studies. By combining CAD modelling, material comparison, static structural analysis, and experimental planning, the study presents a comprehensive methodology that can be extended to other EV components requiring structural optimization.

III. PROBLEM STATEMENT

Conventional brake pedals used in electric four-wheelers are often over-designed, heavier than necessary, and susceptible to stress concentration and long-term fatigue under repeated braking loads. The lack of optimized geometry and material selection can lead to excessive deformation, reduced braking efficiency, and potential structural failure. Therefore, there is a need to design and structurally optimize a brake pedal using Finite Element Analysis (FEA) and validate its strength, stiffness, and safety through Universal Testing Machine (UTM) testing, specifically evaluating Mild Steel and Cast Iron as viable materials for EV applications.

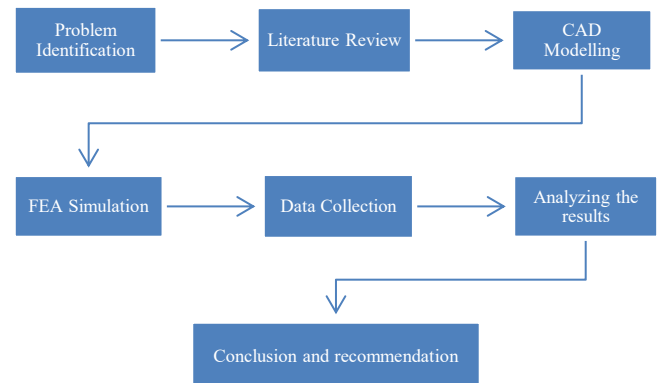
IV. OBJECTIVES

The aim is to design and optimize an EV 4-wheeler brake pedal by applying design optimization and finite element analysis (FEA), with the goal of reducing its weight while

ensuring strength, durability, and safety. The primary objectives are:

- To design a structurally efficient brake pedal model suitable for an electric four-wheeler using CAD tools.
- To perform Finite Element Analysis (FEA) to evaluate stress distribution, deformation, and factor of safety under various braking load conditions.
- To fabricate prototype and conduct Universal Testing Machine (UTM) experiments to validate FEA results for stiffness, yield strength, and ultimate load capacity.

RESEARCH METHODOLOGY



V. FINITE ELEMENT ANALYSIS

Finite Element Analysis (FEA) is a numerical technique used to obtain approximate solutions to complex engineering problems involving stress, deformation, vibration, heat transfer, and multiphysics interactions. The method divides a continuous body into a finite number of smaller, interconnected elements that collectively approximate the behavior of the actual structure under applied loads. Each element is governed by mathematical shape functions that describe how displacement, strain, and stress vary within the element. By assembling the stiffness matrices of all elements and applying relevant boundary conditions and loading conditions, the method produces a global system of equations that represents the response of the entire structure.

The core advantage of FEA lies in its ability to handle geometries, materials, and boundary conditions that are too complex for analytical solutions. It enables engineers to visualize stress concentrations, predict failure modes, and optimize component geometry before manufacturing. For mechanical components such as brake pedals, FEA helps evaluate critical parameters like von Mises stress, total deformation, strain distribution, and factor of safety under static and dynamic loading. The accuracy of FEA largely depends on mesh quality, material modeling, element selection, and the fidelity of applied loads and constraints. Therefore, understanding the underlying theory ensures proper interpretation of results and enhances the reliability of numerical predictions.

ANSYS is a widely used engineering simulation software suite that implements Finite Element Analysis for structural, thermal, electromagnetic, and fluid-related problems. Its structural analysis tools allow engineers to model, mesh, and simulate real-world loading conditions with high accuracy. ANSYS provides a versatile environment for defining material

properties, assigning contacts between parts, applying forces or displacements, and evaluating responses such as stress, deformation, strain energy, and fatigue life. The Workbench interface integrates CAD modeling, meshing, solver settings, and results visualization into a unified workflow, making it well-suited for both academic research and industrial applications.

In this study, ANSYS Static Structural is employed to analyze the brake pedal designed for an electric four-wheeler. The software enables the creation of a refined mesh, realistic boundary conditions, and load cases that reflect actual braking forces. By simulating the pedal made from Mild Steel and Gray Cast Iron under identical conditions, ANSYS assists in comparing their performance objectively in terms of stiffness, stress distribution, and deformation. The ability to evaluate multiple materials and design iterations quickly makes ANSYS a powerful tool for structural optimization. Its numerical accuracy and post-processing capabilities also provide a strong foundation for subsequent physical validation through Universal Testing Machine (UTM) testing.

Static Structural Analysis of MS Brake Pedal:

Geometry:

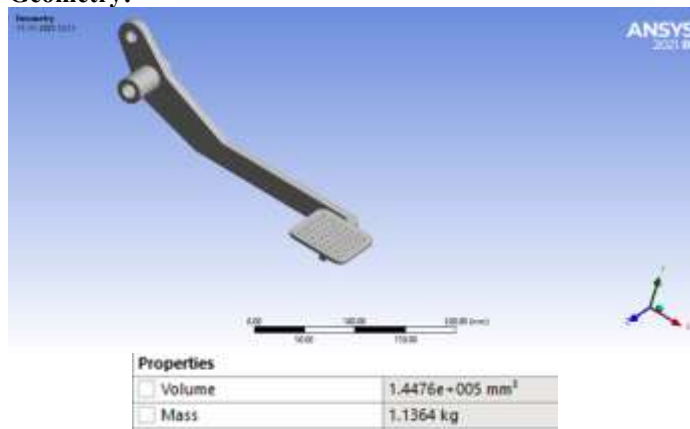


Fig. Geometry of MS Brake Pedal.

Material Properties:

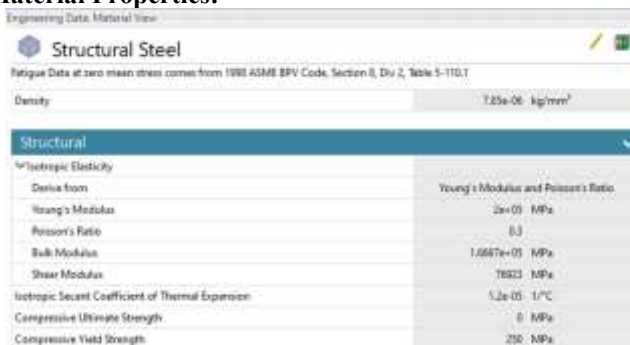


Fig. Material properties of Structural Steel.

Meshing:

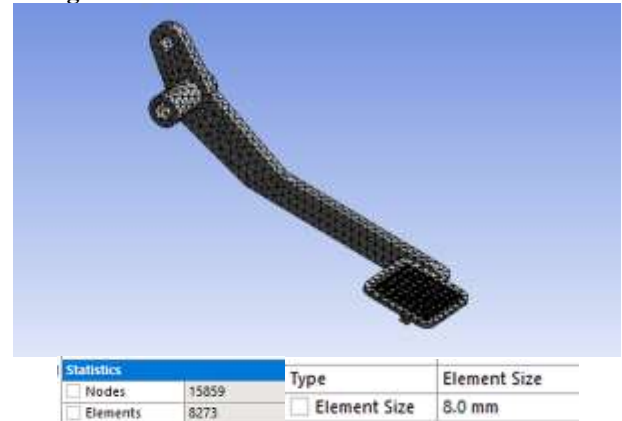


Fig. Meshing Details of MS Brake Pedal.

Boundary Conditions:

Import the modeled braking system, and modify the constraint relationship between components, so that it can carry out the simulation experiment normally, and get the correct simulation results. A braking force perpendicular to the pedal surface is established in the static structural.

In a simple experiment, it is known that for an adult of 60kg, the leg force applied in the sitting position is 200N.

Therefore, in the case of emergency braking, the simulation braking force value is increased to 250N to ensure the stability of pedal structure in case of emergency. Therefore, 250N braking force perpendicular to the pedal face down is applied on the tread surface.

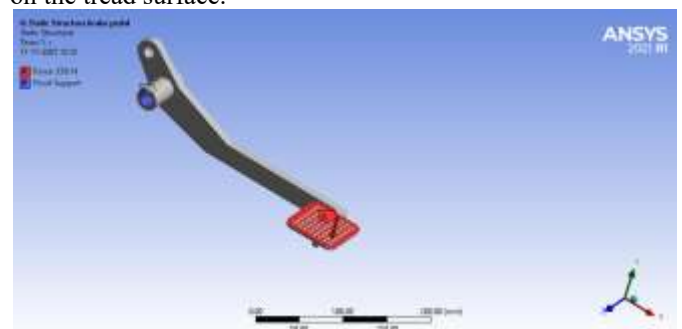


Fig. Boundary conditions for structural analysis of MS Brake Pedal.

Results:

Total Deformation –

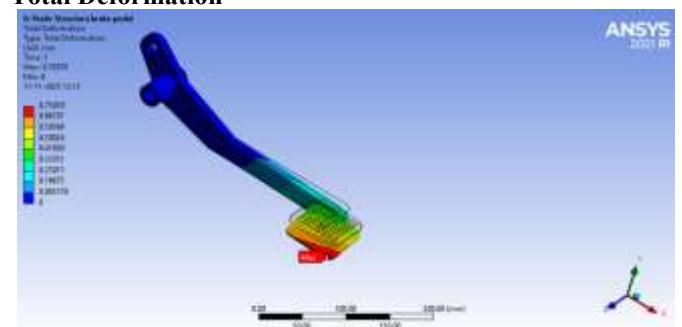


Fig. Total deformation of MS Brake Pedal.

Equivalent Stress –

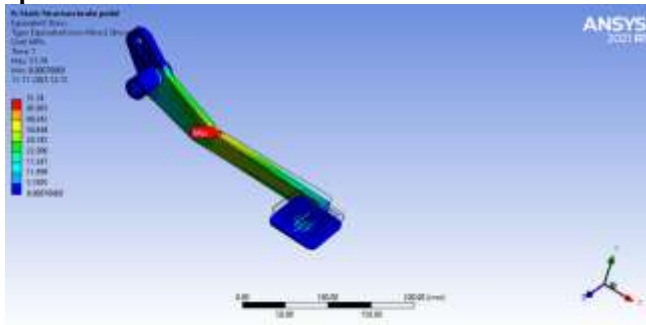


Fig. Equivalent stress of MS Brake Pedal.

Equivalent Elastic Strain –

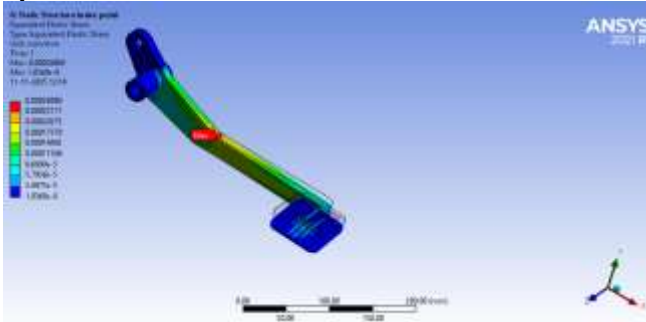


Fig. Equivalent Elastic Strain of MS Brake Pedal.

Static Structural Analysis of Gray Cast Iron Brake Pedal: Geometry:

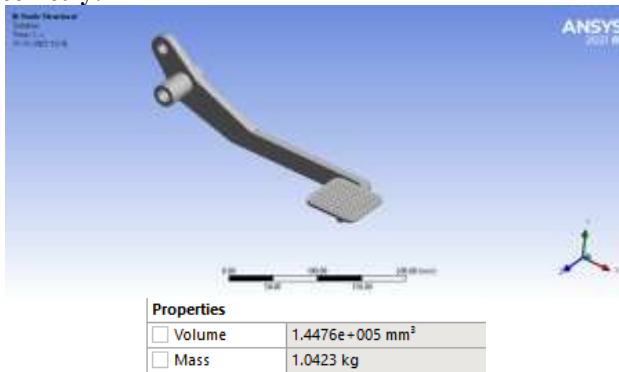


Fig. Geometry of Gray Cast Iron brake pedal.

Material Properties:

Gray Cast Iron	
Density	7.2e-06 kg/mm ³
Structural	
Isotropic Elasticity	
Derivation	
Young's Modulus	1.1e+05 MPa
Poisson's Ratio	0.25
Bulk Modulus	83131 MPa
Shear Modulus	42669 MPa
Isotropic Sequent Coefficient of Thermal Expansion	
Compressive Ultimate Strength	620 MPa
Compressive Yield Strength	0 MPa
Tensile Ultimate Strength	240 MPa
Tensile Yield Strength	0 MPa

Fig. Material Properties of Gray Cast Iron.

Meshing:

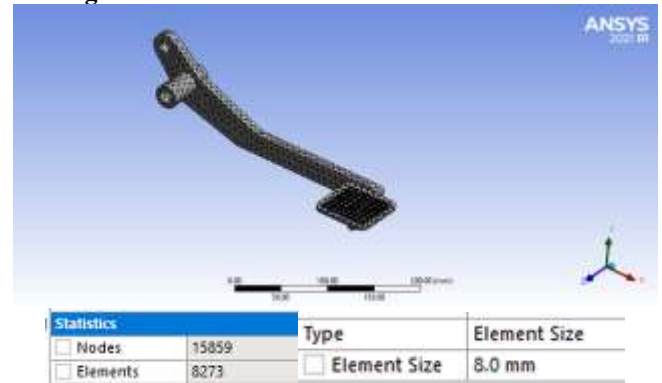


Fig. Meshing Details of Gray Cast Iron brake pedal.

Boundary Conditions:

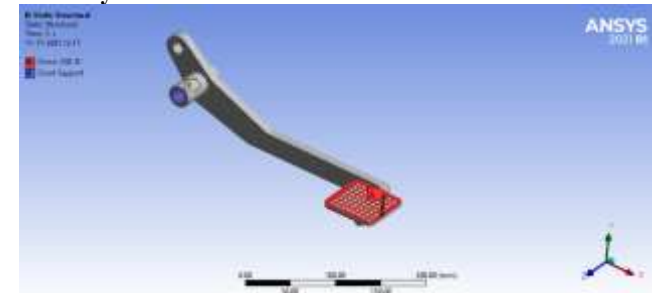


Fig. Boundary conditions for structural analysis of Gray Cast Iron brake pedal.

Results:

Total Deformation –

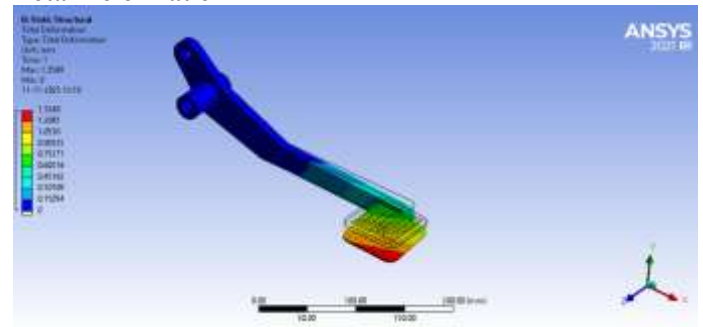


Fig. Total deformation of Gray Cast Iron brake pedal.

Equivalent Stress –

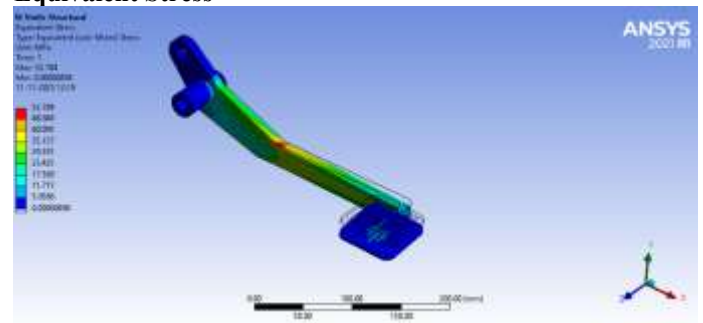


Fig. Equivalent stress of Gray Cast Iron brake pedal.

Equivalent Elastic Strain –

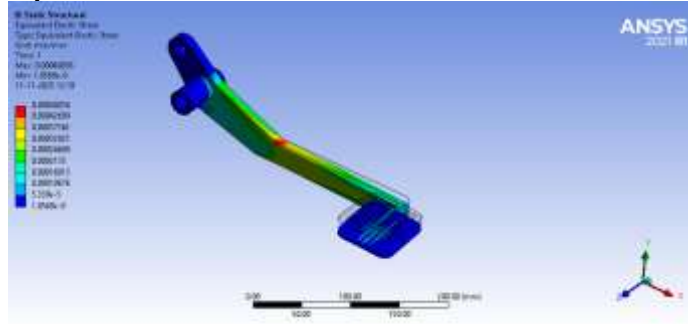


Fig. Equivalent elastic strain of Gray Cast Iron brake pedal.

Static Structural Analysis of Optimized Brake Pedal: Geometry:



Fig. Geometry of optimized brake pedal.

Topology Optimization:

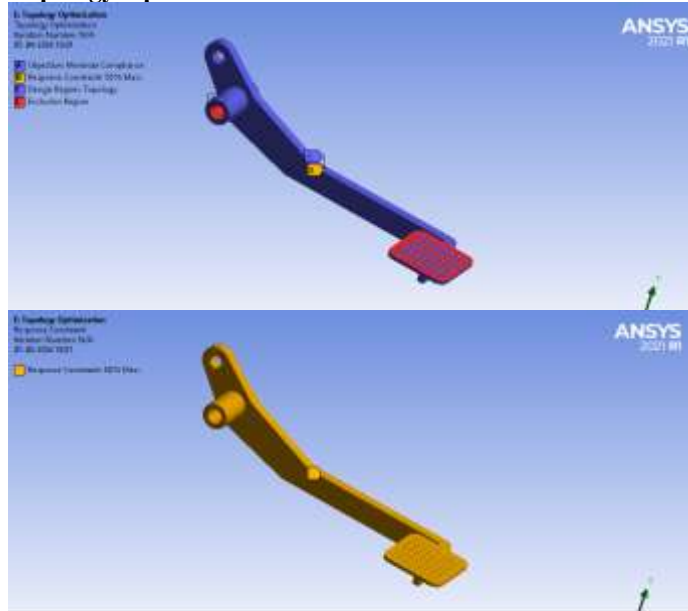


Fig. Conditions for topological optimization.

Meshing:

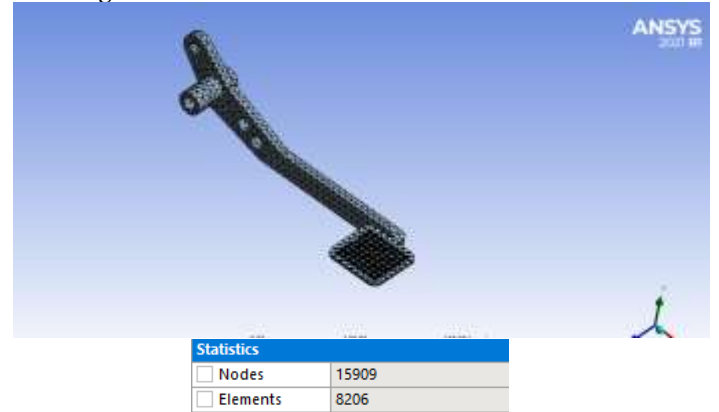


Fig. Meshing Details of Optimized Brake Pedal.

Boundary Conditions:

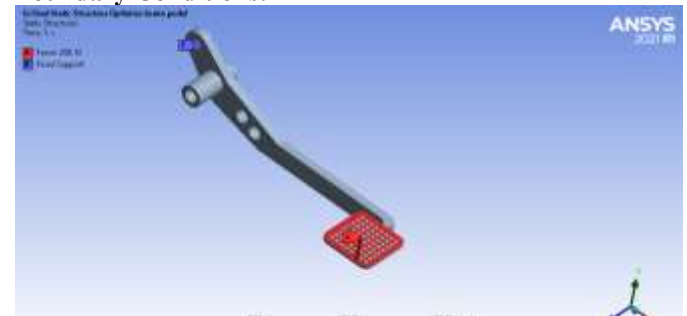


Fig. Boundary Conditions for static structural analysis of optimized brake pedal.

Results:

Total Deformation:

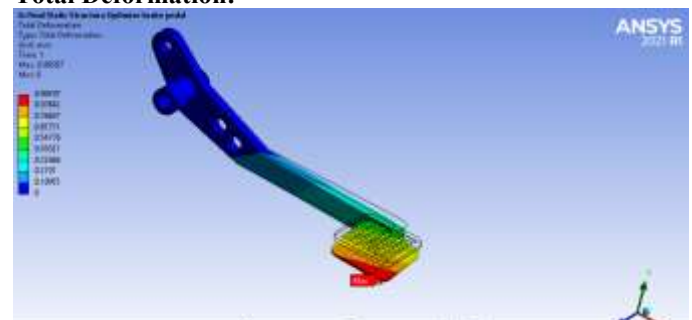


Fig. Total deformation of optimized brake pedal.

Equivalent Stress:

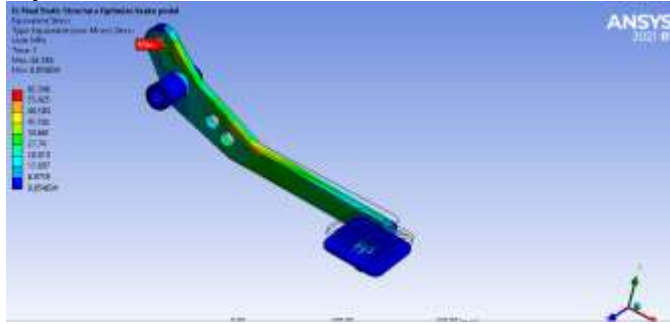


Fig. Equivalent Stress of Optimized brake pedal.

Equivalent Elastic Strain:

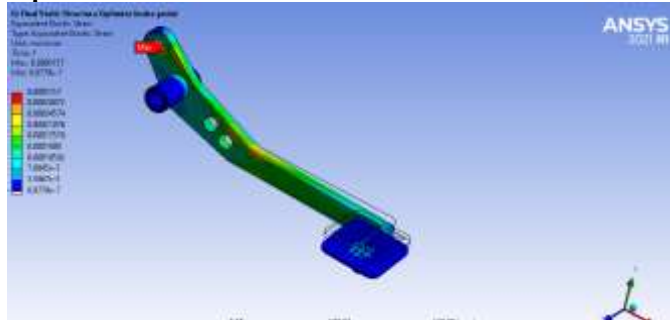


Fig. Equivalent Elastic Strain of optimized brake pedal.

FEA RESULTS

PARAMETER	STRUCTURAL STEEL	GRAY CAST IRON
Weight	1.1364 kg	1.0423 kg
Total Deformation (mm)	0.75039 mm	1.3549 mm
Equivalent Stress (MPa)	51.74 MPa	52.704 MPa

PARAMETER	Original brake pedal	Optimized brake pedal
Weight (Kg)	1.1364	1.0758
Total Deformation (mm)	0.75039	0.98597
Equivalent Stress (MPa)	51.74	62.346

Finite Element Analysis (FEA) was initially performed on the brake pedal using structural steel and gray cast iron to compare their mechanical performance. The structural steel model showed a deformation of 0.75039 mm, whereas gray cast iron exhibited a higher deformation of 1.3549 mm, indicating lower stiffness. The equivalent stress values for both materials were nearly the same (around 52 MPa), while gray cast iron had a slightly lower weight. Based on these results, structural steel was selected due to its superior stiffness and overall performance.

Following material selection, topology optimization was applied to the structural steel brake pedal to achieve weight

reduction. The optimized design reduced the mass from 1.1364 kg to 1.0758 kg. FEA of the optimized model revealed an increase in deformation to 0.98957 mm and an increase in equivalent stress to 80.425 MPa. This increase is attributed to the removal of material from low-stress regions and redistribution of load paths within the structure.

V. MANUFACTURING OF THE COMPONENT

The manufacturing of the optimized brake pedal was carried out by modifying the existing brake pedal based on the topology optimization results. Instead of producing a new component, material was removed from the original pedal to achieve the optimized design.



First, the regions identified for material removal were marked carefully on the brake pedal according to the topology optimization output. These regions corresponded to low-stress areas where material could be safely removed without affecting structural performance.

The main operation performed was drilling. Holes were created in the marked regions using appropriate drill bits and machining parameters suitable for the pedal material. The drilling process was carried out carefully to maintain dimensional accuracy and to avoid any damage or deformation to the component.

VI. EXPERIMENTAL TESTING

Experimental testing is used to validate numerical results by evaluating the actual mechanical behavior of a component under applied loads. In this study, a compression test is performed to determine the load-deformation characteristics and structural strength of the axle specimen.

The test is based on the principle of applying a gradually increasing load and measuring the corresponding deformation to understand stiffness, elasticity, and failure behavior. The resulting load-deformation curve helps identify key properties such as maximum load capacity, elastic region, and structural stability. This experimental approach ensures the reliability of FEA results by providing real-world performance data for comparison and validation.

A Universal Testing Machine (UTM) is a device used to test the mechanical properties of materials and components under different loading conditions such as tension, compression, and bending. It applies a controlled load to a specimen and measures the resulting deformation, helping determine properties like strength, stiffness, and elasticity.

UTMs are widely used in material testing laboratories for validating design and simulation results.

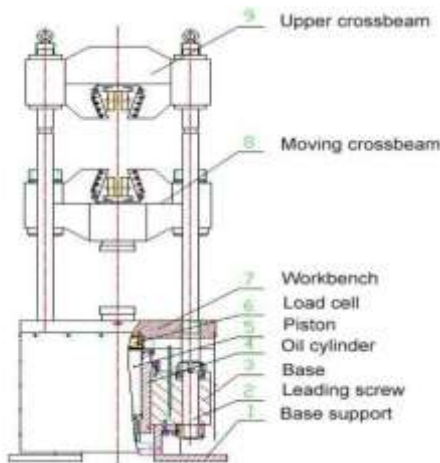


Fig. Schematic Diagram of a UTM.

EXPERIMENTAL PROCEDURE:

1. Measure the width and thickness of the specimen.
2. Mark on the locations where the load will be applied.
3. Place the sample carefully on to the fixture of a universal testing machine.
4. Make sure that the loading point is placed on to the marked location.
5. Carry out the test.

Testing photograph



Fig. Specimen under compression test using a UTM.

Testing graph

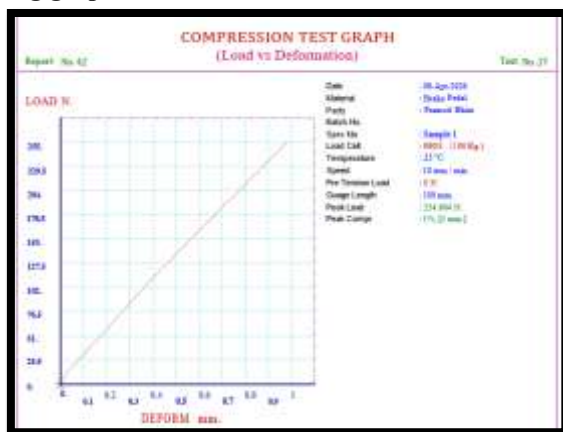


Fig: Graph shows load vs deformation during compression test

Conclusion

- In this study, the structural performance of a brake pedal was evaluated and optimized using Finite Element Analysis (FEA), topology optimization, and experimental validation. A comparative analysis between structural steel and gray cast iron revealed that structural steel provides superior stiffness with lower deformation, making it more suitable for brake pedal applications.
- Topology optimization was then applied to the structural steel model, resulting in a reduction in mass from 1.1364 kg to 1.0758 kg, thereby improving material efficiency. The optimized design exhibited a moderate increase in equivalent stress and deformation; however, these values remained within acceptable limits, ensuring safe operation under applied loading conditions.
- Experimental compression testing using a Universal Testing Machine (UTM) validated the numerical results, showing good agreement between simulation and practical performance. Minor deviations were observed due to real-world factors such as material inconsistencies and boundary condition variations.
- Overall, the study demonstrates that the integration of FEA, topology optimization, and experimental validation is an effective approach for developing lightweight and structurally efficient automotive components. The optimized brake pedal achieves an improved strength-to-weight ratio while maintaining safety requirements, and the methodology can be extended to the design and optimization of other critical vehicle components.

REFERENCES:

1. Shuheng Wang, Xueyuan Liu, Wengang Chen, Jie Tang, Qiping Liu, Taiyu Ning, Haibing Yao "Force Analysis and Optimization of BSC Brake Pedal Based on ANSYS"
2. Oğuz Doğan, Onur Can Kalay, Emirhan Kartal and Fatih Karpaz "Optimum Design of Brake Pedal for Trucks Using Structural Optimization and Design of Experiment Techniques"
3. Muhammad Khafidh, Finny Pratama Putera, Rahmadi Yotenka, Deni Fajar Fitriyana, Rahmat Doni Widodo, Rifky Ismail, Agustinus Purna Irawan, Tezara Cionita, Januar Parlaungan Siregar, Nur Hidayah Ismail "A Study on Characteristics of Brake Pad Composite Materials by Varying the Composition of Epoxy, Rice Husk, Al₂O₃ and Fe₂O₃"
4. Arun J. Kulangara, C.S.P. Rao, Jose Cherian "Topology optimization of lattice structure on a brake pedal".
5. Tanmay Nandanwar, Keyour Waghela, Eshaan Gupta, Narendiranath Babu T. "TOPOLOGY OPTIMIZATION OF THE BELL CRANK & BRAKE PEDAL"
6. M.I.M. Sarginia, S.H. Masood, Suresh Palanisamy, Elammaran Jayamania, Ajay Kapoor "Additive manufacturing of an automotive brake pedal by metal fused deposition modelling".
7. Alexander V. Bobrowskii, Alexey V. Zotov, Dmitry A. Bobrowskii, Mikhail P. Gololobov, and Ekaterina D. Chizhatkina "Application of finite element analysis in the development of the pedal assembly of the Formula Student racing car".

8. Artur de Vito Jr., Gustavo Okada, Ygor Navarro, Raphael Ragozzino, Sandro Vatanabe, André Mendes “Topology Optimization and Experimental Validation of a Baja Competition Brake Pedal Design”.
9. Ruyhan, Abdullah Al Hossain Newaz, Md Mehedi Hassan Melon “Optimization of Brake Pedal Mass and Safety Factor Using Topology Optimization Techniques”
10. Özlem Akçay, Cumali İlkılıç “Structural Optimization of the Brake Pedal Using Artificial Intelligence”.