International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 11 Issue: 05 | May 2024 www.irjet.net p-ISSN: 2395-0072

Optimization of Brake Pedal Mass and Safety Factor Using Topology Optimization Techniques

Ruyhan[1], Abdullah Al Hossain Newaz[2], Md Mehedi Hassan Melon [3]

[1] School of Mechanical Engineering, Xi'an Jiao Tong University, Xi'an 710049, People's Republic of China
[2] School of Mechanical Engineering, University of Bridgeport, USA
[3]Department of Business Administration International American University Los Angeles, USA

***

Abstract - This research paper focuses on optimizing the mass and safety factor of brake pedals by applying topology optimization Techniques. The Original Brake pedal weight was 697 grams. Used as a baseline for comparison. Reference models 1-3 achieve savings weight 41, 85, and 51 grams, respectively. Weight has dropped significantly from 119 grams to 212 grams. Additionally, the safety factors of the original and optimized models were evaluated, while optimized models could have safety factors ranging from exceeding to meeting the industry standards. These results explicitly show the criteria of topology optimization for optimizing weight via placing the brake pedal in the right position while ensuring its safety and structural integrity. Successful application of these techniques improves the design and optimization of lightweight and high-performance automotive components

Key Words: Topology Optimization, Brake Pedal, Mass Reduction, Safety Factor, Finite Element Analysis.

1. Introduction

The automotive component engineering, especially, the safety-focused parts design like the brake pedal, undoubtedly, takes crucial roles in vehicle operation safety and performance. The brake pedal acts as a mechanical linker between the driver and the vehicle braking system, transferring driver input force to the braking system to actuate the brake. To wrap up, the good design of the brake pedal is vital for functionality, efficiency, as well as better vehicle performance. Normally, particular examples of designing car components use the method of iterative and many corrections based on the expertise of engineers. While these methods have succeeded in the production of effective designs, the possibility that their full weight-saving potential or structural efficiency could be attained is still unlikely. Besides, the managers tend to design vehicles with lesser weight so that they can have better fuel efficiency and low emissions as well. Therefore, these innovative design techniques were invented to create light structures with optimal functionality while keeping to the minimal mass at the same time. Topology optimization, the late, computationally wise technique, has allowed designs with material redistribution within a space of a given design to be optimally produced. By approximating the design problem as an optimization task, topology optimization techniques are capable of scanning through a whole bunch of design spaces with certain conditions and parameters, so that the most optimal solution can be obtained. Hence, simulation in virtual environments serves AS a platform for the discovery of new and unconventional design developments that may have been excluded by traditional methods. Here we are going forward and presenting a strategy for the creation and mass topology optimization of a brake pedal using the topology optimization technologies. The goal is to create an optimum weighing and strengthening brake pedal design scheme without increasing mass while meeting or overshooting the performance criteria. The proposed methodology harmoniously combines simulation trials involving topology optimization algorithms and experimental studies to conclude that the approach gives positive results and boosts braking pedal efficiency.

1.2: Brake pedal innovation in design.

The only of the braking system in a car is to slow the speed of moving down by pressing the brake pedal, as the brake pads press/rub against the rotor/friction. The simple task of the brake pedal is to commute the driver's feet power into the braking machine, and this has not been improved for years, contradicting the environment and increasing the cost. This work is made up of the effort to lose the whole weight of the pedals and to achieve a lightweight and beautiful design through Topology Optimization. The pedal ratio, which equals the space from the fulcrum to the pedal's center to the distance to the master cylinder attachment point, is of the utmost importance.

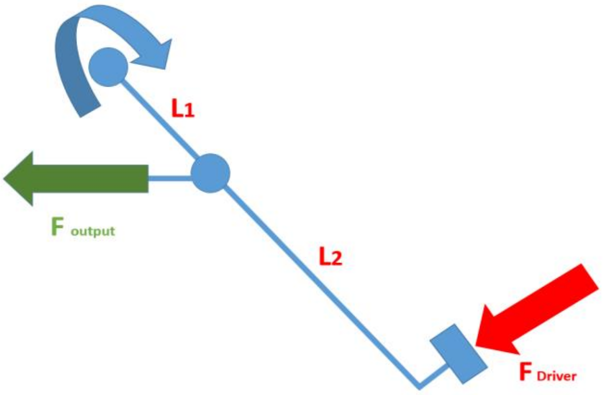


Fig-1: Brake pedal's line diagram illustration.

A recommended ratio of 6.2:Combining the brake-by-wire ratios typically provided by either electrohydraulic or hydro-pneumatic brake-by-wire systems, we propose a simple 5.5:1 ratio should replace the vacuum system's assist, which is usually 3.5 to 4.0:1.[1]. In the word "shortening" the "ing" is altered to get the same effect but the procedure is challenging. Tapping on the mechanical advantage that was being lost by using a smaller master bike or carefully selecting the accessory components complemented the loss. Besides the fact that both, the driver and co-driver, are supported by their race component suppliers in terms of an ergonomically designed pedal system based on dual master cylinders, there are many other reasons to learn. In addition, it is necessary to be cautious with the creation of a custom assembly since this requires the implementation of a very detailed design and structural construction to ensure reliability and maintenance of safety [1].

2. Literature Review

TO proceeds through the use of a mathematical method known as topology optimization to distribute material within a component just by specifying the boundary conditions and loads to be used. With this, engineering shaping of the parts with reduced weight is achievable and the associated costs of material are minimized while the safety from mechanical loading is maintained [2]. For example, a study designed to improve the brake smartphone and increase its shape was conducted. The interesting finding is that the pedal weight was reduced to 50% without using a different type of material [3]. On the same front, another study that is geared towards the reduction of the brake pedal's weight, a crucial part of the braking system, was achieved by using shape and topology optimization, and the result revealed a 31.8% weight reduction resulting from the re-distribution of plastic and the adjustment of dimension [4]. In addition, to topology optimization, topology optimization (TO) is an approach that aims at the removal of excess mass from components while keeping strength and a main functionality, and it differentiates the shape and sizing optimization. Previous creative mindsets incline to material adjustments and to experiment way. This fitness optimization project aims to design an efficient Brake Pedal using topology optimization methods via a direct algorithm. FEA results using the ANSYS 19.0 platform demonstrate a drop in mass with the preservation of strength and support capacity. This scheme assists in the efficiency of vehicles through a reduction of mass and manufacturing costs, which have been finalized in one standard design. For example, revised elements appear lower in yield stress[1]. Secondly, after research in automotive engineering showed an indication that future pedal designs will be based on polymers that can offer features like low weight, high strength, and freedom in shapes, several of such concepts were proposed which resulted in potential weight loss of about 73% as well. [5]. Another aspect is the fact that studies that have looked at polymeric-based composite pedals have demonstrated that the weight reductions are up to 66.7% compared to the conventional metallic ones while maintaining the manufacturer's specifications [6]. While these improvements are mostly indicative of the finding of small and structural automotive components that are more robust and lighter to maximize the efficiency of the car, they also help to contribute to the safety and performance of the car in general. Finite element analysis and topology optimization have been employed by several studies to produce more lightweight brake pedals for automobiles without compromising the hardware's performance and safety standards. The most revealing part of this research is a 16% mass reduction, while the stress levels of the aircraft do not exceed safety limits[7]. Investigated to enhance fuel efficiency and meet road safety guidelines with a new brake pedal design decreasing the overall weight by 12% and increasing the performance, respectively [8]. Likewise, a platform for rebuilding faulty car components using optimization tools resulted in 24% mass reduction, 9% additional stress reduction, and 37% higher rigidity for a clutch fork [9]. The third article presents a methodological synthesis of topology, Topometry, and size optimizations for automotive chassis design by which the researchers lowered weight while maintaining the required standards [10]. Moreover, topology optimization was shown to be also applicable to the concept design, eg. during studies where authors have designed a squeezing device for an automotive compressor bracket, more specifically[11]. Apart from this, a project worked on the lightweight design of the brake without material substitution showed a 22% weight drop with absolute performance, therefore hinting at the need for modifications for industrial application[12]. This research work is constructed based on the innovative use of diverse methods of optimization techniques for automotive components which are presented below. In the first place, the author proves the Optimisation models based on reliability-based design optimization (RBDO) and deterministic design optimization (DDO) for an FSAE brake pedal, setting the RBDO's advantages like in weight reduction and robustness, while exposing the design risk allocation analysis [13]. Besides, it is concerned with the under-availability of full circle tin-containing unit wheels by FEA since a 50% decrease has been reported compared to the solid disc ones and at the stress levels below the stress yield and per the loading conditions [14]. The last study presents the finite element method used for questions in ANSYS software OptiStruct to design a car rim, putting effort into minimizing weight while meeting stress and strain criteria. The validation of the study by experiment in ANSYS for H1 demonstrates that the ANSYS for stress analysis is reliable[15]. An ultimate aim is to set forth multilayer optimization topology optimization with objectives of compliance and Eigen frequencies. This leads to a mass reduction of 4.57% in the optimized wheel disc and hence confirms the effectiveness of the method [16]. This project goes deep into the topology optimization with cast aluminum alloy wheels, conducting the successive impact analysis to predict plastic strain. The modeling, through finite element and impact, aims to reduce rim thickness conduct, which will be an advantageous performance [17].]. Further, the author focuses on magnesium alloy car wheels that are concerned with improving product quality and life span while lowering the risk of fatigue and cost. Finite element analysis taught us thinner wheels will be stronger with less stress, thus giving us better mileage and service life. With this, magnesium alloy can be the optimum choice for our existing materials [18]. Moreover, scientists reveal that the lattice structure technology can be used to create an intricate lattice that lowers the weight of the pedal in particular. The lattice structure is better since it substitutes solid regions approximately 21.2% mass reduction can be obtained with the same displacement and stress levels as solid machining Then, lastly, a study that focuses on the feasibility of low-cost desktop Additive Manufacturing (AM) technologies to make metallic automotive brake pedals and meeting the demands in short periods is carried out as well. The research thesis will encompass the design, materials, and manufacturing process. FDM will be used to produce amalgam filament for the brake pedal. By using Finite Element Analysis (FEA), the design is authenticated, and laboratory testing of metal-based FDM technology validates the dependability[19]. After that, another research assesses the possibility of implementing AM technologies of low cost to produce metallic automotive brake pedals which can be done in a short time. A three-dimensional study was carried out on the design, material, and tools manufacture parts by using the 3D printing (Fused Deposition Modelling; FDM) technique for creating a metal-polymer filament brake pedal. Finite Element Analysis (FEA) aids us in design verification, and physical testing upholds FDM metal-type technology reliability[20].

3. Methodology

The methodology that the study uses for the optimization of the pedal – which is the brake - using the topology optimization approach intends to improve the overall efficiency of the system and to reduce the related mass at the same time. Firstly, the problem is defined by setting objectives, constraints, and design space, with the purpose of loading conditions stated according to regulation standards for operational requirements at the initial stage. A Finite Element Analysis (FEA) is used to reproduce the structural behavior of the Pareto optimal brake pedal layout by applying different loading conditions, and to assess against experimental data the model accuracy is verified. The next step would be the utilization of topology optimization algorithms, which let them automate optimal designs of the brake pedal and the ensuing constraints assure the safety and performance standards of the system. The optimized designs are further subjected to an in-depth process of numerical simulations as well as experimental testing to determine the structures' and the mechanics' integrity and functional performance. The thought-out designs developed during the iterative refinement process are used in the next step to assess the strength of the fracture and minimize the mass. In addition, the methodology features such as a comprehensive analysis of the outcome, comparison of traditional design, and documentation of optimization process alongside possible research proposals for future works, ensure the technical process.

3.1: Utilizing Solidworks for 3D brake pedal design.

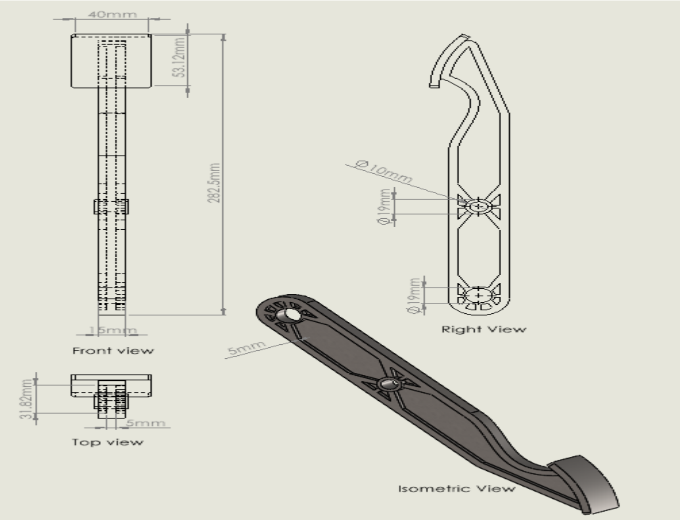


Fig-2: Elaborate brake pedal sketch depiction.

First, the process of Brake Pedal fetching the 3D CAD model involves SolidWorks 2023 software. Firstly, a very rough sketch of the component is created through the use of CAD software. By this, the team measures all the dimensions of each object as correctly as possible to have a precise model before entering into the fabrication phase. The next step is the making of 2D sketches to serve as a base for the development of a 3D full-size model takes place. This redraft gives the rider a chance to estimate the pedal's look and structure, ultimately leading to its implementation into the 3D space. The full-fledged 3-D sketch depicts the flawless accuracy of the pedal's outlook and includes specifications of its features, which act as the very guide throughout the entire modeling process. This structure helps to capture the real view of the pedal geometry and function. The outcome eventually would be that this high-level strategy caused the represented 3D CAD model to be the exact Brake pedal that coped with all the required criteria and specifications. The detailed sketch, as illustrated in Fig: 2, is a visual guide, that operates as a method of pattern by which the digital prototype is developed with great precision and correctness.

3.2: SolidWorks used to create the 3D model.

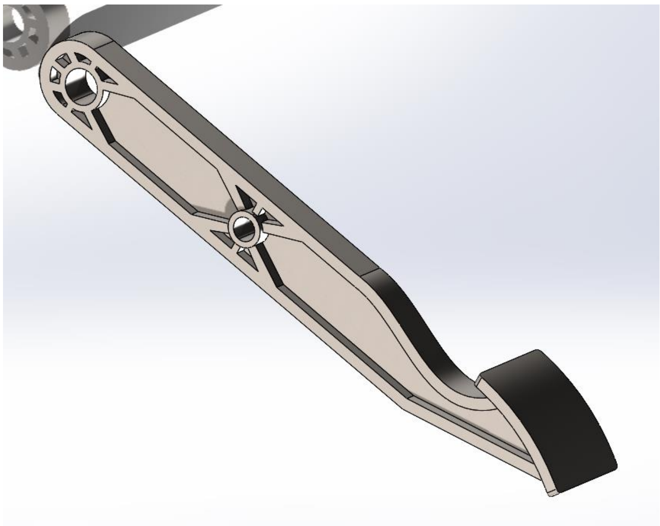


Fig-3: Authentic brake pedal design.

3D modeling is the process of making or designing three-dimensional (3D) digital versions of various objects or scenery using specialized software. It comprises molding and altering different shapes and structures using computer tools to design virtual works which can be simple shapes or complicated, detailed objects. In brake pedal brake pedal designing, the 3D displaying provides reliability for engineers and designers to determine the brake pedal's form, dimensions, and functionality in a digital environment before physical production. Thus, comprehensive testing, investigation, and preparedness of an idea are conducted and the success of the design is supported through reduced time for production and higher standards. With the aid of 3D modeling, it is possible to check the compatibility of the material properties, assembly match, and human anatomy among the several aspects thus resulting in better optimization of brake pedal designs. Also, the 3D models can be transferred, revised, and adjusted to larger systems or simulations without the requirement of additional programs that have made it possible for team members to work together and reduce the product's development time. In total, 3D modeling fills the area of modern engineering and design processes, being a force of visualization, prototyping, and the best working of mechanical components like brake pedals.

4. Analysis of original brake pedal structure, static and transient.

4.1 Pedal's static analysis conducted in ANSYS Workbench.

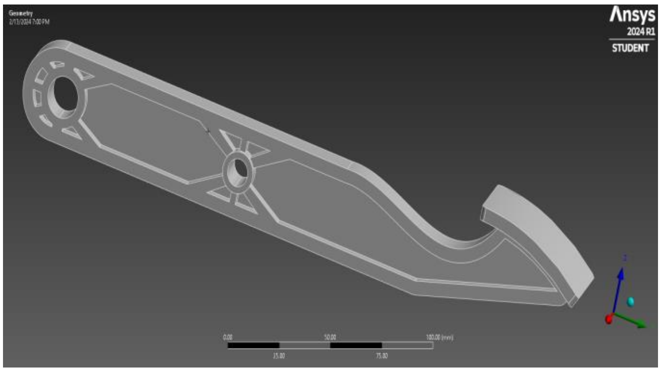


Fig-4: SolidWorks model imported into ANSYS.

ANSYS simulation software, a Finite Element Analysis (FEA). The model that SolidWorks created is imported into ANSYS using the (IGS file format). In ANSYS these mathematical constraints are used on the model to define its behavior all over the variables. This procedure will allow the professionals to undertake an in-depth review and scrutiny of the system and make a better judgment as to factors like load distribution, structural integrity, and functionality.

4.2: Material

In this study, the choice of the FE analysis method assumes the definition of the material properties as the initial phase. Due to the same reason – static structural analysis is employed – the variable section is relevant. The Engineering data section contains information about the coefficients of elasticity and Poisson ratio, which stands as the scientific basis of the results. The special material of the brake pedal, it is stated as A36 (structural steel), makes sure the precise visualization of the FE analysis

Density	7.85e-06 kg/mm^3
Young's Modulus	2e+05 MPa
Poisson's Ratio	0.3
Bulk Modulus	1.6667e+05 MPa
Shear Modulus	76923 MPa
Yield Strength	250 MPa
Tensile Strength	460 MPa

Table 1: Material properties summarized

4.3 Meshing

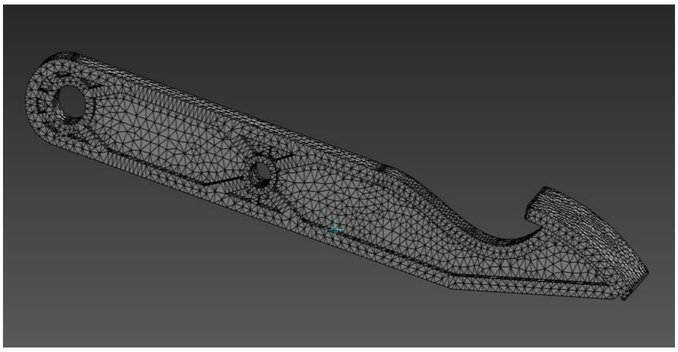


Fig-5: Brake pedal model meshed.

In ANSYS, users can either choose the automatic or manual way of meshing components; elements can be selected according to their shape and size. The pedal uses a configuration of the TRI and HEX elements to improve the stiffness, mainly around the pivot and distorted curvature. The meshing element size is selected as 2 mm for a precise and high-quality simulation. This precise and fine-tuned approach makes sure that the brake's pedal is represented in an actual way during numerical investigation. Through a smart combination of this presentation, engineers can achieve a high level of accuracy, since this technique simulates real-world conditions and acquires performance information like elastic effects and stress distribution.

4.4 Boundary conditions set for analysis.

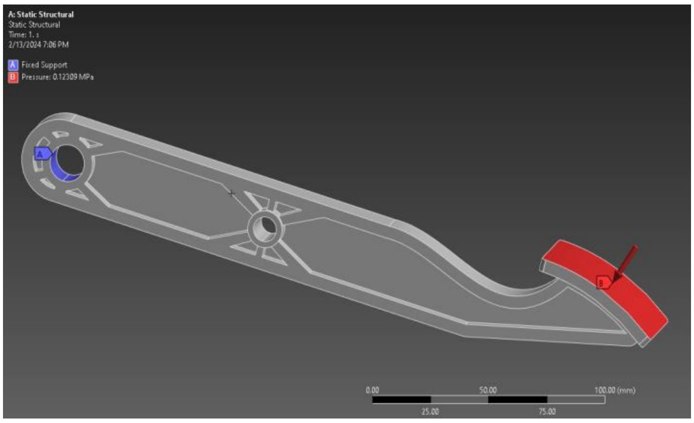


Fig-6: Boundary conditions established for precise analysis.

The equation for the total of the forces (F) that act on the brake pedal represents the maximum force that can arise and that is estimated to be 40 kg. The conversion to the law of gravity (g) gives the result: F = 40 * 9.81 = 392.4 N, which is equivalent to 400 N. The next part is to find.

$$P = \frac{F}{A} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \text{(1)}$$

$$P = \frac{400\text{N}}{3249.1\text{mm}^2} = 0.12309 \text{ MPa [1]}$$

Besides that, the pivot point of half of the pedal and the push rod being attached to the master cylinder do not include drags. These reactive forces are, however, thought to be of negligible proportions in Al's design deliberations.

4.5 FEA Analysis of static structural behavior.

An FEA static structural analysis of a pedal is based on its equivalent Von Misses Stress and total deformation. The graphs consisting of both the research findings are presented below. We will now look at the strain profiles of a selected pedal under static loading conditions. This kind of analysis helps to determine the structural safety of the part as well as deformation characteristics as a difference in cyclic and static loading conditions. The aristocracy such as Von Misses Tension figures out regions that have high effects of stress concentration, while pedal deflection with total deformation stands for the percentage of deflection of pedal under loads. Through the analysis of these dimensional characteristics, engineers try to keep the optimal pedal design in sight from the standpoint of safety and performance. Furthermore, graphical representation is free of any extra confusion, aiding in easy comprehension and communication of findings to technicians and manufacturers involved in the design and the manufacturing process.

Equivalent Von Mises stress in static structural analysis.

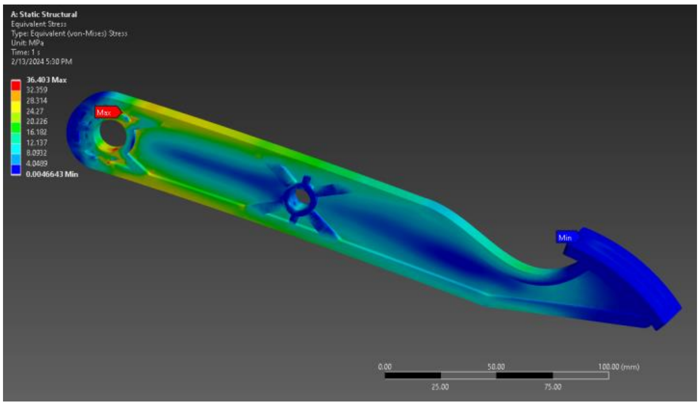


Fig-7: Equivalent Von Mises Stress

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 36.403 MPa and 0.0046 MPa, respectively.

Total Deformation in static structural analysis.

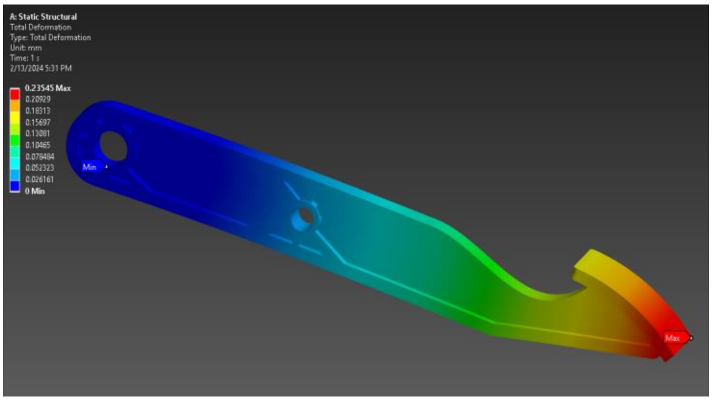


Fig-8: Total Deformation

The total deformation analysis revealed maximum and minimum deformations of 0.23545 mm and 0 mm, respectively.

Safety Factor

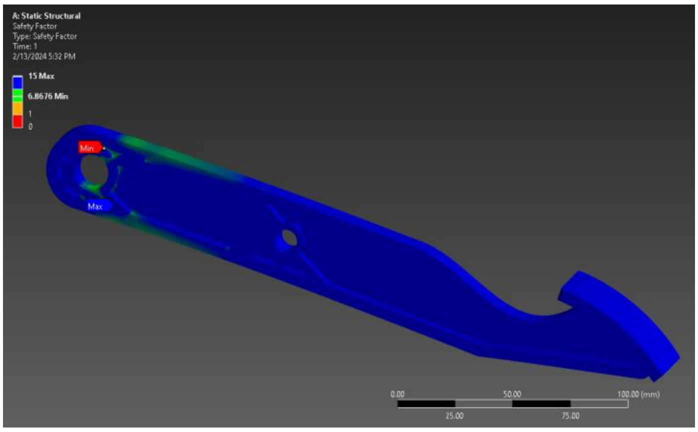


Fig-9: Safety Factor

Safety Factor Max: 15 and Min: 6.8676.

4.6 FEA Analysis of Transient Structural.

Flexible behavioral analysis, which is usually referred to as transient structural analysis, is an essential engineering aid for structural behavior assessment during time-dependent loads such as earthquakes. While the confines of static analysis revolve around how the structures behave being subjected to constant stresses, transient structural analysis explores how the dynamic responses are caused by varying forces over time. This precisely is the very basic requirement for getting to know the extent of how structures behave in natural conditions where loads can unexpectedly change or start fluctuating often. Through the approximation of dynamic loading scenarios, the transient structural analysis will help engineers assess how structures are going to react to forces, for instance; impact, vibration, or seismic instability. Such information should be used when designing structures that are capable of resisting vagaries acutely induced events but at the same time ensuring both safety and reliability.

Overall, the impact of transient structural analysis is significant on the ergonomic behavior of structures; therefore, this study could be used in design, optimization, and safety assessment across various engineering disciplines. FlexML not only can give a complete account of complex dynamic loading conditions but it is also a widely used tool for the validation process of structural integrity of engineering systems in different applications.

4.7 Analysis settings in brief

Number of steps	1.
Current Step Number	1.
Step End Time	0.1 s
Auto Time Stepping	on
Define By	Time
Initial Time Step	1.e-004 s
Minimum Time step	1.e-004 s
Maximum Time step	1.e-004 s
Time Integration	on

Table 2: Specifics regarding analysis parameters provided.

4.7 FEA results for transient structural

Equivalent Von Mises stress in transient structural

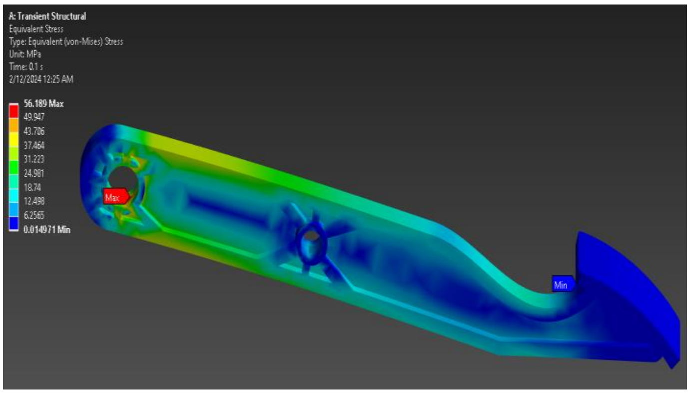


Fig-10: Equivalent Von Mises Stress result

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 56.189 MPa and 0.014971 MPa, respectively.

Total Deformation in Transient Structural Analysis.

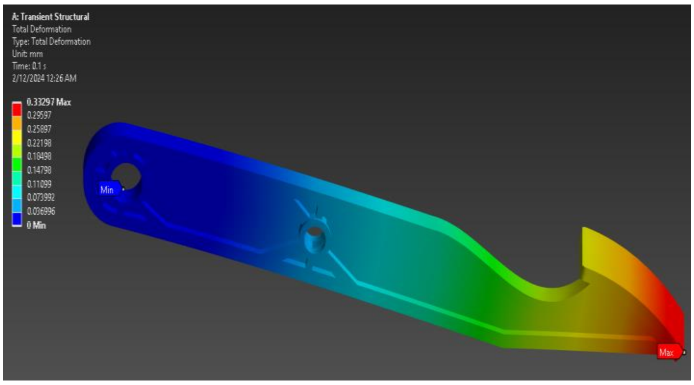


Fig-11: Deformation results

The total deformation analysis revealed maximum and minimum deformations of 0.33297 mm and 0 mm, respectively.

5. Topology Optimization Use Ansys software

Topology optimization stands for a computation technique utilized to design an optimized material placement in a chosen design area to get the best mechanical performance under some set constraint positions. FEA (finite element analysis) is a well-known cost-effective and highly holistic tool utilized in analyzing topology optimization studies. ANSYS provides an entire complex analysis platform that covers many simulation types and lets engineers investigate many design options and analyze their responses under the different conditions of loading. Humanize: ANSYS for topology optimization allows engineers to quantumly go through their design iteration process because of the capacity of the software to automatically adjust the material distribution inside the given design domain to meet performance constraints while satisfying design constraints in the same way. The engine of ANSYS topology optimization of the designing process entails the creation of design space, defining boundary conditions, imparting the loading conditions, defining the material properties, and creating the optimization objectives. Subsequently, ANSYS implements advanced algorithms that recursively allocate material to the design space, and this is a process that seeks the best material configuration that reduces the mass without sacrificing structural integrity and also achieves the design performance requirements. In this repetitious procedure, engineers can investigate a broad range of potential layouts and reveal very original solutions that usually would not have been identified through conventional design methods. For additional contribution, there are ANSYS' visualization tools which are advanced enough to give engineers a chance to analyze and interpret the topology optimization results effectively. Engineers apply various stress distribution, displacement patterns, and other performance metrics to have a view of the dynamic performance of the system with enhanced design. As a result, the experts can make the right decisions and improve the details of the design step by step so that the final result is close to the desired achievement.

Using the layer-by-layer zone designation allocated to the design, the tool inside the ANSYS software for topology optimization took over and produced the optimized free form in the product. The process is to achieve a considerable weight reduction, usually finding that the optimized model typically presents a 50% reduction compared to its original counterpart.

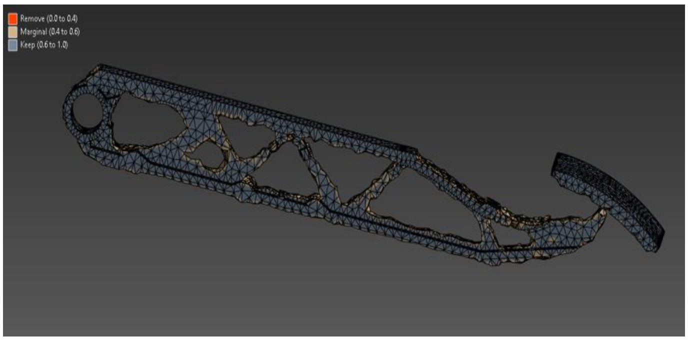


Fig-12: Front view displaying 50% mass optimization.

The defining of the regions within the design space is an integral part of topology optimization because it decides the location where material redesigning takes place to reduce weight as much as necessary for the structure to maintain its integrity and function. Provided the regions are assigned, a topology optimization algorithm is then iteratively applied between these regions to redistribute the material to accomplish the stated objectives, such as mass reduction given that performance criteria are met. Through applying various techniques such as let down, brushing, and filling, the optimized model free form now stands to be the final design of the configuration that conforms to the predetermined functions. Therefore, the model aims to achieve such features as enhanced structural efficiency, decreased mass, and superb performance. Through the deployment of ANSYS software's advanced topology optimization process, it is easy to search and locate the most suitable design layout which possesses the greatest weight-saving advantages but does not compromise the system's structural integrity or functionality.

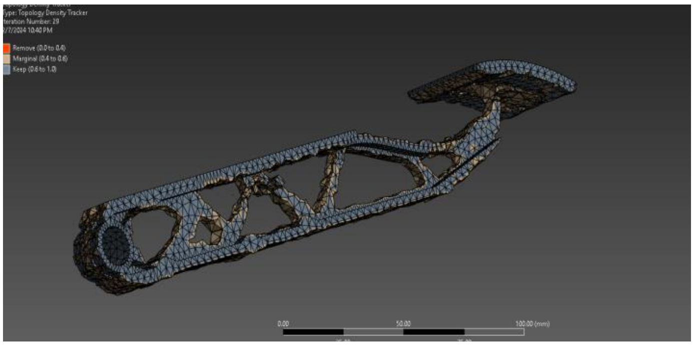


Fig-13: Isometric view displaying 50% mass optimization.

This recurring process of designating regions, executing the topology algorithm on the specified area, and analyzing the free form of the optimized portion continuously enables the engineers to refine and enhance the design until the set light-weighting and performance objectives are reached. To sum up, topology optimization, using ANSYS software, offers engineers in different industries such as automotive, aerospace, and manufacturing an extremely powerful tool for a dual purpose: on the one hand, they can achieve the “optimal structure” of components and on the other hand, they can deploy the suggested designs into reality.

5.1 Topology Optimization Use Solidwork software

In SolidWorks software, topology optimization starts by setting the zones or sections of the area where the main structure will act as the controller of the overall design. After that, the topology tool will run to get the optimized model in free form. This process involves the penetration of coated material layers that deliver major weight reduction to 60% to 80% weight of the original component.

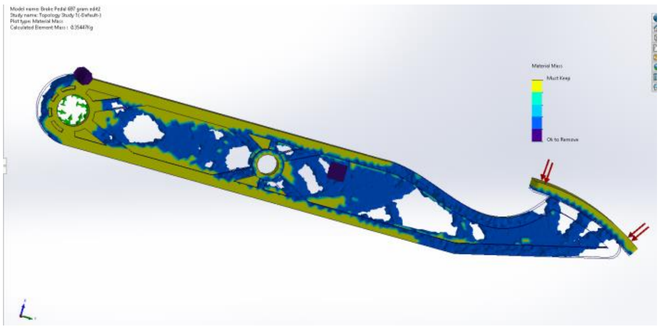


Fig-14: Element Mass: 0.35447 Kg

The utilization of zones within the study space is a salient function of the topology optimization as it controls the parts to which material reduction is affluent in the sense of the weight reduction tensile strength and performance. This is then followed by the regions being distributed. Subsequently, total volume redistribution via SolidWorks throughout those regions is accomplished repetitively to ensure that the specific goals are satisfied, such as having less mass while retaining performance.

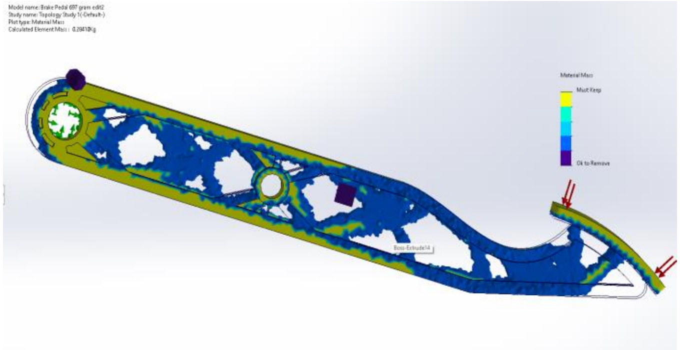


Fig-15: Element Mass: 0.28410 Kg

The free form of the final optimized model standing for the result of its optimization process that fulfills the optimization goals has been changed to meet the targets that were set at its beginning. This model which utilizes structural efficiencies, saved mass and increased performance is referred to as an optimized model. This is a development from the initial product. By using the SolidWorks software capacity in spending on topology optimization, the engineers can thereby rapidly explore and recognize high-weight savings demarches that do not make compromises of either structural integrity or functionality.

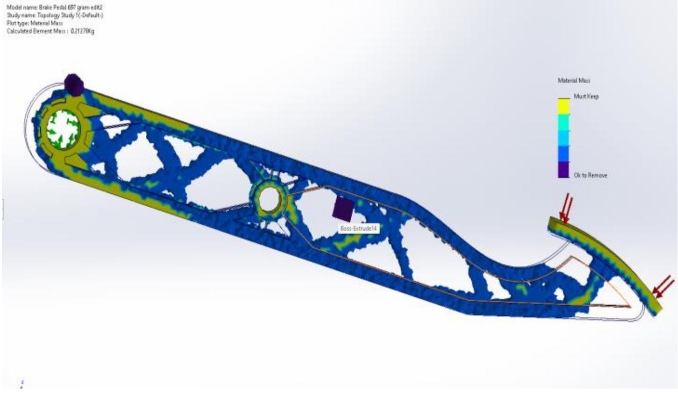


Fig-16: Element Mass: 0.21278 Kg

The iterative process of sectioning, running the topology slots, and analyzing the free-form analysis model which shows the optimized shapes that can be achieved enables the engineers to iteratively refine and improve the designs until the desired mass reduction and performance goals are met. Summing up, topology optimization with SolidWorks software is a very potent tool for engineers to realize the best and most effective imaginable designs which, due to making the components lighter, leads to substantial savings of weight of the component across different industrial areas, such as automotive, aerospace and manufacturing.

6. The Optimized Model Underwent a Redesign and FEA Analysis.

Details on Optimized Model -1 (0.48527 kg)

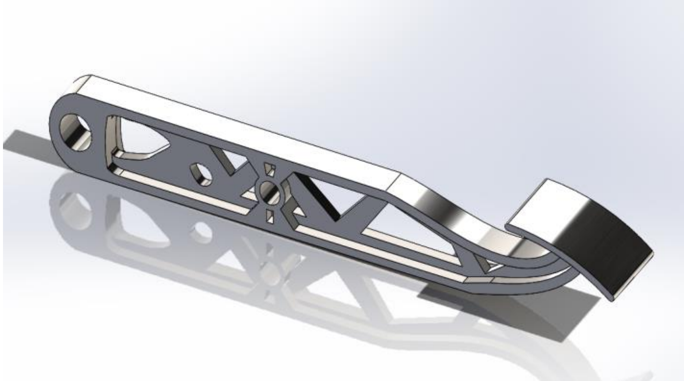


Fig-17: Model -1 (0.48527 kg)

6.1 Static Structural Analysis of Optimized Model-1

In all cases, experiments are undertaken to ensure identical conditions are maintained for both optimized and original brake pedals. The tension of 0.12309 MPa is applied to a particular model until it breaks at the free end while being fixed a the hinged end. Testing has static and dynamic types of loads comprising scenarios. Stress or deformation is measured as the difference between the stress result of the deformed and Undeformed pedal. This procedure involves the simulations involving the optimized designs which must withstand the same operational environment as the original pedal to ensure that the design parameters have been successfully validated. Through comparative stress-strain analysis of the expected outcomes and the original part, engineers discover whether the improved design has the same functionality and safety characteristics as the original one under intense field testing. Such extensive analysis ensures that the optimized models recommended for real use in automotive systems are robust and adequate, which empowers the systems to deliver effective services and perform at a high standard.

6.2 Equivalent Von Mises stress in static structural analysis.

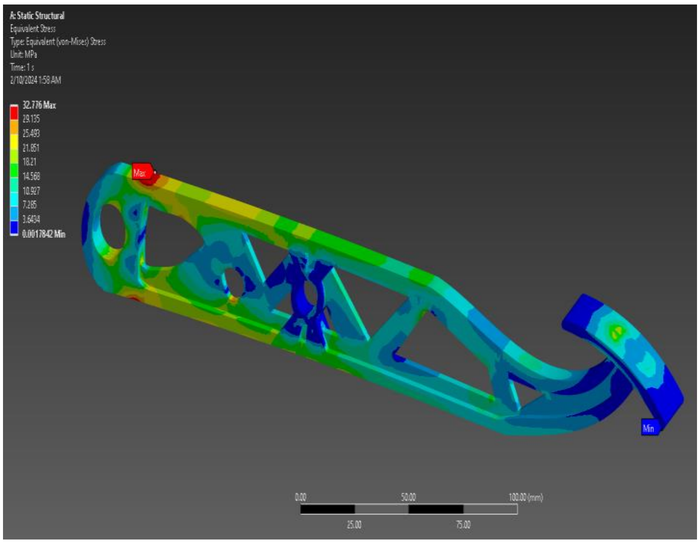


Fig-18: Equivalent Von Mises Stress result

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 32.776 MPa and 0.0017842 MPa, respectively.

6.3 Total Deformation in Static Structural Analysis.

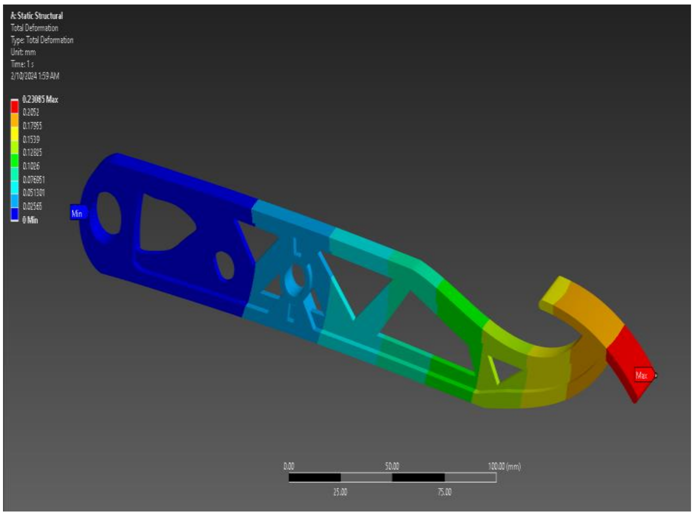


Fig-19: Deformation results

The total deformation analysis revealed maximum and minimum deformations of 0.23085 mm and 0 mm, respectively.

6.4 Safety Factor

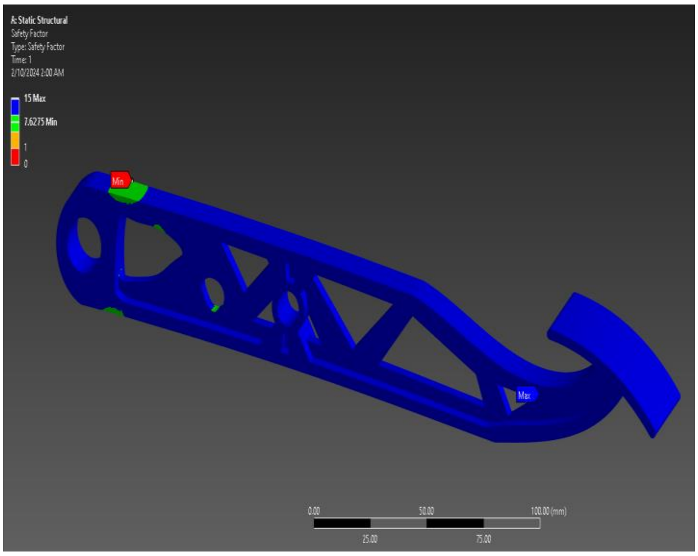


Fig-20: Safety Factor

Safety Factor Max: 15 and Min: 7.6275.

6.5 Transient Structural Analysis of Optimized Model-1

Equivalent Von Mises stress in transient structural

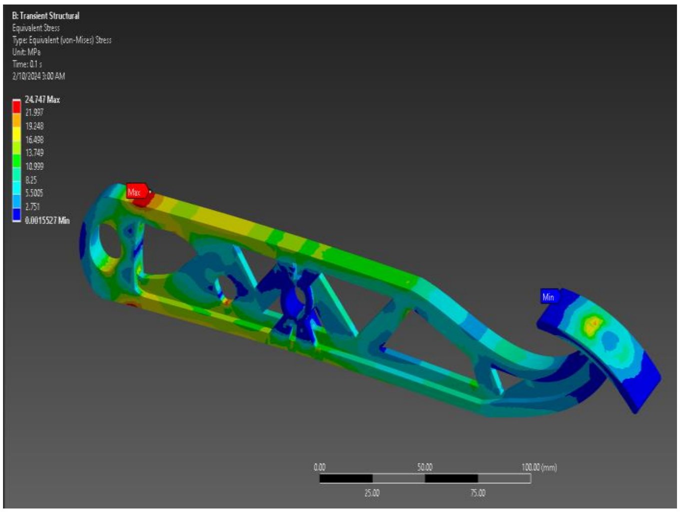


Fig-21: Equivalent Von Mises Stress result

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 24.747 MPa and 0.0015527 MPa, respectively.

Total Deformation in transient structural Analysis.

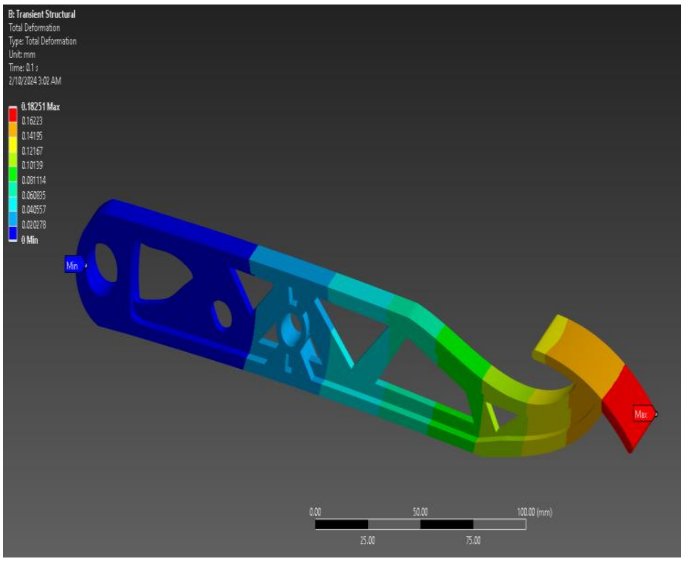


Fig-22: Deformation results

The total deformation analysis revealed maximum and minimum deformations of 0.18251 mm and 0 mm, respectively.

Details on Optimized Model -2 (0.54716 kg)

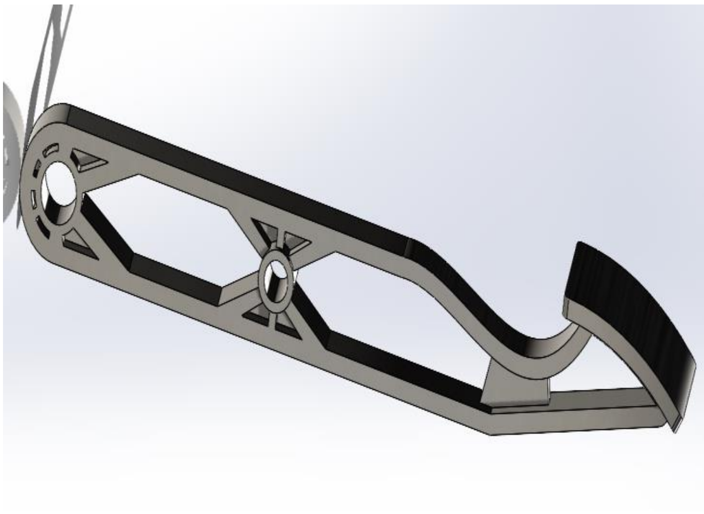


Fig-23: Model -2 (0.54716 kg)

6.2.1 Static Structural Analysis of Optimized Model-2

Equivalent Von Mises stress in static structural analysis.

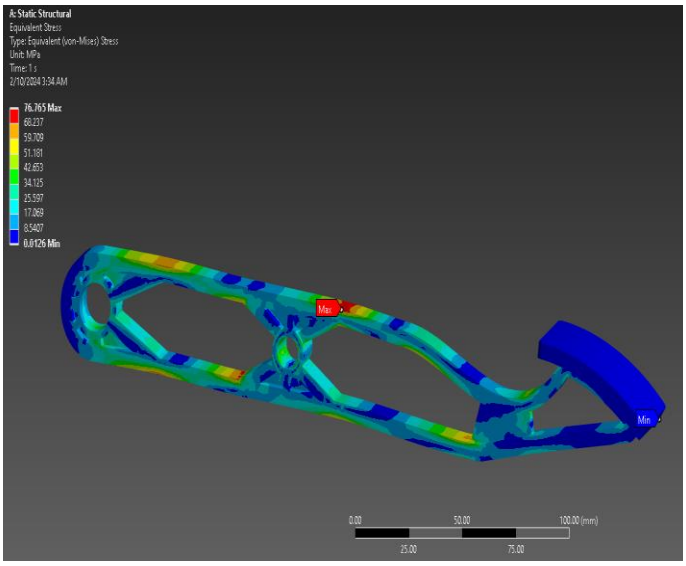


Fig-24: Equivalent Von Mises Stress result

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 76.765 MPa and 0.0126 MPa, respectively.

Total Deformation in static structural analysis.

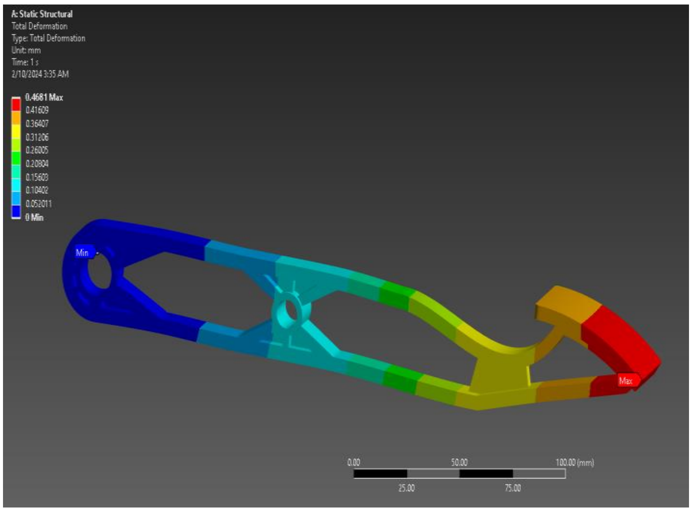


Fig-25: Deformation results

The total deformation analysis revealed maximum and minimum deformations of 0.4681 mm and 0 mm, respectively.

6.2.2 Safety Factor

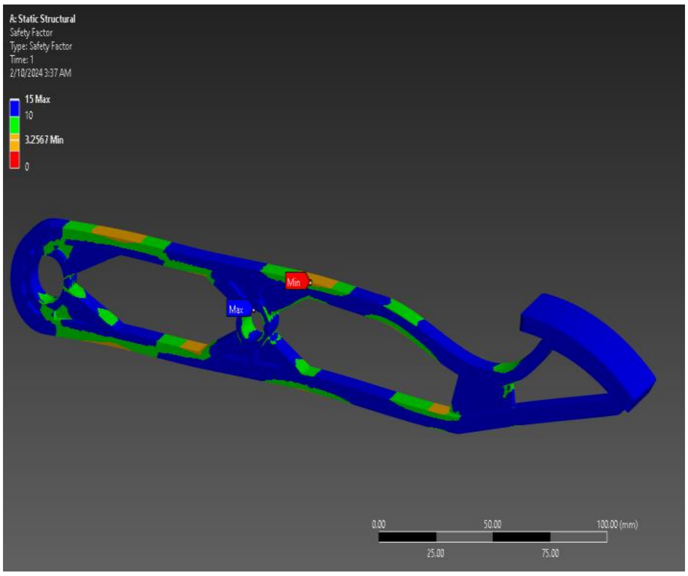


Fig-26: Safety Factor

Safety Factor Max: 15 and Min: 3.2567.

6.2.3 Transient Structural Analysis of Optimized Model-2

Equivalent Von Mises stress in transient structural

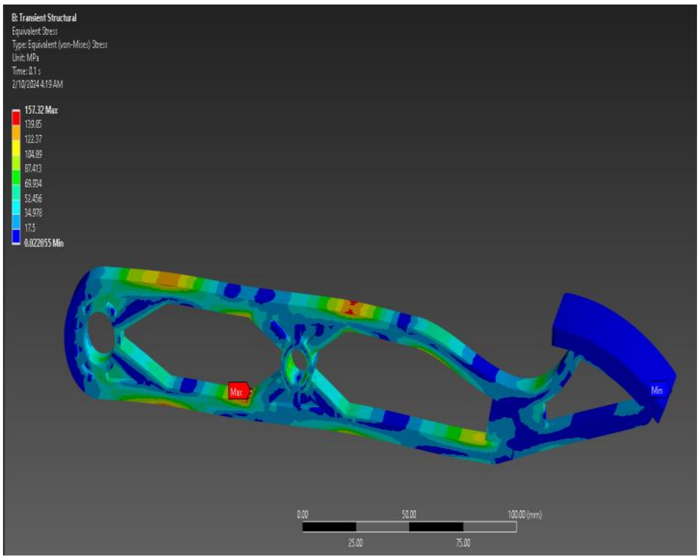


Fig-27: Equivalent Von Mises Stress result

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 157.32 MPa and 0.022055 MPa, respectively.

6.2.4 Total Deformation in transient structural Analysis.

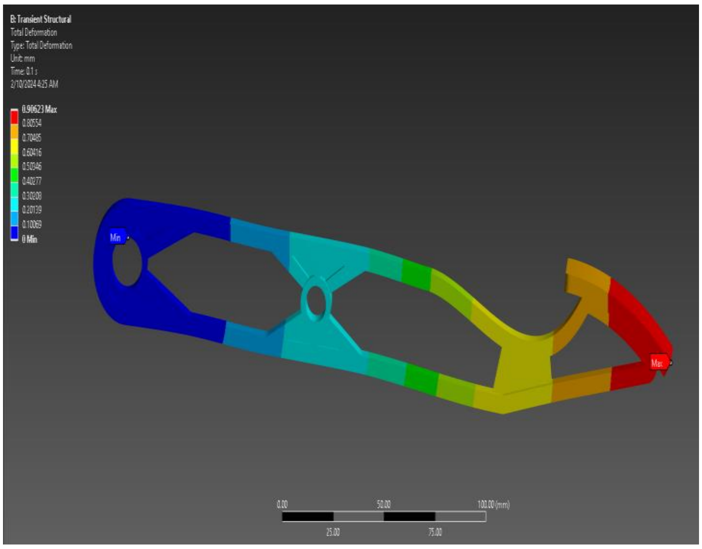


Fig-28: Deformation results

The total deformation analysis revealed maximum and minimum deformations of 0.90623 mm and 0 mm, respectively.

Details on Optimized Model -3 (0.562 kg)

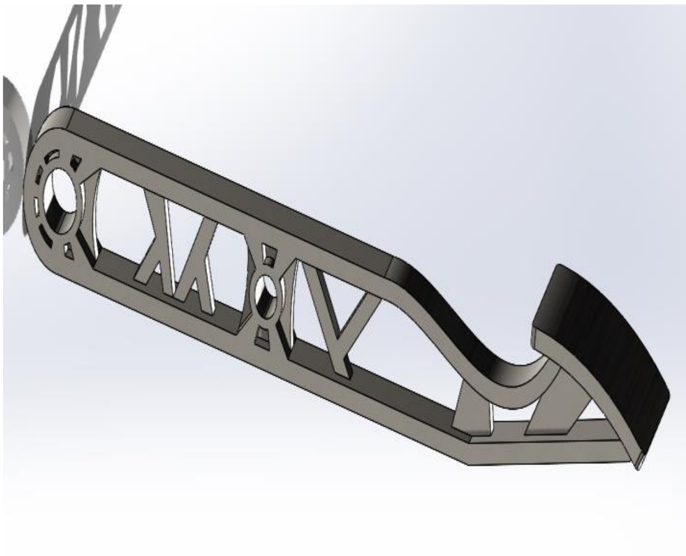


Fig-29: Model -3(0.562 kg)

6.3.1 Static Structural Analysis of Optimized Model-3

Equivalent Von Mises stress in static structural analysis.

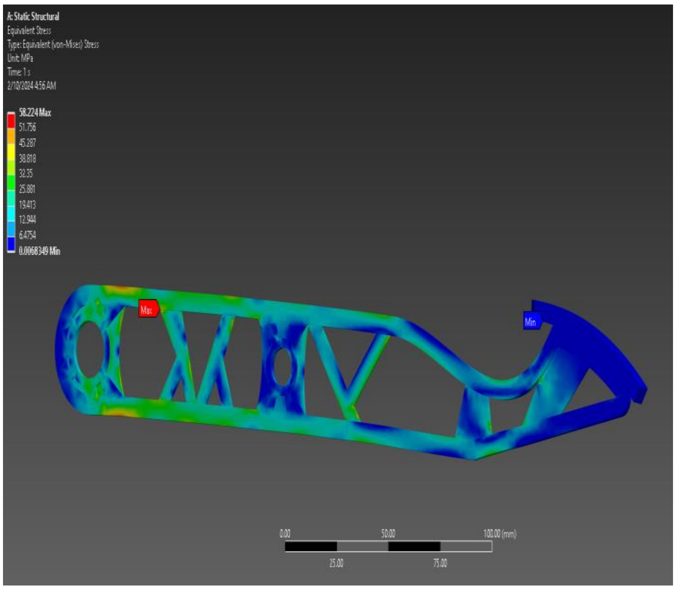


Fig-30: Equivalent Von Mises Stress result

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 58.224 MPa and 0.0068 MPa, respectively.

Total Deformation in static structural analysis.

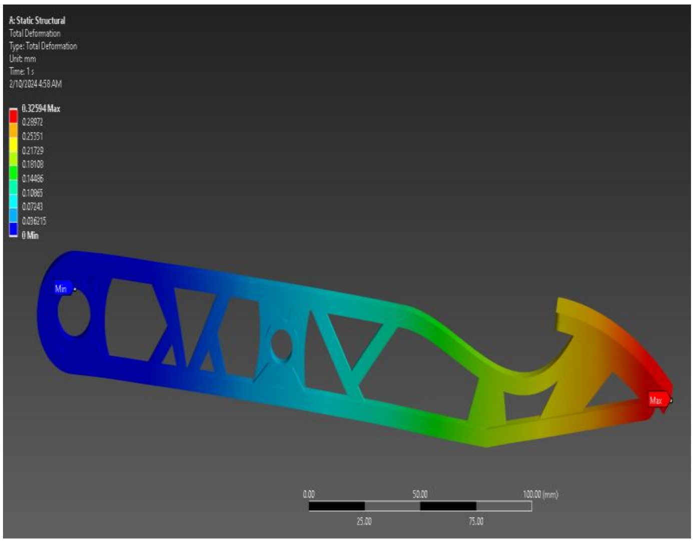


Fig-31: Deformation results total deformation analysis revealed maximum and minimum deformations of 0.32594 mm and 0 mm, respectively.

6.3.2 Safety Factor

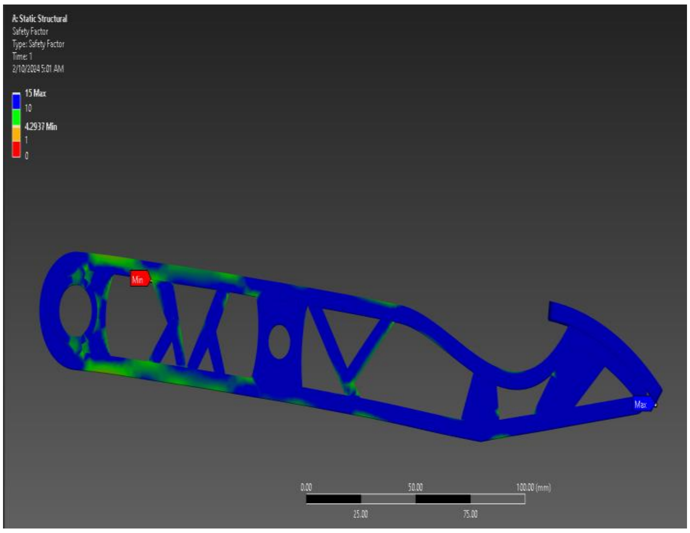


Fig-32: Safety Factor

Safety Factor Max: 15 and Min: 4.2937.

6.3.3 Transient Structural Analysis of Optimized Model-3

Equivalent Von Mises stress in transient structural

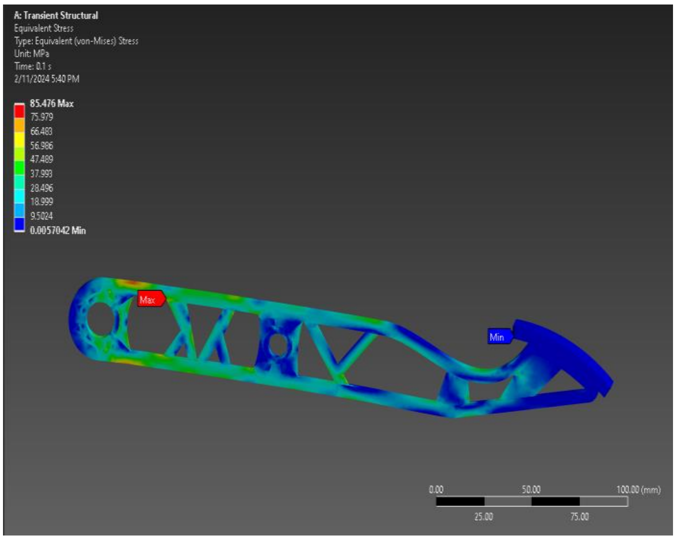


Fig-33: Equivalent Von Mises Stress result

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 85.476 MPa and 0.0057042 MPa, respectively.

Total Deformation in transient structural Analysis.

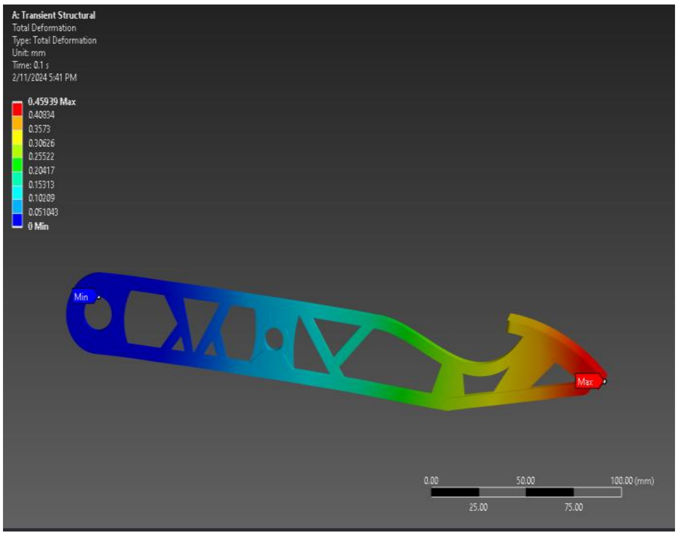


Fig-34: Deformation results

The total deformation analysis revealed maximum and minimum deformations of 0.45939 mm and 0 mm, respectively.

Details on Optimized Model - 4 (0.53694 kg)

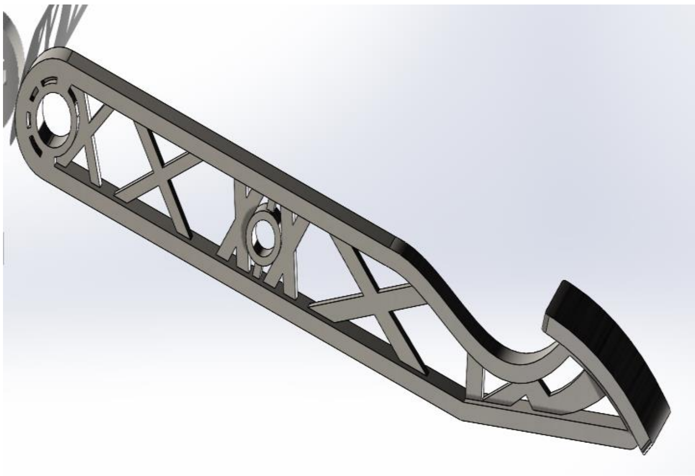


Fig-35: Model – 4 (0.53694 kg)

6.4.1 Static Structural Analysis of Optimized Model- 4

Equivalent Von Mises stress in static structural analysis.

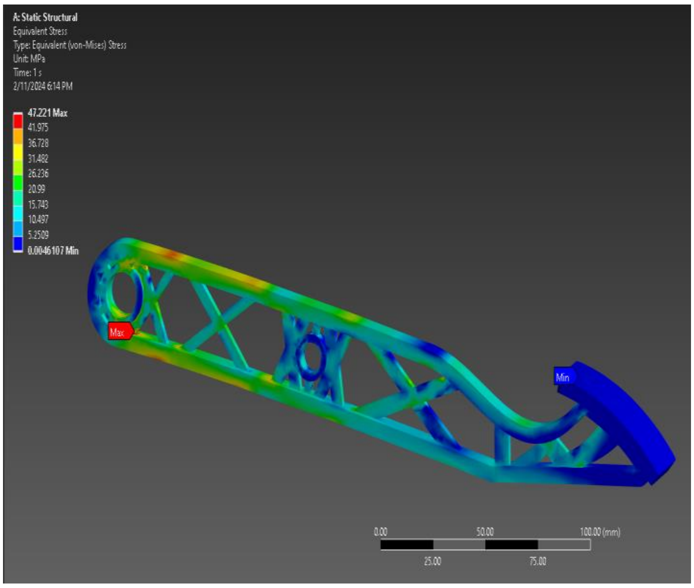


Fig-36: Equivalent Von Mises Stress result

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 47.221 MPa and 0.0046107 MPa, respectively.

Total Deformation in static structural analysis.

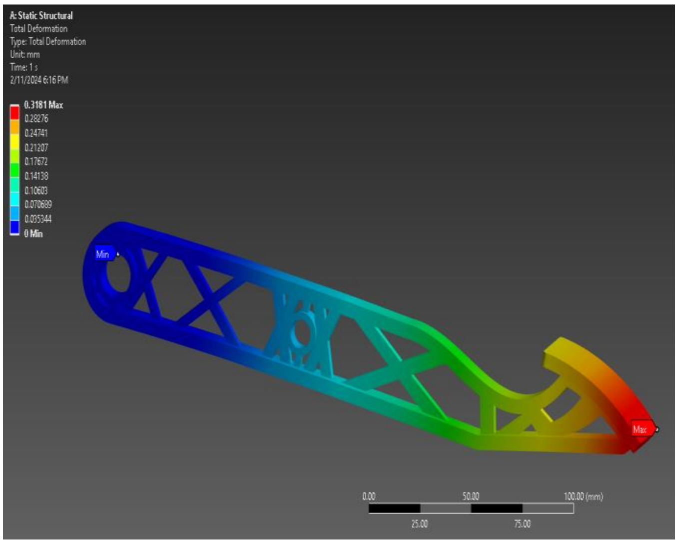


Fig-37: Deformation results

The total deformation analysis revealed maximum and minimum deformations of 0.3181 mm and 0 mm, respectively.

6.4.2 Safety Factor

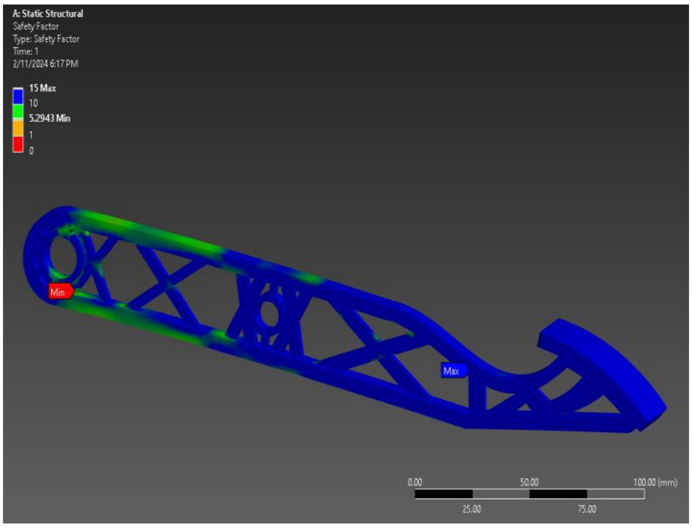


Fig-38: Safety Factor

Safety Factor Max: 15 and Min: 5.2943.

6.4.3 Transient Structural Analysis of Optimized Model-4

Equivalent Von Mises stress in transient structural

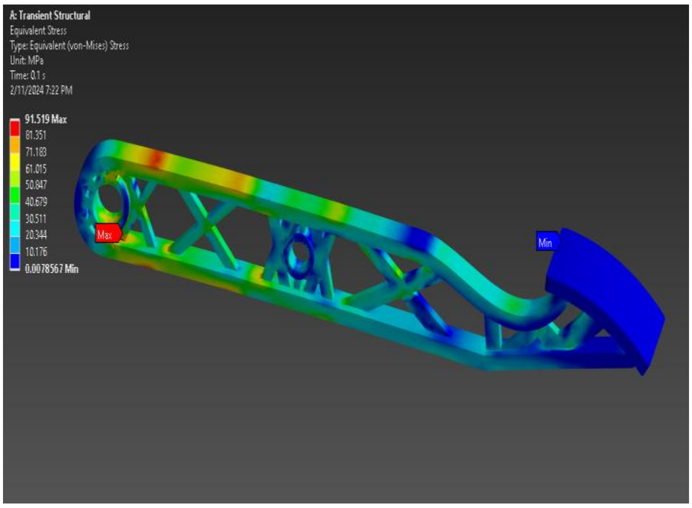


Fig-39: Equivalent Von Mises Stress result

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 91.519 MPa and 0.0078567 MPa, respectively.

Total Deformation in transient structural Analysis.

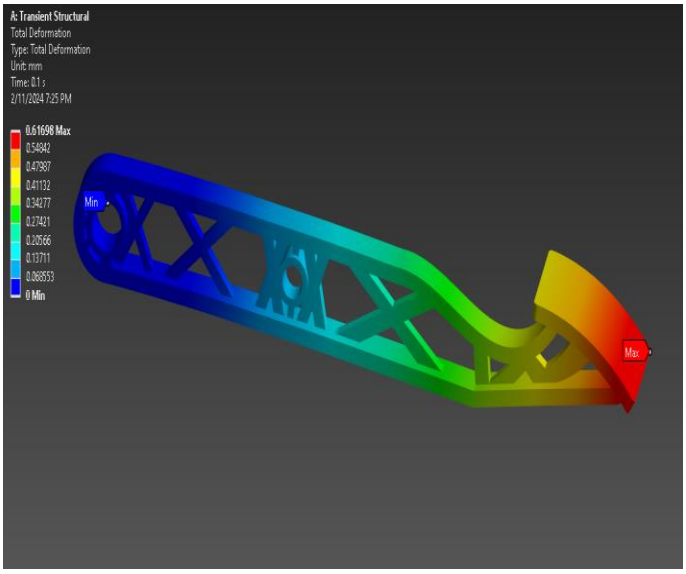


Fig-40: Deformation results

The total deformation analysis revealed maximum and minimum deformations of 0.61698 mm and 0 mm, respectively.

Details on Optimized Model -5 (0.54843 Kg)

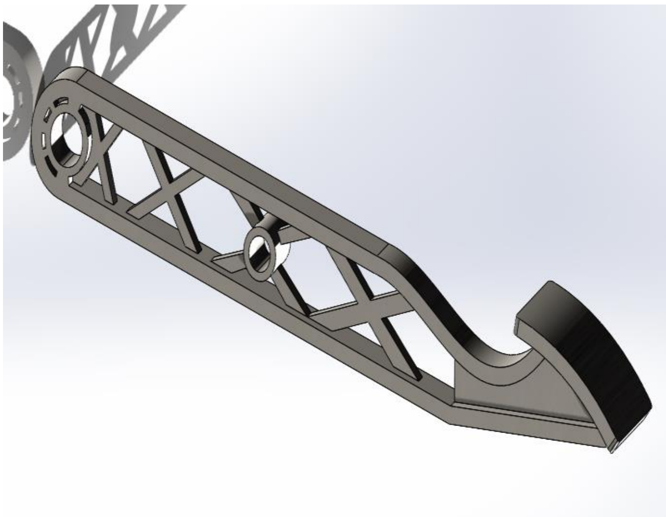


Fig-41: Model – 5 (0.54843 kg)

6.5 Static Structural Analysis of Optimized Model-5

Equivalent Von Mises stress in static structural analysis.

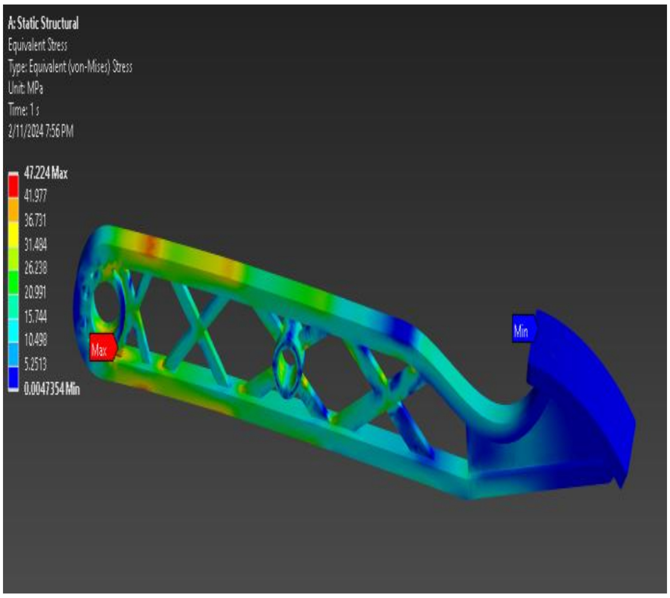


Fig-42: Equivalent Von Mises Stress result

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 47.224 MPa and 0.0047354 MPa, respectively.

Total Deformation in static structural analysis.

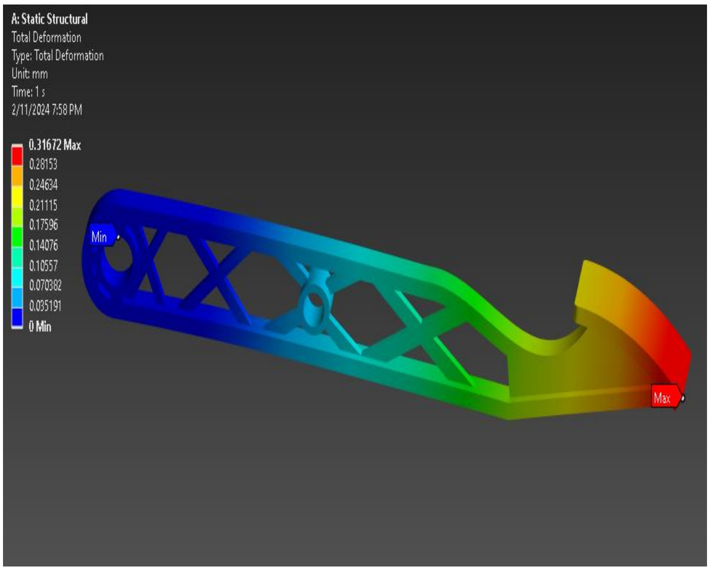


Fig-43: Deformation results

The total deformation analysis revealed maximum and minimum deformations of 0.31672 mm and 0 mm, respectively.

6.5.1 Safety Factor

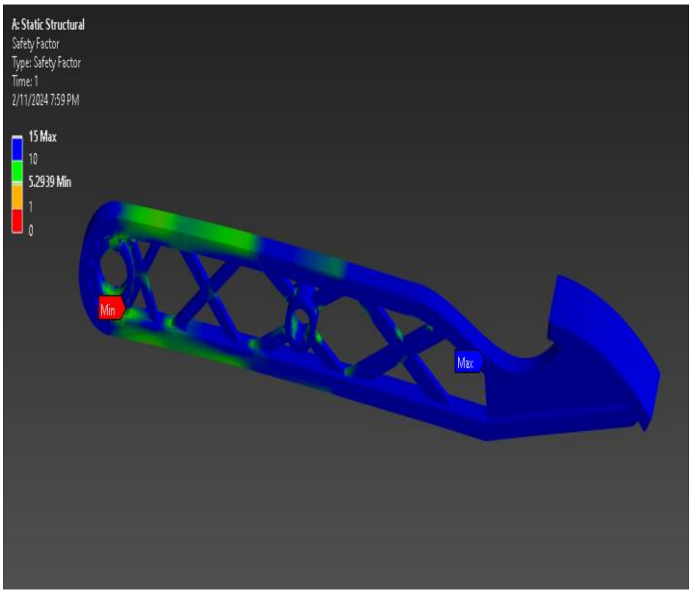


Fig-44: Safety Factor

Safety Factor Max: 15 and Min: 5.2939.

6.5.2 Transient Structural Analysis of Optimized Model - 5

Equivalent Von Mises stress in transient structural

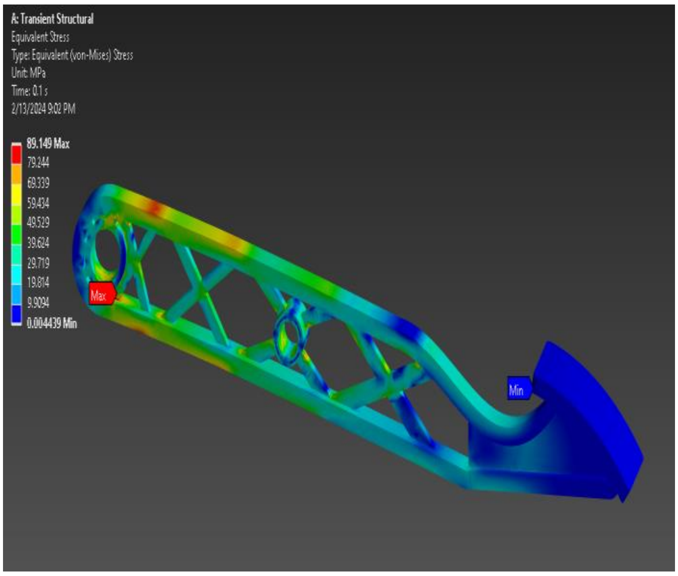


Fig-45: Equivalent Von Mises Stress result

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 89.149 MPa and 0.004439 MPa, respectively.

6.5.3 Total Deformation in transient structural Analysis.

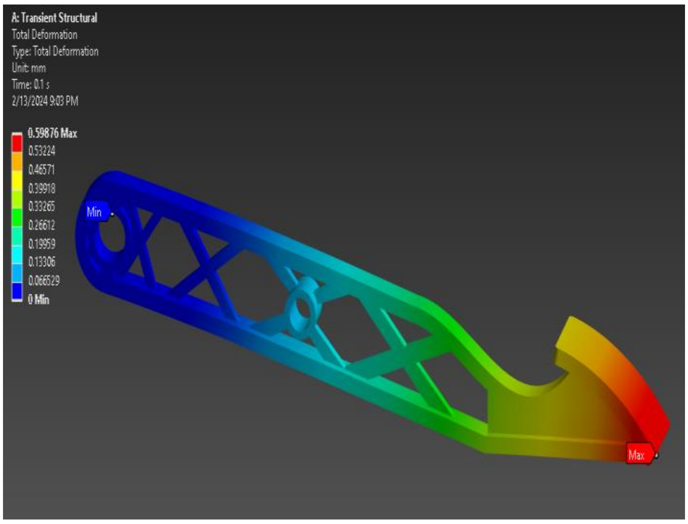


Fig-46: Deformation results

The total deformation analysis revealed maximum and minimum deformations of 0.59876 mm and 0 mm, respectively.

Details on Optimized Model - 6 (0.57803kg)

Fig-47: Model – 6 (0.57803 kg)

6.6 Static Structural Analysis of Optimized Model - 6

Equivalent Von Mises stress in static structural analysis.

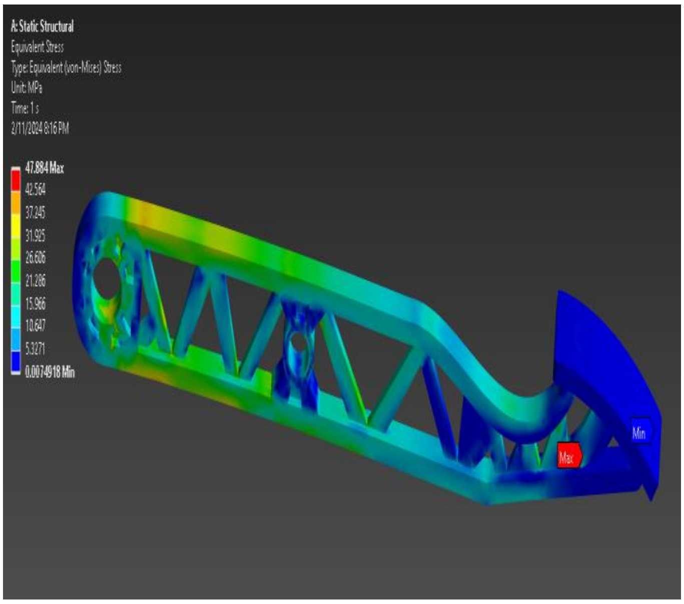


Fig-48: Equivalent Von Mises Stress result

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 47.884 MPa and 0.0074918 MPa, respectively.

Total Deformation in static structural analysis.

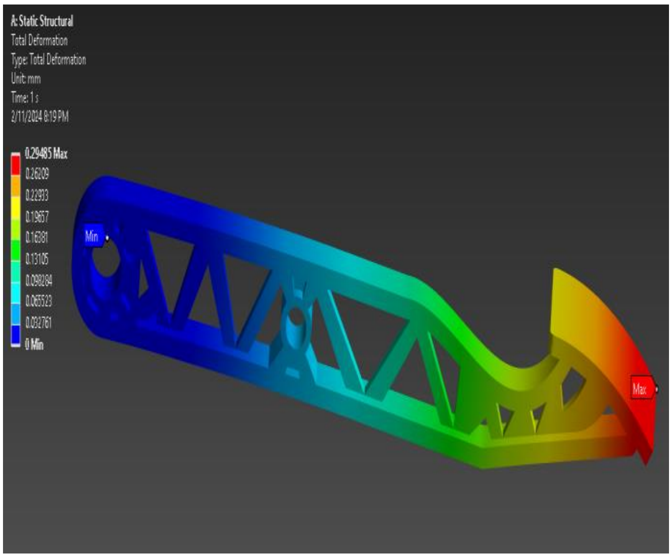


Fig-49: Deformation results

The total deformation analysis revealed maximum and minimum deformations of 0.29485 mm and 0 mm, respectively.

6.6.1 Safety Factor

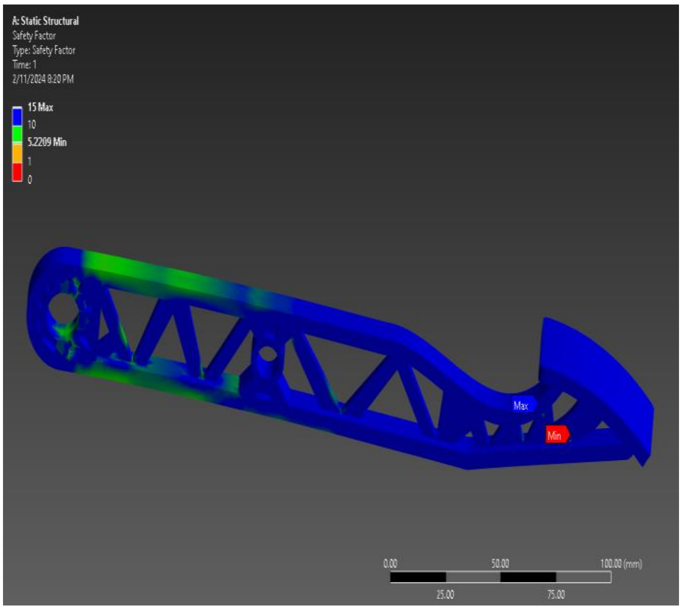


Fig-50: Safety Factor

Safety Factor Max: 15 and Min: 5.2209.

6.6.2 Transient Structural Analysis of Optimized Model-6

Equivalent Von Mises stress in transient structural

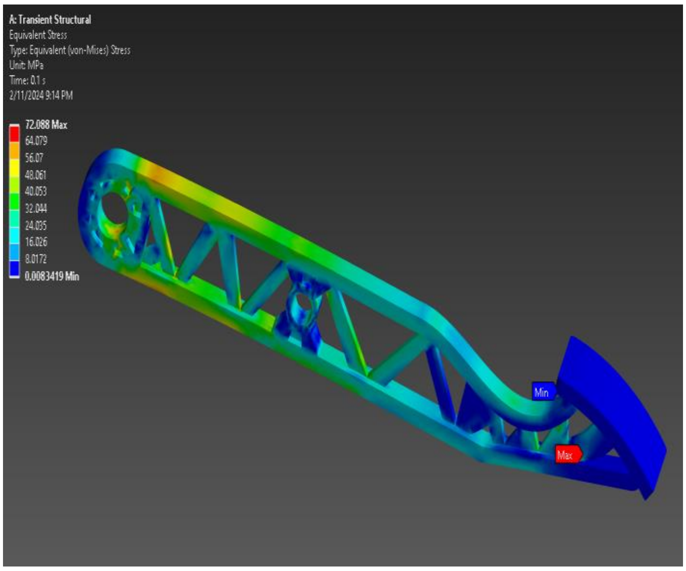


Fig-51: Equivalent Von Mises Stress result

The Von-Mises stress analysis yielded maximum and minimum equivalent stresses of 72.088 MPa and 0.0083419 MPa, respectively.

Total Deformation in transient structural Analysis.

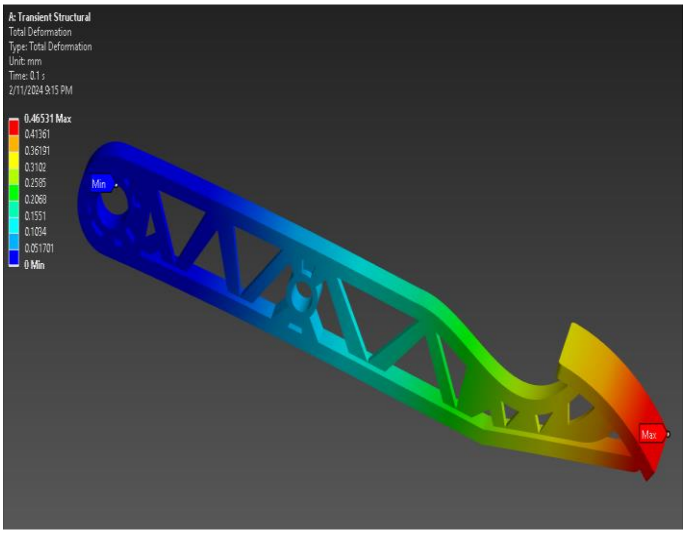


Fig-52: Deformation results

The total deformation analysis revealed maximum and minimum deformations of 0.46531 mm and 0 mm, respectively.

7. Results and Discussion

Table-3: Von Misses stresses (MPa)

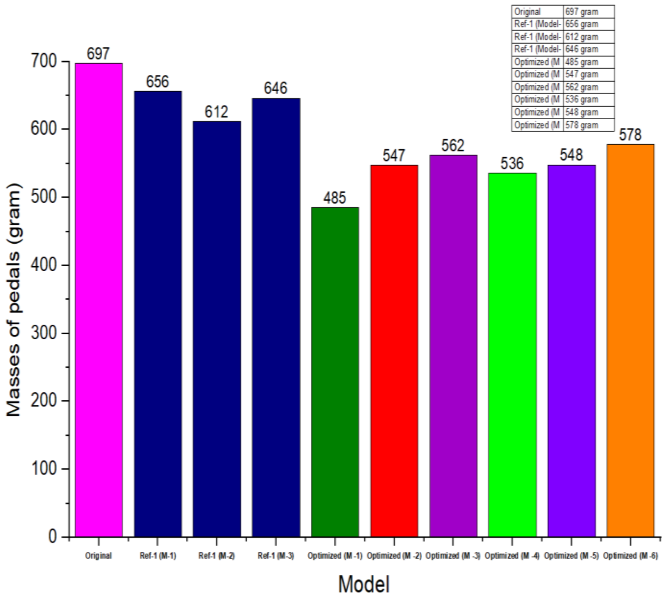
NO	Model	Stress (static) Max	Stress (Transient) Max
1	Original	36.40	56.189
2	Reference-1 (Model-1)	35.65	19.009
3	Reference-1 (Model-2)	50.69	27.321
4	Reference-1 (Model-3)	35.96	21.408
5	Optimized -1	32.776	24.747
6	Optimized -2	76.765	157.32
7	Optimized -3	58.224	85.476
9	Optimized -4	47.221	91.519
10	Optimized -5	47.224	89.149
11	Optimized -6	47.884	72.088

Table 4: Total deformation (mm)

NO	Model	Stress (static) Max	Stress (Transient) Max
1	Original	0.2354	0.33297
2	Reference-1 (Model-1)	0.23	0.1336
3	Reference-1 (Model-2)	0.25	0.16947
4	Reference-1 (Model-3)	0.248	0.15494
5	Optimized -1	0.2308	0.18251
6	Optimized -2	0.4681	0.90623
7	Optimized -3	0.3259	0.45939
9	Optimized -4	0.3181	0.61698
10	Optimized -5	0.3167	0.59876
11	Optimized -6	0.2948	0.46531

Table 5: Masses of pedals (gm)

No	Model	Mass (gm)	Weight saved (gm)
1	Original	697	Reference
2	Reference-1 (Model-1)	656	41
3	Reference-1 (Model-2)	612	85
4	Reference-1 (Model-3)	646	51
5	Optimized -1	485	212
6	Optimized -2	547	150
7	Optimized -3	562	135
9	Optimized -4	536	161
10	Optimized -5	548	149
11	Optimized -6	578	119

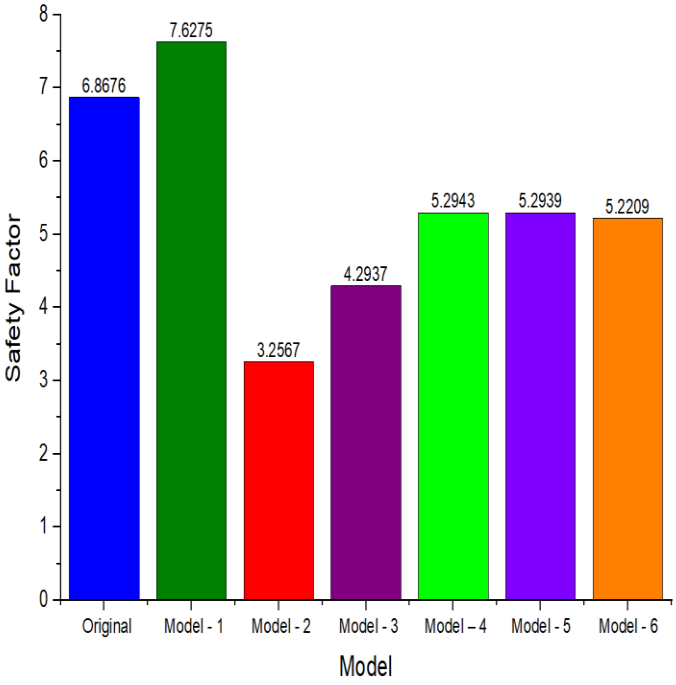


Graph 1: All of the Model Masses (gm) Graph

Table

Model	Safety Factor
Original	6.8676
Model - 1	7.6275
Model - 2	3.2567
Model - 3	4.2937
Model – 4	5.2943
Model - 5	5.2939
Model - 6	5.2209

-6: Safety Factor



Graph 2: All of the optimization model safety factors Graph

In this study, the original brake pedal has a mass of 697 grams, In Reference models 1,2,3 achieves weight savings of 41,85 and 51 grams, respectively. on the contrary, The optimized model presented in this paper (model 1- 6) demonstrates significantly greater weight saving, with saving of 212,150, 135,161,149, and 119 grams respectively. As a result, the safety factor of the original model is calculated to be 6.8676. Upon optimization, we can measure the safety factor for model 1 as 7.6275, for model 2 as 3.2567, for model 3 as 4.2937, for model 4 as 5.2943, and model 5 as 5.2939, for model 6, and 5.2209, respectively. The safety criteria revealed that the models passed all of the limitations and demonstrated a topology optimization process that increased the safety of the pedal of the brake while decreasing the weight simultaneously, which highlights the effectiveness of the topology optimization approach in improving both weight and safety performance of the brake pedal.

8. Conclusion

Finally, it can be said that the paper shows its efficiency as being applied to weight reduction of brake pedal by not only improving but keeping up the efficiency. Through the optimization process, substantial weight savings were achieved in comparison to reference models, with reductions ranging from 119 to 212 grams across six optimized models. On top of this, the safety factors of all optimized models being checked with the safety standards affirm the design and ability of the brake pedal to sustain and withstand high loads and impact. The results in question, thus, verify the efficacy of topology optimization in imparting the next idea of increasing the efficiency and performance of automotive parts. Such optimization processes by engineers with the help of advanced optimization methods can make brake pedals lightweight but tough enough to be efficient for the whole improved vehicle efficiency and safety. The paper above, therefore, demonstrates the role played by topology optimization in the design and optimum configuration of automotive components, and it unveils the possible avenues of development in lightweight and high-performance automotive components for future purposes.

References

1. Kharde, A., et al., *Design and mass optimization of brake pedal using topology optimization technique.* International Research Journal of Engineering and Technology (IRJET), 2021. **8**: p. 2395-0056.

2. Piekło, J., S. Pysz, and M. Małysza, *Zastosowanie optymalizacji topologicznej w kształtowaniu odlewanych konstrukcji szkieletowych.* Prace Instytutu Odlewnictwa, 2014. **54**.

3. Doğan, O., et al., *Optimum Design of Brake Pedal for Trucks Using Structural Optimization and Design of Experiment Techniques.* International Journal of Automotive Science And Technology, 2020. **4**(4): p. 272-280.

4. Akçay, Ö., *Structural Optimization of the Brake Pedal using Artificial Intelligence.* International Journal of Automotive Science And Technology, 2023. **7**(3): p. 187-195.

5. Salit, M.S., et al., *Conceptual design of an automotive composite brake pedal.* Suranaree J. Sci. Technol, 2005. **12**(3): p. 173-177.

6. Dhande, K., N. Jamadar, and S. Ghatge, *Design and analysis of composite brake pedal: an ergonomic approach.* Int J Mech Eng Robot Res, 2014. **3**(3): p. 474-482.

7. GAWANDE, M.A.N., G. KONDHALKAR, and A.R. PAWAR, *STATIC STRUCTURAL ANALYSIS AND OPTIMIZATION OF BRAKE PEDAL.* 2017.

8. Ergenc, A.F., et al., *Reduced weight automotive brake pedal test & analysis.* International Journal of Automotive Science and Technology, 2017. **1**(2): p. 8-13.

9. Kaya, N., I. Karen, and F. Öztürk, *Re-design of a failed clutch fork using topology and shape optimisation by the response surface method.* Materials & Design, 2010. **31**(6): p. 3008-3014.

10. Cavazzuti, M., et al., *High performance automotive chassis design: a topology optimization based approach.* Structural and Multidisciplinary Optimization, 2011. **44**: p. 45-56.

11. Chang, J.W. and Y.S. Lee, *Topology optimization of compressor bracket.* Journal of mechanical science and technology, 2008. **22**: p. 1668-1676.

12. Sudin, M.N., et al., *Topology optimization in automotive brake pedal redesign.* International Journal of Engineering and Technology (IJET), 2014. **6**(1): p. 398-402.

13. Romero, J. and N. Queipo, *Reliability-based and deterministic design optimization of a FSAE brake pedal: a risk allocation analysis.* Structural and multidisciplinary optimization, 2017. **56**: p. 681-695.

14. Das, S., *Design and weight optimization of aluminum alloy wheel.* International Journal of Scientific and Research Publications, 2014. **4**(6): p. 1-12.

15. Yadav, P. and P. Ramadasi, *Optimization of car rim using Optistruct.* J. Environmental Sci., Toxicology, and Food Tech, 2012. **2**(3): p. 10-15.

16. Xiao, D., et al., *Novel steel wheel design based on multi-objective topology optimization.* Journal of Mechanical Science and Technology, 2014. **28**: p. 1007-1016.

17. Kumar, C.P.R. and R.S. Meher, *Topology optimization of aluminium alloy wheel.* International Journal of Modern Engineering Research, 2013. **3**: p. 1548-1553.

18. Ragul, G., et al., *Design, Analysis And Impact Behaviour Of Magnesium Alloys (Zk60a) Of Low Pressure Die Casting For Automotive Wheels By Finite Element Method.* 2006.

19. Kulangara, A.J., C. Rao, and J. Cherian, *Topology optimization of lattice structure on a brake pedal.* Materials Today: Proceedings, 2021. **47**: p. 5334-5337.

20. Sargini, M., et al., *Additive manufacturing of an automotive brake pedal by metal fused deposition modelling.* Materials Today: Proceedings, 2021. **45**: p. 4601-4605.

BIOGRAPHIES

RUYHAN
Master's Student
School of Mechanical Engineering
Xi'an Jiao Tong University
Xi'an, China

Abdullah Al Hossain Newaz
Master's Student
School of Mechanical Engineering
University of Bridgeport, USA

Md Mehedi Hassan Melon
Master's Student
Department of Business Administration,
International American University Los Angeles, USA