International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 05 | May 2021 www.irjet.net p-ISSN: 2395-0072

Design and Mass Optimization of Brake Pedal using Topology Optimization Technique

Aniket Kharde[1], Rajaguru Khande[2], Sagar Khedekar[3], Juned Manchekar[4], Jayesh S. Anavkar[5]

[1,2,3,4]U.G. students, Dept. of Mechanical Engineering, Finolex Academy of Management and Technology
[5]Assist. Professor, Dept. of Mechanical Engineering, Finolex Academy of Management and Technology, Maharashtra, India.

Abstract - *Topology Optimization (TO) is the process of removing the unwanted or excess mass from the components without compromising its strength and Working nature. TO is different from shape optimization and sizing optimization in the sense that the design can attain any shape within the design space, instead of dealing with predefined configurations. The literature review shows that optimization interns of mass was achieved by changing the materials and Trial and error method by changing the design of component. In this project work, Objective is to optimize the Brake Pedal using direct topology optimization technics.*

By the concept of reverse engineering and Reference data, all the design parameters of Brake Pedal are studied and 3D modelling is done using Solidworks 2020 software. Finite Element analysis software ANSYS 19.0 is integrated with Direct Topology optimization techniques that can be used as tool for Mass Optimization.

The FEA result will show that optimized model which will obtained after optimization method will have less mass with same strength and same load sustainability. These types of advance techniques helps the engineering's to enhance the efficiency of the vehicle by removing the excess mass from the different parts of the Automobile which intern reduces the cost of component.

***Key Words*: SOLIDWORKS, ANSYS, Brake pedal, Topology optimization, FEA analysis**

1. INTRODUCTION

Automobile Industries are thriving very hard to make their car light weight and efficient. Many methods are being utilized to achieve this goal of light performance vehicles. As a result, many design engineers are adopting to method known as Topology Optimization in the design phase of auto parts.

1.1 Topology Optimization

IT is a mathematical method that optimizes material layout within a given design space, for a given set of loads, boundary conditions and constraints with the goal of maximizing the performance of the system. TO is different from shape optimization and sizing optimization in the sense that the design can attain any shape within the design space, instead of dealing with predefined configurations.

Topology Optimization has a wide range of applications in aerospace, mechanical, bio- chemical and civil engineering. Currently, engineers mostly use TO at the concept level of a design process. Due to the free forms that naturally occur, the result is often difficult to manufacture.

For that reason the result emerging from TO is often fine-tuned for manufacturability. Adding constraints to the formulation in order to increase the manufacturability is an active field of research. In some cases results from TO can be directly manufactured using additive manufacturing. TO is thus a key part of design for additive manufacturing.

1.2 Implementation Methodologies

There are various implementation methodologies that have been used to solve TO problems.

1. Discrete
2. Solving the problem with continuous variables
3. Commercial Software

1.2.1 Discrete

Solving TO problems in a discrete sense is done by discretizing the design domain into finite elements. The material densities inside these elements are then treated as the problem variables. In this case material density of one indicates the presence of material, while zero indicates an absence of material. Due to the attainable topological complexity of the design being dependent of the amount of elements, a large amount is preferred. Large amount of finite elements increase the attainable topological complexity, but come at a cost. Firstly, solving the FEM system becomes more expensive. Secondly, algorithms that can handle a large amount (several thousands of elements is not uncommon) of discrete variables with multiple constraints are unavailable. Moreover, they are impractically sensitive to parameter variations. In literature problems with up to 30000 variables have been reported.

1.2.2 Solving the problem with continuous variable

The earlier stated complexities with solving TO problems using binary variables has caused the community to search for other options. One is the modelling of the densities with continuous variables. The material densities can now also attain values between zero and one. Gradient based algorithms that handle large amounts of continuous variables and multiple constraints are available. But the material properties have to be modelled in a continuous setting. This is done through interpolation. One of the most implemented interpolation methodologies is the SIMP method Solid Isotropic Material with Penalization This interpolation is essentially a power law. It interpolates the Young's modulus of the material to the scalar selection field.

1.2.3 Commercial Software

There are several commercial topology optimization software on the market. Most of them use topology optimization as a hint how the optimal design should look like, and manual geometry re-construction is required. There are a few solutions which produce optimal designs ready for Additive Manufacturing. Few examples of such Topology Optimization tools are ANSYS TO Module, Hyperworks Optistruct, Simright etc. These software basically combines the above two mathematical methods to directly optimize the product in consideration under proper boundary conditions and give the optimized geometry that can be used to redesign or directly manufacture the part. However, most of the topologically optimized parts have geometries that are impracticable to manufacture directly. Hence, manual redesign is necessary or the part can be directly manufacture using additive manufacturing, neglecting the aesthetics of the product.

1.3 Brake Pedals

The main function of the brake system is to decelerate or decrease the speed of a vehicle. By stepping on the brake pedal, the brake pads compress against the rotor attached to the wheel, which then forces the vehicle to slow down due to friction. Basically, the only work of a brake pedal is to transmit the force applied by the driver with his legs, to the master cylinder of the braking system and eventually to the breaking pads. Even though, a pedal is supposed to transmit a considerably small amount of force, there haven't been any change as such in the pedal design which eventually leads the manufacturers to use up excess material than needed causing material and money wastage. Hence, this work focusses on the mass reduction of the brake pedal using Topology Optimization tool, to obtain the perfect optimized design, low in weight and much more aesthetic in looks.

Fig-1: Brake Pedal

1.3.1 Brake Pedal design

First, let's define the term pedal ratio. The pedal ratio is the overall pedal length or distance from the pedal pivot called the fulcrum to center of the pad your foot will push against (L1+L2) divided by the distance from to the fulcrum to the master cylinder push rod attachment point (L1).

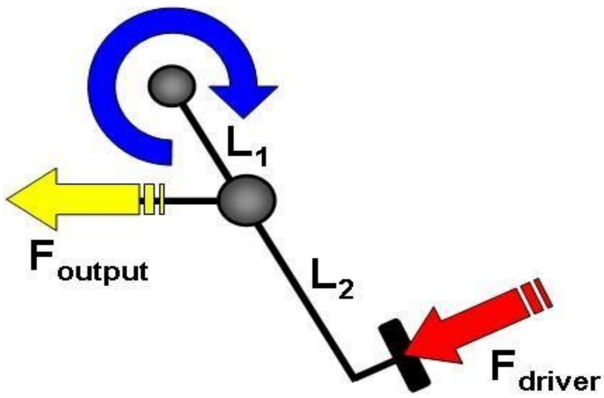


Fig-2 : Line diagram of brake pedal

Experience has shown that a pedal ratio of 6.2:1 is recommended (with 5.5:1 being the recommended minimum) to replace most of the brake force assist that was provided originally by the vacuum assist and the original equipment pedal ratio of 3.5 to 4.0:1. Shortening the distance from the fulcrum to the pushrod would have the same effect, but is a difficult task best left to an expert fabricator. Without this extra mechanical advantage, you would have to use a smaller master cylinder than may be available or you are at least setting yourself up for a difficult time in picking other system components to make up for the missing mechanical advantage. There are manywell- designed and manufactured pedal arrangements with dual master cylinders available from race component suppliers. If you still choose to fabricate your own assembly then take care to design and build a sound pedal.

1.3.2 Brake Pedal failure

The mechanical failure of the brake pedal is very rare in the entire life period of a vehicle because not much thought is given for its service life while designing to save time and the designers end up assigning giving it a very high factor of safety.

Fig-3: Failed Brake Pedal

But sometimes, it may happen that the brake pedal has manufacturing defects which leads to the formation of micro cracks or some other defect which fails the pedal. Hence, it would be a very wrong practice to just randomly remove material from the pedal to make it lightweight. That's why, Topology Optimization is a tool in the hands of design engineers to rectify the material wastage as well as maintain enough strength for infinite life of the pedal.

2. Literature Reviews

Miss. Ashwini N. Gawande, Prof. G. E. Kondhalkar, Prof. Ashish R. Pawar [1]: The existing brake pedal design is considered for structural analysis and a new optimized model is presented. Finite element analysis will be used to apply cantilever load Optistruct solver will be used to perform topology optimization. The model of an existing brake pedal was generated using CATIA V5 solid modelling software. Finally, a new light weight design brake pedal is proposed. The result of the study shows that the weight of a new designed brake pedal was less as compared to an existing brake pedal without sacrificing its performance requirement.

Mohd Sapuan Salit [2]: In automotive industries, metallic accelerators and clutch pedal are replacing with polymeric based composite pedals and the aim of replacement is weight reduction, cost saving of pedals using composites. In this research work, brake pedals have been investigated analytically and computationally from the properties of available and suitable polymeric-based composite, a final design of a composite brake pedal has been made.

Sandeep Ghatge [3]: The automotive industries accelerator and clutch pedal are replacing by lightweight materials such as plastic, polymer composites, aluminum and its alloys, etc. The purpose of replacement is improvement in corrosion resistance and reduction weight, cost. In design aspect; the steel material is replaced by light materials. In this study different lightweight materials of brake pedal are compared with conventional steel. For different sections for different loading and boundary conditions, these materials are analyzed. The purpose of this study is to design and analyze the brake pedal using CATIA and ANSYS software.

Bhagyashri Kurkure [4]: Now a days industries are replacing accelerator and clutch pedal by lightweight materials such as polymer plastic, composites, aluminum and its alloys, etc. The purpose is to reduce weight, cost, and improvement in corrosion resistance without change in material reduction in a commercial vehicle casted brake pedal lever. The FEM and analysis of a brake pedal lever has been carried out. The FE model was generated in CATIA or Pro-E and imported in ANSYS for stress analysis and then optimizing it with the help of Optistruct software. A comparison of baseline and optimized model FEA results have been done to conclude.

Saurav Das [5]: The aim of this paper is to design an aluminium alloy wheel is meeting all the design standards. Topology optimization is carried out on 5 cyclic cases on Abacus software. A new optimised design is analysed under radial, bending & lateral loads. Material used is B.S: LM25 Alloy. Finite element analysis is carried out on Hypermesh Optistruct software. The optimised design had the weight reduction of 52%, cast cutting at Rs 4000/- per component & the endurance stress was found 90MPa satisfying the yield criteria. The new design has less weight, improved fatigue strength, longer life & cost effective.

Mr. P.H.Yadav, Dr.P.G.Ramdasi [6]: The aim of this paper is to optimize the wheel by reducing the weight of the rim using finite element analysis. Altair Hyper works Opistruct software was used for the FE modelling, meshing & analysis of rim. The data was validated by using the ANSYS software, Aluminium casting alloy A356 was used as a material for the rim. Thickness of rim was reduced from 3mm to 2mm (outer) & 2.5mm (inner) which reduced 13.28% of weight of the rim.

S. Chaitanya, B.V Ramana Murty [7]: The aim of this project was to optimize the weight of wheel rim of automobile using Mass Optimisation technique. The author had considered $ different materials as a competitive replacement materials of aluminium alloy i.e. magnesium alloy. Titanium alloy & PEER 30% of carbon reinforce. CATIA V5 RS 1 3D modelling software were used for modelling of wheel rim & ANSYS 15.0 was used for finite element analysis of the component. FEA results on wheel rim for different materials shows that stresses induced in component was almost same in all materials. PEEK with 30% carbon reinforce material weights in 2.74kg as compared to 5.308kg of aluminium alloy. Also PEEK component results in corrosion resistance & increase in fatigue life & low cost of component.

Po Wu, Qihua Ma, Yiping Luo, Chao Tao [8]: The finite element model of the bracket is established according to the structural characteristics of the automobile engine bracket. As the connecting part between the engine and the body, the performance requirements of the automobile engine bracket affect the comfort and the safety of the vehicle directly. Using the RADIOSS solver, the dangerous point of the bracket is analyzed. Under the premise of ensuring its reliability, with the help of Opistruct software to carry out the topology optimization design, to get the optimal material distribution of the bracket and the final design will meet the performance requirements.

Ch. P.V. Ravi Kumar and Prof. R. Satya Meher [9]: The objective of this paper is "Topology optimization of cast aluminium alloy wheel" by increasing the thickness of wheel rim until the plastic strain value is below 4%. Main objective is to generate finite element modelusing Hypermesh V10.0. Impact ANALYSIS is carried out using LS-DYNA software to predict the plastic strain during impact test. A nonlinear elsto-plastic material model used to describe the material behaviour of aluminium wheel.

Vaibhav Pimpalte, Prof. S.C. Shilwant [10]: This project intends to identify the magnitude of the stresses for a given configuration of a two wheeler gears transmitting power while trying to find ways for reducing weight of the gear. The philosophy for driving this work is the lightness of the gear for a given purpose while keeping intact its functionality thus reducing the material cost of the gear. Ease of incorporating the new feature for weight reduction over the existing process of manufacturing and the magnitude of volume of weight reduced could be considered as the key parameters for assessment for this work.

3. Objective and problem definition

3.1 Objectives

•The main objective of this study is to explore weight reduction opportunities for a Brake Pedal using topology optimization.
•Therefore, this study deals with two subjects, first, static load stress and transient analysis of the brake pedal , and second, optimization for weight.
•In this project, finite element analysis of Brake Pedal is taken as a case study, Structural behavior of pedal can be easily analyzed using Finite Element techniques and weight optimization can be achieved using topology optimizing tools.

3.2 Problem Definition

Automobile industries are now mainly focused on weight reduction of the vehicles. A weight optimization opportunity is possible on the brake pedals. Brake pedals are unnecessarily designed with higher factor of safety. Because of which, excess material is used for its manufacturing, costing the manufacturers extra. Hence, Topology Optimization is a very effective tool for this task in order to get rid of that extra material, saving production cost and reducing weight for better vehicle performance.

4. Methadology

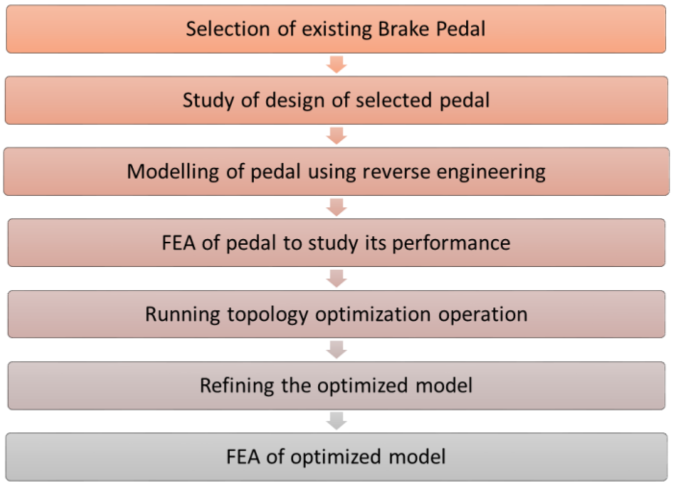


Fig-4 : Methadology

The sole purpose of a brake pedal to transmit the force applied by the driver to the master cylinder of the braking system. Selection of Brake pedal of automobile vehicle is done "Maruti suziki alto 800" and all the parameters of the rim are calculated using manual method of engineering using Vernier caliper and radius gauge. This method of measurement is called reverse engineering. All the measured data are noted down and Tabulated for 3D Modelling of the Brake pedal.Reverse Engineering Method is used to measure all the Parameters of the Brake Pedal. 3D Modeling of Brake Pedal has been done according to the dimensions obtained from Reverse Engineering. Solidworks 2020 is used for 3D designing of the Brake Pedal because of its user Friendly GUI. Analysis of Brake Pedal is done in Finite element analysis software i.e. ANSYS Workbench 19.0. Stresses and deformations have been calculated. Analysis is done for original selected Pedal. Static structural analysis is used to find the structural strength of the Pedal subjected to loading. ANSYS Topology Optimization tool is then incorporated to optimize the Pedal design. All the constraints for pedal design have been accounted for, before running topology optimization operation.The result obtained from ANSYS TO module is a free form design of the actual optimized model, which is very difficult to manufacture. Hence a touch up redesign is necessary in order to easily manufacture the component.

5. Reverse Engineering and 3D Modelling using Solidworks

5.1 Reverse Engineering
Reverse engineering, also called back engineering, is the processes of extracting knowledge or design information from product and reproducing it or reproducing anything based on the extracted information. The process often involves disassembling somethingand analyzing its components and workings in detail.

5.2 Introduction to Solidworks

Solid works is a solid-modelling computer aided design (CAD) and computer aided engineering (CAE) computer program that runs on Microsoft windows. Solid works is published by Dassault systems. Solid works currently markets several versions of the Solid works CAD software in addition to e-drawings, a collaboration tool and a draft sight a 2D CAD product. Building a model in Solid works usually stars with a 2D sketch.

5.3 3D modelling using Solidworks

A 3D CAD model of the Brake Pedal is to be created on SolidWorks 20. The existing Brake model is gauged with Vernier caliper and radius gauge. A detailed sketch of the component is then created on the CAD software. Each dimension is thoroughly measured and a 2D detailed draft is drawn in order to draw a 3D full scale model. The detailed sketch of the pedal is as shown in the Fig:

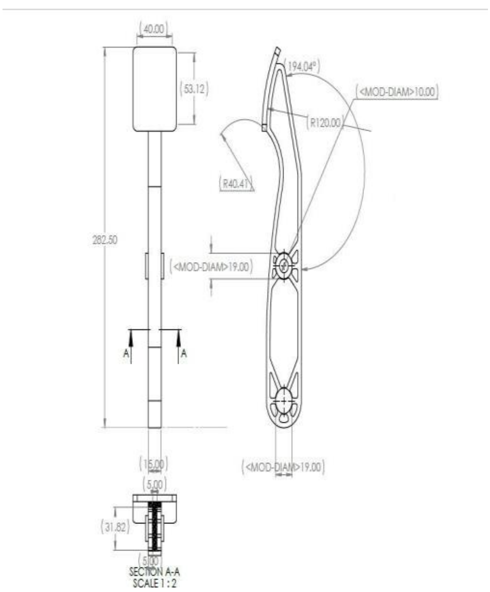


Fig-5: Detailed sketch of brake pedal

5.3.1 Model in SolidWorks 20

On studying the exact dimensions of the pedal using Reverse engineering, a 3D model is drawn on SolidWorks 2020

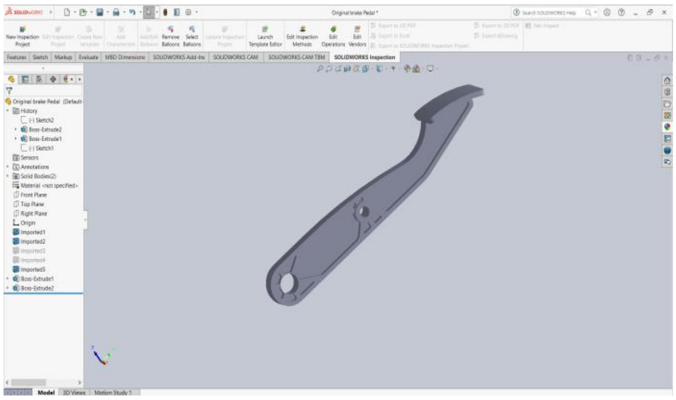


Fig-6: original brake pedal

6 Static and transient structural analysis of original brake pedal

6.1 Static Analysis of Pedal on ANSYS Workbench

Due to the ease of performing FEA, ANSYS simulation software is selected for the purpose. The model that was created on SolidWorks was imported to ANSYS in STP214 file format. The mathematical model defined by bounding constraints are subjected on the model.

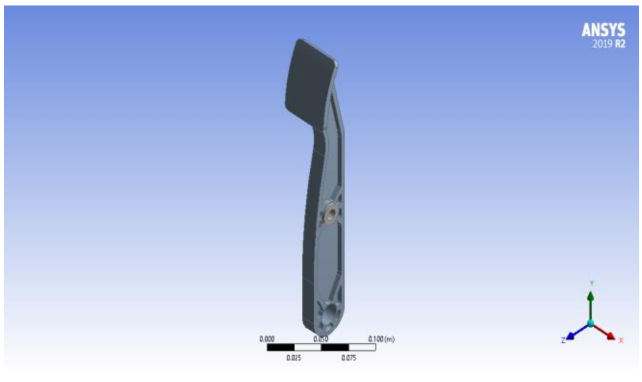


Fig-7: Original Brake Model imported in ANSYS

6.1.1 Meshing

ANSYS allows its users to either mesh the component automatically or manually by selecting the shape and size of each element. Here, the pedal is meshed using TRI and HEX combined elements to give better stiffness matrix in the region of pivots and curves

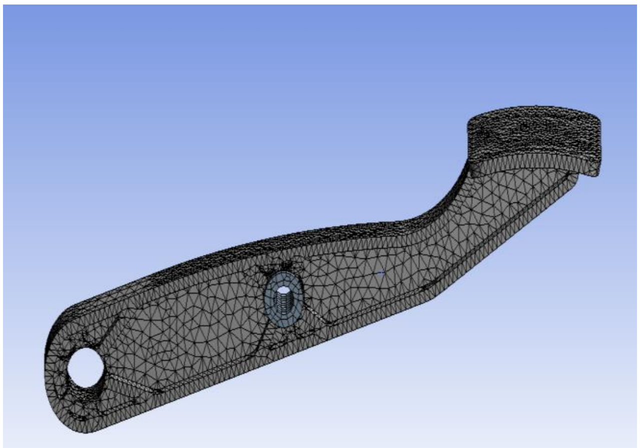


Fig-8: Meshed Model of Brake Pedal

6.1.2 Boundary Conditions

In literature, the max force that a person can apply on a brake pedal is accounted to be 40Kg.Hence Total force acting on brake pedal
F=40*9.81=392.4
Which approximate as 400N. Now total pressure acting on brake pedal.
P=Force/area of surface on which force is acting
Hence, P=400/ 3249.1 =0.12309 Mpa
Also, the pedal is pivoted on end, and the master cylinder push rod is attached to the center of the pedal. . Neglecting the reactive forces due to push rod, as the reactive forces are negligible when considered the design of the pedal.

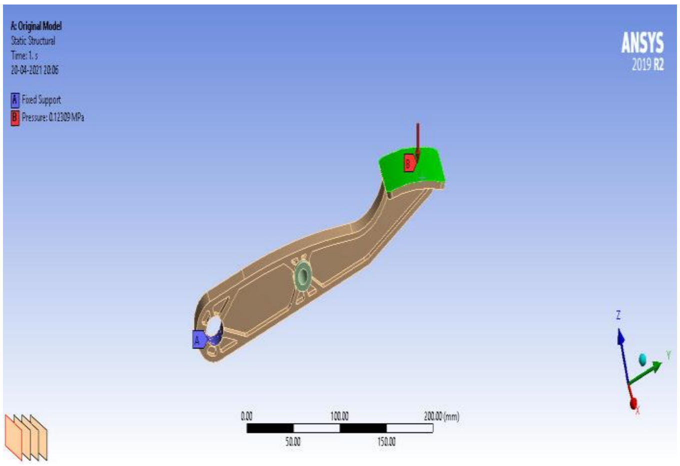


Fig-9: Meshed Model of Brake Pedal

6.2 FEA results for static loading

The pedal is analyzed for equivalent Von Misses Stress and the total deformation. The graphical results are as shown below

6.2.1 Equivalent Von Mises Stress

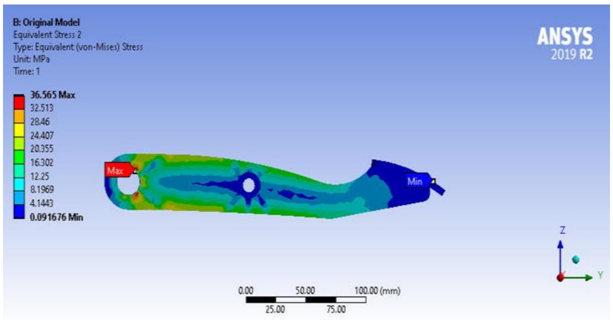


Fig-10: Equivalent Von Mises Stress result

6.2.2 Total Deformation

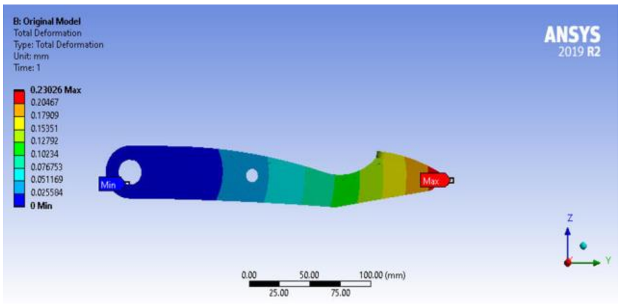
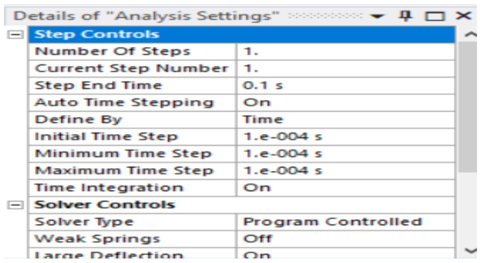


Fig-11: Deformation results

6.3 Transient structural analysis

Transient structural analysis is also referenced as flexible dynamic analysis. It can be used to find out the dynamic response of a structure under the action of any general time-dependent loads.

6.3.1 Details of analysis settings:



Details of "Analysis Settings"

Step Controls	
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
Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Large Deflection	On

Fig-12: Details of analysis

Boundary conditions are same as static structural analysis.

6.4 FEA results for transient structural

The pedal is analyzed for equivalent Von Misses Stress and the total deformation.

6.4.1 Equivalent Von Mises Stress

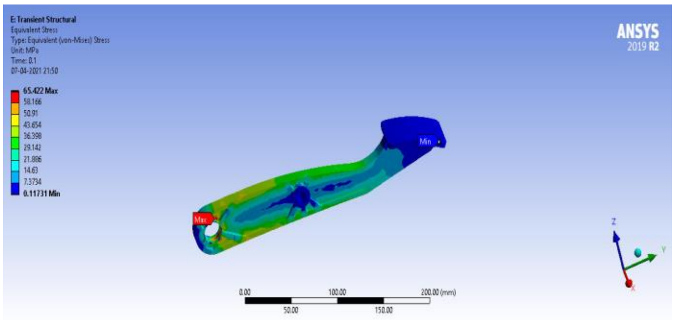


Fig-13: Equivalent Von Mises Stress result

6.4.2 Total Deformation

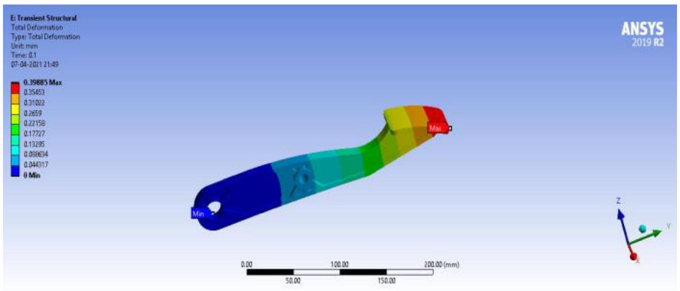


Fig-14: Deformation results

6.5 Results of original brake pedal

Table -1: Results of original brake pedal

Sr. No	Result	Value Static	Value Dynamic	Units
1	Von Misses Stress	36.57	65.422	MPa
2	Deformation	0.23	0.39885	mm

The Pedal is made of Structural Steel whose maximum tensile strength is 250MPa. The induced stress is way less than that of the maximum strength. It simply indicates that a lot of the material is excess and is not needed for the safety of the pedal. Hence, a topology optimization can be performed in order to achieve a better design for the pedal which utilizes less material and hence lower the weight of the component.

6.6 Topology Optimization

For the purpose of Topology Optimization, ANSYS 19.0 TO module is used. Its user friendly interface to run topology optimization on application of constraints is remarkable and gives easy solution to a complicated problem in very short time if the user is familiar with the procedure.

6.6.1 Allocation of design spaces

ANSYS 19.0 Topology Optimization only considers material removal from design space. Hence, it is essential to allocate design and non-design region on the component. The non-design region are mainly the region from where the material cannot be removed like the supporting portion, pivot regions other highly stressed regions. The remaining portion of the part can be considered for material removal freely keeping in mind the basic strength of the component is not lost. The pedal is now assigned its design region as shown in the figure:

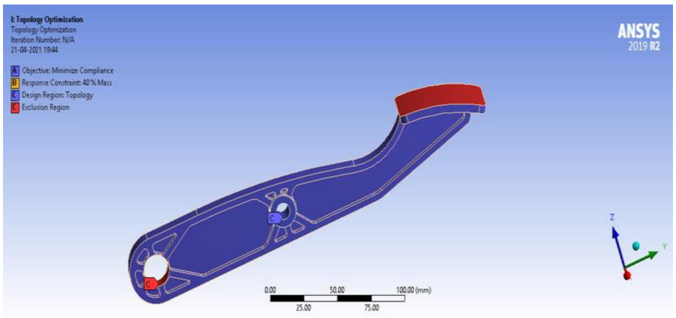


Fig-15: Allocation of design spaces

The blue region is the design region and the red region is the non-design region as they are the pivot and pad, which cannot and need not to be optimized.

6.6.2 Topology Procedure

After the allocation of regions, the topology tool is to be run in order to obtain free form of the optimized model. The free form of the optimized model is a shown in the figure(40% weight reduction):

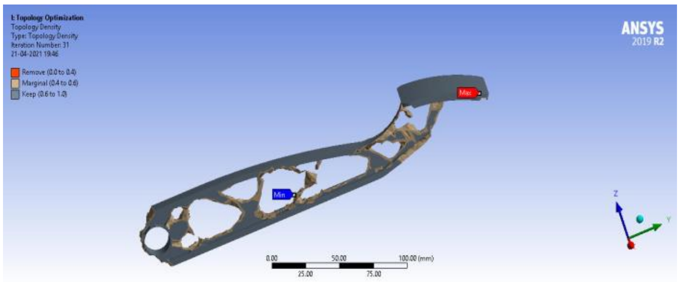


Fig-16: Topology Optimized free from model

This free form model is almost impossible to manufacture by any other method very accurately except for additive manufacturing technique. This free form model will now again be redesigned considering the material removal region and amount.

6.7 Redesign and FEA analysis of optimized model.

The model thus obtained from the Topology Optimization tool will now again be remodeled using SolidWorks 20. Three different Optimized models are generated using Solidworks 3d Modelling software.

Optimized Model 1:

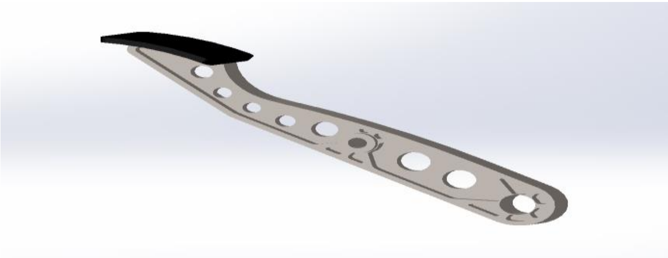


Fig-17: Optimized Model 1

Optimized Model 2:

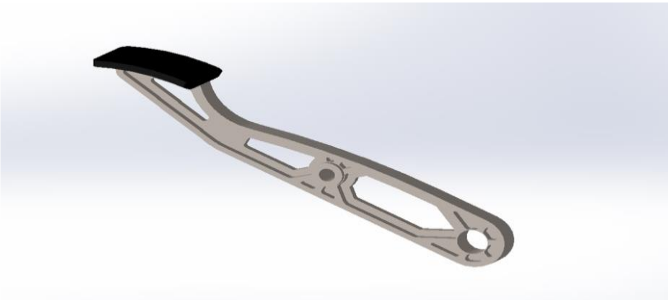


Fig-18: Optimized Model 2

Optimized Model 3:

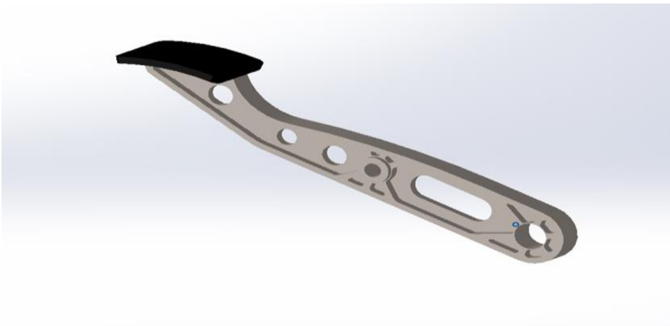


Fig-19: Optimized Model 3

6.7.1 FEA of optimized model

All the three optimized models are tested under same working conditions as that of the original brake pedal condition. The optimized model will now be subjected to pressure 0.12309 Mpa at free end and Fixed at hinged end. All the three models are tested under static and dynamic loading and stress and deformation results are calculated and compared with original result.

Simulation Results:

Optimized Model 1:

Static Results

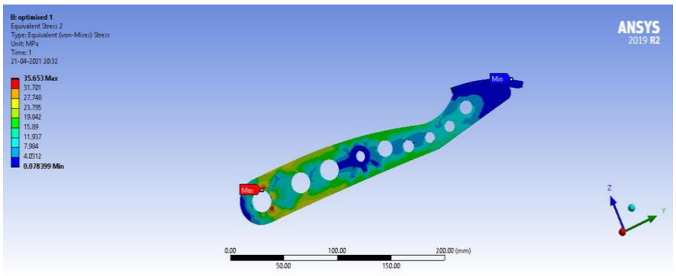


Fig-20: Equivalent Von Mises Stress result

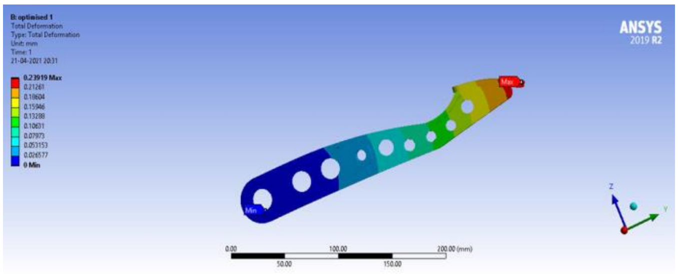


Fig-21: Deformation results

Transient structural :

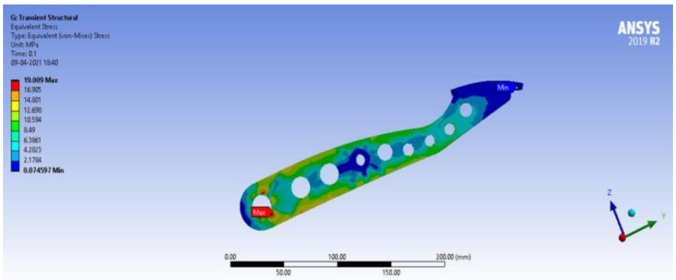


Fig-22: Equivalent Von Mises Stress result

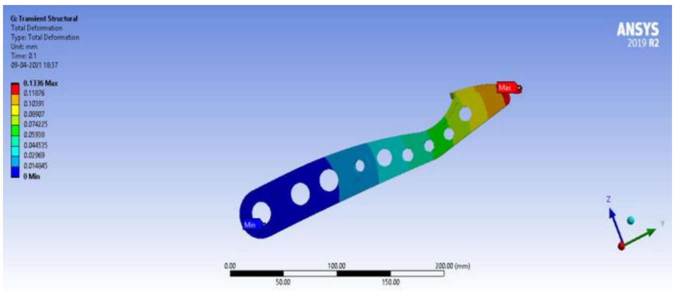


Fig-23: Deformation results

Optimized Model 2:

Static Results

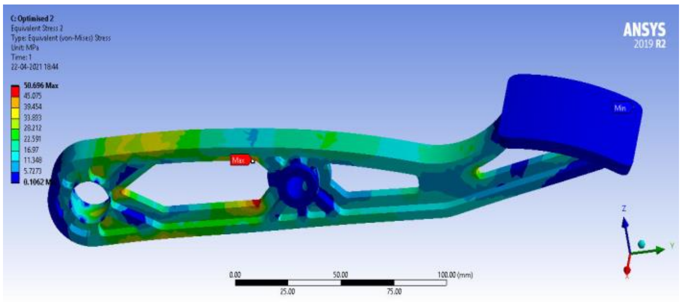


Fig-24: Equivalent Von Mises Stress result

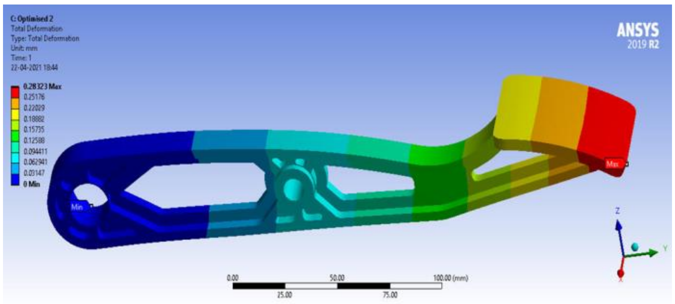


Fig-25: Deformation results

Transient structural:

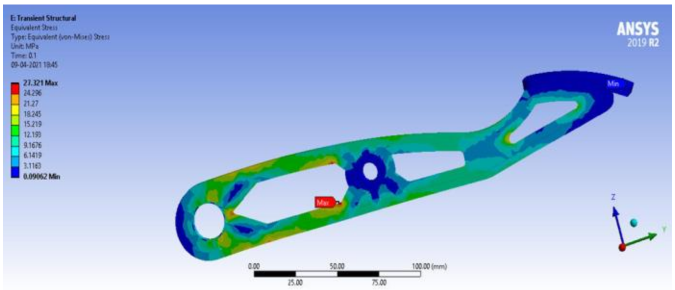


Fig-26: Equivalent Von Mises Stress result

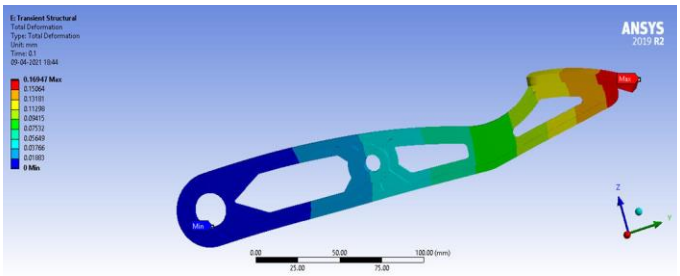


Fig-27: Deformation results

Optimized Model 3:

Static Results:

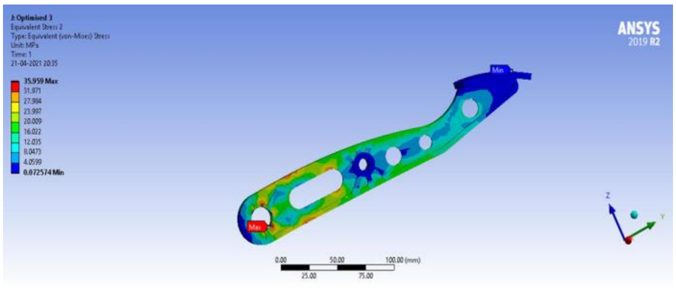


Fig-28: Equivalent Von Mises Stress result

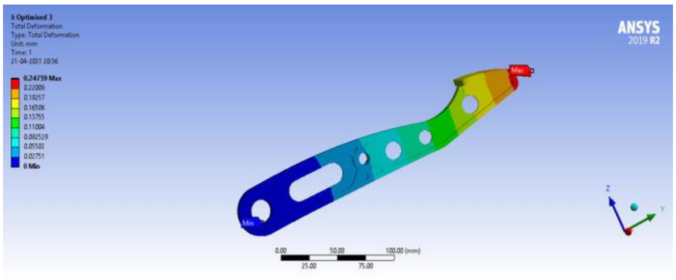


Fig-29: Deformation results

Transient Results:

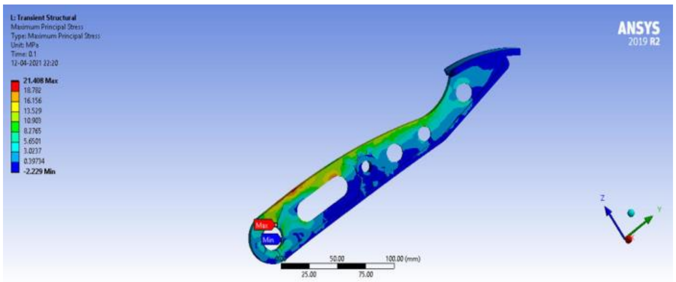


Fig-30: Equivalent Von Mises Stress result

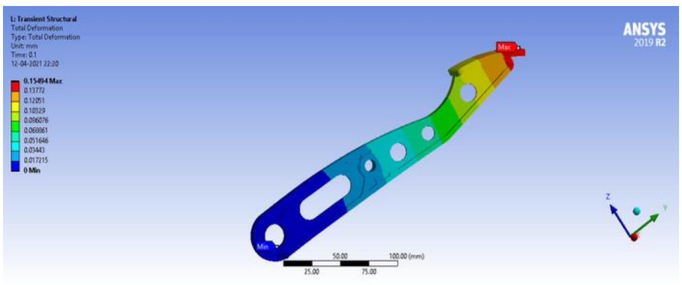


Fig-31: Deformation results

7. Results and Conclusions

7.1 Results

7.1.1 Results Comparison

Table-2 : Von Misses stresses (Mpa)

Sr. No.	Model	Stress (static)	Stress (Transient)
1	Original	36.57	65.422
2	Optimized 1	35.65	19.009
3	Optimized 2	50.69	27.321
4	Optimized 3	35.96	21.408

Table 3 : Total deformation (mm)

Sr. No.	Model	Total Deformation (static)	Total Deformation (Transient)
1	Original	0.23	0.39885
2	Optimized 1	0.23	0.1336
3	Optimized 2	0.25	0.16947
4	Optimized 3	0.248	0.15494

Table-4: Masses of pedals (kg)

Sr. No.	Model	Mass (kg)	Weight saved (gm)
1	Original	0.697	Reference
2	Optimized 1	0.656	41
3	Optimized 2	0.612	85
4	Optimized 3	0.646	51

7.2 Conclusions

- Based on the analysis results, it is evident that the new optimized designs will be help save material usage and also contribute to the weight reduction of the vehicle.
- From the simulation results it found that, optimized model 2, will give material saving of 85grams compared to all other optimized models. And also the stress induced is less than the yield stress of the material which 250MPa.
- With adaption of new design, lot of a manufacturing unit's capital will be saved.

REFERENCES

[1] Miss. Ashwini N. Gawande, Prof. G. E. Kondhalkar, "Design Analysis and Optimization of Automotive Brake Pedal", "International Conference On Ideas, Impact And Innovation In Mechanical Engineering", ISSN: 2321-8169 Volume: 5 Issue: 6

[2] Mohd Sapuan Salit, Mohd Syed Ali Molla and Md Liakot Ali, "Conceptual Design of an Automotive Composite Brake Pedal", Suranaree J. Sci. Technol, Vol. 13 No. 3; July-September 2005.

[3] K K Dhande, N I Jamadar and Sandeep Ghatge, "Design and Analysis of Composite Brake Pedal: An Ergonomic Approach", Int. J. Mech. Eng. & Rob. Res. 2014

[4] Miss. Ashwini N.Gawande, Prof. G. E. Kondhalkar, Prof. Ashishr.Pawar, "Static Structural Analysis and Optimization of Brake Pedal", "International Research Journal Of Engineering And Technology", Volume: 04

[5] Sourav Das, "Design and Weight Optimization Of Aluminum Alloy Wheel", "International Journal Of Scientific And Research Publications", Volume 4, Issue 6, June 2014 1 ISSN 2250-3153

[6] Mr. P. H. Yadav, Dr. P. G. Ramdasi, "Optimization Of Car Rim Using Optistruct", "Journal Of Environmental Science, Toxicology And Food Technology", ISSN: 2319-2402, Volume 2, Issue 3 (Nov. - Dec. 2012), PP 10-15

[7] S. Chaitanya, B.V.Ramana Murty, "Mass Optimization Of Automobile Wheel Rim", "International journal Of Engineering Trends And Technology", Volume 26 Number 3- August 2015.

[8] Denghong Xiao, Hai Zhang, Xiandong Liu, Tian He And Yingchun Shan, "Novel Steel Wheel Design Based On Multi-Objective Topology Optimization", "Journal Of Mechanical Science And Technology", 28 (3) (2014) 1007~1016

[9] Ch. P. V. Ravi Kumar1, Prof. R. Satya Meher, "Topology Optimization of Aluminum Alloy Wheel", "International Journal of Modern Engineering Research".

[10] D. Santhosh Kumar, Jayakumar V, Shajin Majeed, "Modal Analysis and Design Optimization of Automotive Wheel Rim", "Journal of Chemical and Pharmaceutical Sciences".