Design and Analysis of Brake Pedal.

Nilesha.U.Patil[#1], Akshay Sawant[#2], Harsh Patel[#3], . Vishakha Salve[#4] , Vikas Khadk[#5]

Department of Mechanical Engineering SKNCOE,SPPU Pune India

[1]nupatil2@gmail.com [2]futureworld122@gmail.com
[3]harsh.patel1848@gmail.com
[4]vikaskhadke7@gmail.com
[5]Salvekvishakhap5415@gmail.com

Abstract

Brake pedals are widely used in all automotives, which acts as a linkage between occupant and brake mechanism. Existing design seems to be overdesigned as per requirement FEA will be used to apply cantilever load optistruct solver will be used to perform material removing process. The model of an existing brake pedal was generated using CATIA V5 solid modeling. Existing design will be tested using strain gauge technique at tensile loading using UTM machine. Comparative analysis will be done with FEA and Experimental units. Conclusion and future scope will be suggested. The Brake pedal is again a very crucial component of an ATV as it would control the overall braking of the vehicle. Proper braking is very crucial in rough terrains with all four wheels locking so that the stopping distance is minimum. Hence a brake pedal needs to be designed in such a way it provides the necessary leverage and creates enough pressure for the brake fluid to reach all four wheels. A brake is a mechanical component that constrains the motion by absorbing the energy from the motion of the vehicle. When the brake pedal of the hydraulic braking system is pushed against the master cylinder, a piston pushes the brake pad against the brake disc which reduces the speed of the vehicle or stops the vehicle completely. Hence the brake pedal should be able to push the piston in the master cylinder completely with a full-length stroke and thereby provide equal pressure to all wheels. Currently, automotive industry continues to strive for light weight vehicle for improving fuel efficiency and emission reduction. It is crucial to design vehicles with an optimum weight for a better performance of the car. In this field, which has boundaries with fixed standards and regulations, manufacturers continue to work on characteristics, structures and types of the material to develop new components, which have same safety specifications but cheaper and lighter than current products.

Keywords- *Optimization, Stress, FEA,Break pedal,Ansys,Universal testing machine.*

1. INTRODUCTION

In recent year, the material competes with each other for existing and new market. Brake pedals used by driver of a vehicle to operate the brakes. The brake system in car is a sealed hydraulic system and relies on close tolerances between the brake shoes and drums or brake pads and rotors. It is one of the most significant systems of a vehicle. It has some basic roles, it should slow a moving vehicle, It needs to hold a vehicle stationary when stopped, It should bring a vehicle to a stop.

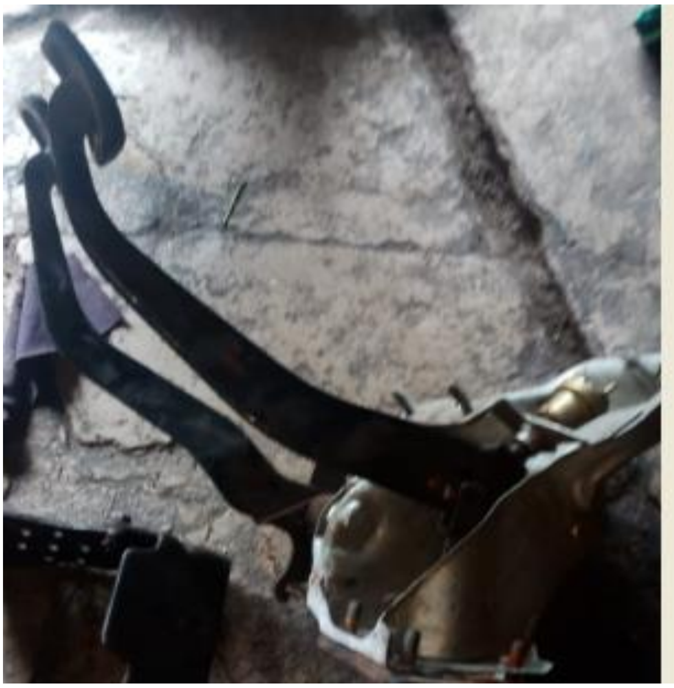


Fig 1 Brake Pedal

Brake pedal of TATA STORME is used as component for study.CAD Model of brake pedal is developed in 3D modeling software CATIA V5.In design of brake pedal, a weight should be minimized. In automobile industry, mass of weight reduction is becoming important issue. To reduce the material, material removal process is used. Which is reducing the weight and cost of product. In automobile industry, it is obligatory to look for cheap & lightweight materials and which should be easily accessible. At present, brake pedals are made from metal but accelerator pedals and composite clutches are successfully utilized in automotive vehicles. This study is concentrated on variable-material for the conceptual design brake pedal profile.The aims and scope of the research paper is to reduce the weight of an existing brake pedal design of a car with the application of topology optimization without the substitution of material. By reviewing the design constraints, load and boundary condition the topology optimization was run and the analysis. However the result of optimization process needs for further refinement as it has manufacturability deficiency.

ISSN: 2233-7857 IJFGCN

2. LITERATURE REVIEW

proposes that in automotive industries, metalic 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.

has illustrated that the automotive industries accelerator and clutch pedal are replacing by light weight materials such as plastic, polymer composites, aluminium 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 softwareproposes that in automotive vehicles, the conventional brake, accelerator and clutch pedals are replaced by polymeric-based composite pedals. The purpose of replacement from metallic pedal to polymeric-based composite material is to improve material degradation by corrosion and reduce the weight, cost. In this research work, as per the design parameters, the four different sections of polymeric based brake pedals are analyzed .The sections are analyzed and arrived at a winning concept based on stiffness comparison. From the winning concept, a full scale model is developed while developing full scale model an ergonomic study has been made on few hatch back and SUVs car's to improve the driver's comfort and due to breaking operation, the fatigue is reduce. The pedal is modeled and analysis using CATIA software & ANSYS software. The results have shown polymeric-based composite material replaced with present metallic pedal. By using composite material, the weight reduction of 66.7% is achieved.

Mohd Nizam Sudin has illustrated that the modern automotive industry is continuing to strive for light weight vehicle in improving fuel efficiency and emissions reduction. To produce a good performance car to design vehicles with optimum weight is important. In order to reduce the weight of vehicle without sacrificing its integrity, this purpose of this project is to employ topology optimization technique to propose an optimal design of component in early phase of product development. The model of an existing brake pedal was generated using CATIA V5 software and Topology optimization by using Altair Optistructsoftware. Finally, a new light weight design of brake pedal is proposed. The result shows that the weight of a new designed brake pedal was 22% less as compared to an existing brake pedal without sacrificing its performance requirement.proposes that now a day's industries are replacing accelerator and clutch pedal by lightweight materials such as polymer plastic, composites, aluminium and its alloys, etc. The purpose is to reduce weight, cost, and improvement in corrosion resistance without change in material reduction. 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. illustrated that the vehicle components of materials are dependent on a supply and demand process, subject to requirements. Metals i.e. steel, aluminium and magnesium are used for elements of the body structure and panels and Plastics are used for exterior attachments to the body. Cars consist of steel and iron but due to the impending use of multi material constructions, it is expected that the amount of steel and iron used is reduced. the steel unibodies are multi material unibodies and aluminium space frames. Magnesium and steel space frame concepts for volume applications are still under development. The materials are replaced by durability and specific strength/ stiffness of high performance carbon fiber composites.proposed that to determine design concept, Concurrent Engineering (CE) approach is used and material of the composite accelerator pedal at conceptual design stage. using the Morphological approach, the Various design concepts are generated. at design stage, CATIA is used and ANSYS is used for analysis. on the basis of past research & specifications, material selection is done .the pedal arm profile on the basis of stress, mass & volume.

ISSN: 2233-7857 IJFGCN

Mohd Sapuan Salit, Sandeep Ghatge, K K Dhande, Mohd Nizam Sudin,

Dr Hossein Saidpour Describes that

3. FINITE ELEMENT ANALYSIS

The first step is to start the analysis with the ANSYS programs. For carrying out detail analysis of brake pedal, the static Structural analysis was selected. The geometry of brake pedal was created in ANSYS by taking all the parameters of brake pedal. After creating geometry, material properties were applied to brake pedal.

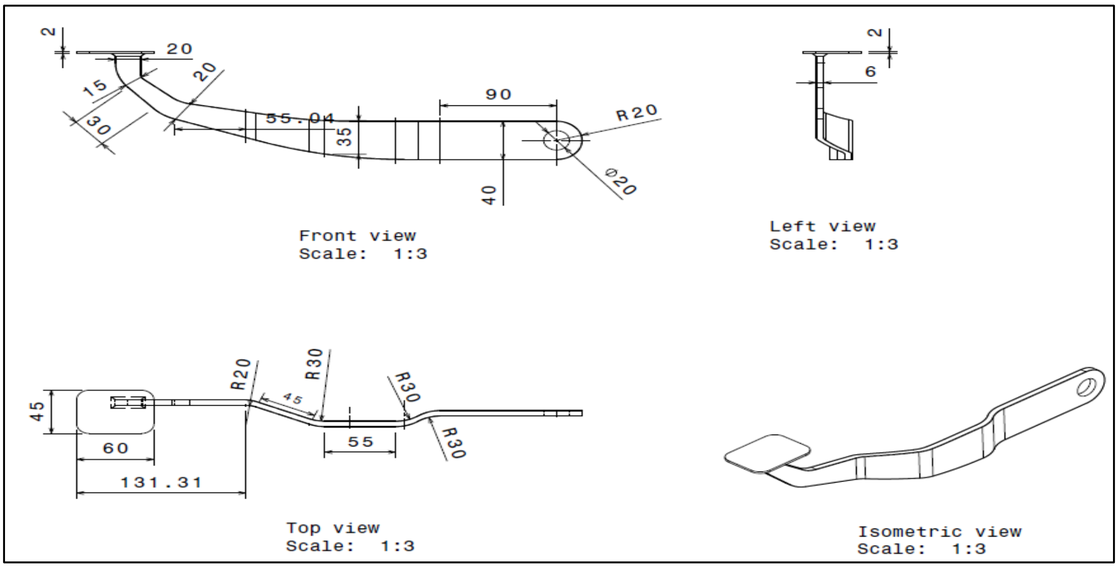


Fig 2 Drafting of Brake Pedal.

3.1 Computer Aided Design (CAD):

Around the mid 1970s, as CAD systems began to provide more capability than a manual drafting with electronic drafting, the cost benefit for companies to switch to CAD became apparent. The benefit of CAD systems over manual drafting are the capabilities one often takes for granted from computer systems today; automated generation of Bill of Material, auto layout in integrated circuits, interference checking, and many others. CAD provided the designer with the ability to perform engineering calculations.

ISSN: 2233-7857 IJFGCN

During this transition, calculations were still performed either by hand or by those individuals who could run computer programs. CAD was a revolutionary change in the engineering industry, where draftsmen, designers and engineering roles begin to merge. It did not eliminate departments, as much as it merged departments and empowered draftsman, designers and engineers. CAD is just another example of the pervasive effect computers were beginning to have on industry. Current computer-aided design software packages range from 2D vector-based drafting systems to 3D solid and surface modelers. Modern CAD packages can also frequently allow rotations in three dimensions, allowing viewing of a designed object from any desired angle, even from the inside looking out. CAD technology is used in the design of tools and machinery and in the drafting.

CAD is mainly used for detailed engineering of 3D models and/or 2D drawings of physical components, but it is also used throughout the engineering process from conceptual design and layout of products, through strength and dynamic analysis of assemblies to definition of manufacturing methods of components. It can also be used to design objects. Furthermore, many CAD applications now offer advanced rendering and animation capabilities so engineers can better visualize their product designs.

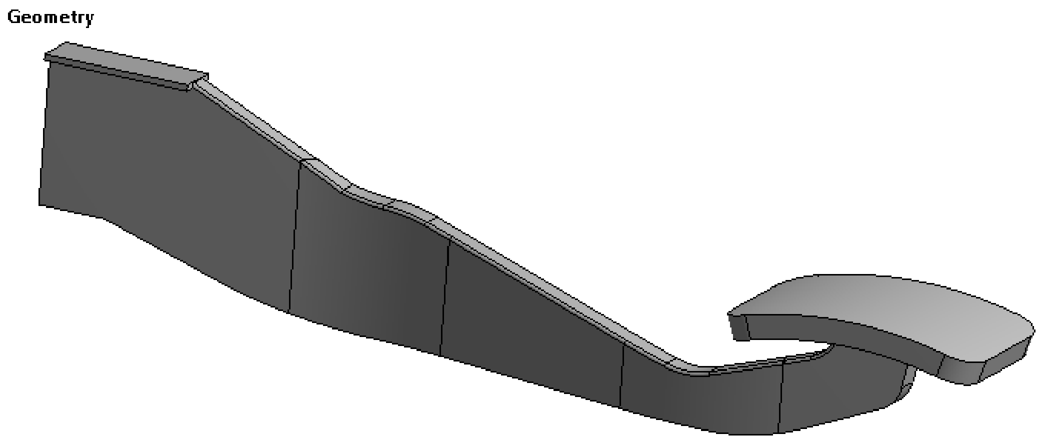


Fig 3 CAD Model of Existing Brake Pedal

ISSN: 2233-7857 IJFGCN

3.2 Material properties

Properties of Outline Row 3: Structural Steel

	A	B	C
1	Property	Value	Unit
2	Material Field Variables	Table	
3	Density	7850	kg m^-3
4	Isotropic Secant Coefficient of Thermal Expansion		
5	Coefficient of Thermal Expansion	1.2E-05	C^-1
6	Isotropic Elasticity		
7	Derive from	Young's Modu...	
8	Young's Modulus	2E+11	Pa
9	Poisson's Ratio	0.3	
10	Bulk Modulus	1.6667E+11	Pa
11	Shear Modulus	7.6923E+10	Pa

Material properties

3.3 CATIA V5R20:

CATIA (computer aided three-dimensional interactive application) is a multi – platform computer - aided design (CAD) / computer - aided manufacturing (CAM) / computer - aided engineering (CAE) software suite developed by the French company Dassault Systems.

CATIA delivers the unique ability not only to model any product, but to do so in the context of its real-life behavior: design in the age of experience. Systems architects, engineers, designers and all contributors can define, imagine and shape the connected world.

a) CAE in the automotive industry:

CAE tools are very widely used in the automotive industry. In fact, their use has enabled the automakers to reduce product development cost and time while improving the safety, comfort, and durability of the vehicles they produce. The predictive capability of CAE tools has progressed to the point where much of the design verification is now done using computer simulations rather than physical prototype testing. CAE dependability is based upon all proper assumptions as inputs and must identify critical inputs. Even though there have been many advances in CAE, and it is widely used in the engineering field, physical testing is still used as a final confirmation for subsystems because CAE cannot predict all variables in complex assemblies (i.e. metal stretch, thinning).

ISSN: 2233-7857 IJFGCN

b) Finite Element Method (FEM):

It is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It is also referred to as finite element analysis (FEA). FEM subdivides a large problem into smaller, simpler, parts, called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. FEM then uses variational methods from the calculus of variations to approximate a solution by minimizing an associated error function.

c) Ansys Meshing:

Over the last 20 years, Meshing has evolved into the leading premier pre-processor for concept and high fidelity modeling. The advanced geometry and meshing capabilities provide an environment for rapid model generation. The ability to generate high quality mesh quickly is one of Ansys Workbench core competencies. Industry trends show a migration to modular sub-system design and continued exploration of new materials; Ansys Workbench has advanced model assembly tools capable of supporting complex sub-system generation and assembly, in addition, modeling of laminate composites is supported by advanced creation, editing and visualization tools. Design change is made possible via mesh morphing and geometry dimensioning. Ansys Workbench is a solver neutral environment that also has an extensive API which allows for advanced levels customization.

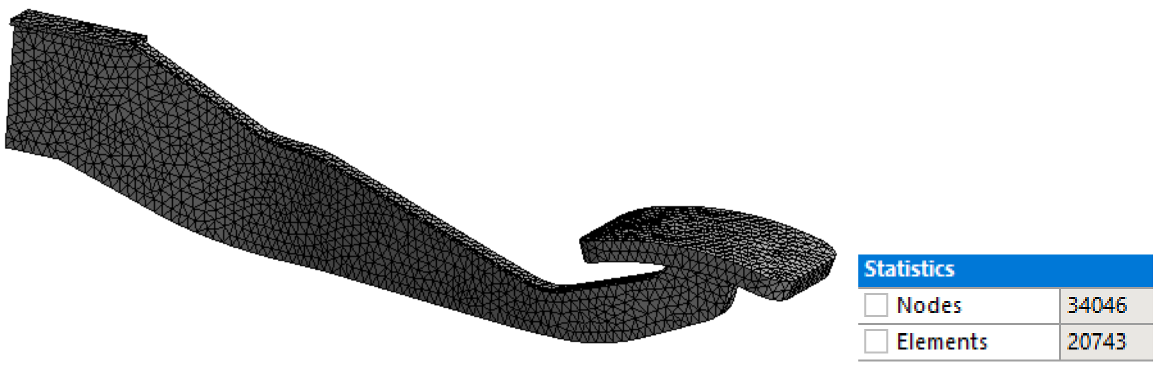


Fig 4 Meshing of Brake pedal

3.4 Meshing

The next step in ANSYS workbench is to generate a meshing, after applying some material properties & creating geometry.

- Element Type: Second order Hexahedron
- Elements count: 20743
- Nodes count:34046

3.5 Loading and Boundary Condition

ISSN: 2233-7857 IJFGCN

The boundary conditions such as loads & constraints are imposed, after meshing the model, It is important to apply correct loads & boundary conditions, To get accurate results.

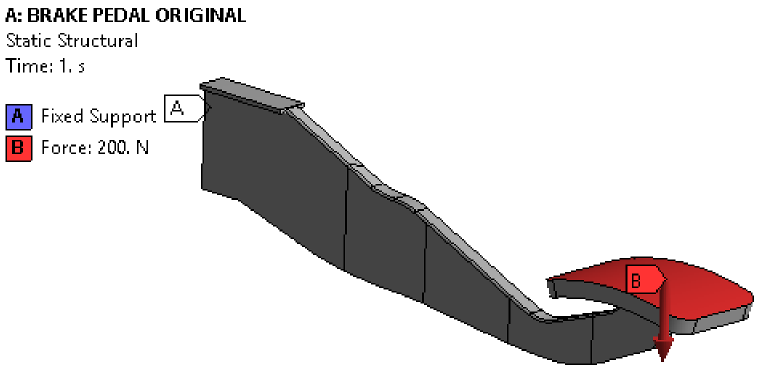


Fig 5Boundary Condition

Existing model of Brake Pedal is fixed at one end at point A.

The Force of 200 N is applied on the other end B.

3.7 FEA Pre Processing

The pre-processing of the connecting rod is down for the purpose of the dividing the problem into nodes and elements, developing equation for an element, applying boundary conditions, initial conditions and for applying loads. The information required for the pre-processing stage of the connecting rod is as follows

- **Material properties**: The values of young’s modulus, poisons ratio, density, yield strength for steel are taken from material library of the FEA PACKAGE.

  Material- Structural Steel

  Young’s Modulus- 200 GPa

  Poisons Ratio- 0.3

  Density- 7850 kg/m^3

  Yield Strength- 520 MPa

ISSN: 2233-7857 IJFGCN

• **Constraints**: The nodes around the connecting rod holes have a rigid element connecting them to the centre of the hole which has of its degree of freedom fixed. The element which is used to fix connecting rod and crank shaft is fixed and used as a rigid element.

The minimum and maximum are set, together with other mesh parameters such as element type and material. The selected object is ready for further analysis.

3.8 Post Processing

The acceptability of the design of the connecting rod needs to be considered from the results of the analysis. The guidance for the modification of the rod need to be available if the design is not considered to be acceptable for the connecting rod are as follows.

Model acceptance criteria: the maximum von-Mises stress must be less than the material yield strength for the duration of the component. The deflection is considered and the maximum Von-Mises stress must be less than the yield strength for abuse load case.

4. TOPOLOGY OPTIMIZATION

4.1 Topology Optimization:

Topology optimization is a mathematical approach that optimizes material layout within a given design space, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets.

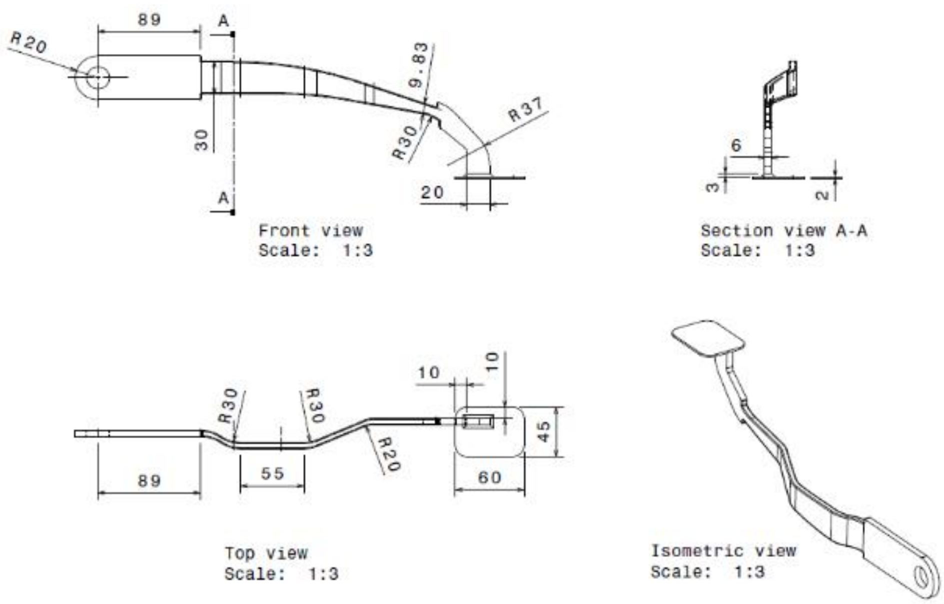


Fig 6 Drafting of Optimized Brake Pedal

ISSN: 2233-7857 IJFGCN

.

Basic Theory:

There are three kinds of structure optimization,

- Size Optimization
- Shape Optimization
- Topology Optimization

Three optimization methods correspond to the three stages of the product design process, namely the detailed design, basic design and conceptual design. Size optimization keeps the structural shape and topology structure invariant, to optimize the various parameters of structure, such as thickness, section size of beam, material's properties; shape optimization maintains the topology structure, to change the boundary of structure and shape, seek the most suitable structure boundary situation and shape; topology optimization is to find the optimal path of material's distribution in a continuous domain which meet the displacement and stress conditions in structure, make a certain performance optimal. Thus, compared to size and shape optimization, topology optimization with more freedom degree and greater design space, its greatest feature is under uncertain structural shape, according to the known boundary condition and a given load to determine the reasonable structure, both for the conceptual design of new products and improvement design for existing products, it is the most promising aspect of structural optimization.

For continuous structure topology optimization, there are some mature methods like: uniform method, evolutionary structural optimization method, variable density method etc. Uniform method introduced cell structure of micro structure (unit cell) in the elements of the structure, each unit cell has three forms, namely non-material voids (size = 1), isotropic-material entity medium (size = 0) and orthotropic-material opening-hole medium (0 < size < 1). Wherein the distribution of each form will be able to describe the form of topology and the shape of structure; evolutionary structural optimization method believe that stress in any parts of the structure should under the same level in an ideal structure. That means the local material with a low stress state is not fully utilized, so you can delete the material artificially. So gradually remove material which in a low stress state, and then delete the update rate, so optimized structure becomes more uniform. Variable density method is used to conduct optimization in this paper. The basic idea is to introduce a hypothetical material which density is variable and range from 0 to 1.

ISSN: 2233-7857 IJFGCN

4.2Process of Topology Optimization:

Based on Hyper Works platform topology optimization holder, first, according to the engine mounting position, we establish the three-dimensional geometric model of engine bracket, and then pretreated in HyperMesh, define design area, objective function and constraints under the optimization panel, finally operate topology optimization which design process is as Figure .

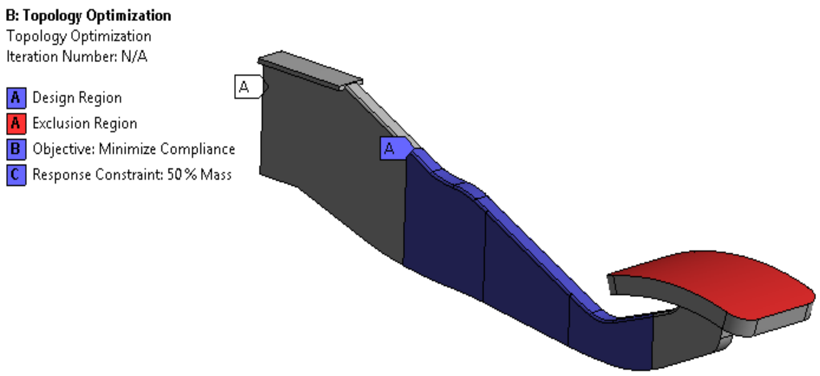


Fig 7.Topology optimization model

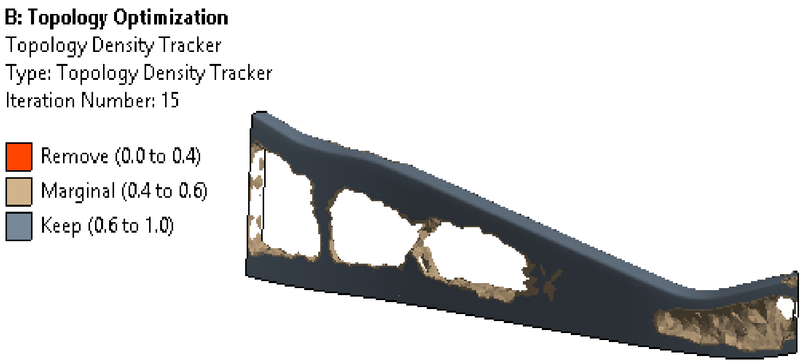


Fig 8. Topology Density Tracker

ISSN: 2233-7857 IJFGCN

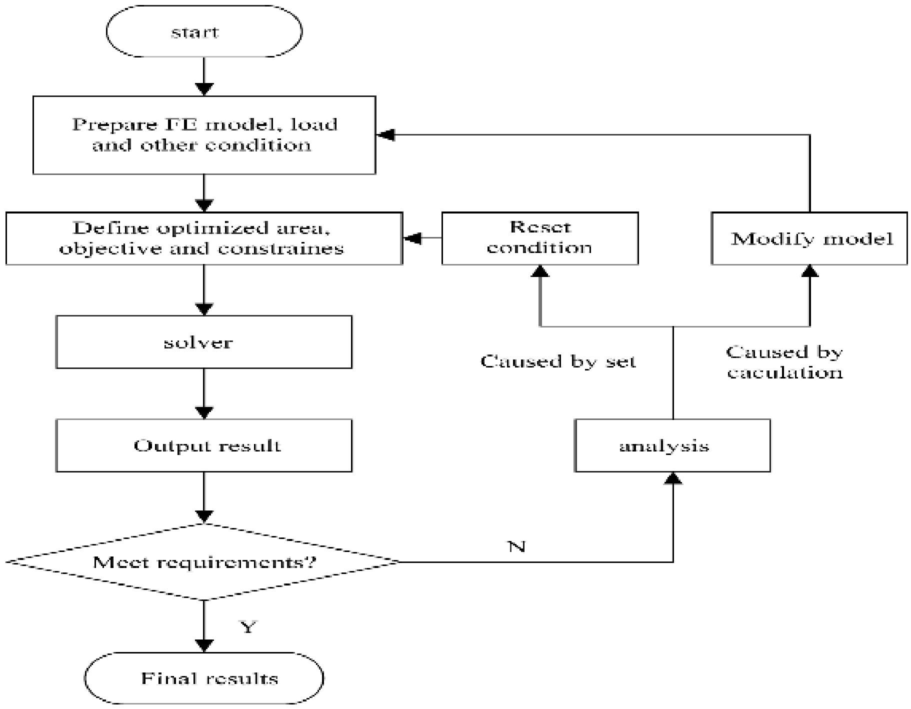


Fig 8.Process of Topology Optimization

Steps in Topology Optimization:

The topology optimization consists of the following sequence of steps.

- Define the design space
- Define optimization parameters
- Material removal process and detail design

4.3Defining the Optimization Parameters:

The aim of topology optimization in this project is to minimize the volume without affecting the bracket stiffness and strength compared to base bracket, so the design objective is taken as to minimize the volume. Following parameters are defined as constrains:

1. Allowable stress limit value is defined as stress constraints from durability point of view
2. Single draw direction is defined as manufacturing constraint.

Material Removal Process and Detail Design:

The optimization process took some iteration to remove the unnecessary material from the design space. The output of the topology optimization, an intermediate model which may be called a topology

ISSN: 2233-7857 IJFGCN

based model, is constructed by removing unnecessary materials from the rough conceptual model.

- Additive Manufacturing

Without manufacturing constraints, optimized components often have complex geometries. Recent developments in Additive Manufacturing (3D printing) enable the fabrication of these optimized designs without compromise. Topology optimization forms the natural design technology for Additive Manufacturing, as it fully exploits its potential.

- Shape Optimization

Where topology optimization excels at automated concept generation, shape optimization allows for efficient final fine tuning of designs. However, every fabrication technique has its shape/property accuracy limitations, particularly at the micro scale. Our activities on shape optimization concentrate on dealing with fabrication inaccuracy, and optimizing for robustness.

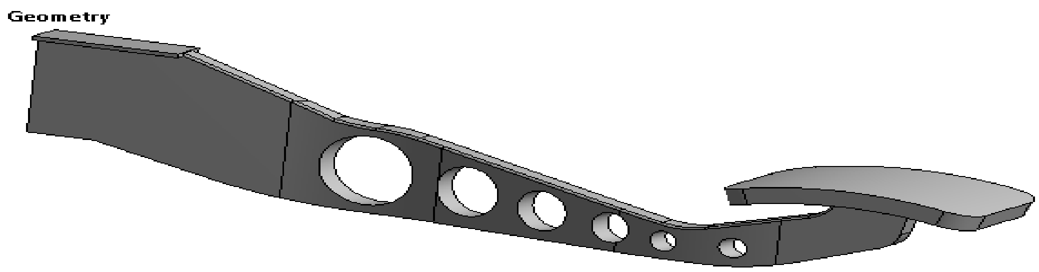
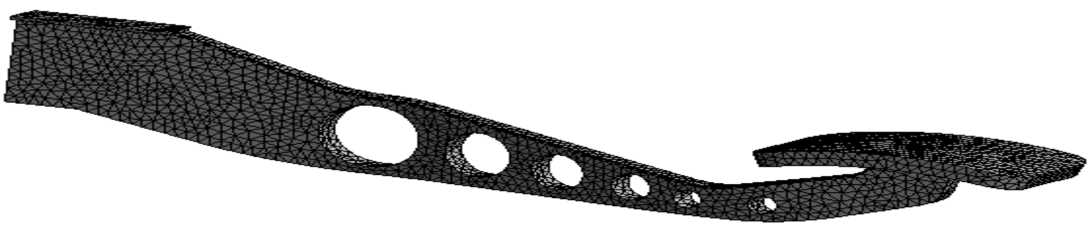


Fig 8.CAD Geometry of Optimized Model.

Meshing



Statistics	
Nodes	31129
Elements	18468

Fig 9.Meshing of Optimized Brake pedal

ISSN: 2233-7857 IJFGCN

Analysis:

To establish the geometry model by CATIA, then input the geometry to the ANSYS to carry out pre-treatment operations like geometry cleanup, meshing, loads, constraints, etc.Initially we need to collect the information regarding different loads acting on the bracket and the packaging data for fixing design space. The base bracket results from testing and finite element analysis (FEA) point of view for evaluating final optimized design.

Boundary Conditions:

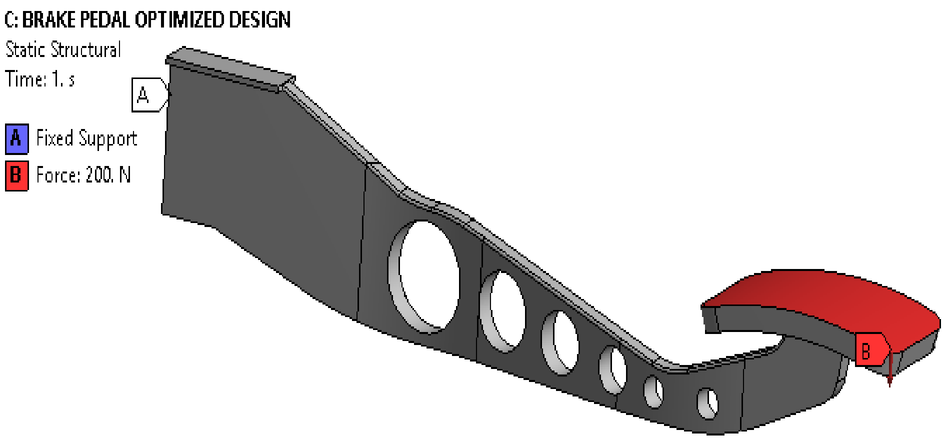


Fig 17Boundary Conditions of Optimized Model.

Optimized model of Brake Pedal is fixed at one end at point A. The Force of 200 N is applied on the other end B.

5. CONCLUSION

From results of finite element analysis it is observed that the maximum stress value is within the safety limit. There is a great potential to reduce material, this safety limit which can be done by removing material from low stressed region thus reducing its weight without affecting its structural behavior. The maximum displacement value is also very less. So, the material from low stressed region is can be removed without affecting its strength and is within the yield strength.

6.ACKNOWLEDGEMENT

All the authors wish to acknowledge our guide and SKNCOE's Mechanical Department and other Contributors for developing and helping us to complete the study.

ISSN: 2233-7857 IJFGCN

REFERENCES

[1] Mohd Nizam Sudin, MusthafahMohd Tahir, FaizRedza Ramli, Shamsul Anuar Shamsuddin, "Topology Optimization in Automotive Brake Pedal Redesign", International Journal of Engineering and Technology (IJET), ISSN : 0975-4024, Volume 6, No 1 Feb-Mar 2014.

[2] MohdSapuanSalit, Mohd Syed Ali Molla and Md Liakot Ali, "CONCEPTUAL DESIGN OF AN AUTOMOTIVE COMPOSITE BRAKE PEDAL", Suranaree J. Sci. Technol. 12(3):173-177, Mar 28, 2004; Revised: Aug 1, 2005; Accepted: Aug 3, 2005.

[3] KKDhande, N I Jamadar and Sandeep Ghatge, "DESIGN AND ANALYSIS OF COMPOSITE BRAKE PEDAL: AN ERGONOMIC APPROACH", International Journal of Mechanical Engineering and Robotic Research, Volume 3, No. 3, July 2014, ISSN 2278 – 0149.

[4] Dr.K.K.Dhande, Prof .N.I.Jamadar, Sandeep Ghatge, "Conceptual Design and Analysis of Brake Pedal Profile", International Journal of Innovative Research in Science, Engineering and Technology, Volume 3, Issue 11, November 2014.

ISSN: 2233-7857 IJFGCN