MANTECH Publications

Journal of Computer Aided Manufacturing and Automation

Volume 7, Issue 1, January-June, 2022

A Study on Critical Front-End Analysis and Topology of Brake Pedal

Parul Srivastava[1], Aman Saxena[2]

Student [1], Lecturer[2]

Department of Mechanical Engineering

Kanpur Institute of Technology

Corresponding Author's Email id: *parulsrivastava134@gmail.com[1]*

Abstract

In today's world, the automobile industry is continually working toward the goal of producing lighter vehicles in order to improve fuel economy and reduce emissions. It is essential to design cars with the appropriate weight in order to have the best possible performance. The automotive industry is now utilising topology optimization and alternative materials in an effort to lower overall vehicle body weight. This is being accomplished through the optimization of vehicle designs. However, the revised design needs to satisfy a number of performance requirements, the most important of which are its stiffness and strength. The purpose of this study is to suggest an ideal design for the brake pedal that is utilised in the three-wheeler market by employing several methods of analysis, including theoretical, numerical, experimental, and topological optimization. This study focuses on finding ways to reduce the weight of an existing brake pedal without changing the material that it is made of, therefore the material that is used for the brake pedal has not been altered. Reverse engineering was used in conjunction with computer-aided design (CAD) software to create a digital model of an existing brake pedal. The pedal in question was used in a vehicle. The linear static stress analysis and modal analysis will serve as the foundation for the finite element analysis and topology optimization that will be carried out with the help of the Altair Opti-Struct programme. At long last, an improved brake pedal design that is less in weight but maintains the same level of performance will be put forth in conjunction with the current brake pedal.

***Keywords**: - Altair opti- struct, CAD, topology optimization.*

INTRODUCTION

In today's world, the automobile industry is seeing explosive growth in the direction of environmentally friendly, cost effective, and lightweight vehicles and vehicle components. One of the possible ways for improving fuel consumption is the lowering of the vehicle's overall weight. When the mass of the vehicle is decreased, the inertial forces that the engine must overcome in order to accelerate are also reduced, which results in a reduction in the amount of work or energy that is necessary to move the vehicle. In reality, reducing the weight of a vehicle may be accomplished in three primary ways: the first is via the replacement of lighter-weight materials; the second is through the downsizing of the vehicle; and the third is through the removal of unneeded material from the structural component. In the past, the brake pedal would be built using iterative approaches and then optimised using a topology that was less than ideal under static load conditions. This strategy calls for a number of design iterations, and the ultimate solution is arrived at through a process of trial and error. This iterative process takes a lot of time, which contributes to an increased cost. In addition, the results of employing this methodology do not always constitute the optimal answer to a design problem.

Therefore, it is essential to use a methodical approach in order to arrive at the ideal design solution at the stage of conceptual design, and this may be accomplished through the application of topology optimization. This project shows the theoretical, numerical, and experimental technique, as well as topological optimization, to remodel the brake pedal of the three-wheeler segment vehicle auto rickshaw with the purpose of reducing the vehicle's overall weight. Before a 3D Model can be made using Solid Works, the component will first be measured using a 3D scan system. By utilising topology optimization, a new brake pedal design will be presented, however there will be no new material substitutions made. The existing brake pedal design as well as the newly proposed design will be assessed with the help of the software programme Altair Hyper Works Opti -Struct. An investigation into how the component reacts when subjected to a tremendous amount of foot load will include the execution of a linear static stress analysis. The findings of the research will inform the development of a new design of brake pedal, which will be offered to replace the present brake pedal while simultaneously achieving a considerable weight reduction.

LITRATURE SURVEY

Review of Papers

Sun Wenlong and colleagues have provided an explanation and study of how energy savings might be achieved with lighter automobiles. This article focuses mostly on the use of a variety of lightweight materials and technological approaches in car applications. Additionally, this article presents a novel strategy for using lightweight materials rather than mild steel. This article has demonstrated, with the assistance of a few real-world examples, the potential for utilising materials in addition to steel in order to achieve the goal of reducing the overall vehicle weight. It has been determined in a piece of research that lightweight materials will see widespread application in the automotive sector even if the prices of lightweight materials are the same as those of traditional materials [1].

Elena Cischino and colleagues have come up with several distinct proposals for lightweight packaging in order to satisfy the need for lightweight electric vehicles. The findings that were produced on the body in white (BIW) were the primary focus of this article. These outcomes concerned design strategies, revolutionary technologies for Al and composite materials, and connecting procedures. They detailed the considerable utilisation of high-strength steel, making full use of the possibilities offered by steel technology, as well as the increased use of lightweight metallic materials, such as aluminium, also for Body in White componentsry [2].

Methods of putting the braking system of a load-carrying vehicle through its paces have been described by Vytenis and others. They have demonstrated the connection between the force applied to the brake pedal and the force applied to the brake wheel and have come to the conclusion that the reliance of the rising braking force on the pedal pressing force is linear [3].

Finite element analysis was used by K. K. Dhande and others to do research on the lightweight material. The decrease in weight, as well as the costs and the lack of increase in corrosion resistance, were the goals of the material substitution. In this investigation, many lighter-weight materials were evaluated alongside the more common material of steel for use in brake pedals. Using CATIA and ANSYS software, an analysis was conducted on these materials for a variety of distinct sections, each with a unique loading and boundary condition [4].

In order to bring down the overall weight of the vehicle, Fredrik Henriksson and his colleagues looked into the possibility of substituting one type of material for another. The results of the experiment further highlighted the difficulties associated with incorporating lightweight materials into automotive BIWs through the process of part-by-part substitution [5].

S M Sapuan has studied the conceptual design for the polymeric based composite automotive pedal box system. This study describes the importance of the concurrent engineering technique in the total design activity. The extensive use of morphological chart enabled designers to identify various sub solutions of some function of pedals. Various methods for generating ides have been used in this study and they proved to be useful [6].

Mohd Sapuan Salit et al have investigated the brake pedal in concept stage. In this research work, possible configurations and geometric profiles of brake pedals have been investigated analytically and computationally. A final design of a composite brake pedal has been made from the properties of available and suitable polymeric-based composite [7].

Pankaj Chhabra et al have proposed the concurrent engineering approach to design pedal. In this work, Concurrent Engineering (CE) approach has been used to determine the most optimum decision on design concept and material of the composite accelerator pedal at conceptual design stage. Comprehensive studies were carried out to prepare the design specifications of composite accelerator pedal. Various design concepts are generated using the Morphological approach. The composite accelerator pedal has been optimized and analyzed for safety parameters and finally prototyped using Selective Laser Sintering. The results reveals the feasibility of composite accelerator pedal with glass fille polyamide providing substantial weight saving and better properties than existing metallic pedal [8].

N I Jamadar et al have analyzed the brake pedal with four different sections of polymeric based materials as per the General Motors design parameters. The results have shown polymeric-based composite material meets the requirements of manufacturer's specification and can be replaced with present metallic pedal. Weight reduction of 66.7% was achieved by using composite material [9].

Kalpesh Khetani et al have performed thermal analysis of welding fixture for brake pedal. Brake pedal assembly has been welded on fixture using MIG welding process. The relative arrangement of different elements on base plate of fixture is vital part of design. The welding fixture will serve in reducing production time, maintain consistent quality, maximize efficiency, and reduce operator error and makes possible mass production of similar parts in very less time [10].

S.Mohammadi et al have explained the applied and theoretical approach for free vibration analysis of structural component. They have introduced two vibration measuring systems, accelerometer and RFID. RFID is easy to use and with respect to its price, they conclude that using RFID is more beneficial than using accelerometer [11].

Jacek Grosel has presented the comparison of Classical and Operational Modal Analysis on the basis of engineering structures. The results of vibration's measurements of selected engineering structures with the use of different methods of data processing were presented and discussed in the paper. Dynamic measurements of a structure were performed with the use of a multichannel PULSE system, produced by the Brüel & Kjær [12].

Rajesh Purohit et al have performed finite element analysis of automotive clutch assembly in FE software. The finite element analysis was carried out in three steps: Preprocessing, Solving and Post processing. The plots for Equivalent von-Mises stress, total deformation and stress tool (factor of safety) were calculated and analyzed. The finite element analysis showed that the designed friction clutch assembly is safe [13].

Ashwani Kumar et al have presented the FEA simulation to investigate the dynamic behaviour of the truck transmission gear box. The objective of this research work was to simulate the relation between dynamic vibrations of transmission and fixed constraint of vehicle frame. The FEA simulation results show that the natural frequency of one bolt unconstraint condition varies from (1637.2 – 2674) Hz. The analysis results were verified with experimental result available in literature [14].

WANG Lu et al have studied the traction battery enclosure.The traction battery enclosure is one of the most significant parts of an electric vehicle. Better structural performance and lightweight design of battery enclosure are extremely important in current situation. This paper focused on a multi-objective topology optimization design method for the traction battery enclosure, in which both the static stiffness and dynamic frequencies were taken into consideration. The optimization is utilized to achieve a new battery enclosure structure with better static and dynamic performances. It has been revealed that the structural topology optimization approach can be a feasible and efficient design methodology for the traction battery enclosure structural design and can provide the designer with detailed guidance in conceptual design phase [15].

B. Lia et al have developed a simple and practical procedure for the optimal design of machine tool bed. In this research work, a simplified model was first defined to characterize the bed structure. The load bearing topology of the bed structure was then identified to represent the optimal layout of the inner stiffener plates. Subsequently, detailed sizing optimization was conducted by using a novel criterion which describes the best solution in terms of weight distribution of both the outer supporting panels and inner stiffener plates. Finally, calculation results were elaborated to demonstrate the effectiveness of the proposed method. The proposed approach involved a three-phase procedure. Firstly, a simplified spring model composed of shell and matrix elements was developed to simulate the real bed structure. With this model, the finite element method can be easily and economically employed to identify the load bearing topology of the bed structure under actual operation conditions. After characterizing the layout pattern of the inner stiffener plates, an analytically based weight distribution criteria is presented to determine the thickness of both the outer supporting panels and inner stiffener plates. Optimization results are finally elaborated to demonstrate the effectiveness of the proposed method [16].

Tsuyoshi Nishiwaki et al described the application of topological optimization method to heel counter in the practical designing process of running shoes. The heel counter has an important role to control the excessive calcaneous eversion in a series of running motion. This control is called as shoe stability, which is one of the most important requirement functions in shoe designing. At the same time shoe weight should be reduced because the runners fatigue can be intimately interrelated with shoe weight. Both functions, stability and lightness are conflicting parameters. The optimization method has been developed to satisfy some requirement functions and widely applied to industrial fields. By using the topological optimization method, the optimized heel counter with enough stability and weight reduction was practically designed. In order to check the validity of the counter manufactured, the stability of running shoes with the heel counter was quantitatively evaluated in the practical running motion analyses and compared with that of the conventional heel counter. In case that the numerical model with accurate analytical conditions is constructed, the optimization results can show the valid designing direction. It has been seen that the numerical simulation can reduce not only designing period but also trial cost. It indicates that the numerical simulation will be more important approach from a viewpoint of sustainability index. Therefore it was analyzed that the topological optimization method was so powerful tool in the practical designing process of running shoes [17].

Kailash Chaudhary et al have explained an optimization method to find link shapes for a dynamically balanced planar four-bar mechanism. An optimization method for dynamic balancing and shape formation of a planar four-bar mechanism is presented in this paper. The shaking force and shaking moment developed in the mechanism due to inertia were minimized by optimally distributing the link masses. The link shapes were then found using cubic B-spline curves and an optimization problem is formulated to minimize the percentage error in resulting links inertia values in which the control points of B-spline curve were taken as the design variables. The effectiveness of the proposed method is shown by applying it to a numerical problem available in the literature [18].

H. Hagenah et al has described the process of introducing modern manufacturing methods into the production of standard parts. Innovative technologies to produce materials suitable for light weight construction and products were introduced in the beginning. The process of adapting a given geometry to the abilities of the new materials was the next step. This has to be done while satisfying the requirements determined by the industrial environment. The solution was sought by means of intelligent application of computers and the tools at hand using them. Finally the results were realized moving them from research to real world manufacture. It has been shown that the combination of modern lightweight materials and the corresponding manufacturing technologies can already result in usable, real world products [19].

Marco Münster et al have worked on holistic development method, in the main areas of development of the Vehicle Concept (Phase 1) and development of the Vehicle Body Structure (Phase 2). The new requirements and boundary conditions of electric vehicles were systematically analyzed and used for the development of new body structures. Given the variation in the shape of the vehicle floor, two different body structural concepts were generated. Using multidisciplinary optimizations, various studies are conducted on the body. The structural components are designed with the selected load case, pole crash. The pole crash was chosen because it is the decisive load case for the floor structure. A partial floor area is constructed and tested as a prototype [20].

R. Rezaie et al have proposed topology optimization method. Applying the proposed methodology; a topology optimized part can be fabricated by a low cost fused deposition modeling (FDM) apparatus with as little as sacrificing the features obtained from the optimization stage. This is an advantage as investigating a proper methodology scheme for applying additive manufacturing (AM) technique for topology optimization (TO) is still an open issue in CAE. Moreover, once more accurate apparatus is applied, more gains from the TO can be achieved [21].

Po Wu et al have explained the topology optimization technique. In this paper, an in-depth research about technology optimization was carried out with introducing basic theory, mathematical models and solution methods. After the optimization, the stiffness has been greatly improved and the bracket has a greater carrying capacity. It has been revealed from the analysis results the mass is reduced about 40%, which fully meets the requirements of static characteristics [22].

Richard Frans et al have considered hybrid genetic algorithms for roof truss optimizations. Practically, roof truss optimizations are unique. In this case, the pitch angles are usually governed by roof covering types. In the optimization process, the pitch angle is set to constant, while the coordinates of the joints are determined by genetic algorithms. The optimization process explained in this paper utilizes hybrid genetic algorithms, i.e., a combination of binary and real coded genetic algorithms.

Genetic algorithms are optimization methods that have been used successfully in this paper for various problems. For the sizing, shape and topology optimizations considered in this research work, the area of cross section and the number of members connected to every node were optimized using binary coded genetic algorithms, while the coordinates of the nodes were determined using real coded genetic algorithms [23].

X. F. SUN et al have represented topology optimization of composite structure using Bi-directional Evolutional Structural Optimization (BESO) method. By redefining the criteria of the optimization evolution progress, the proposed method is able to extend current BESO method from isotropic material to anisotropic composite material. The initial modification of BESO method is to allow for inefficient Material Element String (MES) to be removed while efficient MES to be added in the thickness direction of a composite structure. This modification can reduce the chance of high stress concentration in a composite structure and also produce the geometry which is easy to fabricate using fabrication techniques currently available. The results of a cantilever composite laminate under uniform in plane pressure were presented, showing that the proposed method can produce shape and topology for composite structures with optimal structure stiffness [24].

S. Shojaee et al have proposed an effective algorithm based on the Level Set Method (LSM) to solve the problem of topology optimization. The Hamilton–Jacobi Partial Differential Equation (H-J PDE), level set equation, is modified to increase the performance. They combined the topological derivative with nonlinear LSM to create a remedy against premature convergence and strong dependency of the optimal topology on the initial design.

The magnitude of the gradient in the LS equation was replaced by several Delta functions and the results were explored. Instead of the explicit scheme, which is commonly used in conventional LSM, a semi-implicit additive operator splitting scheme was carried out in this study to solve the LS equation. A truncation strategy was implemented to limit maximum and minimum values in the design domain. Finally, several numerical examples were provided to confirm the validity of the method and show its accuracy, as well as convergence speed [25].

Prashant H. Patil et al have studied the movable jaw of rear vice of horizontal band saw machine and weight reduction has been carried out using topology optimization. The static analysis as well as optimization of movable jaws has been carried out in Hyper-works. From the analyzed results, it has been concluded that the values obtained for the maximum displacement and von-Mises stress of optimized model are lower than existing model.

Topology optimization generates an optimized material distribution for a set of loads and constraints within a given design space. This Optimization reducing weight, manufacturing cost of component fulfilled with all design constraints. Weight optimization of rear vice resulted to 25% of weight reduction than existing model [26].

Purushottam Dumbre et al have presented optimization method such as topology optimization. Topology optimization was used to reduce the weight of existing knuckle component. It has been revealed from the results reduction in weight of the component is 11% while meeting the strength requirement, with limited design space given with or without change in material properties [27].

Isha Tikekar et al have performed weight optimization of chassis. Static structural analysis of truck chassis was carried out. Chassis model was optimized for decreasing the weight by volume reduction and material change. Critical areas in the chassis were identified using topology optimization, where different materials were tried and simulated for weight reduction while retaining the displacement and stresses within the allowable limits. It was found the changing the materials of side members to high strength steel and cross members to carbon fiber weight reduction of 11% is possible without any reduction in strength of the chassis along with reduction in static displacement of the chassis [28].

D. Costi et al have presented a methodology to reduce the weight of an automotive hood substructure. The methodology consists in a loop of different optimization techniques, i.e. topology, size and topography, coupled with a constant re-designing of the model. Without breaking the performance targets expected by Ferrari internal regulation, the mass has been reduced respecting manufacturing constrain [29].

PROPOSED METHODOLOGY

After the critical analysis the following is the proposed methodology which can be used in order to optimize the brake pedal.

LITRATURE SURVEY

It has been revealed from the literature survey most of the research work were belongs to alternative material substitution for light weight component.

Problem Definition

Problem has been formulated with the help of findings (outcomes) in critical literature survey and effort to fill the gap between the research works in literature. Problem definition has been finalized. The aims and scope of the research work is to reduce the weight of an existing brake pedal design of an auto rickshaw with the application of theoretical, numerical and experimental approach and topology optimization techniques without the substitution of material.

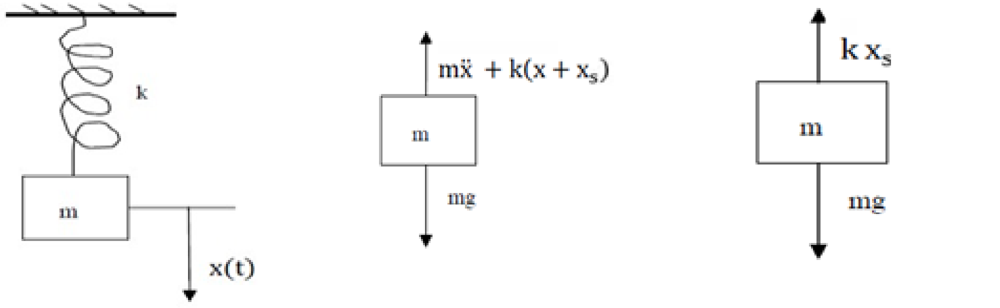
Reverse Engineering

The Reverse engineering tool and techniques such as use of 3D scan, Coordinate measurement machine, measurement instruments like Vernier will be used to extracts the all dimension of the component.

Theoretical Analysis

Theoretical analysis of component will be carried out by using basic engineering principles and formulas Theoretical analysis of brake pedal is carried out by using fundamental theory of vibration of single degree of freedom. Most of the system exhibit simple harmonic motion or oscillation. These systems are said to have elastic restoring forces.

Such systems can be modeled, in some situations, by a spring-mass schematic, as illustrated in Figure 4. This constitutes the most basic vibration model of a machine structure and can be used successfully to describe a surprising number of devices, machines, and structures. This system provides a simple mathematical model that seems to be more sophisticated than the problem requires. This system is very useful to conceptualize the vibration problem in different machine and vehicle components like brake pedal.



a) Spring Mass Schematic
b) Free Body Diagram
c) Free Body diagram in Static Condition

Figure 4- Spring Mass System

We consider the brake pedal as equivalent spring mass system as depicted in Fig.4

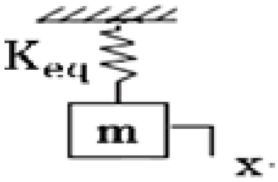


Figure 5- Brake Pedal Equivalent Spring Mass System

- The natural frequency of the brake pedal can be calculated by following relation,
- $\omega n = \sqrt{(K/m_{eq})}$ **(I)**
- Where, K=3EI/l^3
- E is Young's modulus of material in N/m2. (Steel = 210 x 109 N/m2)
- I is Moment of inertia of brake pedal in m4.
- lis length of the brake pedal in m.

Here for calculation we assume the brake pedal structure as tapered beam as shown in fig.5

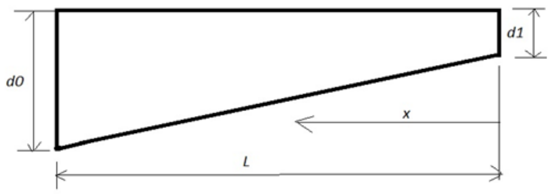


Figure 6- Brake Pedal Equivalent as Tapered Beam

Where d0 = Depth of tapered beam at fixed end. (d0 is 0.035 m)

d1 = Depth of tapered beam at free end (d1 is 0.017m)

L = Length of tapered beam. (L is 0.238m)

x = Length over the variable depth of tapered beam. We can calculate the equivalent depth d of the brake pedal as tapered beam at x= L by using following relation,

d=d1+(do-d1) (x/L) **(i)**

By putting respective values in above equation 6.4 we get,

d = 0.035 m **(ii)**

Moment of inertia of brake pedal can be calculated using following relation,

I(x) = bd3 /12 **(iii)**

Where, b is thickness of brake pedal and d is equivalent depth of brake pedal. By putting respective values from equation (ii) in to equation (iii) we get moment of inertia of brake pedal as

I(x) = 1.7864 x 10-8 m4 **(iv)**

We can calculate the stiffness of brake pedal by using following relation,

K=3EI/l^3 **(v)**

We get the stiffness of brake pedal K=8.348114 x 105 N/m. We need to consider the mass added factor for the self-weight of the brake pedal to account for no load condition of the brake pedal structure, when it is subjected to natural vibration under self-weight consideration. Self-weight of the brake pedal mbp is 0.480 Kg. (actual physical weight).

Equivalent weight of the brake pedal can be calculated by using following relation.

$m_{eq} = m_{af} (m_{bp})$ **(vi)**

Where m_{eq} is equivalent weight of brake pedal, m_{af} is mass added factor (17.5/3) and m_{bp} is self-weight of the brake pedal. [30].

By using equation (I) and equation(vi) we can calculate he natural frequency of brake pedal. We get natural frequency of brake pedal ω_n=546.02 rad/s and in Hertz

$f = \omega_n / 2\pi = 86.90$ Hz

Computer Aided Design (Cad) Modeling

Computer aided design (CAD) will be carried out with the help of CAD software like Solid Works. Necessary changes in design will be carried out with CAD software.

Three Dimensional Laser Scanning Of Brake Pedal

3D Laser Scanning is a non-contact, non-destructive technology that digitally captures the shape of physical objects using a line of laser light. 3D laser scanners create “point clouds” of data from the surface of an object.

In other words, 3D laser scanning is a way to capture a physical object's exact size and shape into the computer world as a digital 3-dimensional representation. 3D laser scanners measure fine details and capture free- form shapes to quickly generate highly accurate point clouds.

3D laser scanning is ideally suited to the measurement and inspection of contoured surfaces and complex geometries which require massive amounts of data for their accurate description and where doing this is impractical with the use of traditional measurement methods or a touch probe.

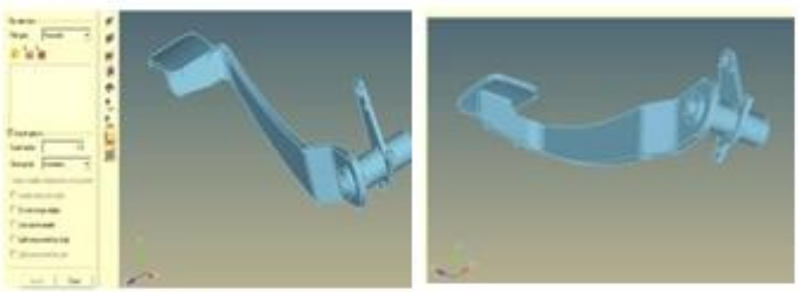
3D scanner used for scanning activity is depicted in figure 7

Figure 7

Physical brake pedal has been scanned by 3D laser scanner and by using cloud points 3D model has been created. Figure 2 show the physical brake pedal model. 3D

\

Figure 8- Brake Pedal Physical Model



i) ii)

Figure 9- 3D Point Clouded Model

Numerical Analysis (Finite Element Analysis)

Linear static stress analysis will be performed to study the behavior of the component when subjected to extreme foot load. Modal analysis of the component will be carried out to investigate the fundamental natural frequency of the component.

Computer Aided Design (CAD) Modeling

An actual brake pedal sample from a three wheeler rickshaw will be used in this project.. A new brake pedal design will be proposed without any new material substitution by using topology optimization.

Topology optimization will give us the optimum material distribution density in the component without sacrificing performance criteria of the component. Both current and new proposed design of brake pedal will be analyzed using Altair Hyper-Works with Opti-Struct solver.

FEA of brake pedal has been carried out using pre- processing tool HyperMesh and Opti-Struct solver

Optimum Design

Optimum design will be analyzed by using finite element analysis to check the behavior of the new lightweight component.

Experimental Analysis

Experimental analysis will be carried out to validate the vibration characteristics of the component.

CONCLUSION

Within the realm of optimization, engineers all around the world are devoting a significant amount of their time and energy. Because of the critical nature of this issue, a research was carried out to investigate ways to reduce the amount of force required to depress the brake pedal.

This study accomplishes the overarching goal of reducing the weight of the item without compromising the material it is made of. In lieu of using a different material, machining results in a weight reduction of a sizeable amount. This is in addition to the fact that the weight shifts with time. There has been a weight reduction of 1.41 grammes in the brake pedal, which is equivalent to a 29 percent decrease in the weight of the weight of the system that was previously in place. This study enables us to save both material and the cost of material, which, in the context of mass production, is a very significant element. It also has the potential to save a significant amount of capital for enterprises.

REFERENCES

1. S.m.sapuan , ‘’ a conceptual design of the concurrent engineering design system for polymeric based composite automotive vehicle"., American applied journal of applied science,2005.

2. Fredrik Henriksson *, Kerstin Johansen, in 26th CIRP Design Conference, On Material Substitution in Automotive BIWs – From Steel to Aluminum Body Sides, Procedia CIRP 50 683 – 688,2016.

3. Kailash Chaudhary*, Himanshu Chaudhary, "Shape optimization of dynamically balanced planar four-bar mechanism" Procedia Engineering 57 (2015) 519 – 526,2015.

4. Mohd Sapuan Salit*, Mohd Syed Ali Molla and Md Liakot Ali in "conceptual design of an automotive composits brake pedal" , 2005

5. Pankaj Chhabra, Puneet Katyal and Visha Gulati ,"Concurrent Design and Prototyping of Composite Accelerator Pedal", International Journal of Advancements in Technology,2009.

6. D. Costi, E. Torricelli, L. Splendi and M. Pettazzoni, "optimization methodology for an automotive hood structure", Proceedings of the World Congress on Engineering 2011 Vol III.

7. K. K. Dhande1*, N I Jamadar1 and Sandeep Ghatge1 ,"Design and analysis of composite brake padel :an ergonomic approach, IJMERR Vol. 3, No. 3, July 2014

8. Elena Cischino a,*, Francesca Di Paolo b, An advanced technological lightweighted solution for a Body in White, Transportation Research Procedia 14 1021 – 1030,2016.

9. Jacek Grosela*, Wojciech Sawickia, Wojciech Pakosa,

"Application of Classical and Operational Modal Analysis for Examination of Engineering Structures" Procedia Engineering 91 (2014) 136 – 141,2014.

10. Sun Wenlonga, Chen Xiaokaia, Wang Lub "Analysis of Energy Saving and Emission Reduction of Vehicles Using Light Weight Materials", Energy Procedia 88 889 – 893,2015.

11. Rajesh Purohita, Pooja Khitoliyab and Dinesh Kumar Kolic, "Design and Finite Element Analysis of an Automotive Clutch Assembly", Procedia Materials Science 6 (2014) 490 – 502,2014.

12. Ashwani Kumara*, Himanshu Jaiswalb, Faraz Ahmadb, Pravin P Patilb, "Dynamic Vibration Characteristics Analysis of Truck Transmission Gearbox Casing with Fixed Constraint of Vehicle Frame Based on FEA", Procedia Engineering 97 (2014) 1107 – 1115,2014

13. Tsuyoshi Nishiwaki and Mai Nonogawa, "Application of topological optimization technique to running shoe Designing" Procedia Engineering 112 (2015) 314 – 319,2015.

14. H. Hagenaha*, W. Böhma, T. Breitsprecherb, M. Merkleina, S. Wartzackb, "Modelling, construction and manufacture of a lightweight robot arm", Procedia CIRP 12 (2013) 211 – 216,2013.

15. Prashant H. Patil, Suresh S. Patil "Weight Optimization of Movable Jaw of Rear Vice of Horizontal Band Saw Machine Using Topology Optimization" , International Journal of Science and Research (IJSR),2007.

16. Purushottam Dumbre, A. K. Mishra, V. S. Aher, "Structural Analysis of Steering Knuckle for Weight Reduction", International Journal of Emerging Technology and Advanced Engineering, Volume 4, Issue 6, June 2014

17. Marco Münster a,*, Michael Schäffer a, Gerhard Kopp a,Gundolf Kopp, Horst E. Friedrich, "New approach for a comprehensive method for urban vehicle concepts with electric

powertrain and their necessary vehicle structures" , Transportation Research Procedia 14 (2016) 3686 – 3695,2016.

18. Isha Tikekar, Abhinav Damle, "weight reduction of heavy duty truck through material optimization", International Journal of Engineering Research and General Science Volume 4, Issue 3, May-June, 2016.

