International Journal of Science and Research (IJSR)
ISSN: 2319-7064
SJIF (2022): 7.942

A Review on Study of Design Optimization Technique of Mechanical Component

Kanchan S. Gaikwad[1], Shekhar G. Shinde[2], Ranjitsinha R Gidde [3]

Abstract: ***This paper is going to present a broad overview of a few researchers who have completed various designs optimization projects during the past couple of years. There are many advantages to designing light weight products, such as lower material consumption, shorter production times, and lower energy consumption, resulting in a more efficient product development process. It is explained in this review article how design optimization and topology optimization may be applied to any machine or automobile component and how they may be used as a means of optimizing performance. Numerical optimization techniques can be used in order to design various components by optimizing their size, shape, and structure using numerical computations. It optmise the material distribution in a design part or product. Basically, Topology Optimization (TO) is the process of removing the unwanted or excessive mass from the components in order to maintain their strength and functionality. A TO design is different from shapes optimization and sizing optimization, in that it can be shaped into any shape within its architectural space, rather than relying on predefined configurations to achieve any shape. In the literature review, it was shown that optimizing the mass interns was possible through the change in materials as well as trial - and - error method by changing the design of the component. Through the process of reverse engineering and reference data gathering, the design parameters of mechanical components are studied and 3D modelling is performed using Solidworks 2020 software in order to find the best solution. There are a number of features integrated within the software package ANSYS 19.0 that allow direct topology optimization to be used as a tool for mass optimization.***

Keywords: Topology optimization, design, FEA analysis, SOLIDWORKS, ANSYS

1. Introduction

In the last decade the rapid growth of the mechanical industry required use the structure optimization method. Structural opyimization is the process of determining the best design shape of the structural part. The shape of the model resulted after applying some optimization criteria such as maximum strength, maximum rigidity, minimal displacement, minimal cost, minimum weight, etc. 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.

I. I. Concept of Optimization Technique

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.

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. Implementation Methodologies

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

a) Discrete
b) Solving the problem with continuous variables
c) Commercial Software

4. 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.

5. Problem solving 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.

6. 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.

7. Objectives

- The main objective of this study is to explore weight reduction opportunities for a mechanical component 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.

8. Methodology

This method of measurement is called reverse engineering. All the measured data are noted down and Tabulated for 3D Modelling of the mechanical component Reverse Engineering Method is used to measure all the Parameters of the mechanical component.3D Modeling of mechanical component 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 mechanical component 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 component. Static structural analysis is used to find the structural strength of the mechanical component subjected to loading. ANSYS Topology Optimization tool is then incorporated to optimize the 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.

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”.

Author Profile

Kanchan Gaikwad, Student Mtech Design Engineering

Shekhar Shinde, MTech in Design Engineering, working as a design engineer.

Dr. R. R. Gidde, PhD in Microfludics, working as dean Administrative in educational institute.
Country - India
Mobile Number - 9356586030
City - Pandharpur