Analysis of Topology Optimization in load carrying element

Satish Kumar P[1], P Magesh Kumar[2], Prithvi Raj[3], Badagala Srikar[4], Nagendra J[5]

[1,2,3,4] *Students, B.E Mechanical, New Horizon College of Engineering, Bangalore, India*
[5] *Assistant Professor, Department of Mechanical Engineering, NHCE, Bangalore, India*

Abstract - The main aim of topology optimization is to remove the material where there is no impact from the load that is applied. This material that is removed from load carrying element will reduce the mass of the component.

This paper presents the simulation of topology optimization in load carrying elements. The main aim of the load carrying element is to withstand and hold the weight of the differential element. But the structure of the load carrying element can be optimized with the help of fusion 360 software. This paper presents how the load carrying element can be optimized.

Key Words: **Topology Optimization, Fusion 360, Shape Optimization, Meshing.**

1. INTRODUCTION

Optimization of topology is a mathematical method for enhancing the layout of materials within a given design space, considering a given set of loads, boundary conditions, constraints, with the goal of maximizing the performance of the system.

The principle of topology optimization is to reduce weight of a Structure to optimal conditions without deterioration of the performance of Structure. The concept of topology optimization holds good only when the constraints, boundary conditions and set of loads are fixed or predetermined. If these conditions are varied the topology optimization leads to failure the structure or reduces the performance of the optimized structure.

Mechanical and civil engineers have been using topology optimization for many years so as to reduce the amount of material used and the strain on structures while maintaining their mechanical strength and performance.

This method has been used in many industries like aerospace, automobile, mechanical, bio-chemical and civil industries. It can reduce predefined cost function and also reduce the carbon footprint in the atmosphere due to significant weight reduction of parts.

2. LITERATURE REVIEW

Simant et al presented a study on increasing the load carrying capacity and the weight reduction of bogie and it's components which requires the improvement and alteration of existing design of critical components of the bogie. The software used for topology optimization of bottom center bearing plate in this study is ANSYS. The central bearing plate of design is an integral component of the bogie. It balances and transfers a variety of forces that are generated during motion of the vehicle. The result obtained after optimization in the FEA tool results in weight reduction of the component which is used to modify the design of the component.

V.B. Hammer et al presented a study that shows a generalized topology optimization of linear continuum structures. The problems regarding the load carrying constraints depends mainly on the design of the structure. It also addresses that the main objective of the design is minimum compliance, with assumptions of boundary conditions and total volume admissible within domain. The presentation of material is provided with a 2D context. In this study, topology optimization is executed using SIMP material model in order to effectively optimize the structure.

Saurabh Bankoti et al addresses that topology optimization is one of the most necessary and challenging part in structural mechanics where there is a necessity to change the topology and the shape of the structure during the process of optimization. There are various design techniques that are used to overcome the difficulties in optimising a structure. This study reviews two most commonly used methods for topology optimization: Material distribution method and Guided weight method. These methods are compared with optimality criteria using ANSYS. Optimises shape and compliance are the parameters used for comparison in this study.

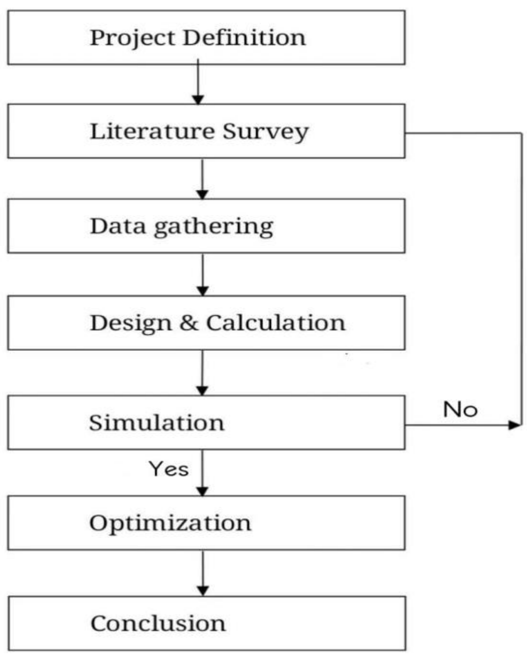
I. Ntintakins et al presented a study on Topology Optimization by the use of 3D Printing Technology in the Product Design Process where inject binder process is used to print furniture models with various wall thickness, and then use compression tests to determine their durability and robustness. The most effective and efficient approach to manufacture prototypes is to employ additive manufacturing. The results from the study shows that hollow printed specimens have good durability in compression tests conducted, and the results are used to asses a design proposal. The thickness of the inner wall varies with the additive manufacturing process employed. To obtain best design solution, topology optimization studies are undertaken under various conditions to evaluate the best approach and to optimize an entire model, several constraints, load scenarios are to be established. This study concludes that, Topology Optimization is a strong tool that may assist designers and engineers in making the best decisions throughout the design phase.

3. OBJECTIVES

- Building weight saving and complete designs.
- To decrease needed time to present and test product.
- Reduction in production and product cost.
- To save the amount of material required.
- Improvement in product performance. For example, better fuel efficiency due to reduction in weight, less wear of sliding components.
- Less carbon footprint.

4. METHODOLOGY

I. Flowchart



II. PRINCIPLE AND WORKING

The Topology Optimization 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

The software that is to be used is Fusion 360 from Autodesk to optimize the component. The first step to optimization is to determine a concept design domain in CAD. The next step is to apply the static load/uniformly distributed load/uniformly varying load in the desired location of the CAD model. There are the regions the where the optimization algorithm shouldn't play around i.e., locations where we have our constraints and where we add the load. In order to do that, we can use the preserve tool. The next step is to create the finite element mesh. The final step is to optimize the component and remove the material where it is not necessary.

The load that is applied on both sides of the load carrying bar is 490 N point load. A load 50 N is applied on the central bar. The size of the mesh given is 3mm.

III. OBSERVATIONS AND CALCULATION

- Volume of the load carrying element, $v = 0.005\ m^3$
- Density of the load carrying element , $\rho = 7870\ kg/m^3$
- Density, ρ = Mass/Volume = m/v
- Therefore , Mass = 0.005 * 7870
- Mass of the load carrying element, m = 40 kg

IV. METHODOLOGY

- **Data Gathering and 2-D Printing:** The data i.e., dimensions with respect to the model that is to be optimized is collected as per the requirements. This data is then processed to get final volume of the model without the model being optimized. This result is used for comparing the final result after being optimized. In order to get the 3-D of the model using Fusion 360 software, 2-D drawing is drawn manually of the required model. The front view, top view and the side view of the model is given below.

- **3-D Drawing of Load Carrying Element:** The 3-D design of load carrying element is done using Fusion 360 software. The first step is to draw the back plate of the load carrying element and then extruded it as per the required thickness. Similarly, the central column and the angular sections are drawn and extruded as needed. The dimensions given here must be accurate in order for the final results to be accurate. The next step is to save the model and go to simulation for shape optimization.

- **Applying Boundary Conditions:** The main types of loading available in FEA include force, pressure and temperature. These can be applied to points, surface, edges, nodes and elements or remotely offset from a feature. The way that the model is constrained can significantly affect the results and requires special consideration. Over or under constrained models can give stress that is so inaccurate that it is worthless to the engineer. It is the process of applying the boundary conditions ie., the area that is to be fixed. The back plate of the load carrying element is fixed. This way with the load being applied on angular section, the load carrying element is considered as a cantilever beam, as the load carrying element is fixed.

- **Applying Load:** The main purpose of load carrying element is to bear the weight of the differential that is placed on the angular section of the load carrying element. In order to go further with the simulation, we have to apply the load of the differential. The load of the differential is assumed to be point load of 980N ie., 100 kg. The direction of load that is to be applied is vertically downwards with a load of 490N on both the angular section.

- **Preserve Region:** The next step is to distinguish the regions where the optimization algorithm can get rid of material and at which locations it should not get rid of material. The region where the optimization algorithm shouldn't play the role is the regions where the boundary conditions are applied and the regions where the load is applied. This is achieved with the help of preserve region tool. This way, we can preserve the regions where the material is not been removed.

- **Meshing:** Meshing is the process in which the continuous geometric space of an object is broken down into thousands or more of shapes to properly define the physical shape of the object. The absolute size of the mesh is 3mm. Finer the mesh, more accurate is the result that is obtained and also the time taken to get the results well also be more. The main aim of the project is to maximize the stiffness and remove the material where it is not needed. After the meshing, the shape optimization is done to attain the final results.

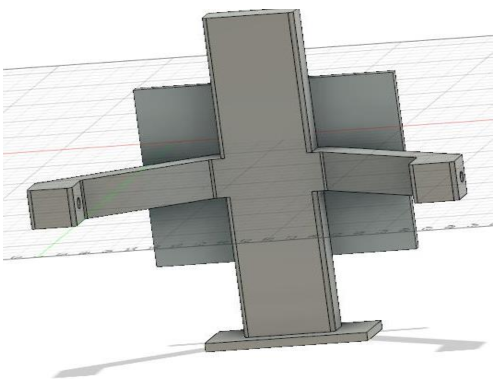


Fig-1: 3-D image of load carrying element.

Fig-2: Load carrying element with Differential being the load.

5. RESULT & OBSERVATIONS

It is observed that after the simulation, the region where the material is not required has been removed. This leads to reduction in the usage of material required for manufacturing and hence reduction in the weight of the component. After the simulation, we will get the material distribution that the algorithm has been calculating. The blue region represents the region where the material is not needed. The red region represents the region where the material for the load to be carried that is the weight of the differential.

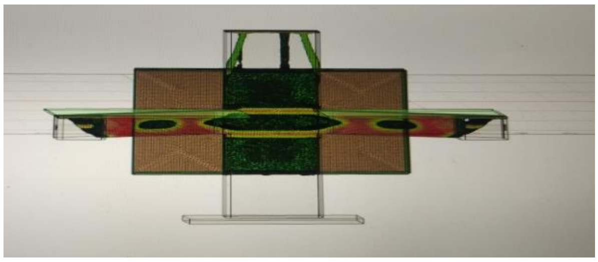


Fig-3: Load carrying element after shape optimization

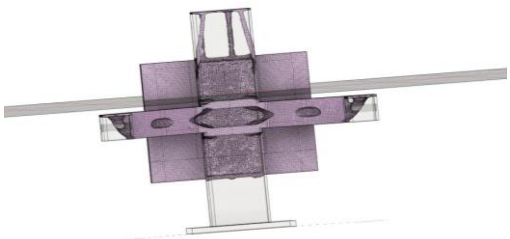


Fig-4: Final model of the load carrying element.

6. CONCLUSIONS

- Designing and simulation of the structure is completed using fusion 360.
- The structure has been optimized resulting in significant weight reduction due to material removal from unpreserved areas while increasing the overall performance of the structure.

7. ACKNOWLEDGEMENT

We wish to express our sincere gratitude to "**Dr. Shridhar Kurse,** Prof. & HOD-Mechanical Engineering" for his constant encouragement and cooperation. We extend our deep sense of gratitude to our teacher and guide "**Dr. Nagendra J**, Associate Professor in the Department of Mechanical Engineering" NHCE, for his valuable suggestions, guidance, care & attention shown during the planning, conduction stages of this work. We thank all the teaching and non-teaching staffs, our classmates and friends for sharing their knowledge and valuable suggestions.

8. REFERENCES

[1] Prashant Kumar Srivastava, Simant, Sanjay Shukla, "Topology Optimization: An Effort to Reduce the Weight of Bottom Centre Pivot'' International Journal of Recent Technology and Engineering (IJRTE) ISSN: 2277-3878, Volume-8 Issue-2, July 2019

[2] V.B. Hammer and N. Olho, "Topology optimization of continuum structures subjected to pressure loading", Struct Multidisc Optim 19, 85-92 Springer-Verlag 2000.

[3] Saurabh Bankoti, Akash Dhiman and Anadi Misra, "Comparative Analysis of Different Topological Optimization Methods with ANSYS'', E-ISSN 2349-7610, International Journal for Research in Emerging Science and Technology, Volume-2 Issue-4, April-2015

[4] Aykut Kentli, "Topology Optimization Applications on Engineering Structures'', Topology Optimization Applications on Engineering Structures, DOI:

http://dx.doi.org/10.5772/intechopen.90474.

[5] I. Ntintakins, G.E. Stravroulakis, N. Plakia, ''Topology Optimization by the use of 3D Printing Technology in the Product Design Process'', HighTech and Innovation Journal, Volume-1, Number-4, December 2020.

[6] Jihong ZHU, Han ZHOU, Chuang WANG, Lu ZHOU, Shangqin YUAN, Weihong ZHANG, "A review of topology optimization for additive manufacturing: Status and challenges", Chinese Journal of Aeronautics 2021.

[7] D. Vlah, R. Zavbi and N. Vukasinovic, "Evaluation of Topology Optimization and generative design tools as support for conceptual design", International Design Conference – Design 2020.

9. BIOGRAPHIES

SATISH KUMAR P

Studying Final year B.E-Mechanical Engineering at NHCE, Bangalore, Karnataka

P MAGESH KUMAR

Studying Final year B.E-Mechanical Engineering at NHCE, Bangalore, Karnataka

PRITHVI RAJ R

Studying Final year B.E-Mechanical Engineering at NHCE, Bangalore, Karnataka

BADAGALA SRIKAR

Studying Final year B.E-Mechanical Engineering at NHCE, Bangalore, Karnataka

Dr. NAGENDRA J

Associate Professor in the Department of Mechanical Engineering at NHCE, Bangalore, Karnataka