INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact factor: 4.295
(Volume 5, Issue 4)
Available online at: www.ijariit.com

Shape optimization of a bracket model using Finite Element Analysis

Ratendra Kumar Srivastava
goldensrivastavame@gmail.com
Dr. A.P.J. Abdul Kalam Technical University, Lucknow, Uttar Pradesh

Santosh Kumar Verma
sv910586@gmail.com
Dr. A.P.J. Abdul Kalam Technical University, Lucknow, Uttar Pradesh

Paveen Kumar Maurya
ppaveenmaurya@gmail.com
Apsinox Design Era Private Limited, Allahabad, Uttar Pradesh

ABSTRACT

The objective of this research is to integrate a method of optimization of structures to the Computer-Aided Design (CAD) process with a mesh quadratic. Uses ANSYS software to perform static finite element analysis on a bracket model. On this basis, the homogenization method is adopted, and the total flexibility of the model is taken as the objective function, and the volume is used as the constraint condition. The topology optimization design of the model under several working conditions is carried out. The problems of basic model establishment, optimization region selection, optimization process control and optimization result analysis and application in topology optimization design process are discussed. The application of topology optimization in the initial design process of automotive structures is realized.

Keywords— *Meshing Element, FEA, Ansys 17, Mesh optimization, Result evaluation*

1. INTRODUCTION

The study of structural optimization is divided into three levels: Sizing Optimization, Shape Optimization, and Topology Optimization. Structural size optimization is basically mature. Structural shape optimization is more difficult than structural size optimization. It is still in the development stage, and structural topology optimization is very difficult. It is considered to be the most challenging subject and is still in the exploratory stage in engineering design.

The basic idea of structural topology optimization is to translate the optimal topology problem of seeking a structure into the problem of finding the optimal material distribution in a given design area. Through topology optimization analysis, designers can fully understand the structure and functional characteristics of the product and can design the overall structure and specific structure in a targeted manner. Especially in the early stage of product design, it is not enough to design parts by experience and imagination. Only under the appropriate constraints, make full use of topology optimization technology for analysis, combined with rich design experience, can design products that meet the best technical conditions and process conditions. The biggest advantage of topology optimization of continuum structure is that it can determine the reasonable structure form according to the known boundary conditions and load conditions without knowing the shape of the structure topology. It does not involve the specific structure size design but can propose the best design. Program. Topology optimization technology provides designers with new designs and optimal material distribution options. Topology optimization is based on the idea of conceptual design, and the resulting design space needs to be fed back to the designer and made appropriate modifications. The optimal design tends to be lighter than the conceptual design and the performance is better. Designers have modified the design to get a better solution through shape and size optimization.

Meshing involves the shape of the element and its topology type, cell type, mesh generator selection, mesh density, Unit number, and geometry voxel. For example, in terms of geometric expression, the beam and the rod are the same, but they are solved from the physical and numerical values. There is a difference in the above. Similarly, the unit solving equations for plane stress and plane strain conditions are also different. Due to the stiffness matrix of different units is different, and the solution method using numerical integration is different. Therefore, in practical applications, it must be adopted. Simulate the solution with a reasonable unit.

2. OPTIMIZATION METHOD SELECTION

2.1 Finite element meshing and refinement

At present, the commonly used topological optimization methods for continuum structures are variable thickness method, variable density method, and homogenization method. The mathematical model of the variable thickness method is simple, but the optimization object is greatly limited. The variable density method is artificially established to establish a relationship between material density and material properties. After the topology optimization calculation, the density value of the unit is 0 or 1, and the topology optimization structure is relatively clear. The homogenization method is the most popular method. After the topology optimization, the density value of the unit is a continuous value between 0 and 1, resulting in a relative topology. The optimal topology structure only considers the strength of the structure. The design of the structure also needs to meet the design requirements of the manufacturing process and assembly relationship. People need to design a structure based on topology optimization. The topology provides a range of values. More conducive to subsequent design.

2.2 Mathematical model of the homogenization method

The basic idea of the homogenization method is to introduce the microstructure-single cell in the material composing the topology. In the optimization process, the unit cell size of the microstructure is used as the topological design variable to establish the relationship between material density and material properties. The addition and deletion of microstructures are realized by the growth and decrease of the cell size, and the composite material composed of the intermediate size unit cells is generated to expand the design space, thereby realizing the structural topology optimization model and the size optimization model, which has a strict mathematical foundation and is a good way.

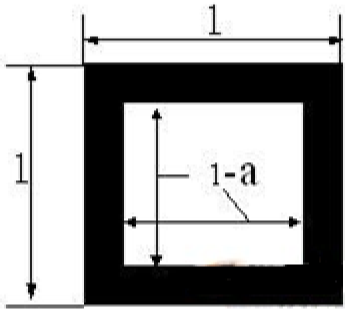


Fig. 1: Microstructure unit cells (unit cells)

The density of microstructure unit cells is $\boldsymbol{\eta = 2a - a^2}$

The mathematical model of the homogenization method is:

$$\boldsymbol{Min\ l(u) = \int pud\Omega + \int tud\Gamma}$$

The constraints are:

$$\boldsymbol{l(v) = \int E_{ijld}\,(a)\varepsilon_{ij}(u)\varepsilon_{ld}(v)d\Omega}$$

$$\boldsymbol{\int \eta d\Omega \leq V}$$

$$\boldsymbol{E_{ijld}(a) = \eta^a E_0}$$

$$\boldsymbol{0 \leq \eta \leq 0}$$

$$\boldsymbol{a \geq 1}$$

$$\boldsymbol{0 \leq a \leq 1}$$

Where:

η: density unit cell density l(u)--structural compliance

$\boldsymbol{l(v)}$: The equivalent bulk force experienced by the structure and the virtual work

$\boldsymbol{p/t}$: Made by the boundary load on the virtual displacement

V: Equivalent volume force and boundary load subjected to the structure

u: node displacement

v: the virtual displacement

$\boldsymbol{\varepsilon_{ij}(u)\varepsilon_{ld}(v)}$of the node: the strain

εkl(v): due to the node displacement u caused by the virtual displacement v of the node the virtual strain

Eijkl(a): the assumed material property is related to the density η and the material property E0 of the actual material used.

E0: the material property of the actual material used

α: the undetermined coefficient

V: the initial volume of the structure

Ω: Integrating

Γ: in the volume domain with volumetric force means that the integral is performed on the boundary domain with area force.

In the above model, equation (2) takes the minimum total flexibility of the structure as the optimization target, and the unit cell size *a* of the microstructure is the optimization design variable; the constraint condition (3) is based on the virtual work principle and the static balance of the structure. Constraints; Constraints (4) Consider that the optimized volume must not be larger than the initial volume, and constraint (5) assumes the relationship between material properties and density.

2.3 The finite element analysis of the bracket body

As the key assembly of the passenger car, the body frame must have sufficient strength and static stiffness to ensure its fatigue life, assembly, and use requirements, and also have reasonable dynamic characteristics to achieve the purpose of controlling vibration and noise. The application practice proves that the finite element method is used to analyze the structure of the model body, and the rigidity, strength, natural frequency and vibration mode of the design drawings can be fully understood before the design drawings become products to understand the stress and deformation of the model body. Timely improvement of the deficiencies, so that the product can meet the use requirements at the design stage, thereby shortening the design test cycle, saving a lot of test and production costs, and it is one of the economical and applicable methods to improve product reliability.

2.4 Generation of finite element model

The geometric model is the basis of the finite element model. In this paper, the Ansys software system is used to build a three-dimensional geometric model based on the design modular two-dimensional design drawings of the body frame structure, and the model is imported into ANSYS using a self-programmed interface program. After importing the geometric model, you need to make some necessary modifications to divide the grid. In order to check the built finite element model, after the node is constrained at the joint of the suspension assembly, the three coordinate axes are given a certain acceleration, and the connection between the beams is checked and modified. The final finite element model is shown in. The scale information of the model: 1288 key points, 2150 lines, body frame finite element model node 31216, unit 16044. The model body skeleton quality is 1kg, the model mass is 0.9998g.

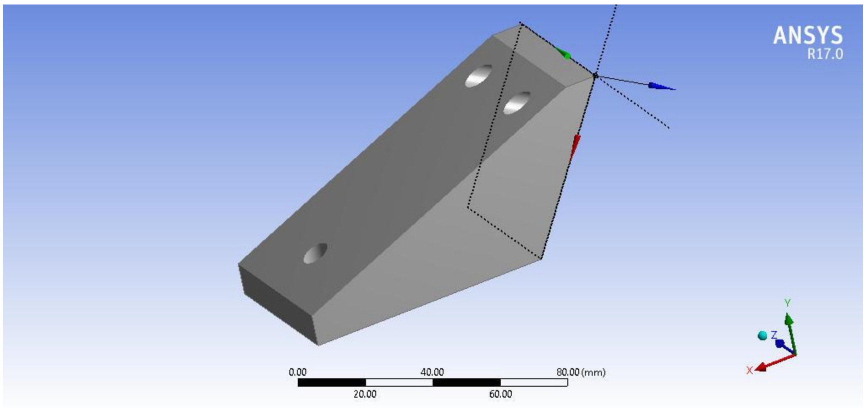


Fig. 2: Bracket body for optimization tropology

2.5 Static finite element load condition analysis of body structure

When the bracket is running, the body bears a lot of loads. In terms of its load characteristics, the main loads on the body are bending, torsion, lateral load, and longitudinal load. The bending load is mainly generated from the mass of the model body, model equipment, passengers and luggage; the torsional load is generated by the asymmetric support of the road surface unevenness to the model body. As a comparative calculation, the static maximum possible torque can be used, that is, a front-wheel is simulated. The limit state of the dangling; the lateral load is mainly caused by the centrifugal action during the steering; the longitudinal load is generated by the inertial force during the acceleration and braking. In order to get a more comprehensive understanding of the stress distribution of the body frame under actual working conditions, horizontal bending conditions (no-load + full load), ultimate torsional conditions (left and right front wheel suspension), emergency turning conditions (left, The right-turning and emergency braking conditions (full load) were analyzed by finite element simulation to analyze the structural strength and stiffness of the model body, which provided a reference for further optimization design.

2.6 Top topology optimization

Topology optimization refers to shape optimization, also known as shape optimization. The goal of topology optimization is to find the optimal material distribution scheme for objects that are subjected to single or multiple loads. This optimization appears as a "maximum stiffness" design in topology optimization. Unlike traditional optimization designs, topology optimization does not require the definition of parameters and optimization variables. The objective function, state variables, and design variables are all predefined, and the user only needs to give the parameters of the structure (material properties, models, loads, etc.) and the percentage of material to be saved. [6] The objective function of topology optimization is to reduce the deformation energy of the structure while satisfying the structural constraints. Reducing the deformation energy of the structure is equivalent to increasing the stiffness of the structure. This technique is achieved by using design variables for the pseudo-density of each finite element.

3. METHODOLOGY

The methodology is given in figure 3.

3.1 Define topology optimization problems

Topology optimization analysis is the same as another finite element analysis. The first thing is to establish its optimization model according to the basic structure of the analysis object. Due to the complexity of the body skeletal structure and the diversity of its load-bearing, it is almost impossible to optimize the topology of the entire body frame. The static analysis results show that the deformation of the top under various working conditions is second only to the engine layout at the rear of the body frame; the modal analysis shows that the vibration amplitude of the top in the middle and high-frequency range is large, which is related to the top arrangement. In order to reduce the scale of the optimization problem, the static analysis results are used as the constraints of the top optimization, and the topological optimization technology of ANSYS is used to optimize the topology of the model body top.

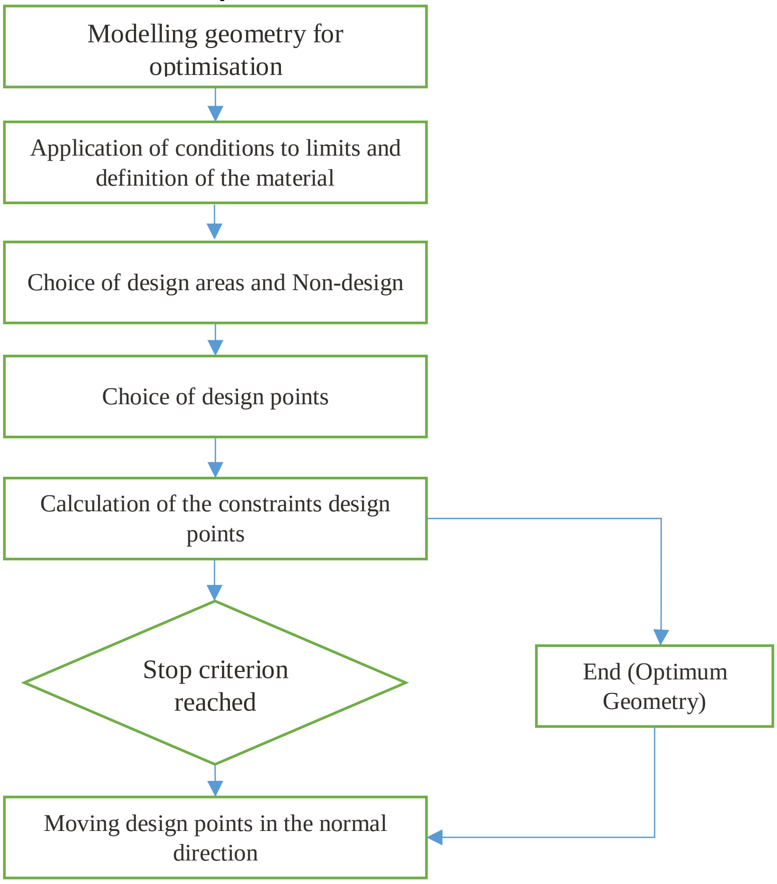


Fig. 3: Flow chart of Methodology

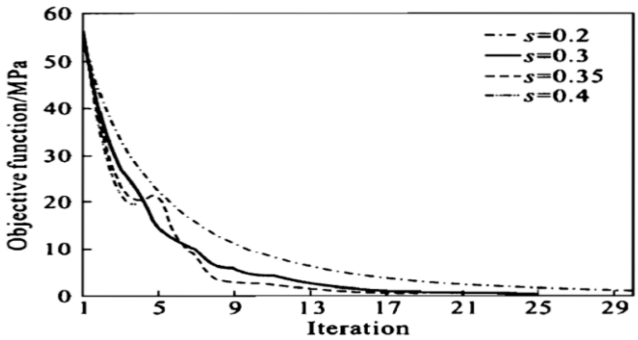


Fig. 4: Graph of Influence du coefficient de convergence

3.2 Selecting the unit type

Based on the analysis of the structure of the car body and its top structure and the force characteristics, according to the setting of the topology optimization design unit properties of ANSYS, the computational capacity of the computer, the convenience of the actual operation of the topology optimization process and the treatment of the optimization results are considered. The SHELL93 [6] unit was selected to simulate the top of the car body for analysis.

3.3 Basic structure

3.3.1 Define and control load conditions: Since the topography does not take into account the skin, it is necessary to convert the skin into a surface load on all sides, and in addition, the top air conditioning load is treated as a concentrated force applied to the response site. As a part of the full load-bearing body, the top is greatly affected by the structural conditions of the model body. For this reason, considering the five load conditions, the displacement of the top and the side-welding joints under various working conditions is analyzed statically. The initial constraints load each solder joint and perform topology optimization. The resulting load distribution is shown in Figure.

3.3.2 Define and control the optimization process: The topology optimization process consists of four parts: defining optimization functions, defining objective functions and constraints, initial optimization processes, and performing topology optimization.

3.3.3 Defining Optimization Functions: ANSYS offers two types of topology optimization, topology optimization based on linear static structural analysis and topology optimization based on structural natural frequency analysis. The topological optimization of the top structure of the car body skeleton is the first type, that is, the structural flexibility is used as the topology optimization function.

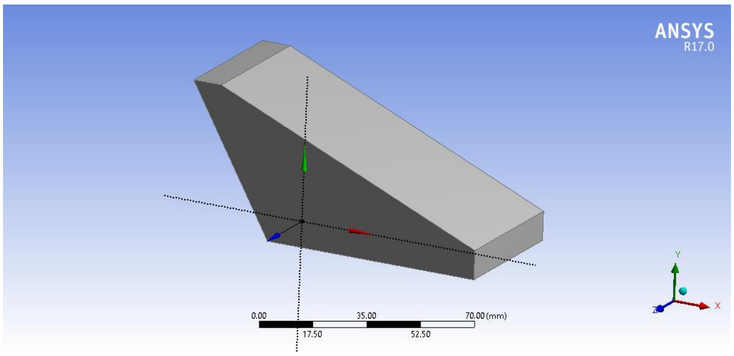


Fig. 5: Top topology optimization model

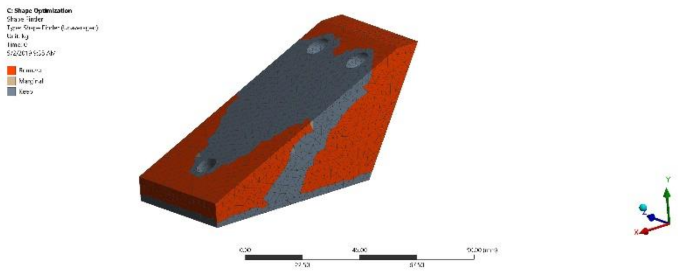


Fig. 6: Top load and boundary condition processing

3.3.4 Defining objective functions and constraints: Before optimization, you must define the objective function of the topology optimization and then define the constraints. ANSYS defines a topology function "VOLUME" (representing the total volume function) for the user by default, so that by adding our custom function F, we get two topological functions.

4. RESULTS

4.1 Optimization Process Initialization

After the optimization problem is defined, the problem must be solved before topology optimization, otherwise, an error message may occur during the first optimization iteration. After clarifying the optimization problem and defining the topology optimization function, you need to choose the appropriate solution method for the optimization calculation process. ANSYS provides users with two optimization algorithms, namely, the selection optimization criterion (OC) or the sequential convex function optimization method (SCP). The principle of the selection method is the choice of OC method with volume as the constraint problem; the SCP method is used for the combination of all legal objective functions and constraints.

4.2 Performing Optimization Iterations

When performing topology optimization, there are two ways to do this: control and execute each iteration, or automate multiple iterations. In this optimization calculation, the structural flexibility under single working condition is first defined as the topology optimization function according to the working condition of the body frame. The function reference name is F; the function F is specified as the objective function of the optimization calculation, and the structure is in the name the total volume VOLUME is specified as a constraint function. After multiple rounds of optimization calculations, the volume removal is selected to be set to 58%; since the function F is defined as the topology optimization objective function in the second step, the function VOLUME is defined. For the constraint condition, according to the principle requirements of ANSYS software, the OC optimization calculation method is selected, the convergence tolerance of the optimization iteration is set to 0.0001 (the default value), and the optimal iteration default maximum number of cycles is 30 times.

4.3 Topology Optimization Results Processing and Analysis

4.3.1 Processing of optimization results: After the topology optimization is completed, the results of ANSYS topology optimization are output as density cloud and node density values. Therefore, the topology optimization results can be processed by two methods, namely, numerical processing of node density values and digital image processing of density cloud images.

It can be seen from Figures below that the lower side of the figure gives the contrast scale of the topology optimization design variable density value, and the position with the density value of 1 corresponds to the red color on the density map, indicating that the structure should be arranged at the time of structural design, and the density value A position of 0.001 corresponds to the blue area on the density map, indicating that there is no need to arrange the structure at the time of structural design. The above are two types of extreme cases, and there are some color areas between the two values. Some necessary processing should be done to classify them into two categories. The figure is the result of image processing of Figure 7, which is processed by ANSYS's own image processing method. The principle of the method is to take a threshold k. When the density cloud image is displayed, when the density value <k is not displayed, only the density cloud map at k is displayed. The threshold of this method is more artificial. Another feasible method is to linearly transform the result density value, and then perform the above classification process on the result of the transformation so that the artificiality can be greatly reduced.

The digital image processing of the density cloud image completely deviates from the practical meaning represented by the color on the image, that is, the problem of density value is not considered, and the image is processed completely. The principle is that the RGB values of each pixel of the image are read first, and then a series of processing such as gradation processing, dot calculation, image equalization, image enhancement, etc. are performed on the color information of each pixel by some digital image processing functions. The resulting map required by the regulations.

The above two methods can deal with the topology optimization results well, but since ANSYS has provided the specific density values of each node after performing topology optimization calculation, the first method is convenient and practical.

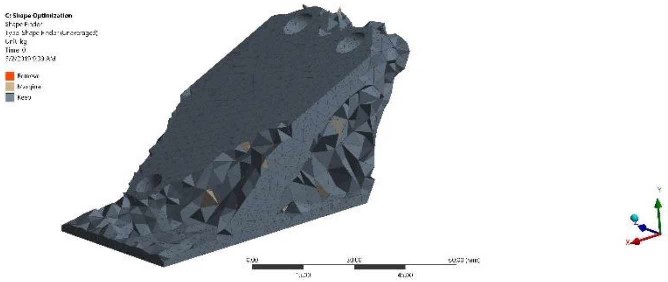


Fig. 7: topology optimization result cloud image output

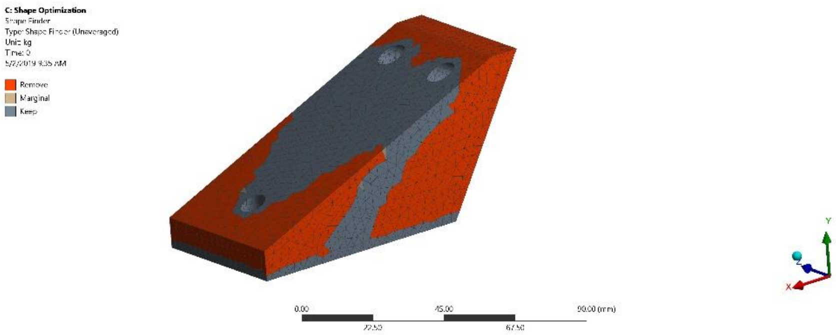
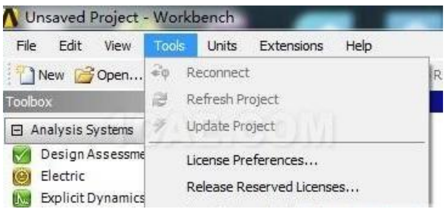


Fig. 8: Topology optimization results in image processing

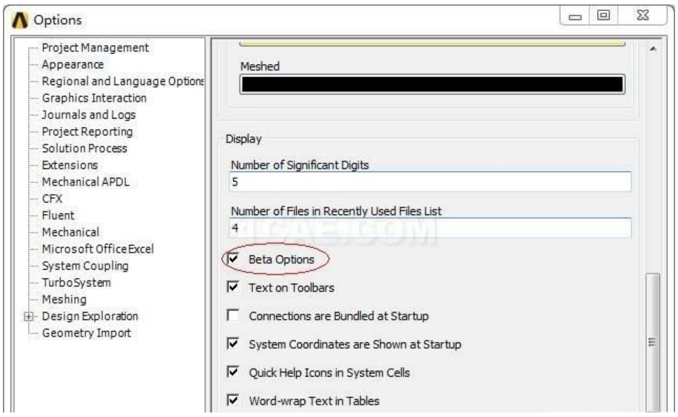
4.3.2 Topology Optimization Results Analysis: In the image processing, the density value in the figure is taken as the material-free distribution area when the density value is less than 0.2, and the area where the density value is greater than 0.2 is the material area. The processed result is shown in Figure 9. The optimal topology form only considers the strength of the structure, and the design of the structure also needs to meet the design requirements of the manufacturing process and assembly relationship. After obtaining the preliminary design parameters of the structure on this basis, further structural parameter optimization is needed. Of course, the structure obtained from the results of the processing is still relatively vague. Although the vertical beam arrangement of the body frame cannot be clearly given, the engineers and technicians can reasonably distribute the position of the beam based on this figure. There are structures to provide reinforcement for reference.

This paper introduces the whole case of ANSYS's optimization analysis of the topological shape of a lifting bar part, the implementation process in ANSYS workbench:

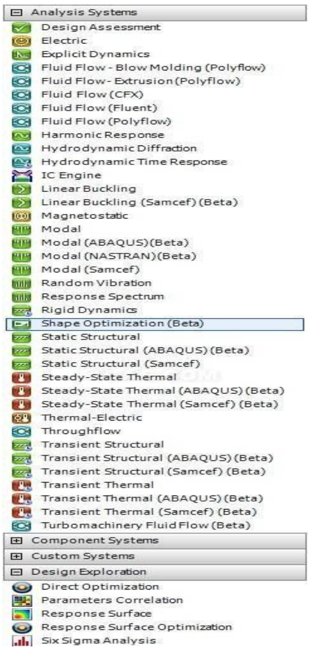
First, start the topology analysis module in ANSYS 17.0
Select Tools-"Options



Select Appearance-"Beta Options



Under Analysis Systems, you can choose Shape Optimization (Beta) analysis.



Second, an example of the analysis of a lifting bar part

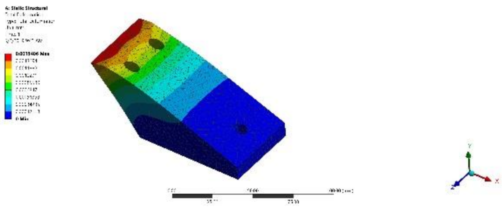
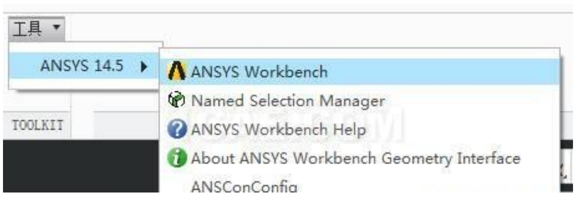


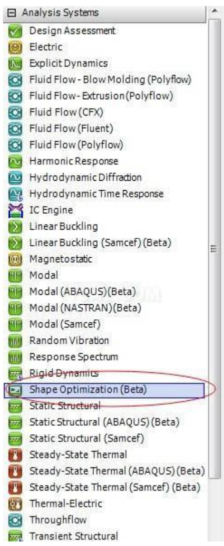
Fig. 9: The material for POM is intended to be optimized using topological analysis.

First, the product should be optimized to be in a regular shape according to the function. As shown below.

Select the tool in the seed, click on ANSYS 17.0, then select ANSYS Workbench to enter the Workbench.



Click on Shape Optimization (Beta) in the Toolbox and hold down the mouse to the right Project Schematic



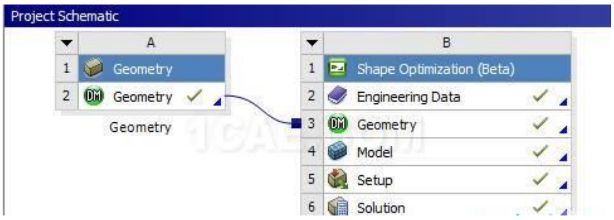


Fig. 10: Double click on Model to enter Mechanically

Set the tension 500N and the fixed constraint as shown.

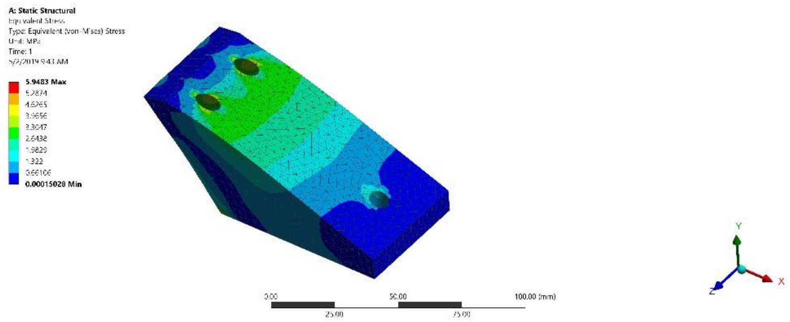
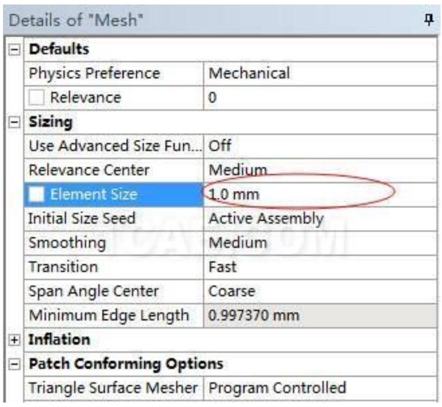


Fig. 11: Stress Calculation of Base model

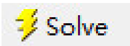
Adjust the mesh density to 1.0mm



Details of "Mesh"	
Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Fun...	Off
Relevance Center	Medium
Element Size	1.0 mm
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	0.997370 mm
Inflation	
Patch Conforming Options	
Triangle Surface Mesher	Program Controlled

Choose the optimization goal to reduce the volume by 58%

Press Solve to start solving.



The calculation is completed.

Click on Shape Finder to see the results

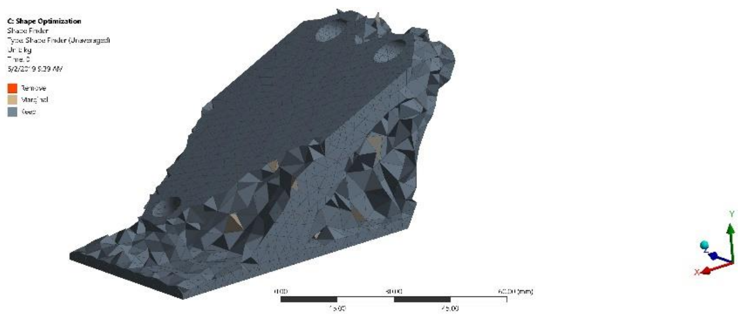


Fig. 12: Choose Capped IsoSurfaces to get a useful location

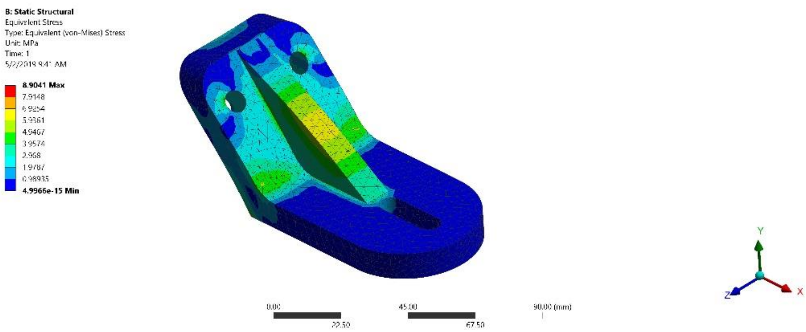


Fig. 13: Stress Calculation of Referring to this shape, the lifting bar was redesigned

As shown below,

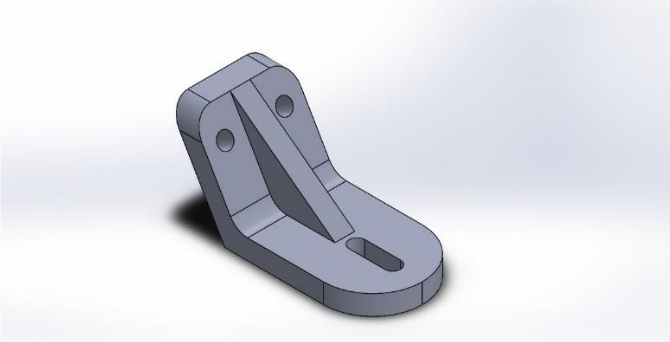


Fig. 14: referring to this shape, the lifting bar was redesigned

4.4 Analysis of the static structure of new and old designs

Summary: For many times, it is difficult to design the most material structure without reference. Through topological analysis, reference can be provided to avoid wasting materials and designing excellent structures at the beginning of the design. Especially for products with relatively high output, reducing the weight by 1g, the cost can be saved hundreds of thousands in the entire product cycle, and this can be completed in half a day. The value of the engineer that will be calculated is reflected here.

Result Optimization and comparison:

	Old	**New**
Weight	999 g	489 g
Total Deformation	0.0035 mm	0.0015 mm
Equivalent (von-Mises) Stress(Max)	8.92 Mpa	5.32 Mpa

5. CONCLUSION

In the present work, In order to reduce the scale of the optimization problem, the static analysis result is used as the constraint condition of the top optimization, which breaks through the limitations of the traditional model body optimization design theory and solves the problem of complex body model and large calculation amount.

Exploring the application of advanced topology optimization technology in model body design, using this technology to carry out local topology optimization analysis of the body frame top, providing design reference for top layout and reinforcement. The application of topology optimization method in the initial design process of automotive structures has very important theoretical and practical value.

At present, the adoption of optimization technology in the development of the bracket body in India is still in its infancy. It can be foreseen that with the further improvement of finite element technology and the introduction of large-scale optimization software, its application potential will be very huge.

6. REFERENCES

[1] Erik Burmany Daniel Elfversonz Peter Hansbox Mats G. Larson{ Karl Larsson “Shape Optimization Using the Cut Finite Element Method” November 18, 2016 AM13-0029, the Swedish Research Council Grants Nos. 2011-4992, 2013-4708,

[2] E. Burman, S. Claus, P. Hansbo, M. G. Larson, and A. Massing. CutFEM: discretizing geometry and partial di erential equations. Internat. J. Numer. Methods Engrg., 104(7):472{501, 2015.

[3] Jrgen S. Dokkeny, Simon W. Funke Y, August Johansson Y, And Stephan Schmidt Z, “SHAPE Optimization Using the Finite Element Method on Multiple Meshes with Nitsche Coupling” Research Council of Norway through a FRIPRO grant, project number 251237.

[4] Nam H. Kim *, Youngmin Chang, “Eulerian shape design sensitivity analysis and optimization with a fixed grid” 6 December 2004 Mech. Engrg. 194 (2005) 3291–3314.

[5] Aman Dutt, “Effect of Mesh Size on Finite Element Analysis of Beam” SSRG International Journal of Mechanical Engineering (SSRG-IJME) – volume 2 Issue 12 – December 2015

[6] M. Such, JR Jiménez-Octavio, A. Butcher, C. Sanchez-Rebollo. Simulation of mobile loads on structures through a mobile finite element meshing. Rev. Int. Method. Numer Calc. Denise. Online, 2014.

[7] Structural Analysis of Tram Car Using Steel and Aluminum Honeycomb by Benyam Adane,2014.

[8] G.W. Jang, Y.Y. Kim, K.K. Choi, Remesh-free shape optimization using the wavelet-Galerkin method, Int. J. Solids Struct. 41 (22–23) (2004) 6465–6483.

[9] J. A. Sethian and A. Wiegmann. Structural boundary design via level set and immersed interface methods. J. Comput. Phys., 163(2):489{528, 2000.

[10] R.R. Salagame, A.D. Belegundu, Distortion, degeneracy, and rezoning in finite elements—a survey, Sadhana—Acad. Proc. Engrg. Sci. 19 (1994) 311–335.

