Journal of Engineering Research and Application www.ijera.com
ISSN : 2248-9622, PP 60-64

RESEARCH ARTICLE OPEN ACCESS

Topology Optimization and Analysis of Crane Hook Model

Thejomurthy M.C[1], D.S Ramakrishn[2]
[1] *Dept. of Mechanical engineering, CIT, Gubbi, 572216, India*
[2] *Dept. of Mechanical engineering, JNNCE, Shivamogga, 577204, India*

ABSTRUCT: *A Numerical optimization method, provide a unique and versatile tool for design optimization. It is defined as the process of finding the conditions that give the maximum or minimum value of a function. To enhance the structure that having a least contribution to the overall stresses or stiffness can be identified. Topology optimization is a mathematical technique that optimizes material layout within a given design space to reduce its weight for a given set of loads, boundary conditions. Hyper works opti-struct solver which uses the density approaches has been used for this purpose. To demonstrate usefulness of the topology optimization approaches, a crane hook has been used to carry out the study. The model of the crane hook with trapezoidal cross section is created with CATIA V5. The finite element analysis reveals the region of low stress and the scope for removal of material. The topology optimization is carried out by Opti-struct solver, the redistribution of material gives the final shape, this gives the weight reduction of 35%. Based on the knowledge of finite element analysis, a simple model with modified topology is also analyzed.*
Key words: *- Opti-struct, Topology optimization, CATIA V5, ANSYS,*

I. INTRODUCTION

The product which is developed needs to improve for a better efficiency of an objective function, i.e. reduce weight, reduce cost etc. this improvement is ongoing process, Design optimization is a means of generating improved designs, a design that possesses some optimal characteristics, such as minimum weight, maximum first natural frequency. The process of design optimization needs formulation of an objective function in terms of design variables. This process is carried out using an optimizer.

Topology optimization is a method of design optimization and it is a process of reducing the weight of the model by better material distribution in a given design domain where the stress level will be minimum by essentially an iterative finite element analysis without affecting strength of the model. Under that crane hook is the one to explain the process of topology optimization, the trapezoidal cross-sectional area of crane hook is created in CATIA V5 software and apply FEA method, stress induced because of deflection in the model. By considering optimization result of the crane hook it clearly shows the stress is minimum in a neutral axis, it goes on increasing the stress value towards the outer fiber of crane hook. The material removal process taken place only in the neutral axis because of minimum stress, using hyper work opti-struct solver software by density method.

The solid model of crane hook in hyper mesh is categorized into two parts. One is designing area and another one is non-designing area. Optimization is carried out on designing area because of applied load is concentrated design area. When load is applied on the crane hook, designing area undergone deflection and stress was induced. But non-designing area is unaffected by the load, it having a minimum stress value it will be removed manually so no need of apply the topology optimization technique. Based on topology optimization result, original design is modified and validated using CAE tools which are equally strong and withstand the constraints of the operation conditions.by considering 2nd iteration of hyper work design, modified the model by knowing analysis result and remove material manually in CATIA V5 software and again it tested in ANSYS software.

II. STRESS ANALYSIS AND NEED FOR TOPOLOGY OPTIMIZATION

2.1Example-crane hook

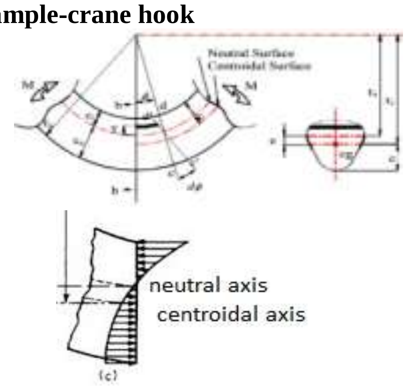

Figure 1. Crane hook

2.2. Theoretical Formulation and numerical

Load acting on crane hook can be calculated as following formulae

Distance of centroid axis, $C_i = \frac{h}{3}\left(\frac{b_i+2b_o}{b_i+b_o}\right)$ (1)

Radius of centroid axis, $r_c = r_i + C_i$ (2)

Radius of neutral axis,

$$r_n = \frac{0.5h(b_i+b_o)}{\left(\frac{b_i r_o - b_o r_i}{h}\right)\ln\left(\frac{r_o}{r_i}\right) - (b_i - b_o)} \quad (3)$$

Distance from neutral axis, $e = r_c - r_n$ (4)

Area of cross section, $A=0.5(b_i + b_o)h$ (5)

Bending moment about centroid axis,

$M_b = Fxl$ (6)

Combined stress at inner fibre

$$\sigma_{ri} = \frac{F}{A} + \left(\frac{M_b C_1}{r_i A_e}\right). \quad (7)$$

b_i=23mm, b_o=(2x5) =10mm, h=35mm, inner radius r_i=19mm, Outer radius r_o=19+35=54mm

TABLE 1: Crane hook terminology

C_i	15.20mm
r_e	34.2mm
r_n	31.42mm
e	2.78mm
A	577.5mm^2
M_b	34.2F
σ_{ri}	0.0188F
F	958N

2.3. FEM Analysis

After developing crane hook model in CATIA V5 is import in to hyper mesh software (opti-struct solver) and all analysis process are performed by applying actual loading and boundary condition. The deformation values are obtained in a FEM analysis.

The data base is used to identified the damage occur in the crane hook due to load condition were the region selected in the crane-hook design. The critical load determination is important to avoid damage on crane hook by comparing simulated and actual deformation result of data base. properties of material and element type is mentioned in the design and the model is import in to hyper work then mesh a created model with fine size of 4mm using a quad element. Load and constraint are applied in to a meshed model, shown in Fig 2, And apply the finite element method, von-mises stress patterns was obtained as shown in the Fig 3.

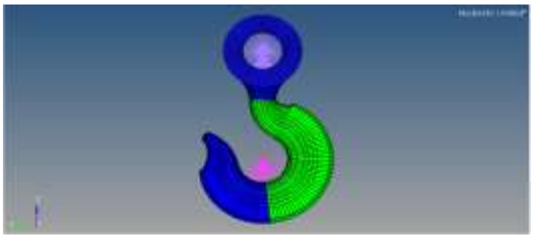

Fig 2. Load and constraint of meshed model

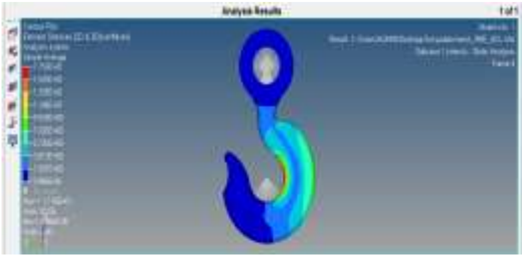

Fig 3. von-mises stress patterns

2.4. Need for Topology Optimization

The von mises stresses due to application of loads. The induced stresses are maximum at inner and outer fiber in the centroidal axis of crane hook and minimum at neutral axis. The model as considered has two regions in that one is designing area, here we optimize the material at this location and other one is non-designing area, no optimization is required. The material at those non-designing locations are safe and hence there is no need of FEM analysis is required.

In the neutral axis, the material which is present will not majorly affected. This material is additional weight to the model, which increases the expense of the model. Hence it can be reducing by topology optimization, there by weight of the crane hook model can be reduced. So, in this topology optimization using hyper works opti-struct solver the additional material that is present in the neutral axis region is majorly removed in such a way that, the net weight of model become lesser without altering or affecting the strength parameters.

III. TOPOLOGY OPTIMIZATION PROCESS

3.1 Hyper Works Opti-Struct

Opti-struct is extraordinarily advanced finite element software for the both design topology optimization and structural analysis. Hyper work Opti-struct is used to design, evaluate and enhance overall performance of machine member, hyper work opti-struct program proposed to a new and better solutions at some point of the whole design system. Effective optimization routines are deeply included with famous evaluation sorts along with linear statics, buckling and frequency analysis - making design optimization clean, automated and accurate.

3.2 Easy Model Setup, Post Processing and Automation

Opti-struct is exceptionally included into the hyper works environment. For that reason, models1can be set up completely in hyper mesh. Animations, contour plots and charts may be generated using the post processing gear in hyper view. By way of the usage of Opti-struct to ensure closed simulation manner chains; moreover, jobs may be easily computerized the use of a powerful automation and statistics control layer to be had in hyper works.

A CAD model of the component is created and meshed with p-solid element of element size 4mm. The model is subjected to a load of 958N as shown in fig 2, and the displacement and stresses at various points of the sheet are found out. The displacement and stress plots are as shown in fig 3.

Hyper mesh is used for modeling and meshing 2D and 3D element configurations. Altair-opti-struct solver is used in analysis and topology optimization. The model is divided in to design area and non-design area. The subjected area which are showed in different color in the fig3. Only design area is subjected for optimization where as non-design area remains unaltered.

3.3 General Design Procedure

The design of crane hook by topology optimization using Hyper Mesh software involves the following steps namely:

- Identification the shape and size of the design area and non-design area of crane hook
- Geometrical shapes can be considered to simplify the meshing of the domain.
- Identifying the Force and constraints applied to the crane hook by choosing a suitable FEA Program (Hyper Mesh software is in the current work)
- Mention all these design area and non-design area specifications to the program to carry out the Finite Element Analysis and to determine stress in each node
- Specifying the percentage of material mass reduction for optimizing the design area based on the convergence criteria.
- Plot the design domain after certain regular intervals like (10, 15, 20, 25, 30iterations) for monitoring the convergence of the crane hook model.
- Stopping the iterations based on the repetition of compliance values.
- The mass in the iterative process is re-distributed in the design area, to improve material stiffness

Converged plot shows the material distribution throughout the area. In general, the regions having higher density (material distribution) are indicated by red colour and the elements in the blue colour are the region that needs to be minimize weight reduction by iterative technique from the original rectangular domain to make the physical compliant amplifier. The output density plot is obtained at the design location by the input load.

The geometry of component performing topology optimization by iteration process in the following shape. material redistribution to achieve final geometry. This design of crushing mechanism has taken 30th iterations to complete the optimization process. The fig 5 to fig 7 illustrate such iterative process carried out in opti-struct solver.

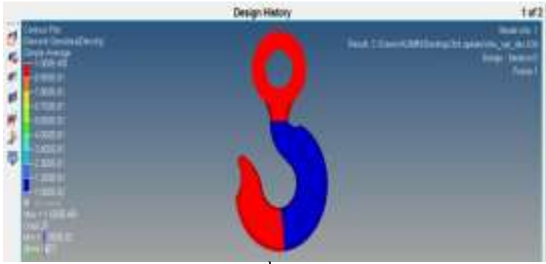

Fig 5. 0th iteration

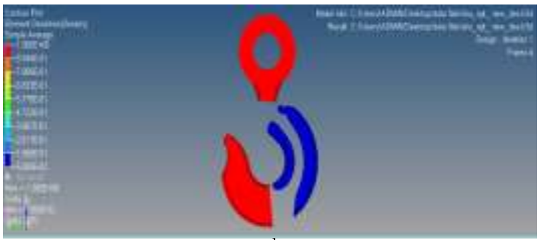

Fig 6. 2nd iteration

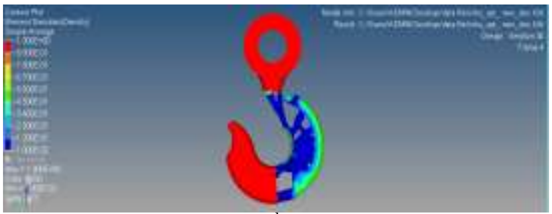

Fig 7. 30th iteration

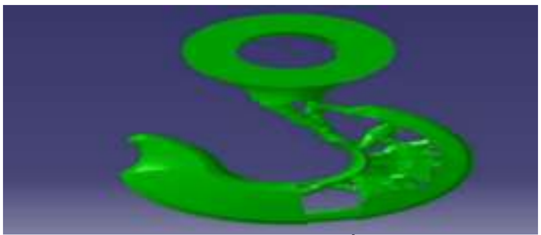

Fig 8. CAD model of 30th iteration

The cad model is extracted hyper mesh for the isovalue of 0.11 for the further analysis. The extracted model is further meshed with p-shell quadratic elements and apply load further and the boundary conditions, same as previous steps. The FE model is subjected to analysis. The displacements and the stress are determined. Based on these details we again apply a topology optimization for simulated model.

IV. FINITE ELEMENT ANALYSIS OF CRANE HOOK BY ANSYS

Now CAD model are introduced with slots by considering 2nd iteration of hyper mesh opti-struct result. The CAD model is created in CATIA V5. There by apply a same load (958N) and boundary, carried out FE analysis to CAD model, the von-mises stress result is obtained with safe design value.

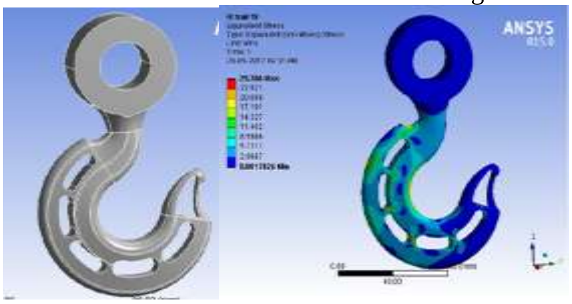

Fig 9. CAD model and ANSYS results

From the observation of optimization result it clear that stresses are minimum at the neutral axis of design domain. The weight optimization taken place only in the neutral axis not in the outer fiber of crane hook. The material present in the neutral axis is some extent not in use, it is present only for retain its shape of the model. material removal taken place on neutral axis in crane hook, this is to compare with the experimental results since single active strain gauge is mounted on the critical section of model to carry the experimentation, this model is named as partially topology optimized model with percentage of material reduction is 29%. From the fig 9, it is clear that. The slot introduced in to the crane hook, the von-mises stress also increased up to safe extent. Hence slots on the hook affect the von-mises stress.

The final model obtained in topology optimization is very complex geometry based on observation, the model is very difficult manufactured by conventional method. The 3D printing technique is an alternate method for manufacturing a complex model. It is digital manufacturing technique helps in producing a component by point to point position, is very useful in preparing a model obtained by topology optimization. Therefore, the 3D printing technique is obtained to preparing a model

V. RESULT AND DISCUSSION

5.1 Weight Reduction

Material removal in the low stress region up to safe design limit. The model obtained in this process, the weight of the 3D printed model is measured by digital weighing machine, weighing machine has a pocket scale, and it is auto calibrated. The 3 models namely before topology optimizes model, partially topology optimized model by FE analysis and after topology optimized model by opti-struct solver have a 148.54grams, 105.77grams and 95.43grams respectively. Redistribution of material there will be significant reduction in weight of the topologically optimized model compared with before optimized model. The percentage of weight reduction, obtained 3D printed model has shown below.

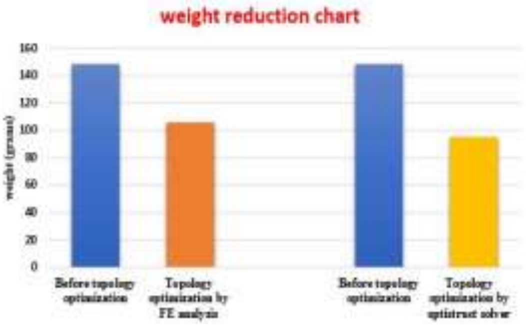

Fig 10. weight reduction chart

Fig 10, compared weight reduction chart, the weight of Topology optimized model by opti-struct solver is 95.43grams and weight of Topology optimized model by FE analysis 105 grams, both have a lesser weight compare with before topology optimized model.

Table 2. Tabulation of % of Weight Reduction

BO Model (grams)	TO Model by FE Analysis (grams)	TO Model by Opti-Struct Solver (grams)	% of Weight Reduction
148.54	105.77	-	28.94
148.54	_	95.43	35.75

5.2 Maximum Stress

Table 3. Tabulation of % Increase in Stress by FE analysis

Load	Stress (MPa) BO Model	Stress(MPa) TO Model by FE Analysis	% of Increase in Stress
958 N	17.16	25.78	33.43
161 N	2.769	5.517	49.8

Fig 10. Stress chart

VI. CONCLUSION

During this dissertation work, topology optimization has been carried out on a machine member to optimize its shape and thus reduce its weight. Hyper works opti-struct solver which uses the density approaches has been used for this purpose. To demonstrate usefulness of the topology optimization approaches, a crane hook has been used to carry out the study.

The model of the crane hook with trapezoidal cross section is created with CATIA V5. The finite element analysis reveals the region of low stress and the scope for removal of material. The topology optimization is carried out by Opti-struct solver. The 30th iteration gives the final shape of the crane hook. The redistribution of material gives the final shape, this gives the weight reduction of 35%. Based on the knowledge of finite element analysis, a simple model with modified topology is also analyzed.

REFERENCES

[1]. Topology optimization of compliant mechanisms with strength considerations, A. Saxena and G. K. Ananth Suresh, Mechanics of structures and mechanisms, 29(2), 199-221(2001).

[2]. Optimization for engineering design, Kalyan Moy deb, prentice hall of India.

[3]. Design of compliant mechanisms using topology optimization, ole Sigmund, mechanics for structures and mechanisms, 25, (4) 493-524(1997).