ISSN 2395-1621

Design, Structural Analysis and Optimization of Crane Hook

#1Prashant R. Mali, #2Dr.K.K.Dhande

1prashantmali0604@gmail.com
2kkdhande@gmail.com

#1P.G. student Mechanical Department, DYPIET Pimpri, Pune,
#2Professor and Head of Mechanical Engg. Department, DYPIET, Pimpri,Pune

ABSTRACT

The structural strength is the key index to response the load bearing ability of the elevating equipment. Crane hook is a curved beam and is widely used for industrial and construction work site for lifting loads. Analytical experimental and numerical methods were used by various researchers to study stress pattern of crane hook in its loaded condition. The stress induced in crane hook must be analyzed in order to reduce failure for safety point of view. Study the different design parameter & stress pattern of crane hook in its loaded condition for different cross section. The stress concentration factors are used in strength and durability evaluation of structure and machine element.

***Keywords*— Crane hook, failure, FEM, Stress Analysis**

ARTICLE INFO

Article History

Received :18th November 2015

Received in revised form :

19th November 2015

Accepted : 21st November , 2015

Published online :

22nd November 2015

I. INTRODUCTION

Crane hooks are the components which are generally used to elevate the heavy load in industries and constructional sites. Recently, excavators having a crane-hook are widely used in construction works site. One reason is that such an excavator is convenient since they can perform the conventional digging tasks as well as the suspension works. Identification of the reason of the damage is one of the key points toward the safety improvement. If a crack is developed in the crane hook, mainly at stress concentration areas, it can cause fracture of the hook and lead to serious accidents. Such an important component in an industry must be manufactured and designed in a way so as to deliver maximum performance without failure. Thus, the aim of the project is to study the stress distribution pattern of a crane hook using finite element method. Optimizations were carried out under static load conditions. The geometry and manufacturing constraints are also considered during optimization process. The optimized geometry is analyzed using FEA Tool. The stress concentration factors are widely used in strength and durability evaluation of structures and machine elements.

LITERATURE REVIEW

AjeetBergaley et.al [1] (2013), conducted study of crane hook, these are highly significant component used for lifting the load with the help of chain or links. In the present paper a crane hook is purchased from the local market for Finite element analysis. The hook was tested on the UTM machine in tension to locate the area having maximum stress and to locate the yield point. The model of hook is prepared in CAE software having dimension and material similar to the crane hook which was purchased from the market. The results obtained were compared with theoretical analysis. Then crosssection in which minimum stress induced for given load was modified through FEM.SantoshSahuet.al [2] (2012), his paper analyzes crane hook of trapezoidal section is modeled in CATIA V5R20, then 2 ton load equivalent to 19620 N on it is applied. The location of maximum stress produced within the member is located and identified using Finite Element Method (FEM). Further Design of Experiment (DOE) is applied by varying the length of two parallel sides of the Trapezoidal Hook and its effects are studied on the basis of Stress , Mass, Displacement and Energy stored within the hook.

Govind Narayan Sahuet.al [3] (2013), this work has been carried out on one of the major crane hook carrying a larger load comparatively. The cad model of the crane hook is initially prepared with the help of existing drawings. It is then followed by implementation of modified cross section of hook in the static structural analysis workbench of catiav5. These results lead us to the determination of stress and deflections in the existing model. In order to reach the most optimum dimensions several models in the form of different dimensions of hook were tested and the most optimum dimension was selected. The selection was based on the satisfaction of several factors in the form of load carrying capacity, stress induced and deflection.

Mr. A. Gopichandet.al [4] (2013), investigated, optimization of design parameters is carried out using Taguchi method, total three parameters are considered with mixed levels and L16 orthogonal array is generated .The optimum combination of input parameters for minimum Von-mises stresses are determined.

Chetan N. Benkaret.al [5] (2014), study the stress pattern of crane hook in its loaded condition, a solid model of crane hook is prepared with the help of ANSYS 14 workbench. Real time pattern of stress concentration in 3D model of crane hook is obtained. Finite Element Analyses have been performed on various models of crane hook having triangular, rectangular, circular and trapezoidal cross sections.

T. Muromakiet.al [6],develop an estimation method of loading conditions based on images of failed structures and an FEM analysis model. Preparing a database that consists of deformation data of the structure corresponding to various load conditions, our system is able to estimate the load conditions that caused structure failure based on the processed images of failed structure samples. Adopting elasto-plastic model of the structure, the magnitude of the load having caused the failure is also estimated in addition to the position and orientation of the critical load. We adopt the EM algorithm to obtain the distribution of the critical load. An optimal design problem that takes account of the distribution of the estimated critical load condition is formulated as a minimization problem with a multi-objective function; the stiffness and the structural weight are also adopted as the evaluation items that make up the objective function. The particle swarm optimization (PSO) is adopted as the optimization algorithm. The approach is applied to crane-hook. The result of estimated critical load distribution and the optimal design based on the load distribution are demonstrated.

Nishantsoni [7], this study is to optimize a low carbon steel crane-hook for its weight. Finite element analysis is used for shape optimization of crane-hook as well as for validation of final geometry. This process is performed in order to have minimum weight under design constraints. These optimizations were carried out under static load conditions. The geometry and manufacturing constraints are also considered during optimization process. The optimized geometry is analyzed under gas force using FEA Tool. The study results in optimized crane hook that is 14% lighter then original crane-hook.

Nishantsoni [8] the objective of this study is to optimize a low carbon steel crane-hook for its weight. Finite element analysis is used for shape optimization of crane-hook as well as for validation of final geometry. This process is performed in order to have minimum weight under design constraints. These optimizations were carried out under static load conditions. The geometry and manufacturing constraints are also considered during optimization process. The optimized geometry is analyzed under gas force using FEA Tool. The study results in optimized crane hook that is 14% lighter then original crane-hook.

III. FAILURE OF CRANE HOOKS

Strain aging embrittlement [5] due to continuous loading and unloading changes the microstructure. Bending stresses combined with tensile stresses, weakening of hook due to wear, plastic deformation due to overloading, and excessive thermal stresses are some of the other reasons for failure. Hence continuous use of crane hooks may increase the magnitude of these stresses and eventually result in failure of the hook. All the above mentioned failures may be prevented if the stress concentration areas are well predicted and some design modification to reduce the stresses in these areas. [5]

IV.METHODS FOR MEASUREMENT OF STRESS ON CRANE HOOK

The principal measurement and analysis procedure for finding the stress in crane hook is given below-

A. *FEM ANALYSIS*
B. *THEORETICAL ANALYSIS*

A. *FEM ANALYSIS*

FEM model of the crane-hook referring to one of its actual designs was constructed. A database was prepared based on the FEM model; it was constructed as a collection of a number of various possible load conditions and the corresponding deformation values, obtained as the results of the FEM analysis. The database was used to identify the load conditions that were fatal to those damaged crane-hooks. Some of the feature points were selected on the crane-hook design; the deformation of a damaged crane-hook can be then obtained based on the feature points detected by means of the image processing. The critical load condition of the damaged crane-hook was calculated by comparing the obtained actual deformation and the simulated deformation values in the database. [2]Profile of the model is drawn in CATIA R20.Material properties and element type are fed. And the model is Import in ANSYS 12 then mesh using smart size option with the global size of the element as 3. Loading and constraint are applied to the meshed model as shown in the Figure 1 and the finite element model is then solved. Principal stress and von-mises stress patterns are thus obtained as shown in Figure 2.

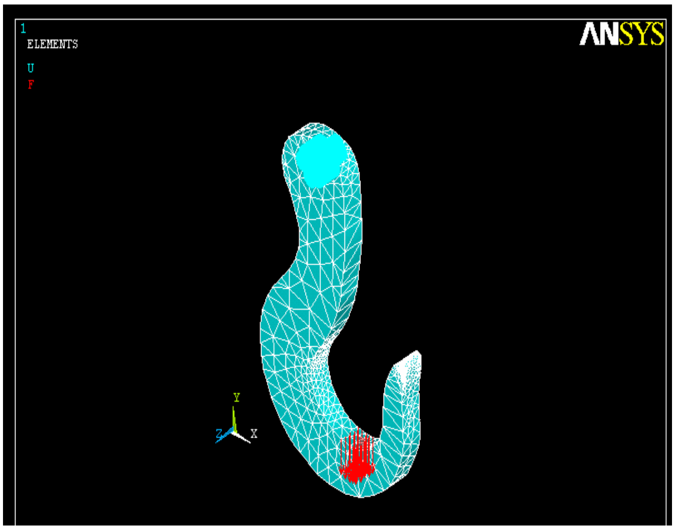


Fig 1. Meshed constraint model

Fig.2 Principal stresses in the model

B. THEORETICAL ANALYSIS

For initially straight beams the simple bending formula is applicable and the neutral axis coincides with the centroidalaxis. A simple flexural formula may be used for curved beams for which the radius of curvature is more than five times the beam depth. For deeply curved beams, the neutral and centroidal axes are no longer coinciding and the simple bending formula is not applicable. [5]

1) Curved Beam

A beam in which the neutral axis in the unloaded condition is curved instead of straight or if the beam is originally curved before applying the bending moment, are termed as "Curved Beams".

2) Straight Beam

A beam is a straight structural member subjected to a system of external forces acting at right angles to its axis.

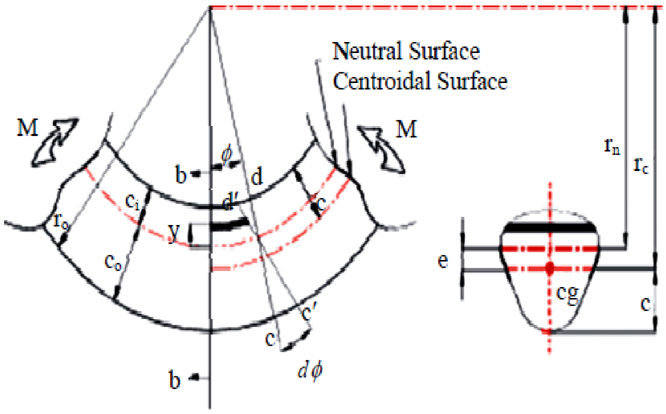


Fig.3 Curved Beam with Cross Section Area

The strain is clearly zero at the neutral axis and is maximum at outer radius of the beam. Using the relationship of stress/strain = E, the normal stress is simply.

$$\sigma = E\epsilon = E(r - r_n)\frac{d\emptyset}{r\emptyset} \ldots\ldots\ldots\ldots\ldots\ldots (1)$$

Maximum stress occurs at either inner or outer surface of hook

Stress at inner surface

$$\sigma_i = \frac{M c_i}{A e r_i} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (2)$$

Stress at Outer surface

$$\sigma_i = \frac{M c_o}{A e r_o} \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots (3)$$

$$(\text{Where } e = r_c - r_n)$$

The curved beam flexure formula is reasonable for beams with ratio of curvature to beam depth $(\frac{r_c}{h})$ >5(Rectangular section) as this ratio in- creases, the difference between the maximum stress calculated by curved beam formula and the normal beam formula reduces.In case of crane hooks, the bending moment is due to forces acting on one side of the section under consideration. For calculations the area of cross section is assumed to be trapezoidal. Values of stresses as shown in Figure 3 are found out at the B-B section as it is the section where maximum stress is induced.

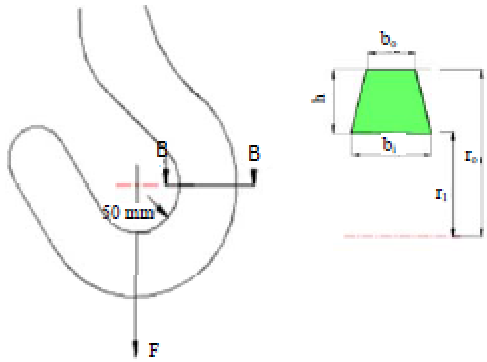


Fig.4 Analysis of hook

V. RESULTS AND VALIDATION

The induced stresses obtained from FEA software are compared with results obtained from analytical calculations.

Maximum value obtained analytically = 12.35N/mm^2

Maximum value obtained from ANSYS = 13.372 N/mm^2

Possible reasons for variation might be the assumptions that

1) Loading is considered as point loading in analytical calculation while it is taken on a bunch of nodes in ANSYS.

2) Cross sectional area is assumed to be trapezoidal and

3) Plane sections remain plane after deformation.

Using analytical calculations the stress variation yields the results.

Maximum tensile stress is 158.72N/mm^2 on the inner surface of the crane hook and on the outer surface of the hook; compressive stress is 44.23 N/mm^2. From the fig.5 stress goes on decreasing from a max value (Tensile stress) to zero and again increases from zero to a certain value (Compressive stress).

Innermost point of section B-B.

Max stress by ANSYS=132.46 N/mm^2; Max stress analytically= 158.72 N/mm^2.

Outermost point of section B-B.

Stress by ANSYS= 45.728 N/mm^2 Stress analytically = 44.23 N/mm^2.

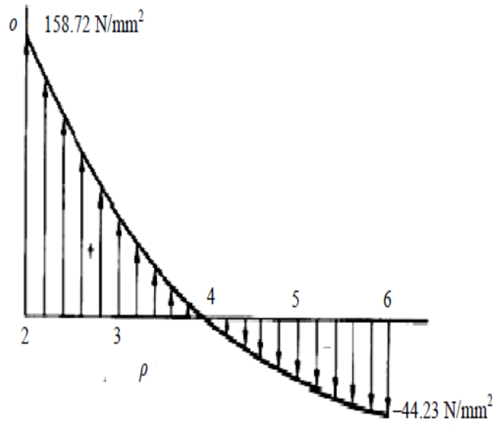


Fig 5.Variation of Stress with depth for the actual model

Here the results of ANSYS are validates with the results of theoretical. After comparing with these two results it is clear that the ANSYS and theoretical results match.

Reasons for variation: Various assumptions made during the analytical calculations (discussed earlier). Profile of the hook obtained from CAD Modeling software may not be exactly the same as actual one.

VI. OPTIMIZATION FOR CRANE HOOK

A. FOR REDUCING FAILURE

Manufacturing process: Forging is preferred to casting as the crane hooks produced from forging are much stronger than that produced by casting. The reason been in casting the molten metal when solidifies, it has some residual stresses due to non-uniform solidification. Thus casted crane hooks cannot bear high tensile loads.

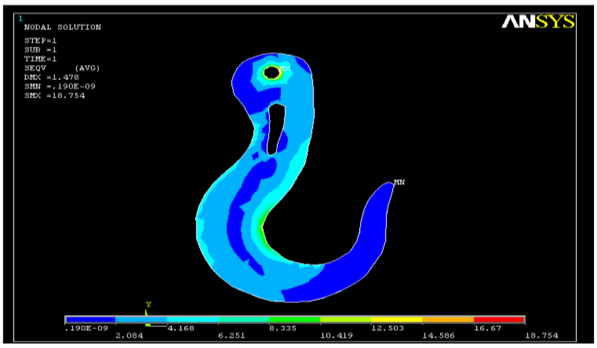
B. GRAIN SIZE

The stress bearing capacity depends on the homogeneity of the material i.e. the relative sizes of the grains in various areas of the component. Smaller the grain size better is the stress bearing capacity. So grain refinement process such as normalizing is advisable after forging.

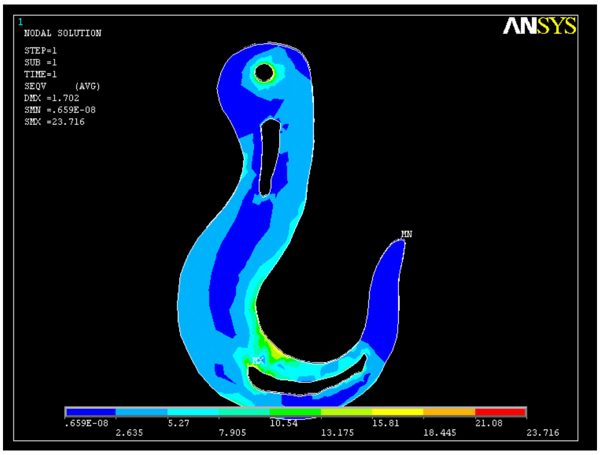
Removal of metal from the hook body is not feasible as it increases the amount of stresses in the hook. This is validated by the following illustration.

It is clear from the fig. 6(a) that removal of a small amount of material from minimum stress concentration areas increases the stress slightly though reducing the cost of material.

The fig.6 (b) validates the fact that when considerable amount of material is removed stresses increase by a good enough margin which is not at all feasible.



(a)



(b)

Fig.6 a and b is Hook with material removed.

C. DESIGN IMPROVEMENT

From the stress analysis it is observed that the cross section of max stress area. If the area on the inner side of the hook at the portion of max stress is widened then the stresses will get reduced. Analytically if the thickness is increased by 3 mm, stresses are reduced by 17%. Thus the design can be modified by increasing the thickness on the inner curvature so that the chances of failure are reduced considerably.

VII.CONCLUSION

The stress concentration factors are used in strength and durability evaluation of machine elements. In order to minimize the failure of the crane hook, the stress induced in crane hook must be studied. The study of Finite Element Method (FEM) is one of the most effective and powerful method for the stress analysis of the crane hook. By knowing the results of ANSYS it is clear that by removing the considerable amount of material stresses increase by a good enough margins which are not at all feasible. If the area on the inner side of the hook at the portion of max stress is widened then the stresses will get reduced.Analytically if the thickness is increased by 3 mm, stresses are reduced by 17%. Increasing the thickness on the inner curvature so that the chances of failure are reduced considerably.

ACKNOWLEDGMENT

The author would like to thank for the valuable suggestions and guidance rendered by Prof. N. I. Jamadar. And special thanks to Dr.K.K.Dhande (HOD-Mechanical Engineering) and Dr. R. K. Jain (Principal) for their extreme support to complete this assignment**.**

REFERENCES

[1] AjeetBergaley et.al "Structural Analysis of Crane Hook Using Finite Element Method" 2319-6386, Volume-1, Issue-10, September 2013 (IJISME)

[2] SantoshSahuet.al "Study of Crane Hook Having Trapezoidal Section by Finite Element Method & Design of Experiments" Vol.2, Issue.4, July-Aug 2012 pp-2779-2781 International Journal of Modern Engineering Research.

[3] Govind Narayan Sahuet.al "Design and Stress Analysis of various cross section of Hook" Vol. 3, Issue. 4, Jul - Aug. 2013 pp-2187-2189

International Journal of Modern Engineering Research.

[4] Mr. A. Gopichand et.al "Optimization of design parameters for crane hook using taguchi method" Vol. 2, Issue 12, December 2013 International Journal of Innovative Research in Science, Engineering and Technology

[5] Chetan N. Benkar (M.E. Scholar) et.al "Finite element stress analysis of crane hook with different cross sections" Volume 1, Issue 9, May-2014 International Journal For Technological Research In Engineering

[6] T. Muromaki et.al "Estimating load condition having caused structure failure and an optimal design taking account of the estimated result"

[7] Nishantsoni "Crane-hook shape optimization and thermal analysis using finite element tool" IJAIR.

[8] E. Narvydas et.al "Circumferential stress concentration factors at the asymmetric shallow notches of the lifting hooks of trapezoidal cross-section", ISSN 1392 - 1207. MECHANIKA. 2012 Volume 18(2): 152-157

[9] Nishantsoni "Crane-hook shape optimization and thermal analysis using finite element tool" IJAIR.

