Experimental Stress Analysis and Optimization of Crane Hook

Snehal A. Dhengle and Prof. Sachin K. Dahake,
MET's Institute of Engg. Bhujbal Knowledge City, Nashik.

ABSTRACT

Lifting mechanism for carrying loads includes hook as the basic structure. Ancient techniques used for designing hooks results into overdesigned structure. Base design and analysis of hook done using FEA and experimental stress analysis techniques. Concept hook design achieved using software like CAD (CATIA). Optimization will be achieved through FEA package like Ansys / Optistruct/ Hyper mesh. Strain gauging installed at high strain location for measuring strain to quantify then within Elastic/yield limit. Machining/EDM used for achieving optimized hook model dimensions. Comparative analysis done between FEA and experimental stress analysis final results. Conclusion and future scope suggested.

Keywords— *ANSYS, Crane hook, Strain gauging, FEA.*

---

1. INTRODUCTION

Crane hooks system operation are extremely liable parts and are always subjected to failure cause OF ACCUMULATION of enormous quantity of stresses which may eventually cause its failure. Crane hook are usually system elevate the significant number of loads, significant machineries, man, material in industries and constructional sites. Recently, excavators having a crane-hook are wide utilized in construction works web site observation. The structure strength is that the necessary characteristic to retort the load bearing capability of the elevating instrumentation. Crane hook could be a curved bar accustomed raise the masses within the cranes. so as to cut back the structure failure within the crane hook, induced stresses are analyzed. Fatigue injury is that the initiation of crack because of unsteady loads. it's caused due to stress levels that are not sufficient to cause injury during a single application. it's an extremely accountable and necessary element used for industrial applications. Winkler batch theory is employed to calculate the theoretical stress. For the straight beams, the censorial axis coincides with neutral axis of the cross section and also the stress distribution within the beam is liner. however just in case of curved beams observation, the neutral axis of the cross-sectional is slightly shifted towards the middle of curvature of the beam inflicting a non-linear distribution of stress. The potential hazards concerned in victimization lifting devices cannot be overcome exclusively by mechanical means operation. The operator should be alert, competent system, and trained within the safe operation of lifters. it's conjointly essential for the operator to exercise intelligence, care and customary sense in anticipating operation the motions that will occur because the load is raised. Hook is used to grab and raise the masses. it's a hoisting fixture designed to have interaction a hoop or link of a lifting chain or the pin of a shackle or cable socket. Crane hooks with trapezoidal, rectangular, circular of triangular cross sections is usually used. The hooks should be designed to deliver highest performance while not failure

1.1 Failure of Crane Hook

Due to continuous operating of crane hook nanostructure of crane hook are changes and a few problems like weakening of hook due to wear, tensile stresses, plastic deformation, overloading and excessive thermal stresses these are another reason of failure Strain aging embrittlement cause of continuous loading and unloading changes the microstructure. Bending stresses combined with tensile stresses, weakening of hook cause of wear, plastic deformation, overloading, and excessive thermal stresses are a number of the opposite reasons for failure. thus, continuous use of crane hooks might increase the magnitude of those stresses and eventually lead to failure of the hook.

2.LITERATURE REVIEW

B. Phani Shankar et. al [1] This paper involves modeling and analysis of Dual Hook Joint. Modeling was done by advanced design software CREO 1.0 (Feature based parametric bi-directional software). Dual Hook Joint was modeled with the use of the feature of this software. Every components of this dual hook joint are designed individually within the part module and assembled within the assembly module. In the assembly module there are 2 ways to assemble the Dual hook joint parts, one is bottom up technique and other one is top down technique and by following bottom up assembly technique the modeling of the dual crane hook was completed. Finally, analysis was done on the ANSYS 14.5 that works on the outstanding FEM technique, as a result of work done on a graphical user interface it had been known as FEA. This software helps to carry out stress analysis and modal analysis with different materials at different loading conditions and therefore the best material is going to be chosen.

Reddy, M. et. al [2] In this paper authors discuss about the cross section of hook as Crane hooks area unit very liable elements that are usually used for industrial applications. Failure of a crane hook all depends on major factors i.e. dimension and material. throughout this paper load carrying capability is studied by variable the cross sections. the chosen sections are Circular, Trapezoidal and I-section. the area remains constant whereas dynamic the dimensions for the three different section. Crane hook is sculptured with the use of SOLIDWORKS software package. the strain analysis is completed with the use of ANSYS 14.0 workbench tutorial version. it's found that I cross section yields minimum stresses at the given load of 6ton for constant cross section area compared to other cross sections. the strain distribution pattern is verified for its correctness on model of crane hook victimization Winkler-Bach theory for curved beams.

Agrawal S. et.al [3] Crane hook may be a mechanical part used for material handing or transfer, observes stresses, induced once different forms of load applied. From the protection purpose the Crane hook fracture should be prevented because of crack fracture developed caused in the main at stress concentration areas. Stress on crane hook depends upon numbers of geometric variables as well as material properties. during this analysis the material properties of hook is constant throughout the analysis and stress is to be reduced by changing the values of different geometric parameters. When optimizing the cross section of crane hook the approach turned towards the material saving throughout manufacturing of crane hook. For material saving the most stress region is to be known by victimization FEM analysis and so material is removed by considering the maximum bending stress at failure point.

Shrivastava, P. et. al [4] Authors talks about Crane hook is extremely major factor used for lifting the load with support of chain or links. Crane hooks are extremely important elements and are perpetually subjected to failure cause of the quantity of stresses concentration which might ultimately result in its failure. to reduce the failure of crane hook, the strain induced in it should be studied. A crane is subjected to continuous loading and unloading process. this could cause fatigue failure of constellation hook. The review of previous publications alter to conclude that elements with complicated geometry as crane hooks need a lot of extensive investigation and also the fact that few articles are revealed up to now relating to stress analysis of this curved beam (crane hook).

Rajaroy M., et.al [5] Crane hook is important component used for lifting the load with the help of equipment like chain or wire ropes. Crane hooks subjected to bending stresses which leads to the failure of crane hook. To minimize the failure of crane hook, the stress induced in it should be well observed. A crane is subjected to load fluctuations. This may result structural failure of the crane hook. In the present work, an attempt has been made by considering four different types of cross sections of crane hooks and by using curved beam theory it is designed theoretically. CATIA software is used for modeling the crane hook and ANSYS software is used to find out the stresses all over the hook. As a conclusion, the results obtained from ANSYS and theoretical calculations are compared.

Suryaprakash K. et.al [6] Hoists and cranes have a necessary role in several sectors of assorted industries just like the logistic, some transport vehicles, in ship building industries, some transport vehicles, steel industries., for lifting loads and carry them. many varieties of hooks having totally different ratings and sizes are used according with the desired objective. the categories of hooks used are depends on type of hoist or the type of crane to that they're connected. This paper aims at building a 245 KN or 25-ton resistant block hook with a hook created from lead alloy within the market, Alloy 1.2367 (X38CrMoV5-3), that have high tensile strength, compression strength, and yield strength compared to the presently used alloy in the manufacturing of crane hooks. the design of block hook and its parts is completed in Solidworks part design and the individual components are assembled within the Solidworks assembly. The analysis on the hook made up of the Alloy

1.2367(X38CrMoV5-3) is administered in Solidworks Simulation. the pictures are rendered using Solidworks Visualize.

Material Handling Equipment [7] Material handling (MH) involves "short-distance movement that typically takes place at intervals the scope of a building like a plant and between a transportation agency and building." It can be used to manage "Time and Place Utility" through the handling, storage, and management of material, as distinct from producing (i.e., fabrication and assembly operations), that creates "form utility" by varying shape, form, and makeup of material.

3.PROBLEM STATEMENT

Lifting mechanism for carrying loads includes hook as the base structure. Traditional techniques used for designing hooks results into overdesigned structure. Base design and analysis of hook done using FEA and experimental stress analysis techniques.

OBJECTIVES & METHODODLOGY

- To achieve optimized design having less mass as compared to traditional design.
- CAD modeling of existing Model using CATIA
- Finite Element discretization using Ansys applying loads at boundary conditions
- Experimental stress analysis of Model using Strain gauging & lifting load applied through UTM
- Comparative analysis between FEA & Experimental results.

METHODOLOGY

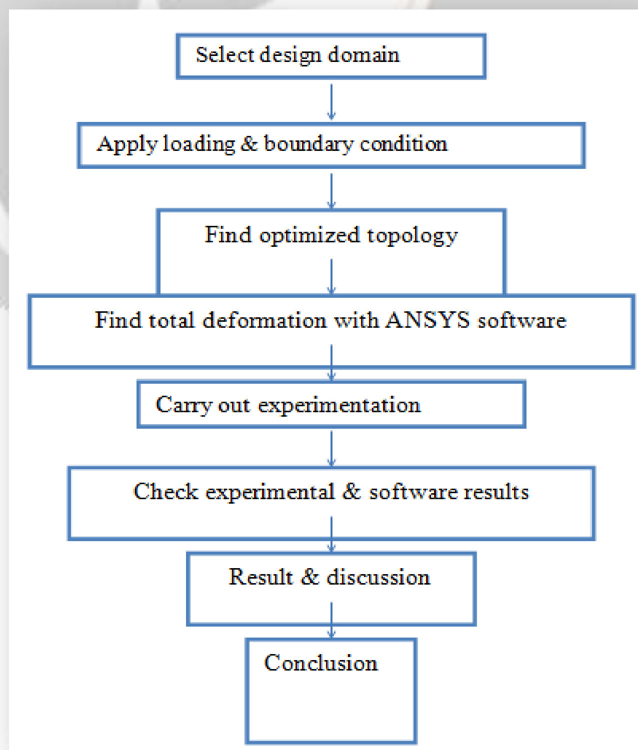


Fig.1 Flowchart for the Methodology

FINITE ELEMENT ANALYSIS

The Finite part method is numerical approximation method, during which Complicated geometry structure divided into many numbers of very small parts and these small parts are referred to as finite elements. These tiny parts connected to every different by suggests that of small points referred to as nodes. As finite element method uses matrix algebra to resolve the simultaneous equation, therefore it's referred to as structural analysis and it’s turning into primary analysis tool for designers and analysts.

The 3 basic FEA phases are
1) Pre-processing
2) process or solution
3)Post process
Modal analysis is carried out to get the natural frequencies and mode shapes of a structure. Natural frequency and in the design of a structure for dynamic loading conditions mode shapes are very important parameters.
1) FEA Pre Processing*:*
The pre-processing of the flat metal sheet is down for the aim of the dividing the problem into nodes and elements, developing equation for a element, and applying boundary conditions, initial conditions for applying forces. the basic information needed for the pre-processing stage of the flat metal sheet is as follows,
• Material properties: The Structural steel (S250) which is used in the study of crane hook has following material properties which are taken from material library of the FEA package.

TABLE 1

Material Properties

Properties	Structural Steel (S250)
Yield Strength	250 N/mm^2
Young’s Modulus	2e+11 Pa
Poisson's Ratio	0.3
Density	7850Kg/m^3
Coefficient of Thermal Expansion	1.2e-0.5 C-^{1}
Bulk Modulus	1.6667e+11 Pa
Shear Modulus	7.6923e+10 Pa

TABLE 2

Chemical composition (ASTM A36/A36M)

Mn	Si	C	P	S	Other
0- 1.6	0.04-0.05	0.22-0.26	0.04-0.05	0.05	-

2) Finite element method (FEM) or Solution:
it's a numerical technique for getting approximate solutions to boundary value problem for partial differential equation it's also mentioned as finite element analysis (FEA). FEM subdivides an oversized problem into the smaller, parts, known as finite element. the equations that are model these finite elements are then assembled into a bigger system of equations that models the complete problem. FEM then the uses variation method from calculus of variations to approximate final solution by minimizing an associated error operate or function.

Constraint: The nodes round the side panel holes have a rigid element connecting them to the center of the opening that has of its degree of freedom mounted. The element that helps to keep the side panel fix and vehicle is also fixed and used as a rigid element. The maximum and minimum are set, along with alternative mesh parameters like element type and material. the chosen object is as prepared for further analysis.

3)Post Processing*:*

The satisfactoriness of the design of the metal sheet has to be considered from the results of the analysis.
Model acceptance criteria: Natural frequency of the structure should to be increase with reinforcement of elastic material

4. MODELING IN CATIA V5

4.1 Original crane hook

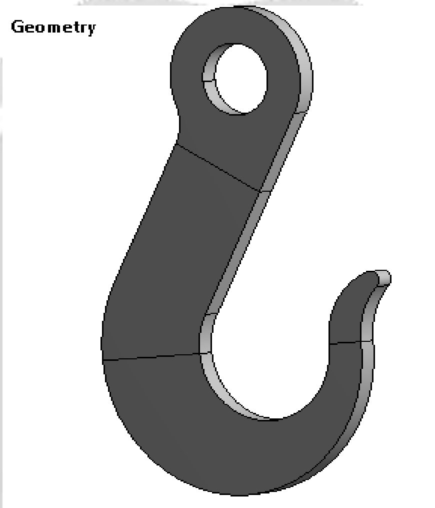


Fig.2 Catia of crane hook

Topology Optimization of Crane Hook

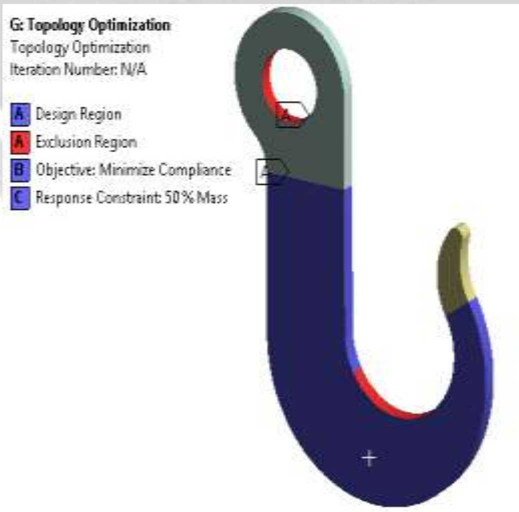


Fig.3 Topology Optimization

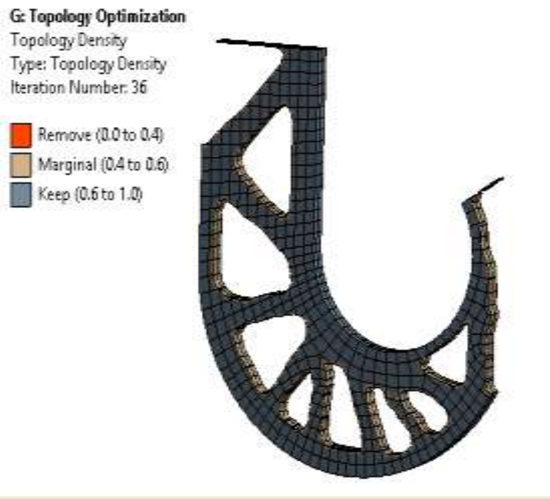


Fig.4 Topology Density Tracker

Elliptical Hole made as per Topology Density Tracker of Crane hook. The weight optimization taken place only in the neutral axis and absolutely 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

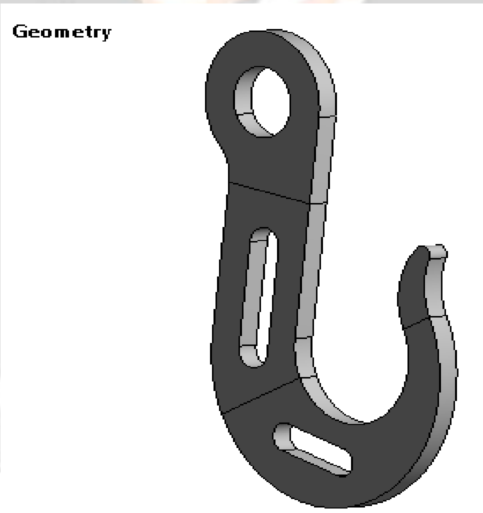
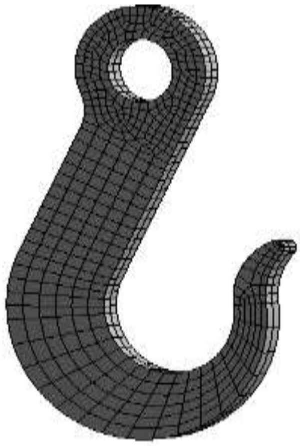


Fig. 5 CATIA model of Optimized Crane hook

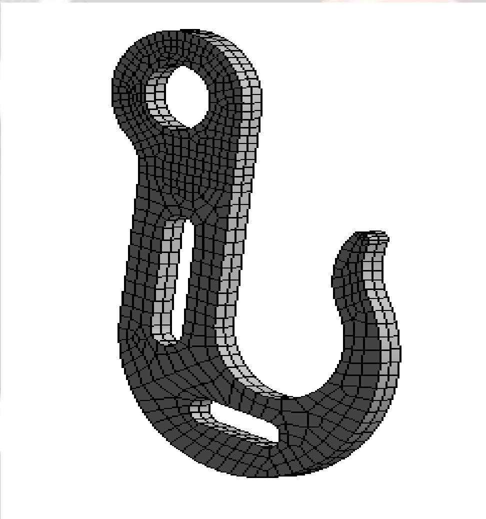
MESH

ANSYS Meshing is a normal purpose, intelligent, automated product with the high performance. It produces the most corrected meshing for correct, efficient Multiphysics solutions. A mesh well matched for a particular analysis are often generated with one mouse click for all parts in a model. Full controls over the choices accustomed generate the mesh are on the market for the professional user who desires to fine-tune it. the power of processing in parallel is automatically used to minimize the time you stay waited for mesh generation.



Statistics	
Nodes	5321
Elements	854

Fig. 6 Meshing of Original Crane Hook



Statistics	
Nodes	6257
Elements	978

Fig. 7 Meshing of Optimized Crane Hook

In the Present work we are generating Hexahedra meshes as are economic with the number of elements because the same degrees of freedom (or for 8 nodes) for one Hexaedron corresponds to six Tetrahedra. Increasing the number of elements defunately will not increase the size of the global finite element matrices but the computations for one hexahedron are generated also for six tetrahedra. This step has to be compared in software time in order to state if it is interesting to use hexahedra than constant strain tetrahedra knowing that curved or linear hexahedra use Gauss intergration points to generate the element characteristics (stiffness, mass) and tetrahedra use exact formula without any integration to get the same characteristics. Researchers have always used tetrahedra elements because they fit very well arbitrary shaped geometries with their simple computations.

Boundary Condition

A boundary condition for the model is that the setting of a well-known value for an associated load or a displacement. For a specific node you'll be able to set either the load or the displacement but not each.

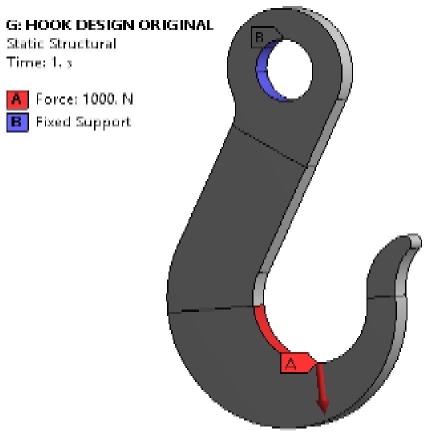


Fig. 8 Boundary Condition of Original Model

Total Deformation

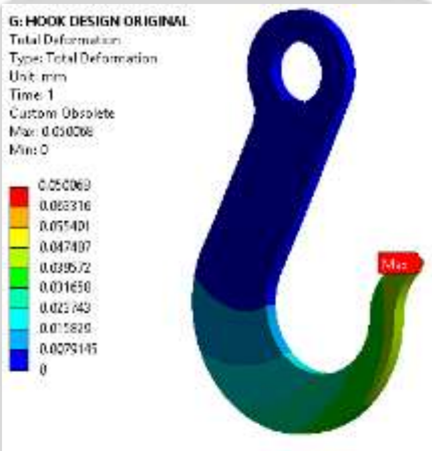


Fig 9 Total Deformation of Crane Hook

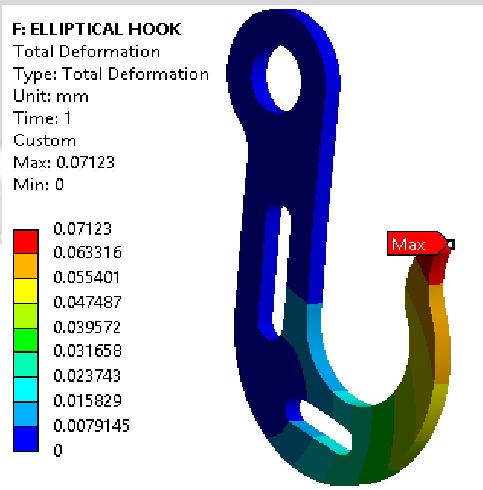


Fig.10 Total Deformation of Optimized Crane Hook

Equivalent Stress

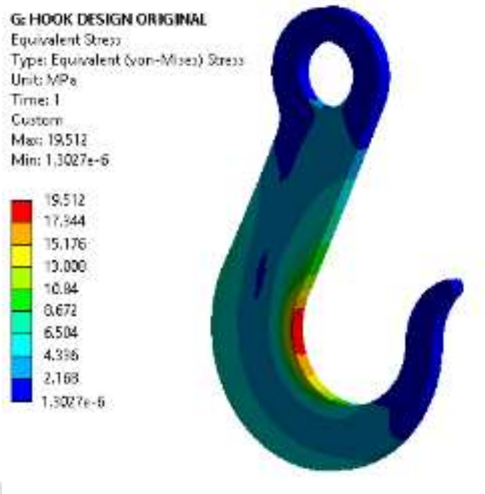


Fig 11 Equivalent Stress of Hook

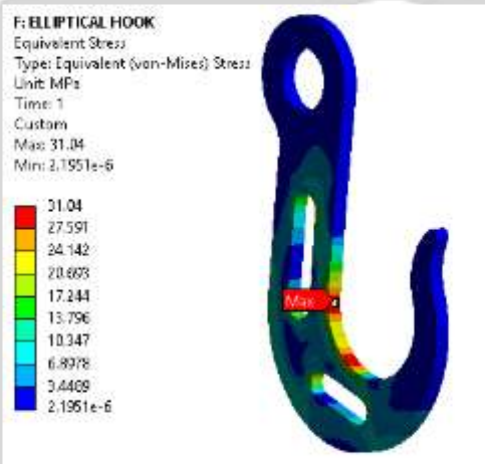


Fig. 12 Equivalent Stress of Optimized Crane Hook

Maximum Principal Elastic Stress

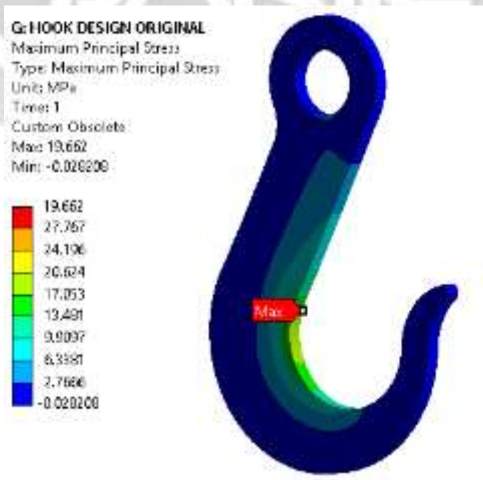


Fig 13. Maximum Principal Stress of Hook

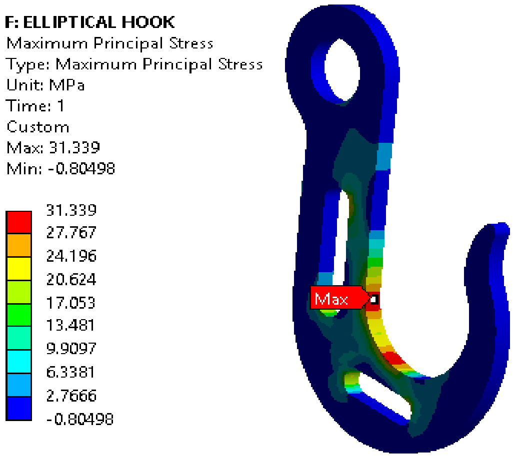


Fig. 14 Maximum Principle Stress of Optimized Crane Hook

Safety Factor of Optimized Hook

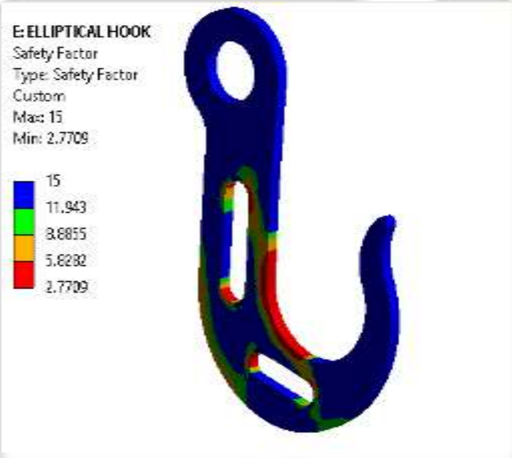


Fig. 15 Safety Factor of Elliptical Hook

Life

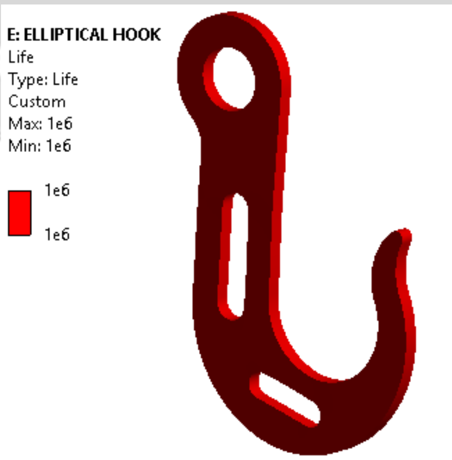


Fig. 16 Life of Optimized Crane Hook

5. MANUFACTURING PROCESS

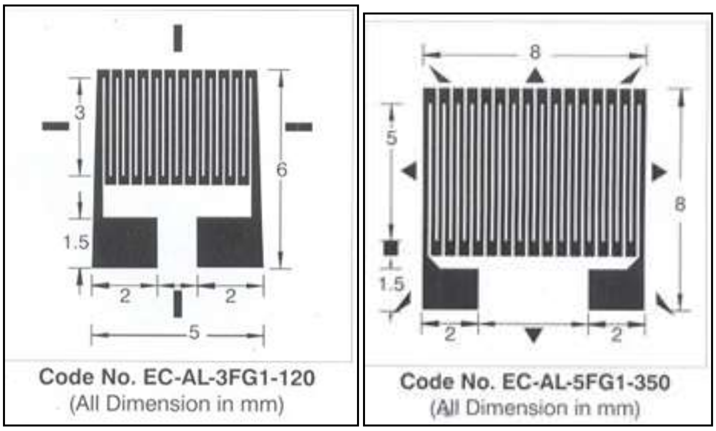
Fig. 17 Laser Sheet Metal Cutting Machine

Fig. 18 Optimized Hook

6. EXPERIMENTAL TESTING

A strain gauge is a instrument used to measure strain on an object. In 1938 it is invented by Arthur C. Ruge and Edward E. Simmons , generally the type of strain gauge consists of an insulating flexible backing which supports a metallic foil pattern. By a suitable adhesive, such as cyanoacrylate the gauge is attached to the object. As the foil is deformed with respect to object, causing its electrical resistance to change. Using a Wheatstone bridge this resistance change is measured, it is related to the strain by the amount known as the gauge factor.

A strain gauge having advantage of the physical property of electrical conductance and its dependence on the conductor's shape and size. When an electrical conductor is gets stretched within the specified limits of its elasticity such that it does not permanently deform or break, it will become narrower and longer, which increases its electrical resistance one end to other end. When a conductor is compressed such that it does not buckle, it will shorten and broaden, which reduces its electrical resistance end-to-end. From the measured electrical resistance of the strain gauge, the amount of induced stress could be inferred.



Strain Gage Foils

Specification of Strain Gage Foils

Type	Foil	Foil	Nickle Guages	
Gauge Resistance	120 +0.5 Ohms	350+0.5 Ohms	25-50 Ohms	20-30 Ohms
Gauge Length	3 mm	5 mm	5 mm	3 mm
Backing Material	Epoxy	Epoxy	Epoxy	Epoxy
Gauge Factor	1.9-2.2	1.9-2.2	-	-
Max. Excitation Voltage	6V	12V	-	-
Temperature Range	5 to100° C	5 to 100° C	5-70 °C	5-70 °C
Temperature Co-efficient	-	-	0.59% / °C	0.59% / °C
Insulation Resistance	5000 Meg. Ohms at 30V		5000 Meg. Ohms at 30V	
Creep Compensated for	Aluminium Alloy - 2024			
Temp. Compensated for	Aluminium Alloy - 2024			
Max. Repeatable Strain Allowed	1500 Microstrains			
Fatigue Life	More than 1 Million Cycles			
Cement	1. Two Component epoxy cement consisting of Adhesive & Hardner. 2. Cyanoacrylate.			

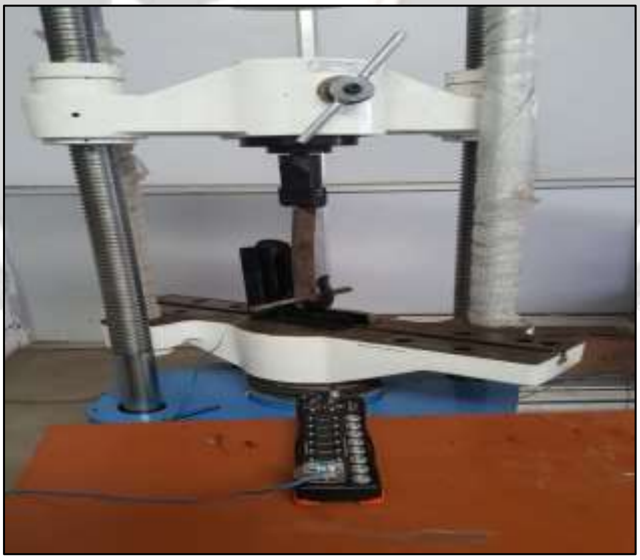


Fig. 19 Experimental Testing of Hook

7.RESULTS & CONCLUSION

Maximum Principal Elastic strain (FEA)

Fig 20. Maximum Principal Strain of Optimized Crane Hook

Experimental strain

Fig 19. Maximum Principal Strain of Optimized Crane Hook from stain gauge test

8.CONCLUSION

1. Topology optimization of Crane hook done.
2. From FEA result it conclude that weight of Crane hook after optimization reduces 12.11% than original Crane hook.
3. The strain value after the analysis is 155 microstrain and strain value after the testing is 148 microstrain.
4. The results obtained after the testing & analysis are nearly equal. So, validation of optimized Crane hook is done.

REFERENCES:

[1] B. Suneel, and B. Phani Shankar, (2015). Structural and Model Analysis of Dual Hook Joint. International Journal of Science and Technology Research (IJSETR), 4.

[2] Krishnaveni, M., Reddy, M. and Nagaraju, B. (2015). Static Analysis of Crane Hook with I-Section Using Ansys. International Journal of Engineering and Management Research, 5, pp.679-684.

[3] Ali, s., Kumar, H., Agrawal S. and Rajurkar, M. (2013). Stress Analysis of Crane Hook with Different Cross Section Using Finite Element Method. International Journal of Science and Research (IJSR).

[4] Shah, S. and Shrivastava, P. (2016). A Review on Study of Crane Hook Design. International Journal of Recent Technology Science & Management, ISSN: 2455-9679.

[5] Nagaraju, B., Rajaroy, M., Reddy, P. And Satyanarayana, K. (2015). Stress Analysis of Crane Hook Using FEA. Technical Research Organisation India, 2, Pp.Issn: 2394-0697.

[6] Bhagyaraj, G., Suryaprakash, K. and Subba Rao, K. (2017). Crane Hook Design and Analysis. International Research Journal of Engineering and Technology (IRJET), 4.
[7] "Material Handling Equipment" Michael G. Kay Fitts Dept. of Industrial and Systems Engineering North Carolina State University.
https://www.theworldmaterial.com/astm-a36-steel/