STRESS ANALYSIS AND OPTIMIZATION OF CRANE LIFTING TACKLE

Suraj N. Kadam*1, Prof. P. K. Parse#1

*1Post Graduate Scholar, #1Assistant Professor
Mechanical Engineering Department
TSSM'S Padmabhushan Vasantdada Patil Institute of Technology, Pune, Maharashtra, India
surkadam28@gmail.com

ABSTRACT
Lifting mechanism for carrying loads includes hook as the base of structure. Traditional technique used for designing hooks results into over designed structure. Design and analysis of hook done using FEA and stress analysis techniques. The concept hook design achieved using CAD (CATIA) software. Optimization will be achieved through FEA like Ansys Strain gauging installed at high strain location for measuring strain to quantify within Elastic/yield limit. Machining/EDM used for achieving optimized hook model dimensions. Comparative analysis done between FEA and experimental stress analysis result. Conclusion and future scope suggested.

Keyword: Hook design, Strain gauging, Topology optimization, EDM, Optimization of hook

1. INTRODUCTION
Crane hook framework activity are exceptionally risk parts and are exposed to failure because of enormous measure of stresses which can in the long run lead to its failure. Crane hook are for the most part framework used to lift the substantial burdens, man, material in ventures and constructional locales. Excavators having a crane-hook are generally used in development works site perception. The structure quality is the significant to react the heap bearing ability of the raising gear. Crane hook is a bended bar used to lift the heaps in the cranes. So as to decrease the structure failure in the crane hook, initiated stresses are examined. It is a capable and part utilized for mechanical applications. For the straight bars, the pivot of the cross area agrees with its censorial hub and the pressure dispersion in the shaft is liner. However, if there should be an occurrence of bended pillars perception, the nonpartisan pivot of the cross-area is moved towards the focal point of flow of the bar causing a non-direct circulation of stress. The potential dangers engaged with utilizing lifting gadgets can't be defeated exclusively by mechanical methods activity. The administrator must be ready, skillful framework, and prepared in the sheltered activity of lifters. It is additionally fundamental for the administrator to practice knowledge, care and presence of mind in foreseeing activity the movements that may happen as the heap is lifted. Hook is utilized to get and lift the heaps. It is a raising apparatus intended to draw in a ring or connection of a lifting chain or the pin of a shackle or link attachment. Crane hook with trapezoidal, rectangular, roundabout of triangular cross segments are usually utilized. The hook must be intended to convey greatest execution without failure.

Failure of Crane Hook
Because of persistent working of crane hook nanostructure of crane hook are changes and a few issues like debilitating of hook because of wear, elastic anxieties, plastic misshaping because of over-burdening and inordinate warm anxieties these are some other explanation of failure Strain maturing embrittlement because of nonstop stacking and emptying changes the microstructure. Twisting burdens joined with pliable anxieties, debilitating of hook because of wear, plastic misshaping due to over-burdening, and exorbitant warm burdens are a portion of different explanations behind failure. Henceforth constant utilization of crane hook may expand the size of these burdens and in the end bring about failure of the hook.

2. LITERATURE REVIEW
Structural and Model Analysis of Dual Hook Model [1]This Project includes in displaying and examination of a straightforward Dual Hook Model. Displaying was finished
by utilizing propelled plan programming CREO 1.0[Feature based parametric bi-directional software]. By utilizing the highlights of this product Dual Hook Model was displayed. Singular pieces of this double hook Model are planned independently in the part module and amassed in the gathering module. In the get together module there are two techniques, one is base up strategy and the other is top down technique and by utilizing base up get together technique the displaying of the double crane hook was finished. At long last, investigation was done on the ANSYS 14.5 which takes a shot at the noticeable FEM strategy, since work done on a GUI it was called as FEA. By utilizing this product stress investigation and modular examination was accomplished for various materials at various stacking conditions and the best material will be picked.

Design and Analysis of Crane Hook with Different Material[2] In this paper the structure of the hook is finished by expository technique and configuration is accomplished for the various materials like fashioned steel and high ductile steel. After the logical technique structure and demonstrating of hook is done in displaying delicate product (strong edge). The demonstrating is finished utilizing the plan estimation from the displaying the investigation of hook is done in FEA programming (ANSYS). This result lead us to the assurance of worry in existing model. By foreseeing the pressure fixation territory, the hook working life increment and r-e-duce the disappointment stress

Stress Analysis of Crane Hook with Different Cross Section Using Finite Element Method[3] Crane hook is a mechanical part utilized for material giving or move, watches stresses, initiated when various types of burden applied. From the wellbeing perspective the crane hook damaged must be forestalled because of break crack created caused for the most part at pressure focus regions. Weight on crane hook relies on different geometric factors just as material properties. In this investigation the material properties of hook kept consistent all through the examination and stress is to be decreased by changing diverse geometric boundaries.

Operational Guide for Lifting Devices[4] The utilization of Operational Guide is warning just and not compulsory. Intentional use is inside the control and attentiveness of the client and isn't planned to, and doesn't in any capacity, limit the creativity, duty or privilege of individual makers to structure or produce electric lifting gadgets which don't agree to this Operational Guide. This Guide gives general methodology to establishment, review, upkeep and fixes, activity and administrator preparing for lifting gadgets. Following this Guide doesn't guarantee consistence with pertinent government, state, and neighborhood laws or guidelines and codes. This Operational Guide isn't authoritative on any individual and don't have the impact of law.

Material Handling Equipment[5] Material taking care of (MH) includes "short-separation development that generally happens inside the bounds of a structure, for example, a plant or a stockroom and between a structure and transportation organization. It can be utilized to make Time and Place Utility through the taking care of, capacity, and control of material, as unmistakable from assembling (i.e., manufacture and get together activities), which makes "structure utility" by changing the shape, structure, and cosmetics of material.

Design and Analysis of Crane Hook with Different Material[6] In this paper the plan of the hook is finished by expository strategy and configuration is accomplished for the various materials like fashioned steel and high elastic steel. After the expository technique structure and displaying of hook is done in demonstrating delicate product (strong edge). The displaying is finished utilizing the plan computation from the demonstrating the examination of hook is done in FEA programming (ANSYS).

2.1 PROBLEM STATEMENT

Lifting instrument for conveying loads incorporates hook as the base structure. Customary methods utilized for planning guides results into over designed structure. Base plan and examination of hook done utilizing FEA and test pressure investigation methods.

2.2 OBJECTIVES

- To achieve optimized design less mass as compared to traditional design.
- CAD modeling of existing Model using CATIA software.
- Topology optimization of existing design using topology module in ANSYS software.
- FEA analysis of model to determine stress and deformation along with strain gauging & lifting load applied through UTM for validation.
- Comparative analysis between FEA & Experimental results.

3. METHODOLOGY

![Flowchart for the Methodology](image)

4. FINITE ELEMENT ANALYSIS OF CRANE HOOK USING ANSYS WORKBEACH

Suraj N. Kadam and Prof. P. K. Parse
The Finite Element Method is a numerical approximation method, in which complex structure divided into number of small parts that is pieces and these small parts are called as finite elements. These small elements connected to each other by means of small points called as nodes. As finite element method uses matrix algebra to solve the simultaneous equations, it is known as structural analysis and its becoming primary analysis tool for designers and analysts.

The three basic FEA process are
a) Preprocessing phase
b) Processing or solution phase
c) Post processing phase

Modal analysis is done to determine the natural frequencies and mode shapes of a structure. The natural frequency and mode shapes are important parameters in design of a structure for dynamic loading conditions.

FEA Pre Processing:
The pre-processing of the sheet metal is down for the purpose of the dividing the problem into nodes and elements, developing equation. for an element, and applying boundary conditions, initial conditions for applying loads. The information required for the pre-processing stage of the sheet metal is as follows,
• Material properties:
The values of young’s modulus, poisons ratio, density, anyield strength for steel are taken from material library of the FEA package.

Material properties
Material- Structural Steel
Young’s Modulus- 200 GPa
Poisons Ratio- 0.3
Density- 7850 kg/m3
Yield Strength- 520 MPa

Finite Element Method (FEM) : It is a numerical technique for finding approximate solutions to boundary value problems for partial differential equation. It is referred to Finite element analysis (FEA). FEM subdivides a large problem into the smaller, parts, called finite elements. The simple equations that are model these finite elements are then assembled into a larger system of equations that models the entire problem. FEM then the uses variation methods from the calculus of variations to approximate a solution by minimizing an associated error function.

Constraint:
The nodes around the side panel holes have a rigid element connecting them to the centre of the hole which has of its degree of freedom fixed. The element which is used to fix side panel and vehicle is fixed and used as a rigid element. The minimum and maximum Value are set together with other mesh parameters such as element type and material. The selected object is as ready for further analysis.

Post Processing:
The acceptability of the design of the sheet metal needs to be considered from the results of the analysis. Model acceptance criteria: Natural frequency of the structure should be increase with reinforcement of elastic material.
5. TOPOLOGY OPTIMIZATION

Topology optimization is the mathematical approach that optimizes material layout within a given design area, for a given set of loads and boundary conditions such that the resulting layout meets a prescribed set of performance targets.

**Basic Theory**

There are three kinds of structure optimization,
- Size optimization
- Shape optimization
- Topology optimization

Topology optimization with more freedom degree and larger design area, its greatest feature is below unsure structural form, in keeping with the well-known condition and a given load to figure out the cheap structure, every for the abstract variety of recent product and improvement design for existing product, it’s the foremost promising side of structural optimization. For continuous structure topology optimization, there are some mature ways like: uniform technique, evolutionary structural optimization technique, variable density technique etc.
OPTIMIZED DESIGN OF CRANE LIFTING TACKLE

Fig. 9 Material removal area of Crane Lifting tackle
Red region indicates material removal area

Fig. 10 Geometry of optimized hook

MESH

Fig. 11 Meshing of optimized Crane Lifting tackle

FEA Results

Fig. 12 Boundary condition for optimized Crane Lifting tackle

Fig. 13 Total deformation of optimized Crane Lifting tackle

Fig. 14 Equivalent stress of optimized Crane Lifting tackle
STRESS ANALYSIS AND OPTIMIZATION OF CRANE LIFTING TACKLE

Fig. 15 Equivalent elastic strain of optimized Crane Lifting tackle

- Strain is observed around 1330 microns using FEA.

6. EXPERIMENTAL SETUP

Universal Testing Machine is used to test the tensile and compressive strength of materials. Universal Testing Machines are capable of performing many different varieties of tests on an equally diverse range of materials, components, and structures. Universal Testing Machines accommodate different materials, ranging from hard samples, such as metals and concrete, to flexible samples, such as rubber and textiles. This diversity makes the Universal Testing Machine equally applicable to virtually any industry. The UTM is a versatile piece of testing equipment that evaluate materials properties such as tensile strength, elasticity, compression, yield strength, elastic and plastic deformation, bend compression, and strain hardening. Different models of universal Testing Machines have different load capacities, some as low as 5kN and others as 2000kN.

In this experimental purpose we used 400KN load capacity UTM.

Table. 2 Specification of UTM

<table>
<thead>
<tr>
<th></th>
<th><strong>Max Capacity</strong></th>
<th>400KN</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>Measuring range</td>
<td>0-400KN</td>
</tr>
<tr>
<td>3</td>
<td>Least Count</td>
<td>0.04KN</td>
</tr>
<tr>
<td>4</td>
<td>Clearance for Tensile Test</td>
<td>50-700 mm</td>
</tr>
<tr>
<td>5</td>
<td>Clearance for Compression Test</td>
<td>0-700 mm</td>
</tr>
<tr>
<td>6</td>
<td>Clearance Between column</td>
<td>500 mm</td>
</tr>
<tr>
<td>7</td>
<td>Ram stroke</td>
<td>200 mm</td>
</tr>
</tbody>
</table>

8 | Power supply  | 3 Phase , 440Volts , 50 cycle. A.C |
9 | Overall dimension of machine (L*W*H) | 2100*800*2060 |
10 | Weight         | 2300Kg |

A strain gauge takes advantage of the physical property of electrical conductance and its dependence on the conductor's geometry. When an electrical conductor is stretched within the limits of its elasticity such that it does not break or permanently deform, it will become narrower and longer, which increases its electrical resistance end-to-end. Conversely, when a conductor is compressed such that it does not buckle, it will broaden and shorten, which decreases its electrical resistance end-to-end. From the measured electrical resistance of the strain gauge, the amount of induced stress may be inferred.

Strain Gage Foils

Table. 2 Specification of Strain Gage Foils

<table>
<thead>
<tr>
<th>Code No.</th>
<th>EC-AL-3FG1-120 (All Dimension in mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Code No.</th>
<th>EC-AL-5FG1-350 (All Dimension in mm)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Experimental procedure

- Fixture is manufactured according to component designed.
- Single force is applied as per FEA analysis and reanalysis is performed to determine strain by numerical and experimental testing.
- Strain gauge is applied as per FEA results to maximum strained region and during experimental testing force is applied as per numerical analysis to
check the strain obtained by numerical and experimental results.

- During strain gage experiment two wires connected to strain gage is connected to micro controller through the data acquisition system and DAQ is connected to laptop. Strain gage value are displayed on laptop using DEWESOFT software.

- Static structural analysis of Simple Crane Lifting tackle is performed to determine deformation and equivalent stress. It is observed that around maximum deformation is 0.248 mm and equivalent stress is 73.78 MPa. An optimized model is obtained from topology optimization technique.

- It is concluded that the region indicated in red region in topology optimization provides information regarding removal of material from that area.

- It depends on us to removal of material by proper design and reanalysis as per existing conditions to sustain boundary condition.

- After removing material from Crane Lifting tackle Static structural analysis is performed to determine deformation and equivalent stress. It is observed that around maximum deformation is 0.746 mm and equivalent stress is 270.26 MPa.

- Weight optimization of around 11% is observed.

- Strain measurement of 1330 microns and 1310 microns by numerical and experimental testing respectively.

### REFERENCES


[8] Rasmid Uddinwadikar “Stress Analysis Of Crane Hook and Validation by Photo-Elasticity” in Scientific Research, 2011, ISSN 935-941


**7. CONCLUSION**