

Experimental Stress Analysis and Optimization of Crane Lifting Tackle

Pravin Rajendra Hajare¹, Prof. S. M. Jadhav²

¹PG Student, Dept. of Mechanical Engineering, STES'S NBN Sinhgad School of Engineering, Maharashtra, India

²Prof. of Dept. of Mechanical Engineering, STES'S NBN Sinhgad School of Engineering, Maharashtra, India

Abstract - 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. Concept hook design achieved using CAD (CATIA) software. 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 results, Conclusion and future scope suggested.

Key Words: Crane hook, Strain gauging FEA, ANSYS.

1. INTRODUCTION

Crane hook framework activity are profoundly at-risk segments and are constantly exposed to failure because of aggregation of huge measure of stresses which can in the end lead to its failure. Crane hook are by and large framework used to hoist the overwhelming burdens, substantial apparatuses, man, material in ventures and constructional locales. As of late, excavators having a crane-hook are broadly used in development works site perception. The structure quality is the significant trademark to react the heap bearing capacity of the lifting hardware. 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 dissected. Weariness harm is the inception of split because of fluctuating burdens. It is caused because of feelings of anxiety which are inadequate to cause harm in a solitary application. It is an exceptionally dependable and significant segment utilized for mechanical applications. Winkler clump hypothesis is utilized to compute the hypothetical pressure. For the straight bars, the nonpartisan hub of the cross area harmonizes with its censorial pivot and the pressure dissemination in the pillar is liner. In any case, in the event of bended bars perception, the unbiased hub of the cross-area is moved towards the focal point of shape of the pillar causing a non-direct conveyance of stress. The potential perils associated with utilizing lifting gadgets can't be defeated exclusively by mechanical methods activity. The administrator must be ready, skilled framework, and prepared in the sheltered activity of lifters. It is additionally basic for the administrator to practice insight,

care and presence of mind in foreseeing activity the movements that may happen as the heap is lifted. Hook is utilized to snatch and lift the heaps. It is a raising installation 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, round of triangular cross segments is generally utilized. The hook must be intended to convey most extreme execution without failure.

Because of consistent working of crane hook nanostructure of crane hook are changes and a few issues like debilitating of hook because of wear, elastic burdens, plastic distortion because of over-burdening and unnecessary warm anxieties these are some other explanation of failure Strain maturing embrittlement because of constant stacking and emptying changes the microstructure. Bowing anxieties joined with malleable burdens, debilitating of hook because of wear, plastic misshaping due to over-burdening, and unreasonable warm anxieties are a portion of different explanations behind failure. Thus, consistent utilization of crane hook may build 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] Venture includes in displaying and investigation of a straightforward Dual Hook Model. Displaying was finished by utilizing propelled structure programming CREO. By utilizing the highlights of this product Dual Hook Model was displayed. Singular pieces of this double hook Model are structured independently in the part module and amassed in the gathering module. In the gathering module there are two strategies, one is base up technique and the other is top down strategy and by utilizing base up get together strategy the displaying of the double crane hook was finished. By utilizing this product stress examination and modular investigation 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 plan of the hook is finished by expository technique and configuration is accomplished for the various materials like manufactured steel and high pliable steel. After the logical technique plan and demonstrating of hook is done in displaying delicate product (strong edge). The demonstrating is finished

utilizing the structure computation from the displaying the examination of hook is done in FEA programming (ANSYS). This result lead us to the assurance of worry in existing model. By anticipating the pressure fixation zone, the hook working life increment and re-duce the failure 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, prompted when various types of burden applied. From the security perspective the crane hook harmed must be forestalled because of split crack created caused essentially at pressure fixation territories. Weight on crane hook relies on different geometric factors just as material properties. In this investigation the material properties of hook kept steady all through the examination and stress is to be diminished by fluctuating distinctive geometric boundaries.

Operational Guide for Lifting Devices[4] The utilization of Operational Guide is lenient and warning just and not compulsory. Deliberate use is inside the control and circumspection of the client and isn't planned to, and doesn't in any capacity, limit the inventiveness, obligation or privilege of individual makers to structure or produce electric lifting gadgets that which don't conform to this Operational Guide..

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

4. METHODOLOGY

Step 1:- To achieve optimized design having less mass as compared to traditional design.

Step2:- CAD modeling of existing Model using CATIA.

Step 3: - Finite Element Discretization using Ansys applying loads at boundary conditions.

Step 4:-Experimental stress analysis of Model using Strain gauging & lifting load applie through UTM

Step 5:-Comparative analysis between FEA & Experimental results.

5. DESIGN

5.1 CAD Part in CATIA Software

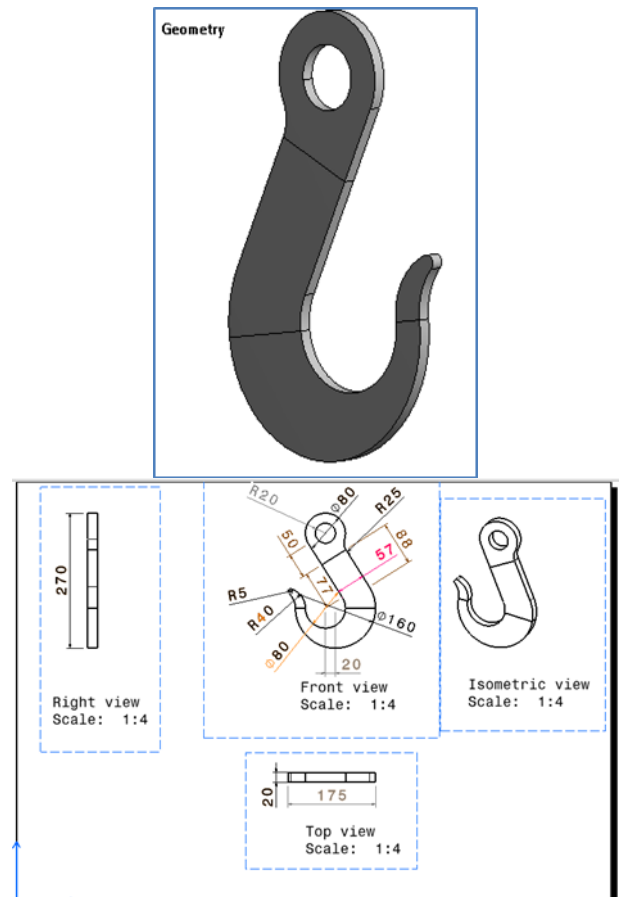


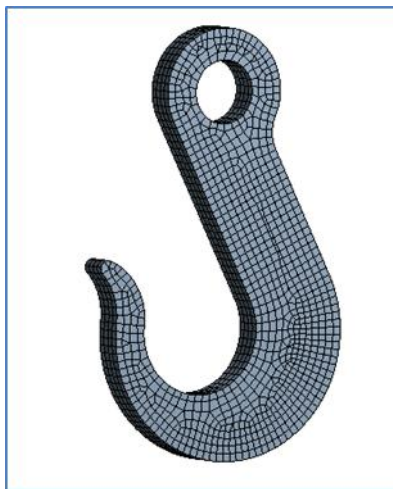
Fig.1. CATIA and drafting of hook

| Properties of Outline Row 3: Structural Steel | | | |
|---|---|------------|--------------------|
| | A | B | C |
| 1 | Property | Value | Unit |
| 2 | Material Field Variables | Table | |
| 3 | Density | 7850 | kg m ⁻³ |
| 4 | Isotropic Secant Coefficient of Thermal Expansion | | |
| 5 | Coefficient of Thermal Expansion | 1.2E-05 | C ⁻¹ |
| 6 | Isotropic Elasticity | | |
| 7 | Derive from | Young... | |
| 8 | Young's Modulus | 2E+11 | Pa |
| 9 | Poisson's Ratio | 0.3 | |
| 10 | Bulk Modulus | 1.6667E+11 | Pa |
| 11 | Shear Modulus | 7.6923E+10 | Pa |

Table.1. Material Properties

5.2 Mesh

ANSYS Meshing is a general-purpose, intelligent, automated high-performance product. It produces the most appropriate mesh for accurate, efficient Multiphysics solutions. A mesh well suited for a specific analysis can be generated with a single mouse click for all parts in a model. Full controls over the options used to generate the mesh are available for the expert user who wants to fine-tune it. The power of parallel processing is automatically used to reduce the time you have to wait for mesh generation.



| Details of "Body Sizing" - Sizing | |
|-----------------------------------|--------------------|
| Scope | |
| Scoping Method | Geometry Selection |
| Geometry | 1 Body |
| Definition | |
| Suppressed | No |
| Type | Element Size |
| Element Size | 5.0 mm |

| Statistics | |
|------------|-------|
| Nodes | 18067 |
| Elements | 3500 |

Fig. 2. Details of meshing

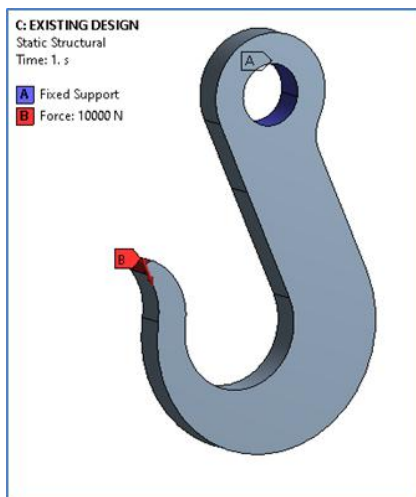


Fig.3. Boundary condition

5.3 Finite Element Analysis

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, so it is known as structural analysis and it's becoming primary analysis tool for designers and analysts. Modal analysis is done to determine the natural frequencies and mode shapes of a structure. The natural freq. and mode shapes are important parameters in the design of a structure for dynamic loading conditions.

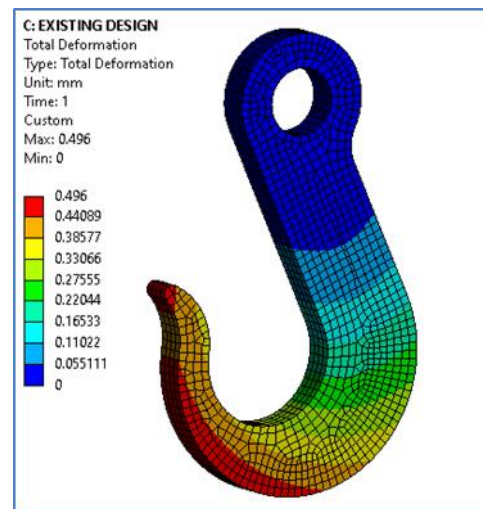


Fig. 4 Total deformation of crane hook

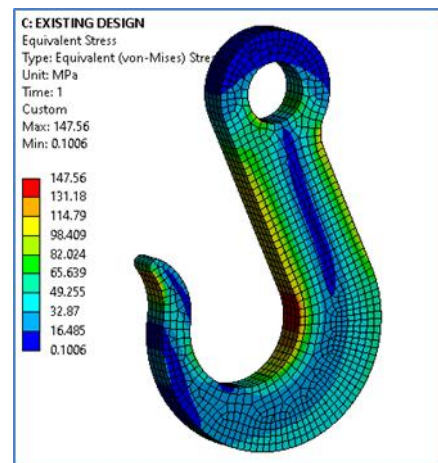


Fig. 5 Equivalent stress of crane hook.

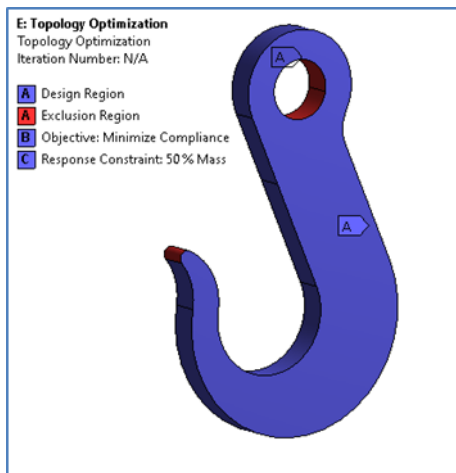
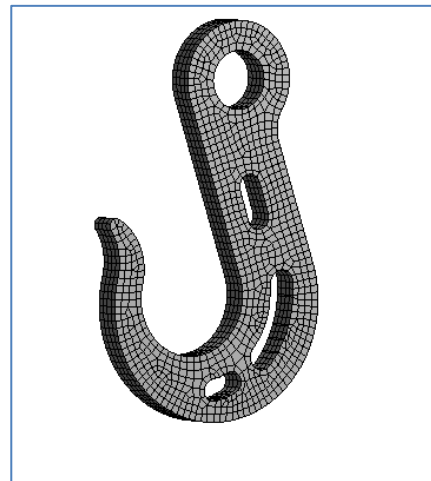


Fig. 6 Boundary condition for topology



| Details of "Body Sizing" - Sizing | |
|---------------------------------------|--------------------|
| Scope | |
| Scoping Method | Geometry Selection |
| Geometry | 1 Body |
| Definition | |
| Suppressed | No |
| Type | Element Size |
| <input type="checkbox"/> Element Size | 5.0 mm |

| Statistics | |
|-----------------------------------|-------|
| <input type="checkbox"/> Nodes | 16571 |
| <input type="checkbox"/> Elements | 3064 |

Fig. 9 Meshing of elliptical hook

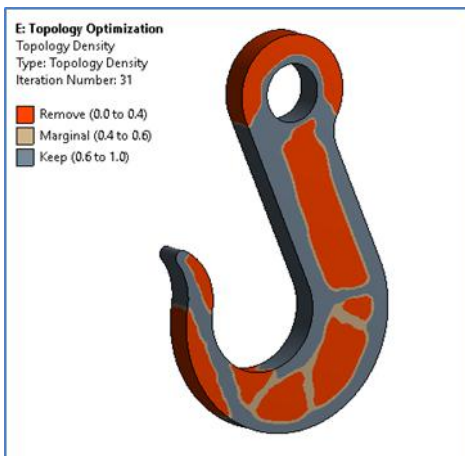


Fig. 7 Material removal area (Red region indicates material removal area.)

5.4 Finite Boundary Conditions For Topology Optimization

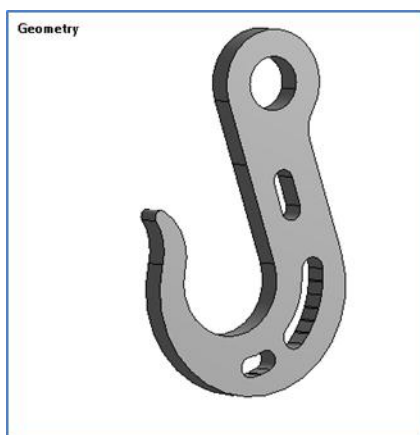


Fig. 8 Geometry of elliptical hook

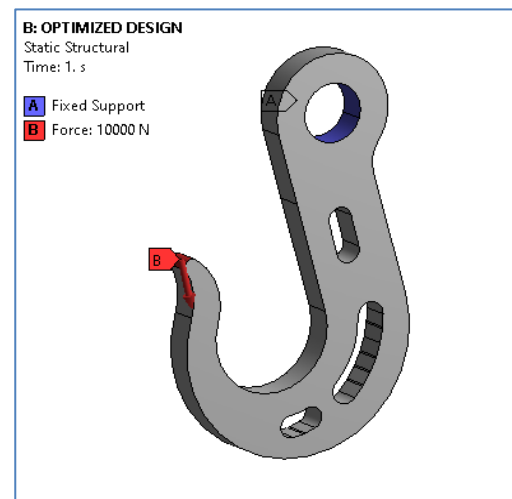


Fig.10 Boundary condition for optimized model

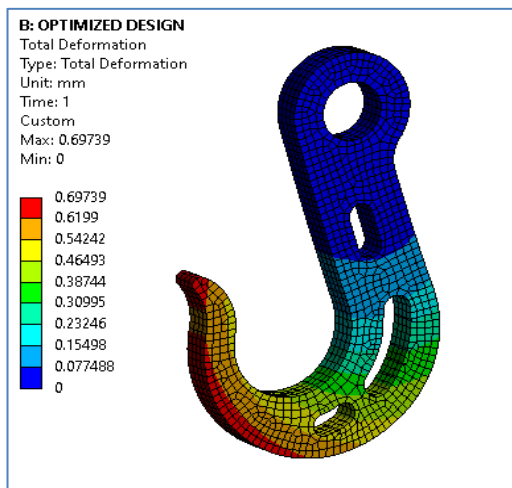


Fig. 11 Total deformation of elliptical hook

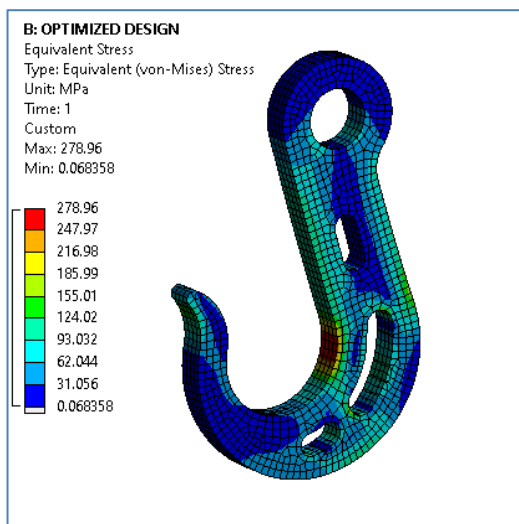


Fig.12 Equivalent stress of elliptical hook

6. EXPERIMENTAL SETUP

A universal testing machine (UTM), also known as a universal tester, materials testing machine or materials test frame, is used to test the tensile strength and compressive strength of materials. An earlier name for a tensile testing machine is a tensometer. The "universal" part of the name reflects that it can perform many standard tensile and compression tests on materials, components, and structures (in other words, that it is versatile). The set-up and usage are detailed in a test method, often published by a standards organization. This specifies the sample preparation, fixturing, gauge length (the length which is under study or observation), analysis, etc. The specimen is placed in the machine between the grips and an extensometer if required can automatically record the change in gauge length during the test. If an extensometer is not fitted, the machine itself can record the displacement between its cross heads on which the specimen is held. However, this method not only records the change in length of the specimen but also all other

extending / elastic components of the testing machine and its drive systems including any slipping of the specimen in the grips. Once the machine is started it begins to apply an increasing load on specimen. Throughout the tests the control system and its associated software record the load and extension or compression of the specimen.

Specification of UTM

| | | |
|----|---------------------------------------|------------------------------------|
| 1 | Max Capacity | 400KN |
| 2 | Measuring range | 0-400KN |
| 3 | Least Count | 0.04KN |
| 4 | Clearance for Tensile Test | 50-700 mm |
| 5 | Clearance for Compression Test | 0- 700 mm |
| 6 | Clearance Between column | 500 mm |
| 7 | Ram stroke | 200 mm |
| 8 | Power supply | 3 Phase , 440Volts , 50 cycle. A.C |
| 9 | Overall dimension of machine (L*W*H) | 2100*800*2060 |
| 10 | Weight | 2300Kg |

6.1 Experimental FEA

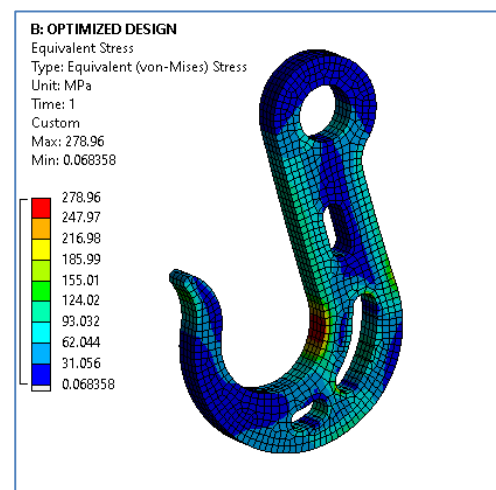


Fig. 13 Equivalent stress

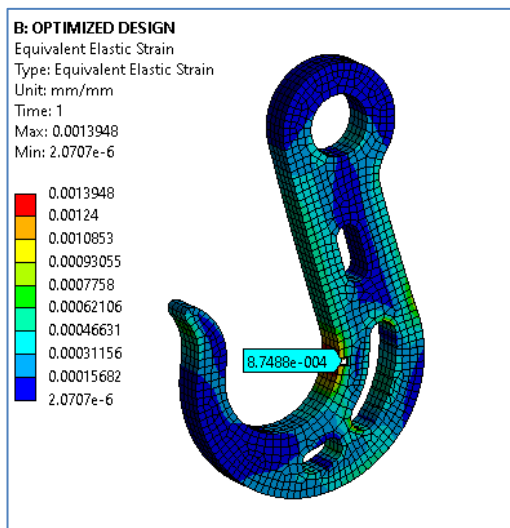


Fig. 14 Equivalent elastic strain (Strain is observed around 874 microns using FEA)

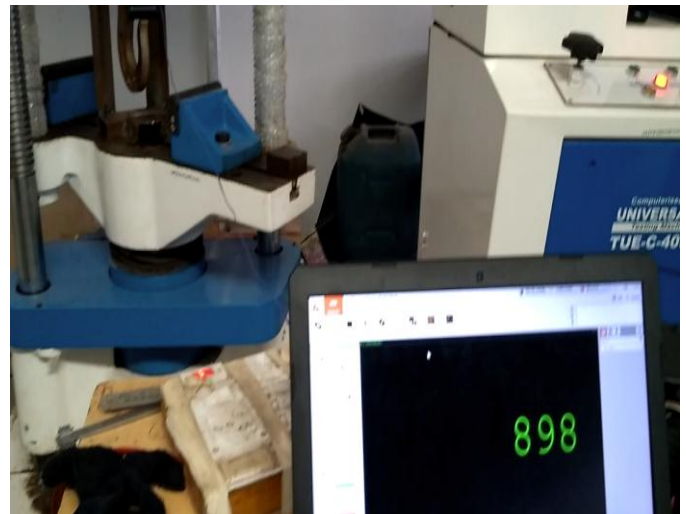


Fig. 16 Experimental result

6.2 Experimental procedure

1) 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.

2) 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.

3) 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 values are displayed on laptop using DEWESOFT software.



Fig. 15 Experimental testing

7. EXPERIMENTAL RESULTS

| Sr.no. | Characteristics | Hook joint | Hook joint after optimization (elliptical hook) |
|--------|-------------------------|------------|---|
| 1. | Deformation (mm) | 0.496 | 0.697 |
| 2. | Equivalent stress (mpa) | 147.56 | 278.96 |

8. CONCLUSION

1. Static structural analysis of Simple hook is performed to determine deformation and equivalent stress. It is observed that around maximum deformation is 0.496 mm and equivalent stress is 147.56 MPa. An optimized model is obtained from topology optimization technique.
2. It is concluded that the region indicated in red region in topology optimization provides information regarding removal of material from that area.
3. It depends on us to removal of material by proper design and reanalysis as per existing conditions to sustain boundary condition.
4. After removing material from hook Static structural analysis is performed to determine deformation and equivalent stress. It is observed

that around maximum deformation is 0.697 mm and equivalent stress is 278.96 MPa.

5. Strain measurement of 874 microns and 898 microns by numerical and experimental testing respectively.

REFERENCES

- [1] B.Suneel, B. Phani Shankar "Structural And Model Analysis Of Dual Hook Joint" International Journal of Science, Engineering and Technology Research (IJSETR), Volume 4, October 2015
- [2] Patel Ravin B,, Patel Dixit H.,Patel hirak "Design And Analysis Of Crane Hook With Different Material" Department of mechanical engineering " ISSN:2319-7900
- [3] Sayyedkasim Ali1, Harish Kumar2, Shishir Agrawal3, "Stress Analysis of Crane Hook with Different Cross Section Using Finite Element Method" International Journal of Science and Research (IJSR) ISSN (Online): 2319-7064
- [4] Sandeep Kumar Shah1, Prof. Ashish Kumar Shrivastava2 "A Review On Study Of Crane Hook Design" International Journal Of Recent Technology Science & Management ISSN: 2455-9679
- [5] Patel Bhakti K, "Operational Guide for Lifting Devices" June 2003.
- [6] Michael G. Kay, "Material Handling Equipment" Dept. of Industrial and Systems Engineering North Carolina State University. July 2008.