

# Finite Element Analysis and Optimization of Crane Hook

Nutan Joshi<sup>1</sup>, Dr. K. B. Waghulde<sup>2</sup>

P.G. Student, Department of Mechanical Engineering, TSSM' P.V.P.I.T, Bavdhan, Pune University, India<sup>1</sup>

Professor, Department of Mechanical Engineering, TSSM' P.V.P.I.T, Bavdhan, Pune University, India<sup>2</sup>

**ABSTRACT:** Crane hooks are designed to connect with chains and ropes attached to point loads like construction beams, and machinery. The basic function of it is to lift the load, to carry it and to transfer it from one place to another. Sometimes an accident may occur due to stress concentration factor in it so stress analysis is necessary before making it applicable. Finite Element Analysis (FEA) is the most powerful technique for strength calculations of the any structures working under known load and boundary conditions. FEA approach can be applied for the optimization. 3D model of a crane hook will be drawn in CATIA V5 R19, Meshing will be carried out in Hypermesh, and Ansys will be used for post-processing. FE Analysis will be carried out on the existing crane hook. Based on those FEA results topological changes will be made to reduce weight and stress concentration. Various parameters of hook will be changed and analysed for optimum design. If necessary different cross sections of hook such as circular section, T-section, rectangular section etc. will be analysed to find better results FEA results of the existing crane hook will be compared with the FEA results of the optimized crane hook. After getting satisfied results a prototype will be fabricated and tested. The two results i.e. FEA and experimental results will be validated.

**KEYWORDS:** Crane Hook, Analysis, FEA, Design, Optimization..

## I. INTRODUCTION

Crane Hooks are highly responsible components that are typically used for handling material in industries. It is basically a hoisting fixture designed to engage a ring or link of a lifting chain or the pin of a shackle or cable socket and must follow the health and safety guidelines. Thus, the aim of the work is to study the stress distribution pattern of a crane hook using finite element method and to verify the results by testing the prototype. The lifting of material generally occurs on construction sites, in factories and other industrial situations. Correct lifting can move large objects efficiently and reduce manual handling operations. Improper design of crane hooks lead to disastrous accidents. Every year, incorrect lifting procedures cause injuries, loss of work time and property. People, machinery, loads, methods and the work environment, are all important factors for correct lifting. Provided that enough safety measures are fully implemented, lifting accidents can be reduced. The Figure shows the general diagram of crane hook. Dimensions of the crane hook, having the lifting capacity up to 5 tons, will be found through reverse-engineering. We will take actual dimensions of the crane hook by measuring with vernier calliper, measuring tape. In our project, analysis will be done on existing crane hook. 3D model will be made in CATIA V5, Hypermesh will be used for meshing and Ansys will be used for post processing. Results will be in the form of stress and displacement plots by ANSYS.

Mainly the effects of dimensional parameters, like radius, thickness and radial width, on the stresses induced and the deformations in the crane hook will be studied. For each parameter variation we will do at least 3-4 iterations on FEA tools. Further, the combination of best results of single parameter variation will be considered and the CAD model will be changed accordingly. This new optimized hook will be analysed by finite element method. If necessary the crane hook will be analysed for different cross sections depending upon the FEA results after topology optimization. After getting suitable results through iterations of FEA, model will be manufactured and then tested for load vs. stress on UTM. Tested results should be in line with the FEA results.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 7, July 2017

## II. LITERATURE SURVEY

Literature available on crane hook.

Amandeep Singh, VinodRohilla, "Optimization and Fatigue Analysis of a Crane Hook Using Finite Element Method"

Stress analysis plays an important role in the design of structures like crane hook under loading conditions. Crane hook is a reliable lifting component being used in industries. Structure failure of crane hook occurs because of the stress induced due to repetitive loading and unloading conditions. In this study, solid modeling of crane hook having trapezoidal cross-section referring to one of its existing design is done using SOLIDWORKS. Further, analyses are carried out in ANSYS Workbench and n-Code Design Life. The lengths of two parallel sides of the cross-section of crane hook are varied and different candidates are obtained for loading capacity of 30 ton on the basis of Mass, total Displacement and Von-Mises stress. This is done to reduce weight and balance economy. Further, out of these candidates, best candidates are considered and fatigue analysis is performed on these candidates. The structure strength is an important signal to response the load bearing capability of the elevating equipment.

Fatigue damage is the initiation of a crack due to fluctuating loading. It is caused due to stress levels which are insufficient to cause damage in a single application [1]. It is highly responsible and important component being used for industrial works. It is a hoisting fixture manufactured and designed to engage a link of a chain or ring or the pin of a cable socket or shackle and safety guidelines must be followed [2]. In the present work, study of different candidates of crane hook of trapezoidal cross section based on design parameters are carried out and weight optimization is carried out by varying the design parameters. The present study shows the deformation and fatigue life contour plots of crane hook using Ansys Workbench and Ansys nCodeDesignLife.

G.E.V.Ratnakumar, B. Jitendra Kumar, Kalapala Prasad, "Design and Stress Analysis Of Various Cross Section of Hook",

Crane hook is a curved beam. Hooks are employed in heavy industries to carry tones of loads safely. These hooks have a big role to play as far as the safety of crane loaded is concerned. With more and more industrialization the rate at which these hooks are forged are increasing. This project is carried out to study the stress variation in crane hooks for different cross sections such as circular and square and for different radii of curvature as well, experimentally and theoretically. Experimentally, the loads are obtained for different crane hooks for 5mm elongation on UTM (Universal Testing Machine). And then the stresses induced in the crane hooks against the loads obtained from experimentation are also calculated theoretically using curved beam theory. Then the different crane hooks are modeled in Pro-E 2.0 and then analysis is done for modeled hooks using ANSYS 12.1 to find the stresses induced in the hooks. Then the stresses evaluated from curved beam theory and ANSYS12.1 are compared and conclusions are made.

Omkar P. Bhatkar, Arun V. Javir, Shinde Ajay Ashok, SawantSubodhSumant, UkardeSurajEknath, KavathankarKunalShivram, "Finite Element Analysis of Crane Hook And Optimization Using Taguchi Approach"

A Crane hook is a device for grabbing and lifting loads of heavy duty by mean of a device such as crane hoist. A crane hook is usually equipped with a safety latch to prevent the disengagement of the lifting rope to which the load is being attach. The CAD model of a lifting crane hook is initially prepared from the Analytical Design. In the present report crane hook of trapezoidal section is modeled using UG-NX 8.0, further, importing the CAD model into ANSYS 15.0 we estimate until 20 tones i.e. equivalent to 200000N load to be gradually applied, eventually we observed the von-misses stress developed in the model by FEM method these results von-misses stresses are being compared with Taguchi L9 orthogonal array for specific results. Thus, these results would lead us to determine various stresses and deflections in the designed CAD model. In order to reach the most optimum model from different materials of hook would be tested and the most efficient material would be selected.

Crane hooks are one of the important components used in industries to carry heavy duty loads; basically crane hooks are designed with pulley elongated by rope or a chain. It is basically a hoisting fixture designed to engage a ring or link of a lifting chain or the pin of a shackle or cable socket and must follow the health and safety guidelines. Thus, such a crucial component in an industry must be fabricated and designed in a way so as to deliver maximum performance without breakdown.

A. Gopichand, Ms. R.V.S.Lakshmi, Mr. B. Maheshkrishna, "Optimization Of Design Parameters For Crane Hook Using Taguchi Method"

## International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 7, July 2017

In the present work 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. From the Taguchi analysis the optimum combination of input parameters for von mises stress were determined using SN ration plots, The optimum combination is TRIANGULAR cross section ,CASTIRON material and 200mmRADIUS .The first model simulation values are compared with analytical values and the error is 9.26% which is acceptable.

In this paper mathematical calculations are done by varying cross sections triangular, trapezoidal, and rectangular with constant area of cross sections for two different materials of steel, cast-iron analysis are evaluated. PRO/E design is prepared, for different cross sections by preparing the trajectory with the parameters taken from the design data book. In this project result of mathematical calculations is compared with the ANSYS with constant area of cross section, modifying the type of material optimization of better results were done in Taguchi Method.

RashmiUddanwadiker, "Stress Analysis of Crane Hook and Validation by Photo-Elasticity"

To study the stress pattern of crane hook in its loaded condition, a solid model of crane hook is prepared with the help of CMM and CAD software. Real time pattern of stress concentration in 3D model of crane hook is obtained. The stress distribution pattern is verified for its correctness on an acrylic model of crane hook using Diffused light Polari scope set up. By predicting the stress concentration area, the shape of the crane is modified to increase its working life and reduce the failure rates.

In this paper to study the stress pattern of crane hook in its loaded condition, a solid model of crane hook is prepared with the help of CMM and CAD software. Real time pattern of stress concentration in 3D model of crane hook is obtained. The stress distribution pattern is verified for its correctness on an acrylic model of crane hook using Diffused light Polariscope set up. By predicting the stress concentration area, the shape of the crane is modified to increase its working life and reduce the failure rates. Max value obtained analytically=12.35 N/mm<sup>2</sup> while value obtained from ANSYS = 13.372 N/mm<sup>2</sup>. The results are in close harmony with a small percentage error =  $(13.372 - 12.35)/12.35 = 8.26\%$  Possible reasons for variation might be the due to assumption 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.

Patel Ravin B, Patel Bhakti K., Patel Priyesh M, Patel Dixit H., Patelhirak S, "Design And Analysis Of Crane Hook With Different Material"

In this paper the design of the hook is done by analytical method and design is done for the different materials like forged steel and high tensile steel. After the analytical method design and modeling of hook is done in modeling software (solid edge) .The modeling is done using the design calculation from the modeling the analysis of hook is done in FEA software (ANSYS). This result lead us to the determination of stress in existing model. By predicting the stress concentration area, the hook working life increase and reduce the failure stress.

A crane is a machine, equipped with a hoist, wire ropes or chains and sheaves used to lift and move heavy material. Cranes are mostly employed in transport, construction and manufacturing industry. Overhead crane, mobile crane, tower crane, telescopic crane, gantry crane, deck crane, jib crane, loader crane are some of the commonly used cranes. A crane hook is a device used for grabbing and lifting up the loads by means of a crane. It is basically a hoisting fixture designed to engage a ring or link of a lifting chain or the pin of a shackle or cable socket. Crane hooks with trapezoidal, circular, rectangular and triangular cross section are commonly used. So, it must be designed and manufactured to deliver maximum performance without failure. Thus the aim of this research is to study stress distribution pattern within a crane hook of various cross sections using analytical, numerical and experimental methods.

The results of stress analysis calculated from FEA analysis for various different material such as Forged Steel, Wrought iron/MS, Forged steel Alloy. For the different Material, It is observed that keeping the tone are same with different Material topology we will get different results, from the results it is found that the Forged Steel Material gives minimum stress.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 7, July 2017

## III. METHODOLOGY

### Literature Survey

Using the knowledge from literature review, we can know how the CAD model is to be prepared. The conditions required for applying various constraints and how the loads are applied is briefed about in the technical papers referred.

### CAD Model Generation

- Getting input data on dimensions crane hook from market.
- Creating 3D model in CATIA.

### Determination of loads

Determination of different loads and boundary condition acting on the component by studying various ref papers, and different resources available

### Testing and Analysis

- Meshing the CAD model and applying the boundary conditions.
- Solve for the solution of meshed model using Optistruct.

### Re-Design, Analysis and Results

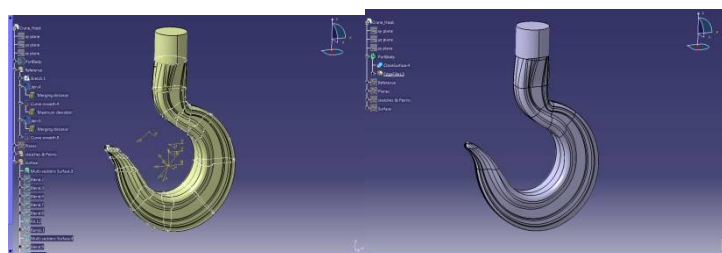
- Making changes in CAD model for optimization in different sections like circular section, T section, rectangular section etc.
- Carrying different iteration by removing material based on FEA results.
- Check the maximum stress ensuring it is well within the safe region.

### Fabrication, Experimental validation and Result

- Fabrication of prototype.
- Suitable experimentation and comparison with present crane hook.

## III ANALYSIS OF THE EXISTING CRANE HOOK

ACE Hydra Crane is used for the project work. Dimensions are taken through reverse engineering. Dimensions for crane hook were measured from site. These dimensions taken from the actual model of crane hook were used for 3D modelling of crane hook. Below are some of the images taken at site. A rough hand sketch was drawn showing all the dimensions of crane hook. Dimensions are required for calculating of boundary conditions. Hence its CAD model is necessary. CAD model then is made by the commands in CATIA of Pad, pocket, fillet, and geometrical selections in part design module. Parametric generation of drawings will help to get the dimensions useful in forces calculations in static loading conditions.



Finite element analysis of the existing hook is performed

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

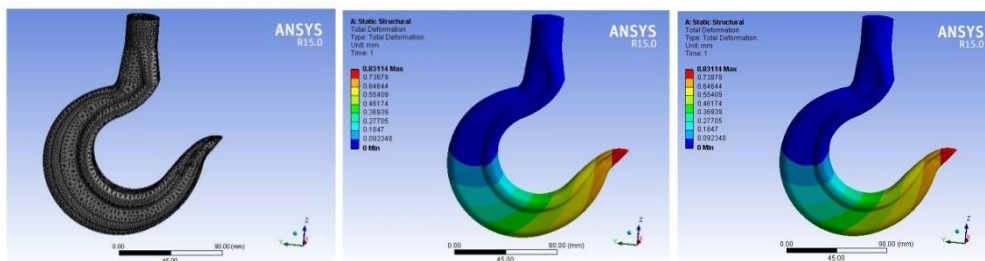
Vol. 6, Issue 7, July 2017

- Meshing the CAD model and applying the boundary conditions.
- Solve for the solution of meshed model using Optistruct.

Number of nodes: 85301

Number of elements: 50463

Element size = 3mm



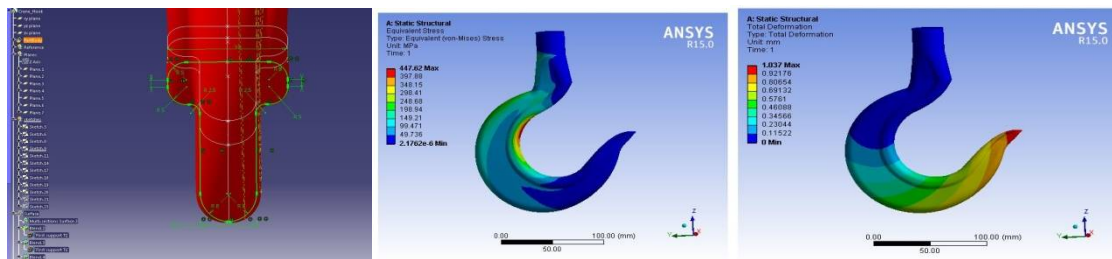
## IV. ITERATIONS WITH ALTERNATE MODELS USING FEA

Here iterations with alternate models of the hook based weight optimization of crane hook is carried out. Also, several iterations will involve, modifying the thickness at certain places in structure to reduce its deformation and check for its stress. Also it includes the comparative study of different models.

For the analysis purpose, solver deck is prepared. Geometry was meshed and material properties were assigned. After applying forces geometry in Ansys for solving the problem and viewing the results.

- Making changes in CAD model for optimization in different for different dimensional parameters
- Carrying different iterations by removing material based on FEA results.
- Check the maximum stress ensuring it is well within the safe region.

### Iteration 1:



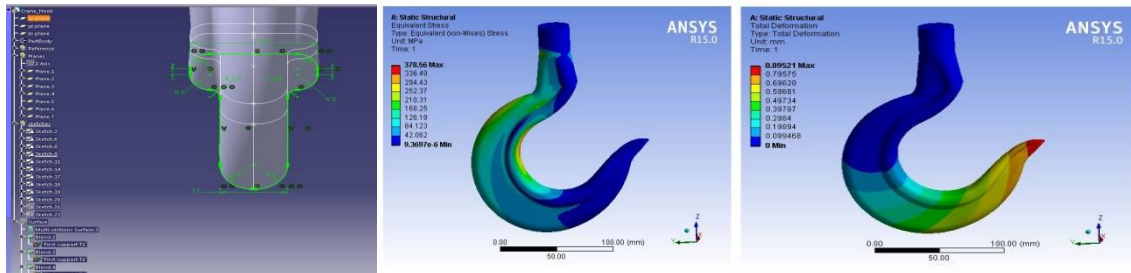
# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

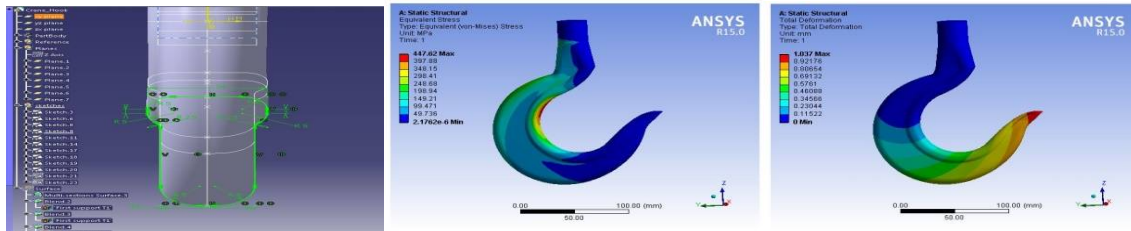
Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 7, July 2017

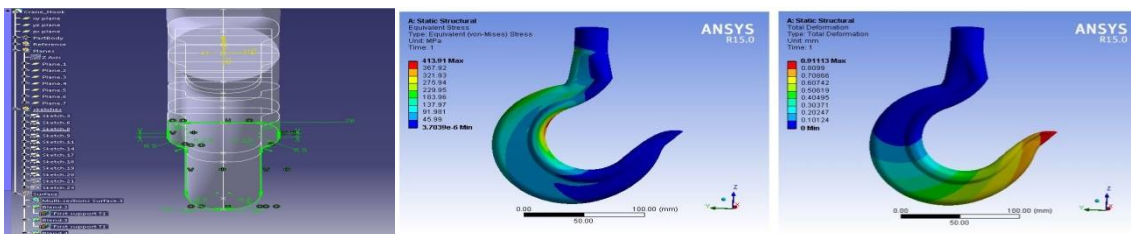
## Iteration 2:



## Iteration 3



## Iteration 4:



Comparisons of all the iterations in tabular form are as given below

Sr.No	Iterations	Stress (MPa)	Deformation (mm)	Weight (Kg)
1	Original	388.29	0.83	1.797
2	iteration 1	447.62	1.037	1.641
3	iteration 2	413.91	0.911	1.718
4	iteration 3	378.56	0.895	1.708
5	iteration 4	378	0.895	1.8

Crane hook of dimensions in iteration 3 is used for experimental validation as there is weight reduction.

# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 7, July 2017

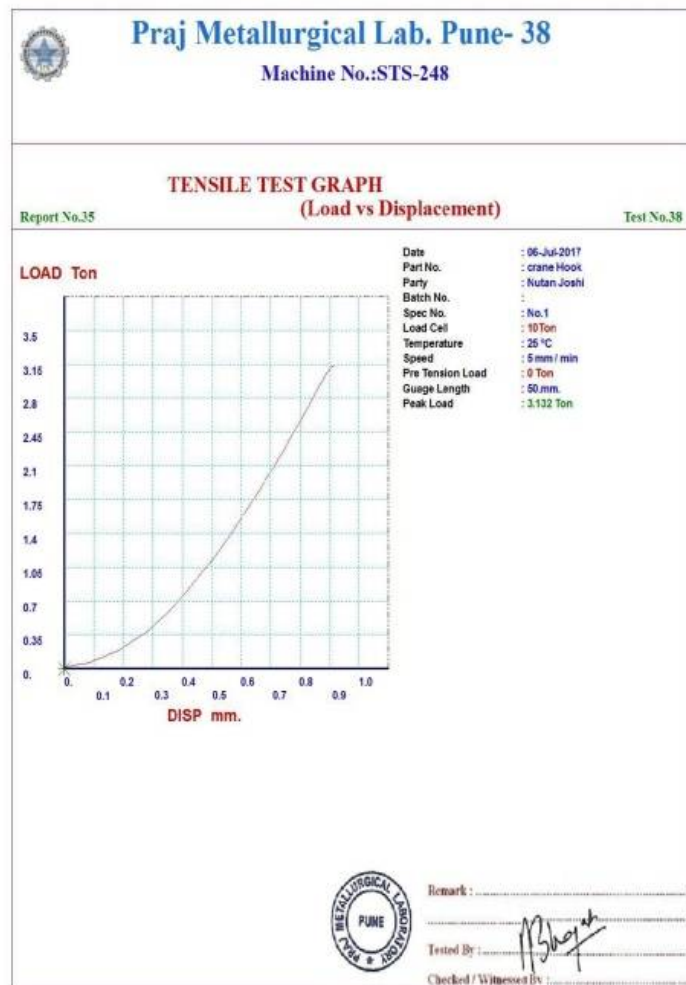
## IV. EXPERIMENTAL RESULTS

Optimised model of dimensions in iteration3 are tested on UTM. Outcomes from the FEA and experimental results are presented in tabular form as shown below.

Sl.No.	Deformation(mm)		Error(%)
	FEA	Experimental	
1	0.895	0.92	2.7%

### Lab Report:

When the testing is carried out in the laboratory i.e. when crane hook of dimensions in iteration 3 was subjected to the loads the tensile test graph was generated. The results obtained in the laboratory are as shown below.



# International Journal of Innovative Research in Science, Engineering and Technology

(An ISO 3297: 2007 Certified Organization)

Website: [www.ijirset.com](http://www.ijirset.com)

Vol. 6, Issue 7, July 2017

## V. CONCLUSION

### Conclusion and future scope

In the present work, the geometry of the crane hook is modelled and finite element analysis (FEA) is applied on the model. The results of static structure analysis are obtained from the FEA software Ansys for different dimensions. From above project the FEA and experimental results are validated. In this study, we finally conclude that out of four candidates obtained, best candidates is obtained in iteration 3 is considered for validation on the basis of minimum weight criteria. The error in FEA result and Experimental result is 2.7%.The stresses and deformation are in permissible limit in above design. In future different cross sections are to be used and altered .It can be studied for different type of materials also. Further it is advisable to conduct photo elasticity test for the crane hook under investigation in order to get better insight for stress concentration.

## REFERENCES

- [1] Amandeep Singh, VinodRohilla, "Optimization and Fatigue Analysis of a Crane Hook Using Finite Element Method", International Journal of Recent advances in Mechanical Engineering, Vol.4, No.4, November 2015. 4403
- [2] G.E.V.Ratnakumar, B. Jitendra Kumar, Kalapala Prasad, "Design and Stress Analysis Of Various Cross Section of Hook", International Journal of Innovations in Engineering and Technology, ISSN: 2319 – 1058, Volume 4 Issue 4 December 2014
- [3] Mr. A. Gopichand, Ms. R.V.S.Lakshmi, Mr. B. Maheshkrishna, "Optimization Of Design Parameters For Crane Hook Using Taguchi Method", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 12, December 2013, ISSN: 2319-8753.
- [4] Omkar P. Bhatkar, Arun V. Javir, Shinde Ajay Ashok, SawantSubodhSumant, UkardeSurajEknath, KavathankarKunalShivram, "Finite Element Analysis of Crane Hook And Optimization Using Taguchi Approach", Journal of The International Association of Advanced Technology and Science, ISSN-3347-4482, Vol. 16 | MARCH 2015
- [5] Patel Ravin B, Patel Bhakti K., Patel Priyesh M, Patel Dixit H.,Patelhirak S, "Design And Analysis Of Crane Hook With Different Material", International Journal of Advanced Computer Technology, ISSN:2319-7900.
- [6] RashmiUddanwadiker, "Stress Analysis of Crane Hook and Validation by Photo-Elasticity", Scientific research, Engineering, 2011, 3, 935-941