Design & Analysis of various cross section of Crank hook by using ANSYS Software

[1]MD IRSAD ANSARI, [2]DR. MANISH GANGIL

M.Tech.Scholar[1], *Professor*[2]

Department of Mechanical Engineering, RKDF, University Bhopal, (M.P.) India.

[1]mdirsadansari786@gmail.com, [2]rkdfbhojpal@gmail.com,

Abstract

These Crane hook is very significant component used for lifting the load with help of chain or wire ropes. Crane hooks are highly liable components that are typically used for industrial purposes. Failure of a crank hook mainly depends upon various geometric variables as well as material properties. In this analysis the material properties of a hook kept constant throughout the analysis and stress is to be reduced by varying different geometric parameters. The selected sections are Circular, Trapezoidal and Tsection. The area remains constant while changing the dimensions for the three different sections. The crane hook is modeled using CATIA V5 software. The stress analysis is done by using ANSYS 14.5 workbench Educational version. It is found that T cross section yields minimum stresses at the given load for constant cross section among three cross sections. The stress distribution pattern is verified for its correctness on model of crane hook using Winkler-Bach theory for curved beams.

Keywords:- ANSYS 14.5, CATIA V5, Crane Hook, Static analysis, Winkler-Bach Theory, Stress.

.

I. Introduction

The structure strength is the important characteristic to respond the load bearing capability of the elevating equipment. Hook of a crane is a curved type of bar used for lifting the heavy loads in the cranes. In order to reduce the structure failure of a crane hook, induced stresses are analysed. Fatigue damage is the initiation of crack due to fluctuating loads. It is caused due to stress levels which are insufficient to cause damage in a single application. It is a highly responsible and important component used for industrial applications Crane hooks are generally prone to failure due to concentration of stress. It is an element to elevate the loads in constructional sites and industries.

A crane is a kind of machine, loaded with a hoist, wire and ropes or chains and sheaves used to lift and move heavy materials. Some of the types of cranes are: overhead crane, mobile crane, telescopic crane, gantry crane, jib crane, deck crane, and loader crane. Hook is used to grab and lift the loads. It is a hoisting fixture designed to engage a ring or link of a lifting chain or the pin of a shackle or cable socket. These are designed for maximum performance without failure.

1.2 History of Finite Element Analysis

The Finite Element Analysis (FEA) commenced its journey from 1943. R. Courant, used the Ritz method of numerical analysis and minimization of variation calculus to generate approximate solutions to vibration systems. The paper published in 1956 by M. J. Turner, R. W. Clough, H. C. Martin, and L. J. Topp established a broader definition of numerical analysis. The paper centred on the "stiffness and deflection of complex structures".By the 70's, finite element analysis had its limitations to expensive mainframe computers generally owned by the aeronautics, automotive, defence, and nuclear industries. Due to rapid decrement in the cost of computers and the phenomenal increment in computing power, finite element analysis has been developed to an incredible precision. Finite element method consists of a soft model of a design that is stressed and analysed for specific results. It s used n design of new product and existing product refinement. A company is able to verify a proposed design's ability to perform the client's specifications prior to manufacturing or construction.

Modifying the design of a product or structure is made to make the product or structure capable for a new service condition. For structural failures, finite element analysis is used to determine the modifications in design to meet the new requirements.

Generally two types of analysis are used in industry: 2-D modeling, and 3-D modeling.

2-Dimensional modelling has simplicity and permits the analysis to be run on a relatively normal computer, yielding a less accuracy in results.

3-Dimensional modelling generates highly accurate results while sacrificing the ability to run on all but the fastest computers effectively. Within each of these modelling schemes, the programmer can insert numerous algorithms (functions) which may make the system behave linearly or non-linearly. Linear systems are not much complex and usually do not take into account for plastic deformation. Non-linear systems do account for plastic deformation, and many also are capable of testing a material all the way to fracture.

1.2.1 Finite Element Analysis

FEA uses a complex system of points called nodes which make a grid called a mesh (Figure 31). This meshing is generated to contain the material and structural properties which defines the structure to certain loading conditions. The nodes are assigned at a certain density throughout the materials depending on the anticipated stress levels of a particular area. Regions receiving

large amounts of stress usually have a higher node density than those which experience little or no stress. These points of interest may consist of fracture point of previously tested material, fillets, corners, complex detail, and high stress areas. The meshing behaves like a web from each node, and here extends a mesh element to each of the adjacent nodes. This web of vectors carries material properties to the object, creating many elements.

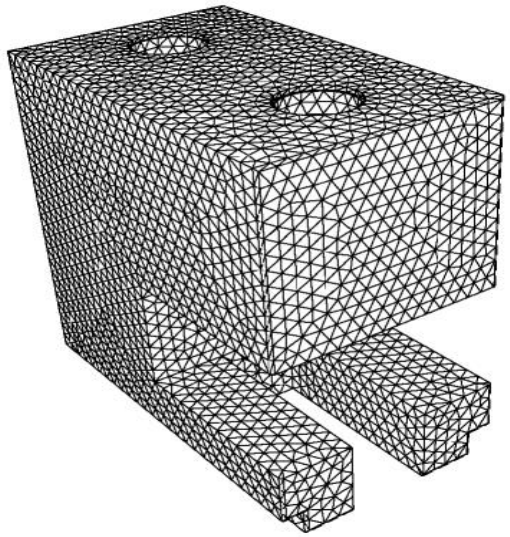

Fig 1: - Tetrahedral Mesh

2 Literature Review

Kunjan B. Vanpariya , Vishal Pandya , (2016) Lifting hooks are the components which are generally used to elevate the heavy load in industries and constructional sites. It is the member used for lifting the heavy mass using wire ropes and crane. The main objective of this paper is to establish a methodology to design and stress analysis of a lifting hook using stress distribution and changing the cross sections where the stress is minimum. A survey of literature for this subject has shown that, various CAD and FEA software used for modeling and analysis of a hook. The lifting hook is given by company and prepared the model in in pro-e wildfire 5.0 and analysis done in ansys 14. Using FEM, weight optimization done in ANSYS software and remove the material, where the stress is minimum. For Weight Optimization Of Lifting Hook, MATLAB software is used for Genetic Algorithm and Prepare a new Model According to Optimization And Analysis done in Ansys 14.0 for 1.5 tonne load. [1]

B. Nagaraju, M. Rajaroy, P. Venkatesh Reddy, K. Satyanarayana (2015) Crane hooks are highly liable components that are typically used for industrial purposes. Thus such components in an industry must be manufactured and designed in a way to deliver maximum performance without failure. Failures of a crane hook mainly depends on three major factors i.e. dimension , material , overload .The project is concerned towards increasing the safe load by varying the cross sectional dimensions of the three different sections .The selected sections are rectangular ,triangular, and trapezoidal . The area remains constant while changing the dimensions of the three different sections. The crane hook is modelled using PTC CREO software. The stress analysis is done using ANSYS 14.5 workbench. The normal stress along y direction, deformation along y direction and strain is considered. It is found that trapezoidal cross section yields maximum load of 700 kg for constant cross section area among three cross sections. [2]

Santosh Shaun and Ritesh Dewangan (2012), movable cranes of the hand operated type were in use in the 1880's. About this time complicated designs of powered motion were offered by English and American builders, involving a driving shaft along the runway and multiple clutches for transferring the power of the driving shaft to the hoist, trolley or bridge motions. The study of crane hook for trapezoidal cross section, finite element analysis done by santosh Shaun , in this paper CAE is used to shorten the development FEM concept is used in the locking system of the crane hook stress factor is distributed uniformly [3].

Govind Narayan Sahu1, Narendra yadav (2011) Hooks are employed in heavy industries to carry 4 tonnes of loads safely. These hooks have a big role to play as far as the safety of the crane loaded is concerned. With more and more industrialization the rate at which these hooks are forged are increasing. 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 catia v5. 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. [4]

M. Shaban et. al. (2013), studied stress influence of crane hook in its loaded condition, a solid model of crane hook is prepared with the help of ABAqUS design software. Real time influence of stress concentration in 3D model of crane hook is obtained. The stress distribution is verified for its correctness on an acrylic model of crane hook using shadow optical method (Caustic method) set up. [5]

E. Narvydas et. al (2012), investigated circumferential stress concentration factors with shallow notches of the lifting hooks of trapezoidal cross-section employing finite element analysis method. The stress concentration factors were widely used in strength and durability evaluation of structures and machine elements. The FEA results were used and fitted with selected

generic equations. This yields formulas for the fast engineering evaluation of stress concentration factors without the usage of finite element models. The design rules of the lifting hooks require using ductile materials to avoid brittle failure; in this respect they investigated the strain based criteria for failure, accounting the stress triaxiality. [6]

SpasojeTrifkovic' et. al (2011), this paper analyzes the stress state in the hook using approximate and exact methods. They calculated stresses in various parts of the hook material firstly by assuming hook as a straight beam and then assuming it as a curved beam. Analytical methods were used with the help of computers, using FEM.[7]

Bhupender Singh et al (2011), Work presented involves the solid modeling and finite element analysis of crane boom has been done using PRO/E WILDFIRE 2.0 and ALTAIR HYPER MESH with OPTISTRUCT 8.0 SOLVER Software to get the variation of stress and displacement in the various parts of the crane boom and possible actions are taken to avoid the high stress level and displacement. By using Finite Element Analysis the following objectives have been achieved.

- Weight Reduction, (4.86 kg, approx.5kg).
- Stresses are within limits (at higher load points).
- Price cutting (Rs-180/- for a single component).
- The analysis also concluded that maximum stress is coming near the fixing position. [8]

Y. Torres et. al (2010), initially studied the probable causes which led to a failure of the crane hook in service. The study of accident includes: details of the standards governing the manufacturing and use of lifting hooks, experimental analysis, mechanical behaviour of steel of reported hook and simulation of the thermal history of the hook. It was concluded that the accident was caused by the strain-aging embrittlement of the used steel. The brittle fracture was originated from a crack in the material, generated during the welding performed on the lifting hook.[9]

Takuma Nishimura et. al (2010), studied the damage estimation of crane-hooks. They judge the value the load conditions which were assumed to be crucial to the crane-hook damages. 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 deadly 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 simulated deformation values in the database. On the basis of these calculated load conditions, the critical load condition for the crane-hook was estimated as a statistical distribution based on the Bayesian approach.[10]

C. Oktay AZELOGLU et. al (2009), this paper presents the different methods of stress calculation for lifting hooks based on different assumptions. They applied curved beam theory, Finite Element Method and photo elasticity experiments to obtain the stress field on the hook. As a result, different methods used to obtain the stress field on the hook are compared. Some recommendations were suggested for lifting hook calculations on the field applications.[11]

Yu Huali et. al (2009), the structure-strength is the key index to response the load-bearing ability of the elevating equipment. Researching and analyzing the static characteristic of the hook that functions at the limited load has an important meaning to design larger tonnage hook correctly. In this study, hook of drill well DG450 were analyzed. Firstly, based on the characteristic modeling technology, the 3-D entity model of the hook was built using Pro/E. Secondly; the static analysis on three dangerous work conditions at ultimate load of the hook was preceded by FEM software ANSYS. This work illuminates the instructional meaning and engineering application value to the design and development of the larger tonnage drill well hook.[12]

Bernard Ross et. al (2007), This paper describes the comprehensive engineering analysis of the crane accident, undertaken to disprove the Mitsubishi theories of failure as confirmed by jury verdict. Among the topics discussed were: wind tunnel testing, structural analyses of the boom, metallurgy of failed parts from a critical king-pin assembly, and soils engineering work related to ground loads and displacements during the lift. Crucial role of the SAE J1093, 2% design side load criterion and Lampson's justification for an 85% crawler crane stability criterion were presented. [13]

3. Modeling

A CAD model of crane hook has been drafted and designed with the suitable parameters for the cross sections like:

- Circular
- Triangular
- Trapezoidal
- Rectangular

The drafted CAD model is imported to FEA (ANSYS) software and analysed for stress concentration and deformation behaviour of the material with the given cross section.

The CAD model of the EN9 steel is manufactured; the following are the properties of the material:

Table 3.1 Properties of the material used

PROPERTY	VALUE
Young's modulus (MPa)	2.e+005
Bulk modulus (MPa)	1.6667e+005
Poisson's ratio	0.26
Shear modulus (MPa)	76923
Tensile yield strength (MPa)	250
Tensile ultimate strength (MPa)	460
Density	7.85 χ 10-6 kg /mm3
Coefficient of thermal expansion	1.2 χ 10-5 / 0C
Specific heat	4.345 mJ /kg/ 0C
Thermal conductivity	6.05-2 W /mm/ 0C

Table 3.2 Values of link radius for various cross sections

CROSS SECTION	LINK RADIUS
Circular	$h^2 = \frac{d^2}{16} + \frac{1}{8} \times \frac{d^4}{16R^2}$
Triangular	$h^2 = \frac{R^3}{A} \times \frac{B}{D}\left[2.3R_2 \log \frac{R_2}{R_1} - D\right] - R^2$
Trapezoidal	$h^2 = \frac{R^3}{A}\left[2.3\left(B_2 \frac{(B_1 - B_2)R_2}{D}\right)\log \frac{R_2}{R_1} - (B_1 - B_2)\right.$
Rectangular	$h^2 = 2.3 \frac{R^3}{D} \log\left(\frac{2R + D}{2R - D}\right)$

To analyse the stresses and deformation in the crane hook, initially a CAD model is generated on the computer and it is imported to FEA software. Through finite element analysis meshing is generated on the model to prepare the nodes. Hook is made fixed from the top end and the load is applied on it to study the flow behaviour of the material.

4.2 **Modelling and designing of crane hook of circular cross sections showing forces on it**

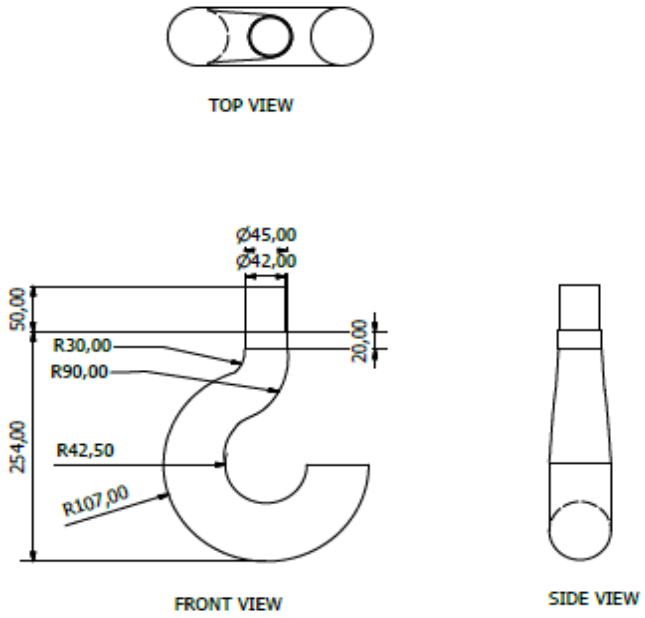



Fig. 2 2D model of circular cross section crane hook

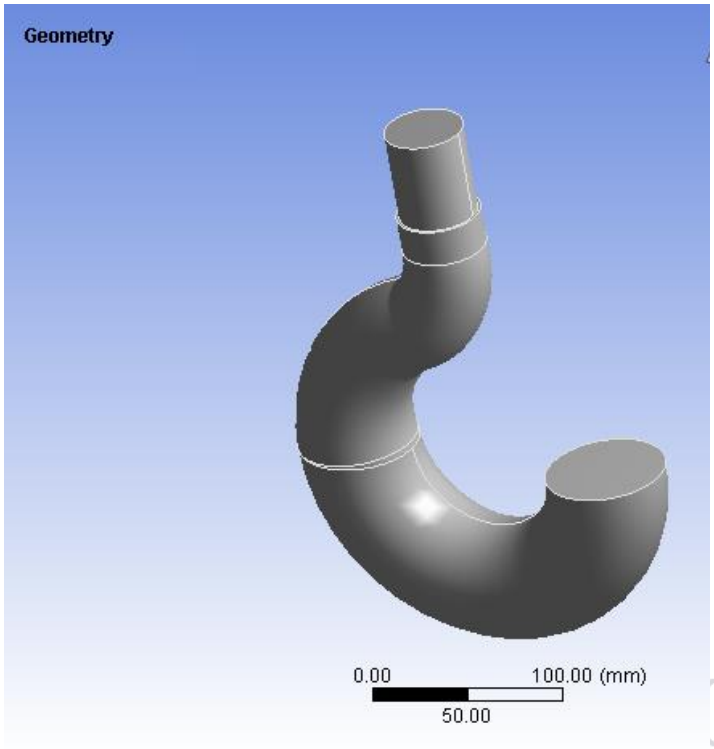

Fig.3 CAD model of circular cross section crane hook

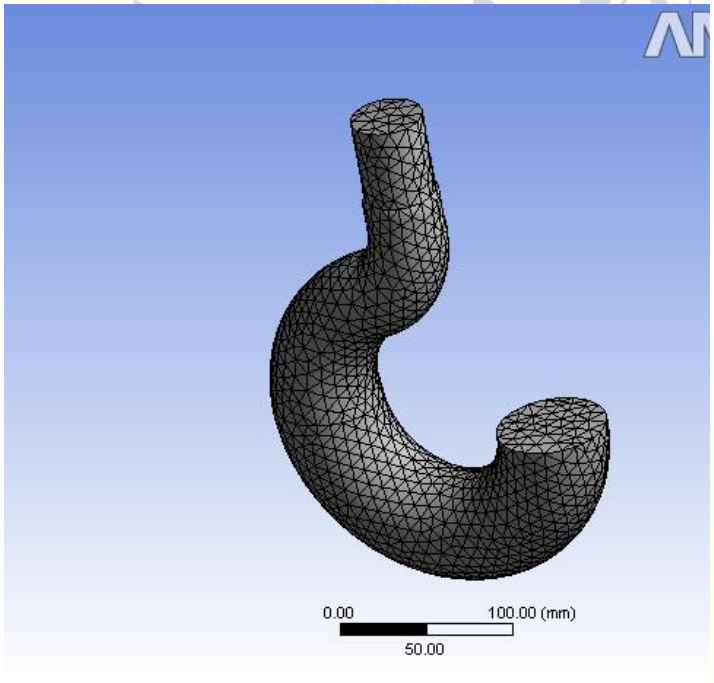

Fig.5 Meshing shown in circular cross section crane hook

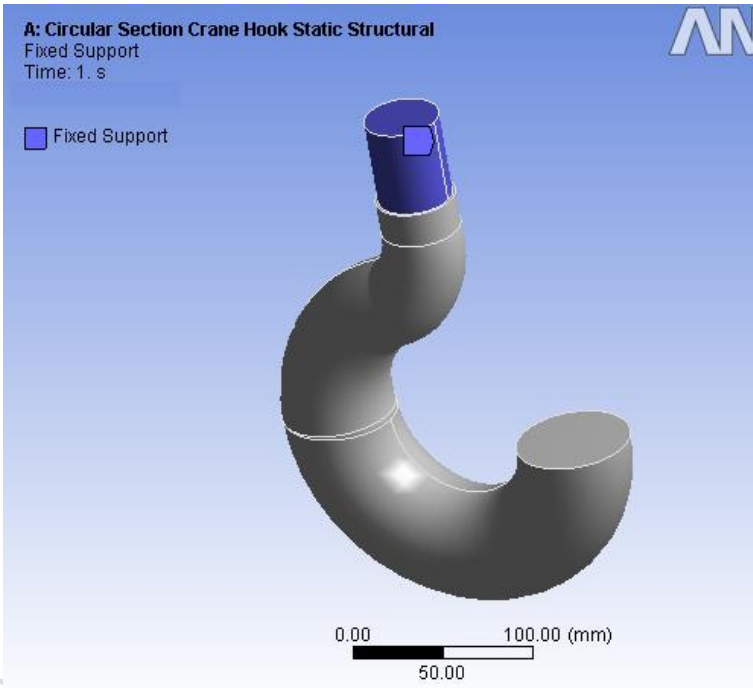

Fig 4 Fixed supports shown in circular cross section crane hook

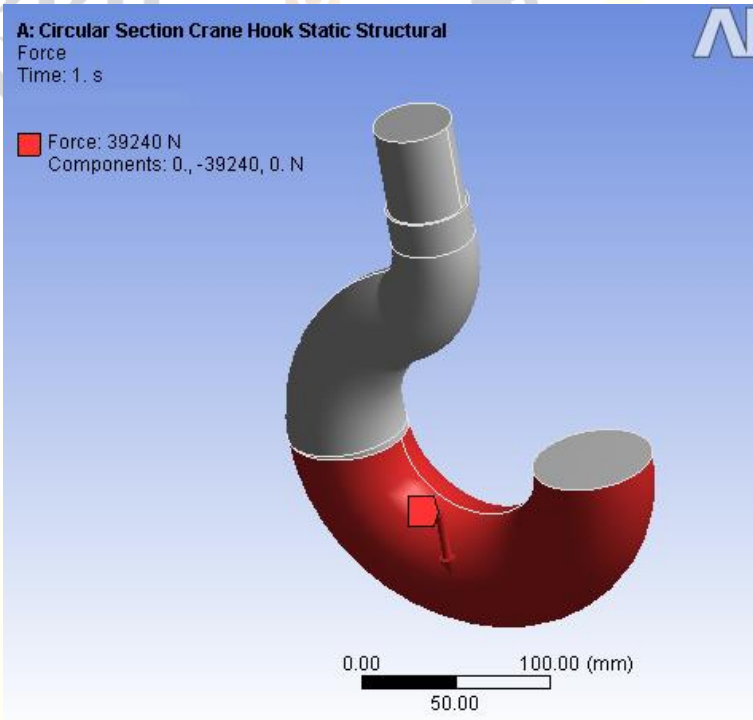

Fig.6 Force shown in circular cross section crane hook

4. Result & Discussion

5.1 Stress analysis of various geometries through finite element method

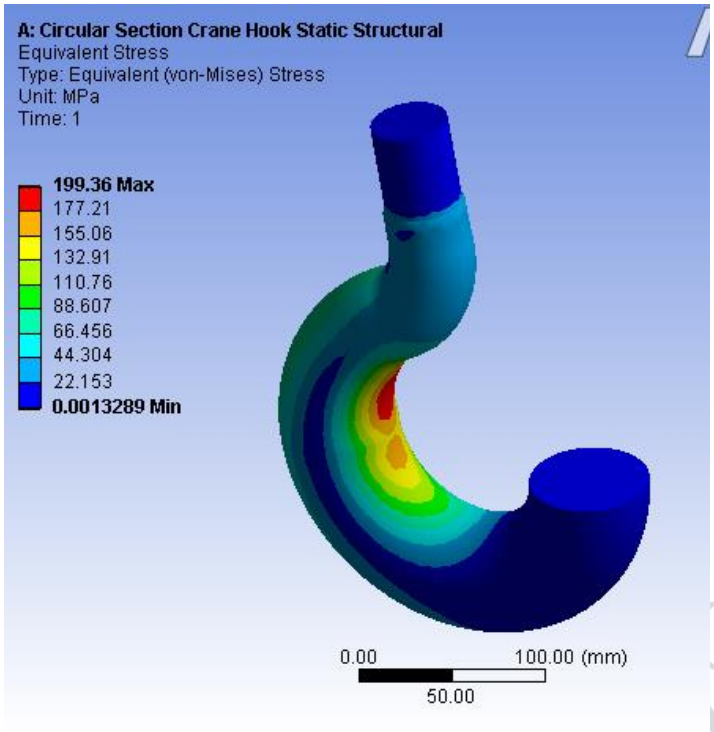

Fig 7 Stress analysis of circular cross section crane hook

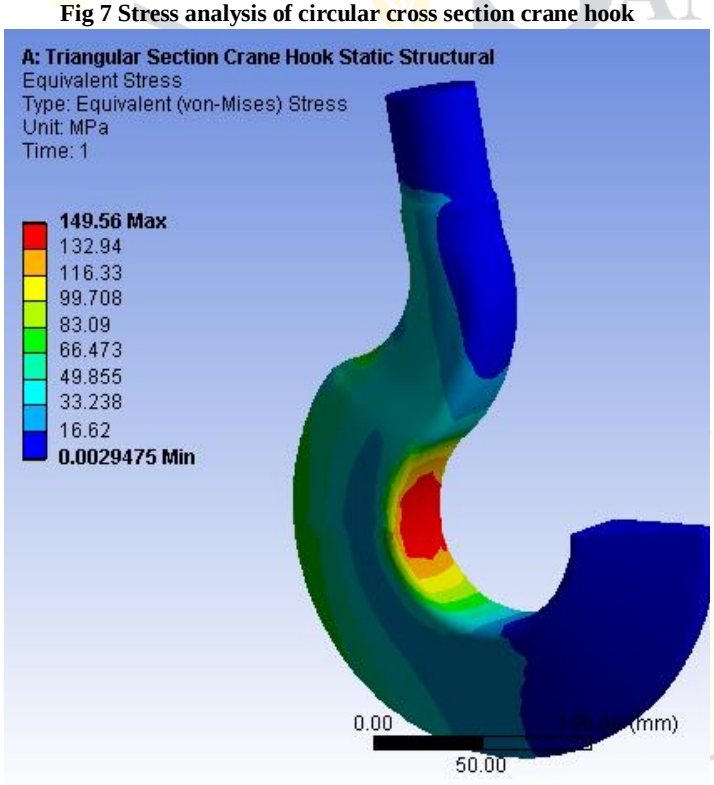

Fig. 8 Stress analysis of triangular cross section crane hook

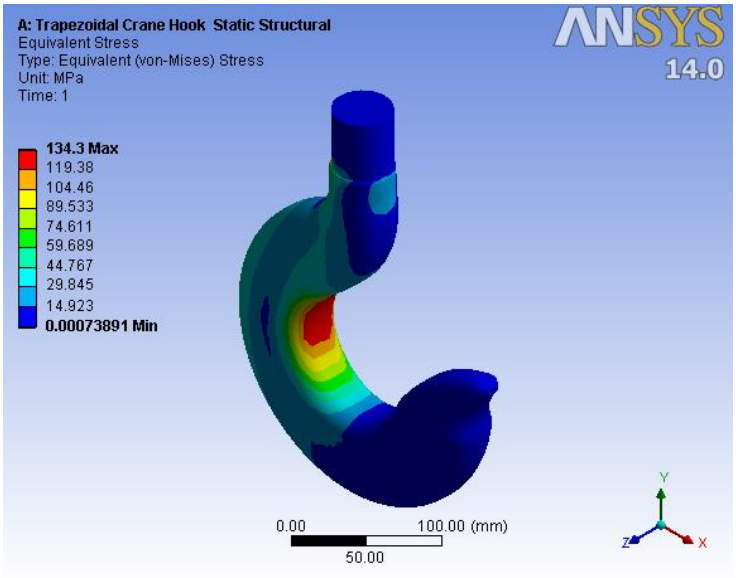

Fig 5.3 Stress analysis of trapezoidal cross section crane hook

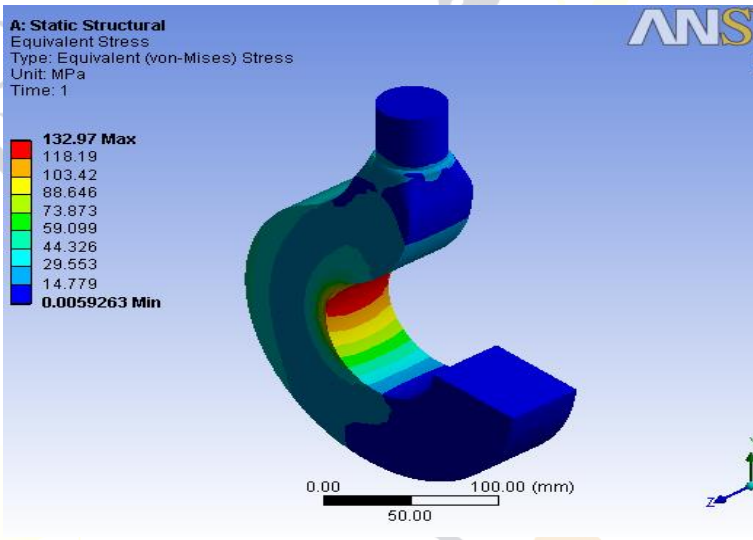

Fig.9 Stress analysis of rectangular cross section crane hook

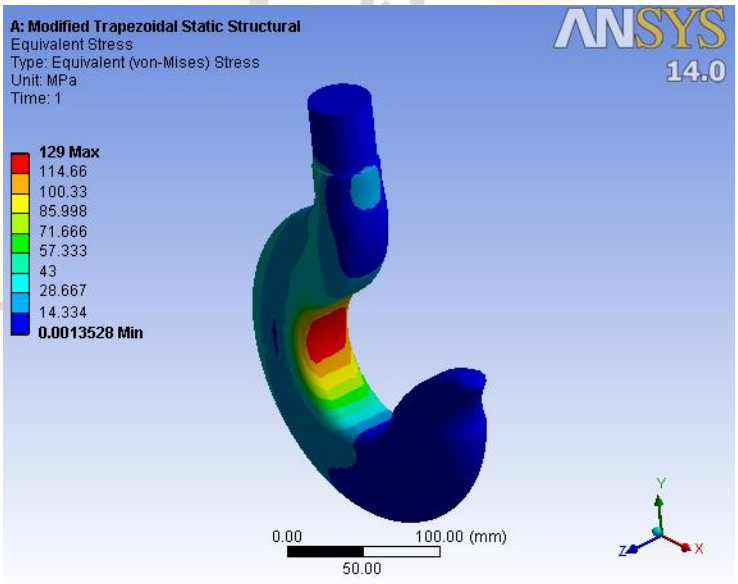

Fig.10 Stress analysis of modified trapezoidal cross section crane hook

Table 1 Von Mises stress values for various geometries

S.No.	Geometry	Von Mises Stress (Max.)
1	Circular	199.39 MPa
2	Triangular	149.56 MPa
3	Trapezoidal	134.3 MPa
4	Rectangular	132.97 MPa
5	Modified Trapezoidal	129 MPa

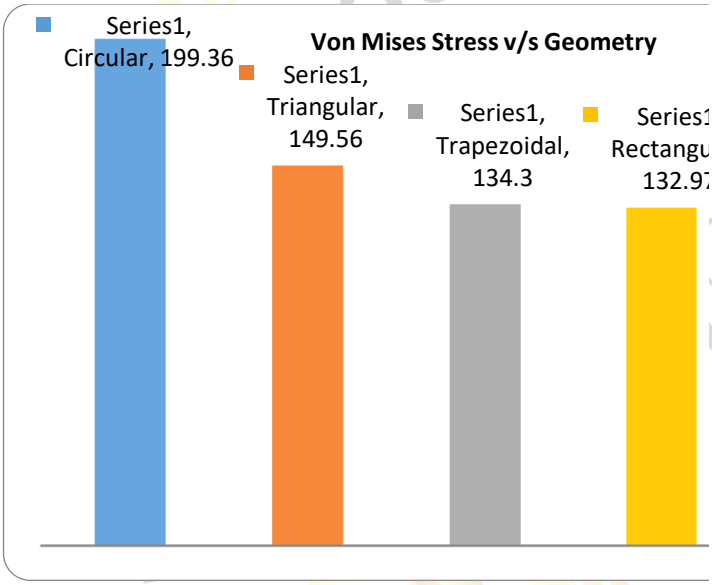

Fig.11 Graphical comparison of Von Mises stresses of various geometries

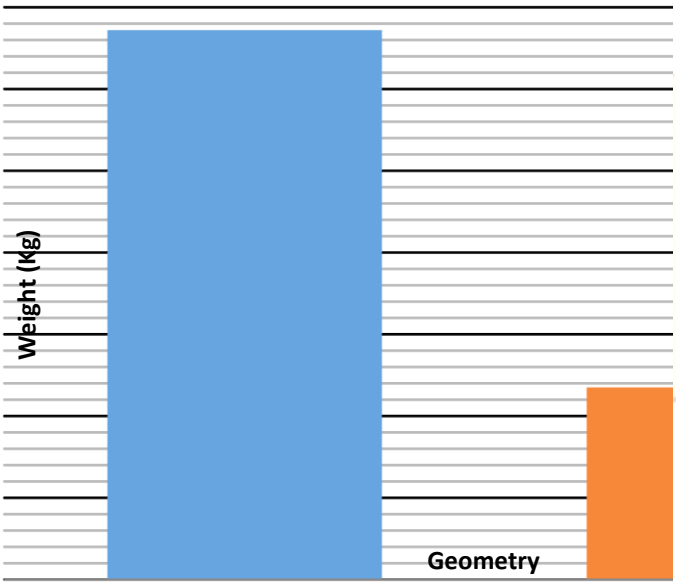

Fig.12 Graphical comparison of weights of trapezoidal and modified trapezoidal cross section

5.2 Deformation analysis of various geometries through finite element method

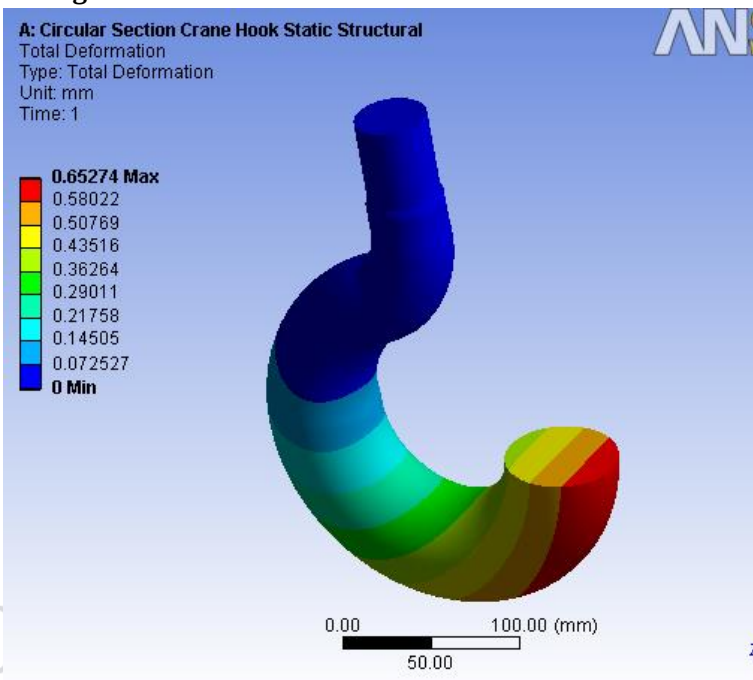

Fig.13 Deformation analysis of circular cross section crane hook

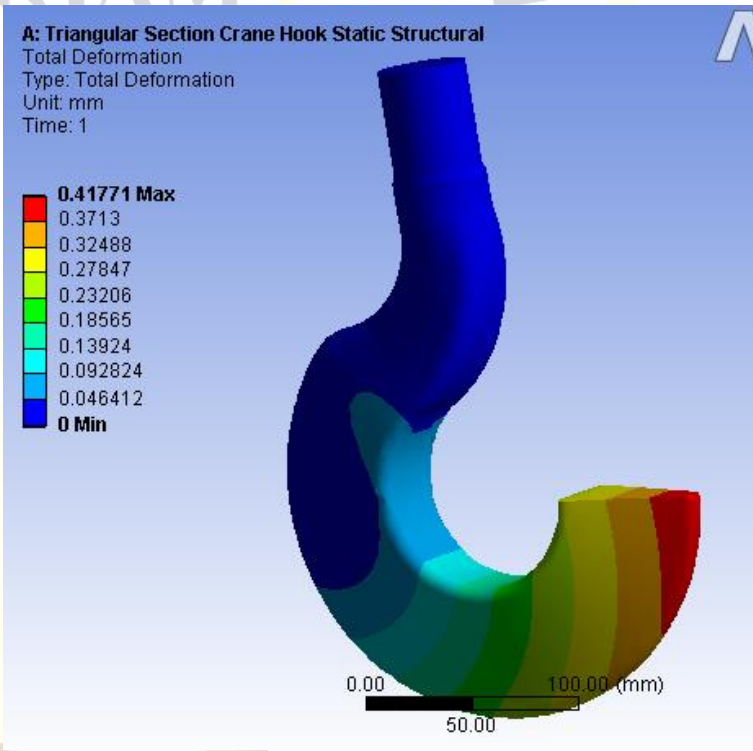

Fig.14 Deformation analysis of triangular cross section crane hook

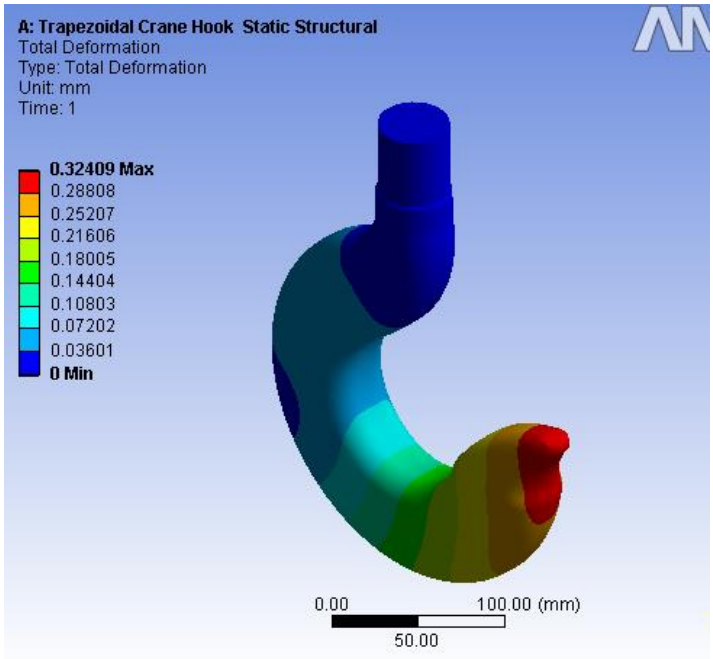

Fig.15 Deformation analysis of trapezoidal cross section crane hook

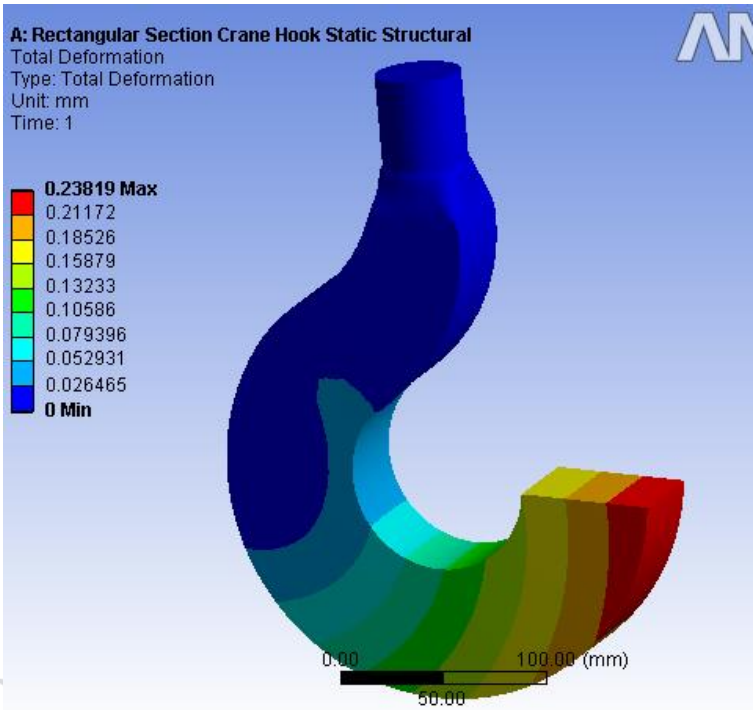

Fig 5.12 Deformation analysis of rectangular cross section crane hook

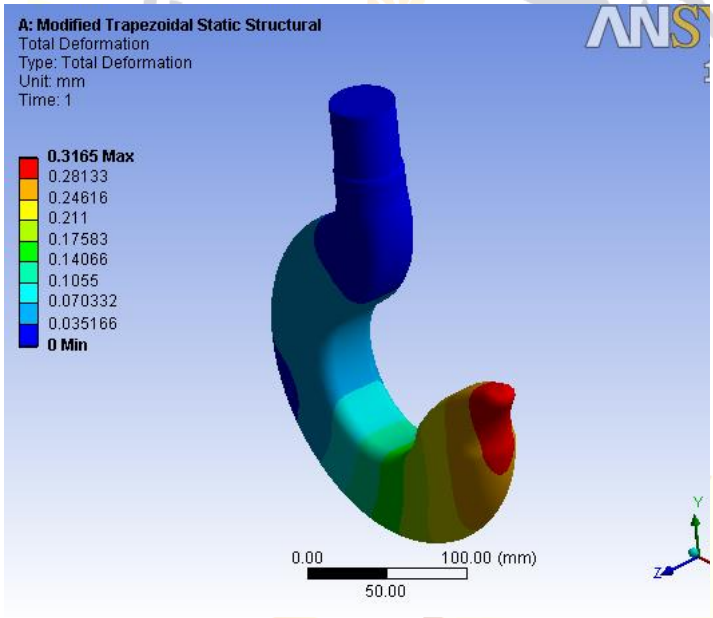

Fig.16 Deformation analysis of modified trapezoidal cross section crane hook

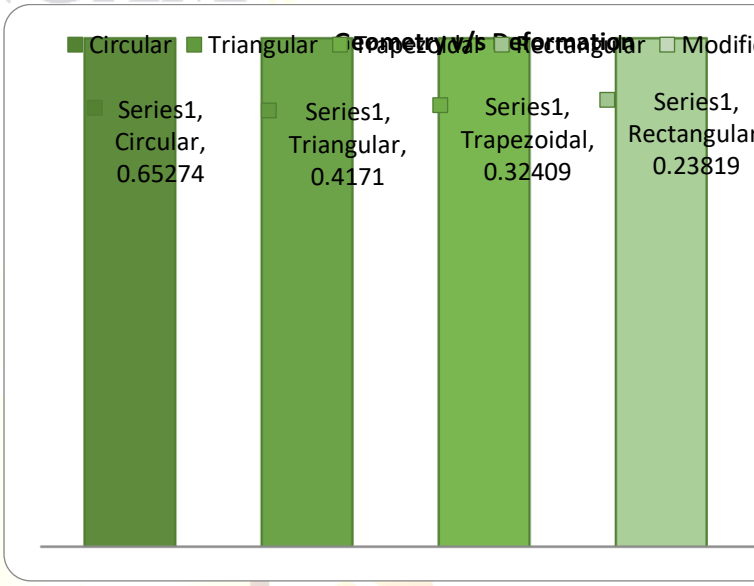

Fig.17 Graphical comparison of the deformation in various geometries

4.3 Cost analysis of the material used

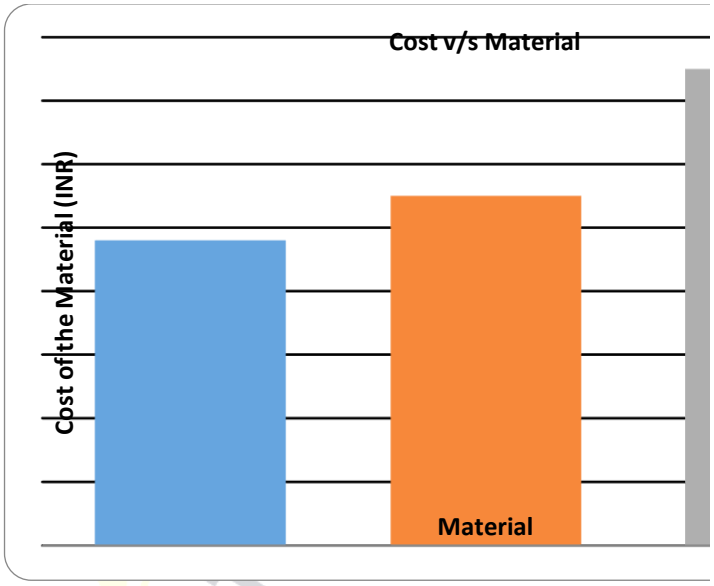

Fig.17 Cost comparison of various materials to design a crane hook

Designing of a crane hook needs a higher level of analysis to study the flow behaviour of the material i.e. stresses induced in the material due to applied loads. Analysis has been performed on various cross sections to observe and optimize the effect of cross sections on the induced stress levels. Cost is also a key player in a mechanical component design. It is highly needed to optimize the cost of material being at the same strength. Optimization of cost requires a thorough knowledge of the materials.

For the given loading conditions, three materials are being compared:

- High strength low alloy steel
- Carbon steel
- Alloy steel

Cost of the materials is analysed as these are at par with each other when strength is compared. It was calculated that the high strength and low alloy steel is best suited for the crane hook design as it exhibited same strength as of others but shows a considerable reduction in cost, reducing the overall cost of manufacturing a crane hook.

High strength low alloy steel has better mechanical properties. It exhibits higher resistance to corrosion and wear. The carbon content varies from 0.05% to 0.25%. They can be alloyed with other elements to have desired properties.

4.4Comparison study with base paper

In this study we are comparing our design product by huck which result are indicated stress induced and deformation in "Modified Curved Hook" is least because of curved shape and fillet edges as well as stress concentration are distributed uniformly. It have less mass due to this we are able to save the material and balance economy.

But our design is safe and result are indicated stress induced and deformation in "Modified Curved Hook" is least because of curved shape and fillet edges as well as stress concentration are distributed uniformly. It have less mass due to this we are able to save the material and balance economy. We have done our study same material like steel and same load 4 tonne.

Table. 2Comparative Analysis Our model to Exiting modal

S.No	Cross section	Our model Design Deformation (mm)	Exiting model Design Deformation (mm)
01	Curved Hook	0.32	0.32
02	Modified Curved Hook	0.31	0.326

So we can say that our design is safe and because it has less deformation Value of deformation is our design 0.31 mm and exiting model value of deformation is 0.32 mm.

Conclusion

The analysis of the crane hook has been carried over FEA software. Drafting of the hook model is carried over ANSYS and analysed for maximum principal stresses and deformation in the hook. A load of about 4 ton is applied on the hook. Cross sections like circular, triangular, trapezoidal and rectangular are used to apply the load. A study was done on these models to see the effect of stresses and flow behaviour of the material due to change in cross sections and following result is obtained:

- Trapezoidal cross sections obtained with the minimum value of the principal stress of 129 MPa.
- It is also seen that the material flow in the case of trapezoidal cross section is also comparatively lower.
- Trapezoidal cross section is also studied along with some design modifications in it. Fillet radius is changed from 6mm to 14mm and it is observed that the magnitude of stress decreased considerably to 129 MPa and a decreased value of deformation in hook is also observed.

References

1. Design Analysis And Weight Optimization Of Lifting Hook", , Volume 3, Issue 12, December 2016
2. B. Nagaraju, M. Rajaroy, P. Venkatesh Reddy, K. Satyanarayana, "Stress Analysis of Crane Hook Using FEA", International Journal, of Current Engineering And Scientific Research (IJCESR), 2394- 0697, Volume-2, ISSUE-2, (2015)
3. Santosh Shaun, Ritesh Dewangan, Manas Patnaik, Narendra Yadav; " Study of Crane Hook Having Trapezoidal Section by Finite Element Method & Design of Experiments" International Journal of Modern Engineering Research Vol.2 Issue pp-2779-2781 ISSN: 2249-66454 July-Aug 2012.
4. Govind Narayan Sahu1, Narendra yadav2 "Design and Stress Analysis of various cross section of Hook" International Journal of Modern Engineering Research (IJMER), Vol. 3, Issue. 4, Jul - Aug. 2013 ISSN: 2249-6645, pp-2187-2189
5. M. Shaban, M. I. Mohamed, A. E. Abuelezz and T. Khalifa, "Determination of Stress Distribution in Crane Hook by Caustic", International Journal of Innovative Research in Science, Engineering and Technology, Vol. 2, Issue 5, May 2013.
6. E. Narvydas, N. Puodžiūnienė, "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
7. SpasojeTrifković, NebojšaRadić et. al, "Stress analysis of crane hook using FEM", INFOTEH-JAHORINA Vol. 10, Ref. C-2, p. 244- 248, March 2011.
8. Bhupender Singh, Bhaskar Nagar, B.S. Kadam and Anuj kumar, "Modeling and Finite Element Analysis of Crane Boom", International Journal of Advanced Engineering Research and Studies, Vol. I/ Issue I/October-December ,pp 51-52, 2011
9. Y. Torres , J.M. Gallardo , J. Domínguez , F.J. Jiménez E, "Brittle fracture of a crane hook", Engineering Failure Analysis 38–47,2010
10. Takuma Nishimura, Takao Muromaki et. al, "Damage Factor Estimation of Crane Hook (A Database Approach with Image, Knowledge and Simulation)", 4th International Workshop on Reliable Engineering Computing (REC 2010).
11. C. Oktay AZELOĞLU, Onur ALPAY, Investigation Stress of A Lifting Hook with Different Methods, "Verification of The Stress Distribution with Photo elasticity Experiments", Electronic Journal of Machine Technologies, Vol: 6, No: 4, pp (71-79),2009
12. Yu Huali, H.L. and Huang Xieqing, "Structure-strength of Hook with Ultimate Load by Finite Element Method", Proceedings of the International Multi Conference of Engineers and Computer Scientists, 2009 Vol II IMECS 2009, March 18 - 20, 2009, Hong Kong.
13. B. Ross, B. McDonald and S. E. V. Saraf, "Big Blue Goes Down. The Miller Park Crane Accident," Engineering Failure Analysis, Vol. 14, No. 6, pp. 942-961, 2007.
14. R. S. Khurmi, "Strength of materials" 23rd Edition Chapter 33 (2009).

1.