ISSN : 2455-9679
Impact Factor : 1.389

IJRTSM

INTERNATIONAL JOURNAL OF RECENT TECHNOLOGY SCIENCE & MANAGEMENT

"A REVIEW ON DESIGN ANALYSIS & WEIGHT OPTIMIZATION CRANE HOOK"

Ankit Sahu[1],Dr. S.S.Chauhan[2],Asst. Prof. Ranjeet Kumar[3]

[1]M. Tech Scholar, Mechanical Engineering, Vaishnavi Institute of Technology & Science
Bhopal,(M.P.), India
[2]Guide & Head, Mechanical Engineering, Vaishnavi Institute of Technology & Science
Bhopal,(M.P.), India
[3]Co-guide & Assistant Professor, Mechanical Mechanical Engineering, Vaishnavi Institute of Technology & Science
Bhopal,(M.P.), India

ABSTRACT

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. Fatigue failure 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. A survey of literature for this subject has shown that, various CAD and FEA software used for modeling and analysis of a hook and optimizes the performance of the crane hook based on stress, geometry, and weight.

KEYWORDS : *Crane hook, , Fatigue failure, FEA, Stress, CAD, Weight*.

I. INTRODUCTION

The structure strength is the important characteristic to respond the load bearing capability of the elevating equipment. Crane hook is a curved bar used to lift the loads in the cranes. In order to reduce the structure failure in the 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. [1]
Crane hooks are highly liable components and are always subject to failure due to accumulation of large amount of stresses which can eventually lead to its failure. Crane hooks are the components which are generally used to elevate the heavy loads in industries and constructional sites. [2]
A crane is a machine, equipped with a hoist, wire ropes or chains and sheaves used to lift and move heavy materials. Overhead crane, mobile crane, telescopic crane, gantry crane, jib crane, deck crane, loader crane are some of the commonly used cranes.[3]
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. Crane hooks with trapezoidal, rectangular, circular of triangular cross sections are commonly used. The hooks must be designed to deliver maximum performance without failure. [4]

ISSN : 2455-9679
Impact Factor : 1.389

Hoisting is the process of lifting something or some load or person from lower position to higher position with the help of some device or mechanisms known as hoisting devices or mechanisms. A hoisting device is used for lifting or lowering a load by means of a drum or lift-wheel around which rope or chain wraps. It may be manually, pneumatically of electrically operated or driven and may use chain, fibre or wire ropes as its medium. [5]

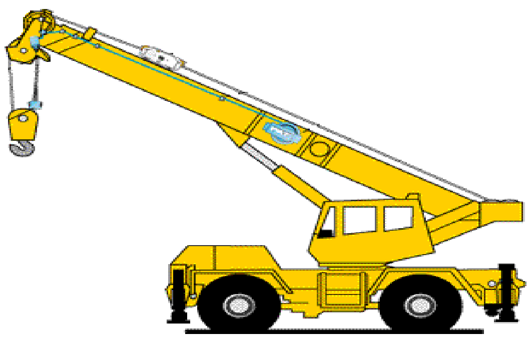

Figure.1

Hoisting part of a crane includes:

- Hoist motor
- Gear box
- Drum
- Pulleys
- Wire rope
- Hook

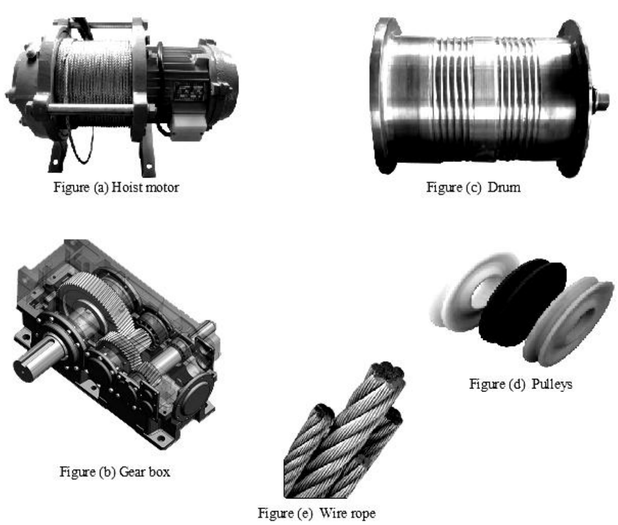

Figure.2

II. FAILURE OF CRANE HOOKS

Due to continuous working of crane hook nanostructure of crane hook are changes and some problems like weakening of hook due to wear, tensile stresses, plastic deformation due to overloading and excessive thermal stresses these are some other reason of failure. Hence continuous working of crane hook may increase the magnitude of these stresses and eventually result in failure of crane hook. Due to some design modification, and changing the materials. All the above mentioned failures may be prevented.[1]

III. LITERATURE SURVEY

In this section, contribution of different researchers is discussed.

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 proe wildfire 5.0 and analysis done in ansys 14. Using FEM, weight optimization done in ansys 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. Failure 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 differentsections. 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), Travelling 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. Study of crane hook for trapezoidal cross section finite element analysis 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]

M. Shaban et. al (2013), studied the stress pattern of crane hook in its loaded condition, a solid model of crane hook is prepared with the help of ABAQUS 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 shadow optical method (Caustic method) set up. [4]

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 (FEA). 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 equation. 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.[5]

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.[6]

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).
- Cost cutting (Rs-180/- for a single component).

The analysis also concluded that maximum stress is coming near the fixing position. [7]

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.[8]

Takuma Nishimura et. al (2010), studied the damage estimation of crane-hooks. They estimated 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 fatal 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 the 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.[9]

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

ISSN : 2455-9679
Impact Factor : 1.389

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.[10]

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.[11]

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.[12]

IV. ANALYTICAL METHOD FOR STRESS CALCULATION

Curved beam flexure formula is used when the curvature of the member is pronounced as in case of hook for different cross sections mathematical analysis of stress [13].

$$\sigma = \frac{F}{A} + \frac{M \times y}{I}$$

Where, *M*=maximum bending moment.

Y=Distance between centroidal axis to neutral axis.

I=Moment of inertia for different cross sections.

Table: 1 Values of Radius of neutral surface for various cross

S.NO	GEOMETRY	VALUES
1.	RECTANGLE	$R = h / In(r_2 /r_1)$
2.	CIRCLE	$R = 1/2 (\bar{r} + / \{ \bar{r}^2 \}^{1/2} - \{c^2\}^{1/2})$
3.	TRIANGLE	$R = h^2 /2 / \{ r_2 In (r_2/r_1) - h \}$
4.	TRAPEZOID	$R = h^2/2 \times (b_1 +b_2) / (b_1r_1 – b_2r_2)$ $In (r_2/r_1) – h (b_1 – b_2)$

V. CONCLUSION

Static analysis is helpful for understanding and improving the operating performance of the crane hook. Static analysis has very important significance for the life of the specific parts. Specific process will be no longer introduced. This concept of Static analysis is followed by number of researches for their application and Finite Element Method (FEM) is one of the most effective and powerful method for the stress analysis of the crane hook. This review provides the background of crane hooks to carried out further research work in same era.

REFERENCES

1. Kunjan B. Vanpariya , Vishal Pandya, "Journal of Emerging Technologies and Innovative Research (JETIR)" " 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. 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.
5. 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.
6. SpasojeTrifković, NebojšaRadić et. al, “Stress analysis of crane hook using FEM”, INFOTEH-JAHORINA Vol. 10, Ref. C-2, p. 244- 248, March 2011.
7. 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
8. Y. Torres , J.M. Gallardo , J. Domínguez , F.J. Jiménez E, “Brittle fracture of a crane hook”, Engineering Failure Analysis 38–47,2010
9. 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).
10. 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
11. 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.
12. 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.
13. R. S. Khurmi, “Strength of materials” 23rd Edition Chapter 33 (2009).