ISSN : 2394-2975 (Online)
ISSN : 2394-6814 (Print)

Study and Analysis of Crane Hook in Loading Area

[I]Vivek Mahadev Thakur, [II]Vaibhav Pawar, [III]Dr. M.D. Nadar, [III]Sanjay Ghorpade, [III]Sahil Patil
[I,II,III]Pillai HOC College of Engineering, Rasayani, Panvel, Maharashtra, India
[I]E-mail : vivekthkr619@gmail.com

Abstract

Crane hook are using in industry like nuclear power plant, thermal power plant, atomic power plant, etc for holding, shifting heavy loads from one place to another place and some heavy load accessories in various types of plants etc., Life of the crane hook is defined as heat flux deformation, crack of the swing cross section etc., life of crane hook mainly depends on crane hook material mechanical properties, load, frequency of loads. In this paper heat flux deformation is studied under heavy load material handling process. The analysis of heat flux deformation is carried out by using theoretical experimental simulation process. In this simulation process is carried out by developing crane hook solid mode by using solid works, analysis of solid model based on heat flux load deformation by using software.

Keywords

Crane Hook, Loading Area, Nanostructure, Failure Heat Flux.

I. Introduction

Crane hook are generally used to elevate the heavy loads, heavy machineries, man, material in industries and constructional sites. Recently, excavators having a crane-hook are widely used in construction works site. Excavator is used to perform the conventional digging tasks as well as the suspension works. There are many cases that the there are cases that the crane hook are damaged during the some kind of work. From safety point of view, such damage must be prevented. First identify the reason of the damage is one of the key points toward the safety improvement. Mainly at loading area due to friction heat are generated and crack is developed in the crane hook. It can cause fracture of the hook and lead to serious accidents. Crack propagates continuously. In ductile fracture, and is more easily detectable and hence preferred over brittle fracture. There is sudden propagation. In brittle fracture, and the hook fails suddenly. This type of fracture is very dangerous as it is difficult to detect.

Failure of Crane Hook

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.

Literature Survey

Y. Torres et. al **(2010)**, initially studied the probable causes of failure of crane hook. It includes the manufacturing and lifting of crane hook, experimental analysis mechanical behavior of material of crane hook. It was concluded that the brittle fracture was originate from crack in the material

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 variations.

Rashmi Uddanwadiker **(2011)**, studied stress analysis of crane hook using finite element method and validated results using Photo elasticity. Photo elasticity test is based on the property of birefringence. To study stress pattern in the hook in a loaded condition analysis was carried out in two steps firstly by FEM stress analysis of approximate model and results were validated against photo elastic experiment. Secondly, assuming hook as a curved beam and its verification using FEM of exact hook. The ANSYS results were compared with analytical calculations, the results were found in agreement with a small percentage error = 8.26%. Based on the stress concentration area, the shape modifications were introduced in order to increase strength of the hook

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

Bernard Ross et. al **(2007)**, this paper describes the comprehensive engineering analysis of the crane accident, Under taken 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

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

Methodology Adopted

A virtual model of crane hook is created using solid work software and then model was imported to ABAQUS for Finite element stress analysis and the result of stress analysis of different materials are compared with each other.

Abaqus Finite Element Method

ABAQUS it is design and analysis tool. It is regarded by many researchers and engineers as a modern, accurate, robust and visually sensible tool to provide solutions for numerous engineering and scientific problems

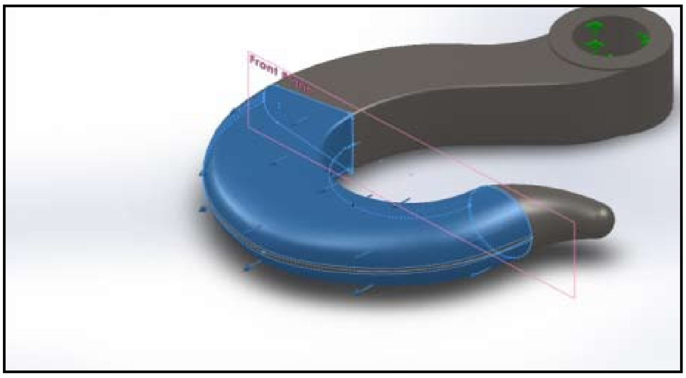
Analytical Method For Stress Calculation

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

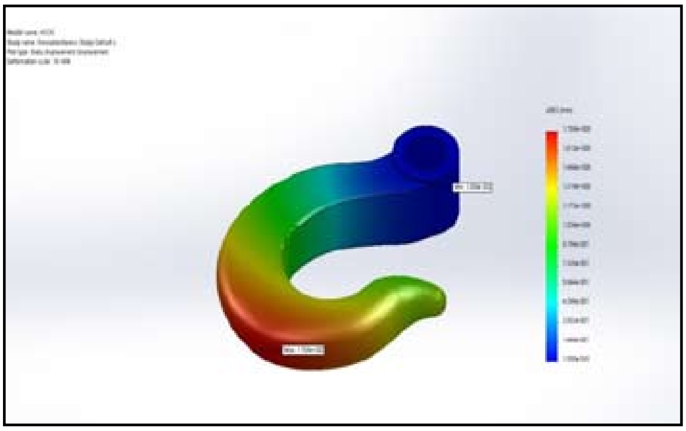
STANDARD SOLID MODEL OF CRANE HOOK

Crane hook are selected with standard design for this study

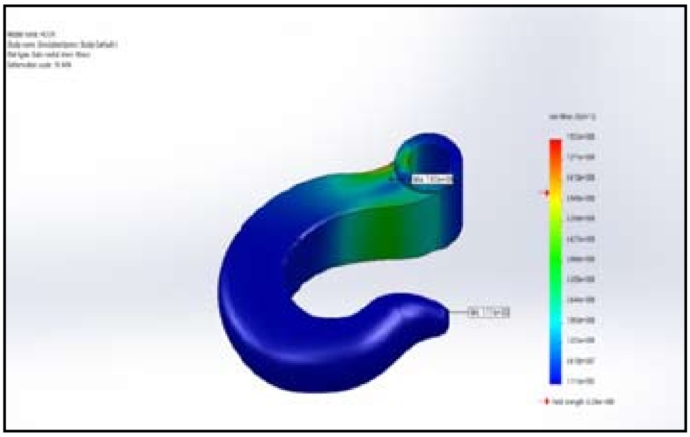
ACTUAL CONTACT AREA OF CRANE HOOK



TOATL HEAT FLUX



LOAD DEFORMATION



Conclusion

From this study the cranes hook. Are generating more heat at the actual loading area due to friction. The study of earlier publication enables us to conclude that it is possible to remove unwanted heat with proper material where heat dissipation cofactor is low and for that Finite Element Method (FEM) is one of the most effective and powerful method for the heat dissipation of the crane hook.

References

[1] *Mr. A. Gopichand, Ms. R. V. S. Lakshmi, Mr. B. Maheshkrishna "Optimization of design parameter for crane hook using taguchi method" in international journals of innovative research in science ,engineering and technology, vol. 2, Dec 2013, ISSN: 2319-8753.*

[2] *Chetan N. Benkar, Dr. N. A. Wankhade "Finite Element stress Analysis of Crane Hook With Different Cross Sections" in International Journal For Technological Research In Engineering, volume 1, Issue 9,May-2014, ISSN 2347-4718.*

[3] *M. Shaban, M. I. Mohamed, A. E. Abuelezz and T. Khalifa, "Determination Of Stress Distribution in Crane Hook by Caustic" in International Journal of Innovative Research in science, Engineering and Technology, Vol. 2 Issue 5, May 2013, ISSN: 2319-8753*

[4] *Apeksha. K. Patel, Prof. V. K. Jani, "Design and Dynamic Analysis of 70T Double Girder Electrical Overhead Crane" in Journal of Information, Knowledge and Research in Mechanical Engineering Vol.2, Oct-2013, ISSN-975-668X.*

[5] *Pradyumnakesharimaharana, "Computer Aided Analysis and Design of Hoisting Mechanism of an EOT Crane", .Thesis, National Institute of Technology Rourkela, May-2012.*

[6] *E.Narvydas, N.Puodziuniene, "Circumferential Stress Concentration Factor at the Asymmetric shallow Notches Of The lifting hook of trapezoidal cross-section" in Mechanika, vol. 18(2), 2012, ISSN 1392-1207*

[7] *C. Oktay Azeloglu, Onur Alpay"Investigation of a Lifting Hook with Different Method, Verification of the Distribution with Photo elasticity Experiments", in electronic journal of Machine Technology, vol-6, 2009, ISSN 1304-4141.*

[8] *Takuma Nishimura, Takao Muromaki, Kazuyuki Hanahara, "Damage factor Estimation of Crane Hook (A database approach with Image, Knowledge and simulation)" in Research publishing services, 2010, ISBN: 978-981-08-5118-7.*