Design and Analysis of Crane Hook using Finite Element Approach

Tukesh Sahu[1] G.R Kesheorey[2]
[1,2]Vindhya Institute of Technology & Science, Indore, India

Abstract— In this paper a three dimensional CAD model of crane hook has been developed using Design modular of ANSYS. The crane hook has been subjected to a fatigue load and a comparative analysis has been made between different materials of crane hook. The stress across the crane hook has been analyzed for all the material and it has been observed that the hook made up of carbon steel has minimum stress level. The obtained results helps in enhancing the hook working life increase and reduce the failure stress.
Key words: Crane Hook, FEA, Stress

I. Introduction

(Crane hooks are highly liable components and are always subjected 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 load in industries and constructional sites. 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. [1]

A crane is a machine, equipped with sheaves, wire ropes or chains and hoist 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, loader crane, jib crane, are some of the commonly used cranes. A crane hook is a device used for lifting and grabbing up the loads by means of a crane. It is basically a hoisting fixture designed to engage a link of a lifting chain or the pin of a cable socket. Crane hooks with circular, trapezoidal, rectangular and triangular cross section are commonly used. So, crane hook must be designed and manufactured to deliver maximum performance without failure.[2]

The crane hooks are vital components and are most of the time subjected to failure due to accumulation of large amount of stresses, which are ultimately leading to failure. Fatigue of the crane hook is happens due to continuous loading and unloading of crane. If the crack is detected in the crane hook, it can cause fracture of the hook. Due to this there are chances of serious accident. Bending stress, tensile stress, weakening of the hook due to wear, plastic deformation due to overloading, excessive thermal stresses are some of the other reasons of failure [3]. Fig 1. Shows the general diagram of crane hook.

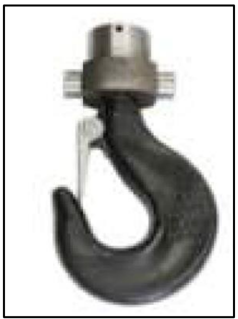

Fig. 1: General diagram of crane hook

II. Literature

Apichit et al. 2017 aims for 2-ton lifting hooks designed to be the appropriate size for using. Both of the cost, the strength, the security and the aesthetics by using the optimal designed method together with the highest weight method. In this study is divided a methodology to design the hook into three-step processes. The first step is to create a lifting hook by ISO 7597: 2013 standard (Forged Steel Lifting Hooks with Latch, Grade 8) to be an initial lifting hook. Then analyzed the axial load of 2 tons by the finite element method. The second step is to analyze by the optimal design to design the lifting hook with the higher safety value by the weight of the lifting hook remains the same or difference. The third step is to analyze the results of the second step by deciding on proper lifting hooks by highest weight method for sizing of the lifting hook. The results of the analysis showed that lifting hook that has been designed by the method presented in this research can be able to reduce the production cost of the materials by 27.5 percents per piece with increasing the safety equal to 42.16 percents. It is good looking and can be easily produced by comparing with the initial lifting hooks.

Qin et al. 2016 mechanical property of a crane hook is analyzed by using the boundary interpolated reproducing kernel particle method (BIRKPM). It is deduced by combining the interpolated reproducing kernel particle (IRKP) method with the boundary integral equation (BIE) method which aims to solve elastic mechanics plane stress. In the BIRKPM, the shape function constructed by the IRKP method possesses interpolation character at any scatter node. Because of this property, the boundary conditions can be applied directly for the BIRKPM and the new method has high precision and less computing time. When using this method to analyze a laminated crane hook, the results could agree well with the results concluded from the finite element method, and this could prove the validity of the new method.

Chijoo et al. 2012 rotation-controllable tower-crane hook block can control the horizontal rotation angle of a steel beam being lifted by a tower crane using a mechanical apparatus. It is expected that the mechanized hook block can relieve safety concerns; however, at the same time, there are several concerns: Is precise control of the rotation angle possible? Won't it be slower than rotating a beam by hand? If a mechanized system is faster, how fast is it? In order to test these, experiments were conducted. The precision of the rotating angle and the time for rotating and stopping the rotation controllable hook block were measured using an orientation sensor and a laser point. The results were compared to those of the conventional manual method. Since a steel beam usually has a symmetrical shape, $\pi/2$ rad (= 90°) is the maximum angle for placing a beam in the target location.

Torres et al. 2010 objective of this work is to identify the causes that led to a failure of the crane hook in service. The study of the accident includes: (1) a summary and analysis of the peculiarities inherent to the standards that determine the manufacture and use of this type of device, (2) metallographic, chemical and fractographic analyses, (3) assessment of the steel mechanical behaviour in terms of Vickers hardness profile, its tensile strength and fracture energy, and (4) simulation of the thermal history of the hook.. All the gathered evidences are in agreement with a strain-aging process triggering the embrittlement of the material, with the fracture starting from a crack generated at the heat affected zone of an uncontrolled welding of the hook.

Ajeet and Anshuman 2013 present paper a crane hook is purchased from the local market for Finite element analysis. The hook was tested on the UTM machine in tension to locate the area having maximum stress and to locate the yield point. The model of hook is prepared in CAE software having dimension and material similar to the crane hook which was purchased from the market.

Naresh and Bhatt 2012 paper is to analyze the stresses and strains condition in the power structure of overhead crane, presenting a fast and evaluated computer aided solving method for complex static indeterminate structures. The analysis of the stresses and strains state of the power structure of overhead crane bridge for increasing its toughness is made using the NX NASTRAN.

III. MATHEMATICAL MODEL

Winkler-Bach Formula for Curved Beams: We shall study the bending of beam which is initially curved. We consider the case where bending takes place in the plane of curvature (Figure: 1). This is possible when the beam section is symmetrical (Figure: 2) about the plane of curvature and the bending moment M acts in this plane. The curved beam flexure formula is used when the curvature of the member is pronounced as in case of hook for different cross section mathematical analysis of stress.

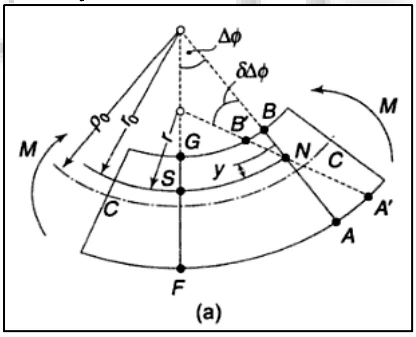

Fig. 2: Geometry of Bending of Curved Beam

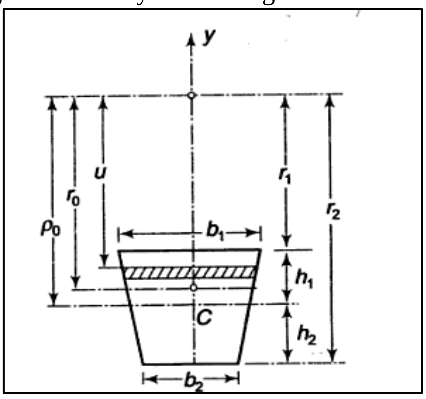

Fig. 3: Parameters for a hook Section

Winkler-Bach Formula for Curved Beams

$$\sigma_x = -\frac{M}{Ae}\frac{y}{r_0 - y}$$

M = Uniform bending moment applied to the beam, positive when tending to increase curvature

y = Distance of neutral axis

A = Area of the cross section, mm2

e = Distance of the centroid from natural axis, mm

r_0 = Radius of curvature of centroidal axis, mm

IV. METHODOLOGY

CAD modeling is used by many designers to create elaborate computerized models of objects before they are physically produced. Computers allow them to visualize their designs and confront problems before they have expended any of the resources necessary to put them into physical form. The model of cross section of hook is prepared by using ANSYS Design module. The 3D model of the Crane Hook shown in figure 4 and the material used for making hook are as follows Structural Steel, Forged Steel, cast iron

Carbon Steel, 20MnCr5

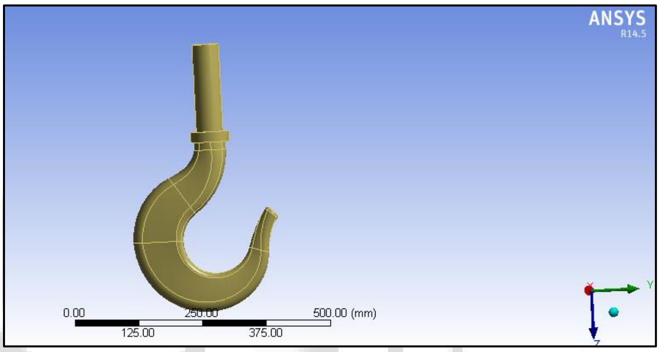

Fig. 4: CAD model of hook

V. RESULT & DISCUSSION

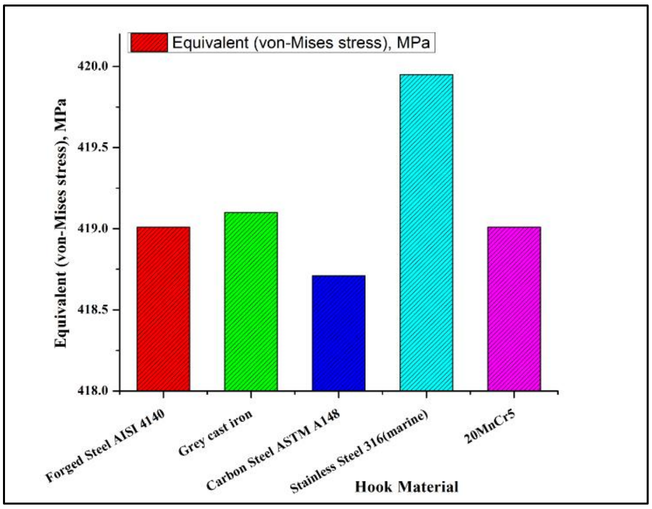

Fig. 5: Effect of Hook material on Equivalent (von-Mises Stress)

Figure 5 shows the effect of Hook material on Equivalent (von-Mises Stress). It has been seen that the maximum Equivalent (von-Mises Stress) takes place in hook made up of Stainless Steel 316(marine) and minimum Equivalent (von-Mises Stress) takes place in hook made up of Carbon Steel ASTM A148. This is because the Carbon

Steel ASTM A148 has higher ultimate and yield stress as compared to other material. Therefore, in order to perform heavy and cyclic load operation Carbon Steel ASTM A148 based hook is preferred so that the breakdown of the lathe can be avoided and the production can be enhanced.

Form the above it can be concluded that maximum Equivalent (von-Mises Stress) takes place in hook made up of Stainless Steel 316(marine) and minimum Equivalent (von-Mises Stress) takes place in hook made up of Carbon Steel ASTM A148. It can also be concluded that Carbon Steel ASTM A148 made hook has 0.29% less Equivalent (von-Mises Stress) as compared to the hook made up of Stainless Steel 316.

VI. Conclusion

After this analysis of the crane hook we get the induced stresses as obtained from Winkler-Bach theory for curved beams for using ANSYS 14.5. There are probable reasons for varying due to following assumptions which are considered in this research work. The most important assumption is that the , loading is considered as point load in case of Winkler-Bach Formula calculation while it is taken on a bunch of nodes in ANSY 14.5. Principal cross section is taken as perfect in all section, which are initially plane remain plane after bending. The complete study is an initiative to establish an ANSYS 14.5 based on Finite Element Analysis Method. From the analysis the stress across the crane hook has been analyzed for all the material and it has been observed that the hook made up of carbon steel has minimum stress level. The obtained results helps in enhancing the hook working life increase and reduce the failure stress.

References

[1] Patel Ravin B, Patel Bhakti K., Patel Priyesh M, Patel Dixit H., “DESIGN AND ANALYSIS OF CRANE HOOK WITH DIFFERENT MATERIAL DESIGN AND ANALYSIS OF CRANE HOOK WITH DIFFERENT MATERIAL”, International Journal of Advanced Computer Technology (IJACT), ISSN:2319-7900, 89

[2] Mahesh Solanki, Antriksh Bhatt, Anilkumar Rathour, Smit Thakkar, “Weight Optimization in Crane Hook”, IJRDO-Journal Of Mechanical and Civil Engineering ISSN: 2456-1479

[3] Joseph Leo A., Arut Pranesh K. , Balasubramani V. , “Structural Analysis of Crane Hook”,International Journal of Emerging Technology in Computer Science & Electronics (IJETCSE) ISSN: 0976-1353 Volume 12 Issue 2 –JANUARY 2015

[4] ApichitManee-ngam, Penyarat Saisirirat and Patpimol Suwankan, “ Hook Design Loading by The Optimization Method With Weighted Factors Rating Method”, Energy Procedia Volume 138, October 2017, Pages 337-342

[5] Y. X. Qin, W. T. Xie, H. P. Ren, X. Li, “Crane hook stress analysis upon boundary interpolated reproducing kernel particle method”, Engineering Analysis with Boundary Elements, Volume 63, February 2016, Pages 74-81

[6] Chijoo Lee, Ghang Lee, Suyeul Park, Joonbeom Cho, “Analysis of field applicability of the rotation-controllable tower-crane hook block”, Automation in Construction, Volume 21, January 2012, Pages 81-88

