

Study and Review on the Analysis of Crane Hook with Different Cross Section Area & Materials

Mukesh Sonava¹, Vishal Wankhade²

¹P.G. Scholar, Swami Vivekanand College of Engineering, Indore

²Assistant Professor, Swami Vivekanand College of Engineering, Indore

Abstract - Crane Hooks are highly responsible components that are typically used for handling material in industries. Crane are amongst material handling equipment which find wide application in the field of engineering. Crane hook is the curved bar which is used for lifting loads in cranes. In order to reduce structure failure of crane hook, induced stress in crane hook is analysed properly. The analysis of stress plays a significant role in the design of structures like crane hook in loading conditions. For getting maximum strength without excessive weight we should make proper design, careful forging, and precision controlled quench and tempering. For selecting the crane hook following information need to find out: Capacity (tonnage). Material (carbon, alloy or bronze). Shank diameter, Shank length, Throat opening

In this paper we would like to review some previous research work performed on crane hook for increasing its strength by optimization of its design parameter and changing its cross-section area and different material combination.

Keywords: crane hook, material handling equipment, Ansys, curved bar

Key Words: Cross-sectional Area, Material, FOS, Deformation & stress.

1. INTRODUCTION

Crane Hooks are most responsible components that are used for industrial purposes. It is generally a hoisting fixture designed to engage a ring or link of a lifting chain or the pin of a shackle or cable socket and it must follow both health and safety guidelines. It is used to lift heavy loads in industries like marine industry, transportation etc. it is mainly used for lifting heavy loads and transporting them to other places. It uses one or more simple machines to create mechanical advantage and moves the load beyond the normal capability of a human. Hooks are generally available in the following different types of cross section area. Circular Cross Section area, Rectangular cross section area, Triangular cross section area & Trapezoidal cross section area





Circular	Rectangular	Triangular	Trapezoidal
			

Table 1: Different cross section area of crane hook [12]

Crane hook are generally made of steel but due to issue of weight and factor of safety for heavy duty operation most of the company manufacture the hook with different composite material. Generally, hook is available with eye and shank



Fig-1: hook with eye and shank [12]

So here in this paper we try to review all these previous journals for better understanding the work done in all the previous research, the researcher work on analytical method a software for finding out the stress and load bearing capacity of hook. Material based study is used for finding the reduction of weight of hook with more factor of safety. The paper also include the study and work performed by the researcher in previous journals. The details of my study on previous papers are present in next section (Literature Survey).

2. LITERATURE SURVEY

The section here shows the review of previous journals based on study and analysis of crane hook M.Amareswari Reddy et.al [1] studied the load carrying capacity of crane hook by varying the cross sections. The selected sections were I-section and T-section. He kept area constant while changing the dimensions of two different sections. The crane hook is modelled using SOLIDWORKS software. He did stress analysis by using ANSYS 14.0 workbench. He found that T cross section yields minimum stresses at the given load of 6 ton for constant cross section area among two cross sections. For validation of stress distribution pattern for its

correctness of the used Winkler-Bach theory for curved beams. The theoretical stresses calculated based on Winkler-Bach theory and the corresponding crane hook model stresses determined by using ANSYS were tabulated and found that the deviations are at minimum of 5%. The FOS for crane hook in static analysis is based on yield stress value. The theoretical stresses results and ANSYS results are compared and deviation is at minimum 5%. For 6 Ton loading the stresses induced in I-section and T section of Structural Steel, carbon steel, grey cast iron and Nickel Alloy (Wrought N06230) are below the allowable stresses. The stresses induced in the T section are little less than the other sections for same area of I-cross section and loading. It determines that T section of Nickel Alloy compromises good results for optimization of the crane hook design.

Ms. Mamta .R. Zade et.al [2] She designed the hook by analytical method and analysis was done for the different Cross sections by applying same load on crane hook like rectangular and trapezoidal. Out those two area Trapezoidal area is selected for further static structural analysis with different materials. Because it gives better results in comparison with other one as because stresses induced are less in trapezoidal cross section. After the analytical method of design and the modelling of crane hook is done in modelling software. The modelling is done using the design calculation from previous work the analysis of hook is done in the FEA software. The aim of the research was the selection the best material from aluminium alloy, structural steel, wrought iron for crane hook from analysis result and fatigue analysis. The results of stress analysis calculated from FEA analysis for different material such as Structural Steel, Wrought iron, Aluminium Alloy. For the different Material, it was observed that keeping the tone are same with different Material topology we will get different results, but it is found that the Structural Steel and Wrought Iron gives minimum stress. Further Fatigue analysis was done on the materials by that it was found that wrought iron can withstand the maximum number of fatigue cycles before failure. Hence we can use wrought iron materials for crane hook.

Vivek Mahadev Thakur et.al [3] studied the heat flux deformation under heavy load material handling process. The analysis of heat flux deformation is carried out by using theoretical experimental simulation process. In that simulation process was carried out by developing crane hook solid model by using solid works, analysis of solid model based on heat flux load deformation by using software. From this study the cranes hook is generating more heat at the actual loading area due to friction. The study 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 effective and the powerful method for heat dissipation of the crane hook.

Rashmi Uddanwadiker [4] studied the stress distribution pattern of crane hook in its loaded condition, a solid model of crane hook was prepared with the help of CMM and CAD software. The real time pattern of the stress concentration in 3D model of crane hook is found. The stress distribution pattern is verified for its rightness on a model of acrylic for crane hook using the Diffused light Polariscope system. By predicting the area of stress concentration, for increasing the working life the shape of the crane is modified and reduce the failure rates. The complete study was an initiative to establish a FEA procedure, by validating the results, for the measurement of stresses. For decreasing the failures of crane hooks the estimation of stresses, their magnitudes and possible locations are very important. Analytical calculation becomes complex as the newer design is too complicated. From the stress analysis it was observed the cross section of max stress area. If area on the inside of the crane hook at the portion of max stress is expanded, then the stresses will get reduced. Analytically if the thickness of hook is increased by 3 mm, the stresses are reduced by 17%. Thus the design can be altered by increasing the thickness value on the inner curvature so that the chances of failure are reduced considerably.

Takuma Nishimura et al. [5] studied the damage estimation of crane-hooks. It has been estimated that loading conditions were crucial to the crane-hook damages. CAD model of the crane-hook referring to one of its actual designs has been constructed. The database has been arranged based on the FEM model; it was constructed as a collection of a number of various possible loading conditions and the corresponding deformation values, obtained in the results of the FEM analysis. The database has been used to identify those loading conditions that might damage those crane-hooks. For the crane hook design some feature points have been selected; the deformations of the damaged crane-hook could be obtained based on the feature points detected by means of the image processing. The critical loading state of the broken crane-hook has been calculated by comparing the obtained actual deformations and the simulated deformations values in the database. On the basis of these calculated loading conditions, the critical load condition for the crane-hook has been estimated as a statistical distribution based on the Bayesian approach

Govind Narayan Sahu et al. [6] presented design and stress analysis of various cross sections of the simple crane hook. The CAD model of crane hook has been prepared with the help of existing drawings. This CAD model has been imported in the ANSYS to perform static stress analysis. Stresses and deformation has been obtained in the existing model of simple crane hook. The cross sectional areas have been changed to analyse the effect of maximum stresses induced. The different types of cross sectional areas used were circular, square and curved. Then curved portion has been again modified to improve maximum stresses induced. It has been found that induced stresses & deformations were minimum in modified curved hook. This was because of the fillet as well as stress concentration effect has been

distributed uniformly. Material and mass economy has been obtained by this modified design of crane hook.

Bhupendra Singh et al. [7] presented the solid modelling and finite element analysis of using PRO/E WILDFIRE 2.0 and ALTAIR HYPER MESH with OPTISTRUCT crane boom. The solid modelling and FEM analysis has been done 8.0 SOLVER Software. The variation of stresses and displacement in the various parts of the crane boom have been determined by using HYPER MESH. The design has been modified to reduce the maximum stresses and displacements. The analysis also concluded that maximum stress was coming near the fixing position. The following objectives have been achieved by using the Finite Element.

- i. Weight reduction (5 kg)
- ii. Stresses were within safe limits
- iii. Cost has been reduced (Rs-180/- for a single component)

R.K. Rathore [8] proposed the technique for the optimization of crane hook, which was used to calculate the best combinations for control function, used the design of experiment with central composite design method, artificial neural networks to calculate the responses for every parameter with respect to the output function and also utilized the multi objective genetic- algorithm for optimize the responses created with artificial neural networks. The 29 crane-hook the shape responses are estimated for the mass and the factor of safety. By comparing the new design and initial design results in raising the safety factor from 0.54 to 1.1 and the mass has been reduced from 752 grams to 669 grams. The design of a crane hook has been utilized to demonstrate the use of the multi objective genetic algorithm on response surface artificial neural network models as an alternative approximation technique for multidisciplinary design optimization because after analysed the crane hook, the implementation of multi objective genetic algorithm on response surface artificial neural network meta model gives the more optimized results.

Torres et al. [9] studied the objective of his work is to identify the causes that led to a failure of the crane hook in service. The study of the accident includes: (1) the summary and the analysis of peculiarities inherent to the standards that determine the manufacture and use of this type of device, (2) fractographic, metallographic and chemical 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. The visual and the microstructural inspections of the hook reveal some evidences that a weld bed was deposited on the crane hook surface. Several cracks raised from that area into the material. The Fracture surface shows the features typical of brittle failures (Trans granular cleavage fracture). The unalloyed, low-carbon steel comprises a relatively low aluminium (<0.025%) and high non-combined nitrogen (>0.0075%) content. All the collected confirmations are in agreement with a strain-aging

process causing 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.

Apichit Manee-ngam et al. [10] He designs the hook by divide the study into three-step processes the design variables to create the optimize crane hooks design process following the Figure. 2. This crane hooks design for support 2 tons loading and material made from high carbon steel AISI 4340. The design by ISO 7597 standard that define factors to study by the Finite Element Analysis method compatible with the Optimum Design approach and the Weighted Factors Rating Method. From all method that was found the best result thus can support more loads and reduce weight. The category of crane hook that was designed by this method is the 9th hook model. This size has enhanced to be easy to mold. It can reduce production costs by up to 27.49% compared to material costs per piece and safety factor increase for 42.16% consider to appearance and the new design process is more convenient than the standard design all so. From crane hooks can carries over load and the condition of weight is less than the standard type. However, to be develop the design and increase safety factor the reason why selects the optimum crane hooks design to find out and define dimension to create prototype and useful for future.

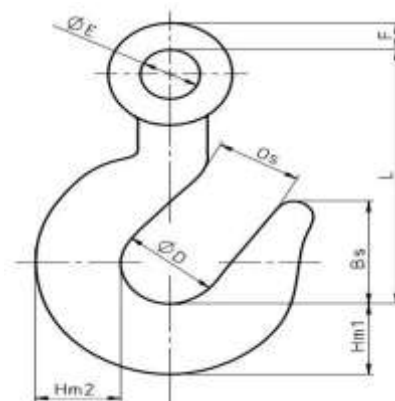


Fig- 2. Show variables of crane hook[10]

Patel Ravin B et.al [11] used analytical method and designed the hook for the different materials like forged steel and high tensile steel. After using the analytical method, he used modelling soft-ware (solid edge) for the design of crane hook. Through the modeling the analysis of hook is done in FEA software (ANSYS). This result leads to the determination of stress in existing model. By forecasting the stress concentration area, the working life of the hook is increase and reduce the failure stress. The results of stress analysis calculated from FEA analysis for various different material such as Forged Steel, Wrought iron/MS, Aluminium Alloy. For the different Material, it was observed that keeping the tone are same with different Material topology we will get different results, but it is found that the Forged Steel Material gives minimum stress.

3. PROBLEM IDENTIFICATION

The following problem has been observed from literature review of the crane hook:

- I. A lesser amount of work has been done for taking diverse cross section area to reduce weight and reduce stress without disturbing ability of hook.
- II. Fewer works has been done for designing the hook with greater factor of safety (FOS).

As per literature presented, there is a lot of possibility of weight optimization and cost saving of crane hook by varying the design parameters, cross section area and material of crane hook.

4. CONCLUSIONS

The study done by us here offers a review on previous literature and journals based on different ideas and modification with the help of analytical and computational methods. Now in this section we conclude here the main parameters analyse by us by the study of these previous literature.

1. The main area focuses on the various cross section of crane hook (trapezoidal, T section, I section, Triangular section)
2. Focus was also given on the stress induced in the crane hook when comes under loading
3. The literature also shows the use of different materials used for manufacturing of crane hook
4. Work is also done on reduction of weight of hook
5. Some experimental method is also performed for finding out the deformation under stress on the solid model of hook.

REFERENCES

- [1] M.Amareswari Reddy1 ,M.N.V Krishnaveni2, B.Nagaraju3 , M RajaRoy4 "Static Analysis of Crane Hook with I- Section and T- Section using Ansys" , IJETT – 2015
- [2] Ms. Mamta .R. Zade "Finite Element Analysis And Fatigue Analysis Of Crane Hook With Different Materials" , IRJET 2017
- [3] IVivek Mahadev Thakur, IIVaibhav Pawar, IIIDr. M.D. Nadar, IIISanjay Ghorpade, IIISahil Patil, Study and Analysis of Crane Hook in Loading Area IJARET 2016
- [4] RashmiUddianwadikar, "Stress Analysis of Crane Hook and Validation by Photo Elasticity", Scientific Research, 2011
- [5] Takuma Nishimura, Takao Muromaki, Kazuyuki Hanahara "Damage factor Estimation of Crane Hook (A database approach with Image, Knowledge and simulation. " Research publishing services" .2010
- [6] Govind Naryan Sahu, Narendra Yadav "Design and stress analysis of various cross section of hook" International Journal of modern engineering research.2013
- [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.2011
- [8] Ram Krishna Rathore, AmitSarda, RiturajChandrakar "An Approach to optimize ANN Meta model with Multi Objective Genetic Algorithm for multi-disciplinary shape optimization". International Journal of Soft Computing and Engineering (IJSCE) 2012
- [9] Y. Torres, J.M. Gallardo, J. Dominguez, F.J. Jimenez E, "Brittle fracture of a crane hook", J-GATE Engineering journals, 2008
- [10] Apichit Manee-ngam*, Penyarat Saisirirat, Patpimol Suwankan "Hook Design Loading by The Optimization Method With Weighted Factors Rating Method"International Conference on Alternative Energy in Developing Countries and Emerging Economies ELSEVIER 2017
- [11] Patel Ravin B, Patel Bhakti K., Patel Priyesh M, Patel Dixit H, Patel hirak s. Design And Analysis Of Crane Hook With Different Material IJACT
- [12] Mahesh Solanki1 Antriksh Bhatt2 Anilkumar Rathour3 Design, Analysis and Weight Optimization of Crane Hook: A Review IJSRD 2014