**International Journal of Research Publication and Reviews**

Journal homepage: www.ijrpr.com ISSN 2582-7421

# Review of Design and Analysis of Crane Hook Using Different Profile and Material

***MOH Ahtesham Mansuri[1], Kamlesh Gangrade[2]***

[1],*PGStudent,DepartmentofMechanicalEngineering,SagarInstituteofResearch&TechnologyIndore,India*
[2]*Professor,DepartmentofMechanicalEngineering,SagarInstituteofResearch&TechnologyIndore,India*

**ABSTRACT**

Crane hooks are extremely important components that are used to hoist heavy loads with the aid of chains or links. Hooks on cranes are extremely prone to failure and are always at risk of failure due to the high concentration of stresses they are subjected to, which might eventually result in their collapse. It is necessary to investigate the tension induced in the crane hook in order to lessen the likelihood of failure. The constant loading and unloading of a crane is a part of its job. This has the potential to cause fatigue failure of the crane hook. Because only a few articles have been published to date addressing stress analysis of this curved beam (crane hook), it is possible to conclude from the evaluation of prior publications that components with difficult geometry, such as crane hooks, require a more comprehensive research (crane hook).

**Keywords**:AISI 1010 STEEL, Gray Cast Iron, Structural Steel, fatigue failure, Strength of material, Factor of saftey

## 1. Introduction:

Many industries, oil rigs, automobiles, and other machinery rely on hoists and cranes with hooks to raise huge cargo. The load rating of the hook affects its performance. A hook failure can be caused by several factors including the type of fastening system used to attach the hook to a hoist or a crane, materials utilised in the design of the hook, and more.

Large, massive, huge, and heavy hooks are also required for heavy loads if they are to work securely at the imposed weights. These factors have led to a growth in hook manufacturing research, particularly in the areas of production process and metallurgy. The goal is to lower hook size and manufacturing costs while also increasing the versatility of hooks that can handle heavy loads.

The goal of this article is to create a hook that can bear heavier weights than the hooks now available on the market that meet the same standards and ratings by employing a tiny form in its design.

**Figure1:Single crane hook**

## 2. LiteratureReview

**M. Shaban et. al [1]**Crane hook stress distribution data was utilised to generate a solid model of it using ABAQUS software. In a crane hook 3D model, stress concentration patterns may be determined in real-time. The stress distribution pattern on an acrylic model of a crane hook is verified using a shadow optical technique (Caustic method) setup. By predicting the stress concentration region, the crane's shape may be changed to maximise service life and minimise failure rates. Ultimately, the researchers hope to create a stress measuring FEA procedure by confirming the simulations they've run. Hook failures can be prevented by assessing the stresses, their magnitudes, and the sites where they could occur. In the stress investigation, they've discovered the cross section of the greatest stress area. At the point of greatest tension, the hook's inner side can be widened to reduce stress. The caustic method is a great method for assessing stress distribution in mechanically complicated components such as hooks. By drilling a succession of small holes in the hook, the caustic method can accurately predict the stress value at each hole site.

**E. Narvydas et. al** [2], finite element analysis was used to evaluate circumferential stress concentration parameters for trapezoidal cross-sectional lifting hooks with shallow notches (FEA). Structural and machine components were often evaluated using stress concentration factors. Results from the finite element analysis (FEA) were used and fitted using a generic equation. Finite element models aren't necessary to derive formulae for stress concentration factor assessment. Using ductile materials to avoid brittle failure is required by the design regulations of the lifting hooks; in this regard they examined the strain-based criteria for failure, taking into consideration the stress triaxiality.

**Ram Krishna Rathore et. al.[3]**It is necessary to discuss The correlations between response functions and control variables may be calculated using regression models as a generic method to optimising various response situations. This is followed by the use of an objective function, a mechanism for compiling numerous response functions into a single number. Finally, an optimization approach is employed to determine the optimal control function configurations. This research proposes an alternative approach for calculating parameter response functions: using an artificial neural network (ANN). A multi-objective genetic algorithm (MOGA) and objective functions are utilised to find the best circumstances for the control functions during the optimization stage. Multiple form parameter responses to a new loading situation have been optimised using an example of a crane hook. The findings demonstrate that the suggested technique for optimising multi-disciplinary form optimization issues reduces bulk and has a high enough safety factor. An optimization strategy for multi-response is shown here. Unrestrained objective functions are used to aggregate several replies into a single single response, and a multi-objective genetic algorithm (MOGA) is used to optimise across multiple disciplines using an artificial neural network for each response function. In three ways, the proposed technique is unique. A core composite design technique is used to begin with. Second, it makes use of artificial neural networks to figure out how each parameter will affect the output function in some fashion. When it comes to artificial neural network replies, it uses a multi-objective genetic algorithm. Using the crane hook example, the shape responses to mass and factor of safety may be predicted using this method. The projected optimization strategy, in particular, consists solely of estimating the outcomes of the reactions. This means that the suggested approach might be extended to incorporate other parameters for the answers. Constrained constraints are needed to predict the varied responses for various control factor values in this situation...

**Rashmi Uddanwadiker [4]**finite element technique and photo elasticity were used to verify the findings of the stress analysis of the crane hook. The birefringence feature is the basis for the photo elasticity test. FEM analysis was used to examine stress patterns in the hook in a loaded situation, and findings were confirmed by photo elastic testing. FEM of an accurate hook is used to verify that the hook is a curved beam. It was determined that the findings from ANSYS and analytical calculations were in accord with a tiny error of 8.26%. In order to boost the hook's strength, form alterations were made based on the stress concentration area.

**SpasojeTrifkovic' et. al [5]**In this study, we use both approximate and accurate approaches to examine the stress condition in the hook. Assuming hook as a straight beam and then as a curved beam, they estimated the stresses in various sections of the material. FEM (Finite Element Method) was utilised in conjunction with computer-aided analytic approaches.

**Bhupender Singh et al [6]**Using PRO/E WILDFIRE 2.0 and ALTAIR HYPER MESH with OPTISTRUCT 8.0 SOLVER Software, the authors have done solid modelling and finite element analysis of a crane boom to determine how stress and displacement vary in different parts of the boom and to determine what steps can be taken to avoid excessive stress and displacement. The following objectives have been met via the use of Finite Element Analysis.

• Losing Pounds (4.86 kg, approx.5kg).

• There is a limit to the amount of stress that can be tolerated (at higher load points).

•To save money, a single component costs Rs. 180/-. Additionally, it was determined that the fixing location is the point at which the most tension is placed.

**Y. Torres et. al [7]**At first, the most likely causes of the crane hook failure in service were investigated. Experimental examination, steel mechanical behaviour, and a modelling of the hook's thermal history are all part of the accident investigation. The standards controlling lifting hook production and usage are also examined. In the end, it was determined that the mishap was caused by the utilised steel becoming brittle due to strain-aging. The lifting hook welding caused a split in the material, which led to the brittle fracture.

**Takuma Nishimura et. al [8]**learned about crane-hook damages. For the crane-hook damage, they calculated what they thought were the most critical load circumstances. Crane hook FEM model based on a real design was created. The FEM model was used to build a database that contains the deformation values generated from the FEM study for a variety of different load circumstances. It was utilised to determine the conditions that killed the crane-hooks that were damaged by the database. Using the image processing, the deformation of a crane-hook may be determined based on specified feature points on the crane-design. hook's In order to determine the crane-critical hook's load condition, we compared the measured deformation to the database simulations. The critical load requirement for the crane-hook was approximated as a statistical distribution based on the Bayesian technique using these computed load conditions.

**C. Oktay AZELOGLU et. al [9]**Based on various assumptions, this study gives various ways of calculating the stress of lifting hooks The stress field on the hook was determined using curved beam theory, the Finite Element Method, and photo elasticity studies. As a consequence, the stress field on the hook is compared using various ways. Lifting hook estimates for field applications were given some advice.

**Yu Huali et. al [10]**The load-bearing capacity of the elevating equipment is mostly determined by the structure's strength. To properly build a greater tonnage hook, it's critical to look into and analyse the static characteristics of the hook that operates at a restricted load. The hook of the DG450 drill well was examined in this study.

To begin, a 3-D model of the hook was created in Pro/E utilising the typical modelling technique. Secondly, ANSYS FEM software was used to perform a static analysis of three risky work circumstances at the ultimate load of the hook. This paper demonstrates the significance of the design and development of the bigger tonnage drill well hook in terms of both instruction and engineering application.

**Bernard Ross et. al [11]**A full engineering examination into the crane disaster was conducted to disprove the Mitsubishi explanations of failure, and the results of that research were presented in this publication. Besides wind tunnel testing, structural and metallurgical analyses of the boom were given, as well as a soils engineering study connected to ground stresses and displacements during the lift. SAE J1093, a 2 percent design side load requirement, and Lampson's justification for an 85 percent crawler crane stability criteria will be discussed in this presentation.

**Rashmi Uddanwadikerin [12]**Photo elasticity is used to verify the stress analysis of a crane hook. Crane hooks are notoriously prone to breaking due to excessive stress accumulation, which can eventually cause the component to fail completely. In order to determine the stress distribution on the crane hook at full load, a solid model of the crane hook is built using a CMM and CAD software. A 3D model of a crane hook has been able to show a real-time pattern of stress concentration. On an acrylic model of a crane hook, the correctness of the stress distribution pattern is evaluated using a Diffused light Polari scope set up. The crane's design is tweaked to make it last longer and have fewer failures. An estimation of the stress concentration zone is used to achieve this.

**Ajeet Bergaley and Anshuman Purohit [13]**Using finite element analysis, they performed a thorough structural investigation of crane hooks. A crane hook is purchased from the local market for use in the finite element analysis. It was done using the UTM machine to determine the area of maximum stress and where the hook's yield point was located. To test the crane hook model, the CAE programme is used to produce a model with dimensions and materials that are similar to those of the crane hook purchased from a store. The results were compared against theoretical predictions to see if they agreed. In order to minimise stress under a given load, the cross section was recalculated using FEM.

**A.Gopichand [14]**The crane hook's design parameters were optimised using the Taguchi approach, which was finished. In the case of a substantial weight accumulation, crane hooks are prone to failure because of the force they face. The design requirements include the cross-sectional area, material, and radius of the crane hook. The Taguchi method is employed in this study to optimise the design parameters. L16 orthogonal array is generated as a consequence of examining three parameters at mixed levels. Input parameter combinations with the lowest possible Von-misses stresses are shown to be the best ones.

**G.E.V.Ratnakumar, B. Jitendra Kumar, KalapalaPrasad[15]**, Various Hook Cross Sections Stress Analysis and Design," It's referred to as "Hook Design and Stress Analysis." hook cross-sections: design and stress analysis Researchers in this research want to use experimental and theoretical approaches to better understand how stress varies in crane hooks with varied cross sections, including squares and circles, as well as different radii of curvature. Crane hook loads with 5mm elongation on UTM are empirically calculated with a variety of crane hooks (Universal Testing Machine). For this purpose, theoretical and experimental results are used to calculate stresses in the crane hooks as a result of the loads gathered via the experimentation process. ANSYS 12.1 is used to analyse the modelled crane hooks and calculate the stresses placed on them. This is done after the different crane hooks have been modelled in Pro-E 2.0. After that, the results of the stress calculations made with ANSYS12.1 and the curved beam theory are compared.

## 3. Conclusion

Among other things, stress concentration parameters are commonly used in the evaluation of machine elements' strength and durability. A crane hook failure may be minimised by looking at the stresses placed on the crane hook. It's reasonable to determine from a review of previous papers that curved beams like crane hooks need further research because there aren't many studies done in this area yet. According to the literature evaluation, the Finite Element Method (FEM) is one of the most effective and powerful methods for stress analysis of the crane hook among the various ways available.

## References

[1] 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.

[2] 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.

[3] Ram Krishna Rathore, Amit Sarda and Rituraj Chandrakar; “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), ISSN: 2231- 2307, Volume-2, Issue-1, March 2012.

[4] Rashmi Uddanwadiker, “Stress Analysis of Crane Hook and Validation by Photo-Elasticity”, Engineering, 2011, 3, 935-941.

[5] SpasojeTrifković, NebojšaRadić et. al, “Stress analysis of crane hook using FEM”, INFOTEH-JAHORINA Vol. 10, Ref. C-2, p. 244- 248, March 2011.

[6]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, 2011/ 51-52.

[7] Y. Torres , J.M. Gallardo , J. Domínguez , F.J. Jiménez E, “Brittle fracture of a crane hook”, Engineering Failure Analysis 17 (2010) 38–47.

[8] 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).

[9] 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, 2009 (71-79).

[10] Yu Huali, H.L. and Huang Xieqing, “Structure-strength of Hook with Ultimate Load by Finite Element Method”, Proceedings of the International MultiConference of Engineersand Computer Scientists, 2009 Vol II IMECS 2009, March 18 - 20, 2009, Hong Kong.

[11] Bernard Ross, Brian McDonald, S.E. Vijay Saraf, “Big blue goes down. The Miller Park crane accident”, Engineering Failure Analysis 14 (2007) 942–961.

[12] Rasmi Uddanwadikar, Stress Analysis Of Crane Hook and Validation by Photo-Elasticity in Scientific Research, 2011, ISSN 935-941

[13] Patel Ravin B, Patel Bhakti K., Patel Priyesh M, Patel Dixit H, Design and Analysis of Crane Hook with Different Material, (International Journal of Advanced Computer Technology (IJACT) ISSN:2319-7900.

[14] M.Amareswari Reddy1, M.N.V Krishnaveni, B. Nagaraju, M RajaRoy4, Static Analysis of Crane Hook with T-Section Using ANSYS, International Journal of Engineering Trends and Technology (IJETT) – Volume 26 Number 2- August 2015

[15] Abhijit Devaraj, Design of a Crane Hook of Different Materials and Stress Analysis Using ANSYS Workbench, International Journal for Research in Applied Science & Engineering Technology (IJRASET Volume 3 Issue VII, July 2015 IC Value: 13.98 ISSN: 2321-9653.