DESIGN AND ANALYSIS OF A CRANE HOOK

Dr. P.Ravikanth Raju[1], Somagani Upendar[2]
[1,2]Department of Mechanical Engineering, Anurag Group of Institutions

Abstract

In this paper the design of the hook is done by analytical method and design is done for the different materials like forged steel, wrought iron and high tensile steel. Crane hook is very significant component used for lifting the load with the help of chain or wire ropes. Crane hooks are highly liable components and are always subjected to bending stresses which leads to the failure of crane hook. To minimize the failure of crane hook, the stress induced in it must be studied. A crane is subjected to continuous loading and unloading. This may causes structural failure of the crane hook.After the analytical method design and modeling of hook is done in modeling soft-ware .The modeling is done using the design calculation from the modeling the analysis of hook is done in FEA software .This result lead us to the determination of stress in existing model. By predicting the stress concentration area, the hook working life increase and reduce the failure stress.

Key words: UG, ANSYS 14.5. Forged Steel, Wrought Iron.

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. A crane hook is a device used for grabbing and lifting up the loads by means of a crane. It is basically 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, circular, rectangular and triangular cross section are commonly used. So, it must be designed and manufactured to deliver maximum performance without failure. Crane hooksmostly employed in transport, construction and manufacturing industry. Overhead crane, mobile crane, tower crane, telescopic crane, gantry crane, deck crane, jib crane, loader crane are some of the commonly used cranes. A crane hook is a device used for grabbing and lifting up the loads by means of a crane. It is basically 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, circular, rectangular and triangular cross section are commonly used. So, it must be designed and manufactured to deliver maximum performance without failure. Thus the aim of this research is to study stress distribution pattern within a crane hook of various cross sections using analytical, numerical and experimental methods.

II. LITERATURE SURVEY

The comparative study by Mr. A Gopichand. Et al. [1] has shown that taguchi method can be used for optimization of crane hook. In his work optimization of design parameters is carried out using Taguchi method. He considered total three parameters and made mixed levels a L16 orthogonal array. The optimum combination of input parameters for minimum Vonmises stresses Are determined. From that array he found optimum combination of area radius for minimum Vonmises stress. Ram Krishna rathour. et al. [2] has worked on a general approach for the multiple responses. He started optimization with the regression models to calculate the correlation between response function and control function. An objective function is generated with the help of system for collecting various response functions together. By using artificial neural network (ANN) to find out the response function. He used multiple objective genetic algorithms (MOGA) to optimize shape function of the crane hook for same capacity by considering combination of objective function to find out the optimize shape of crane hook. The result shows that the reduction in mass as well as safety of factor is not disturbed. Nishant soni et al. [3] has worked on the optimization of low carbon steel for its self-weight. The self-weight and component load coming on the crane–hook hence he worked with objective of the optimization of the mass for cane hook-under the effect of static load comprising the peak pressure load. He used finite element analysis for the shape optimization of crane hook as well as for validation of final geometry. Chetan N. Benkar.et al. [4] worked on crane hook for the optimization. He estimated the stress pattern of crane hook in its loaded condition by preparing a solid model with the help of ANSYS 14 workbench. He calculated stress pattern for various cross section topology such as rectangular, triangular, trapezoidal, and circular by keeping the area constant and found that rectangular cross sectional area gives minimum stress and deformation level. Rashmi Uddanwadiker.et.al. [5] Has calculated the stress pattern produced due to the load on hook. He compared the analytical result of stress and the stress estimated from the FEM analysis and found that there was 8.26% percent error between them. Photo elasticity test is based on the property of birefringence. From the analysis he found the area at which high stress concentration occurs. For the design improvement if the inner side of hook at the portion of maximum stress is widened then the stress will get reduced. He estimated that the stress is reduced up to 17% if the thickness of the inner curvature is reduced by 3mm C.

Oktay Azeloglu.et al. [6] has studied the method for the calculation of stress based on the different assumption. He adopted Timoshenko‟s curved theory and Bach approximation on the simple hooks calculation. He used finite element method to estimate the stress and compared it with different method. M. Shaban. et al [7] prepared a solid model of crane hook to estimate the pattern of stress in the crane hook. They used ABAQUS software and obtained real time pattern of stress concentration. The value and location is very much important factor in reducing the failure. If the inner curvature of hook is widened the stress will be reduced. For complicated mechanical element it is suitable to use caustic method. In caustic method several small several holes are drilled to predict accurate stress value. Takuma Nishimura.et al. [8] studied damage factor estimation of crane hooks to recognize the tendency of the load condition. They used FEM to estimate the relation between the load condition and its deformation. First, load –deformation database that has the relation between the load condition of crane hook and its deformation using numerical calculation is constructed. After the completion of study they found that load acts in downward position and tip –end position and load direction is not downward normal in damaged hook. Santosh Sahu.et al. [9] made a model of crane hook of trapezoidal using CATIA V5R20.Then estimated the location of stress after Appling the 2 ton load using FEM. They also analyzed the effect of variation in length of two parallel sides of trapezoidal hook on stress. Apeksha K Patel.et al. [10] has worked on reduction of weight of girder which has reduced the cost of girder and also life of girder is increased. They made a mathematical design for crane component by using ANSYS workbench V12.They also optimized hook by using Trapezoidal cross sectional area. Pradyumnakeshrimaharana. [11] Has estimated hook dimensions for various cross section topology by keeping the depth and cross section area. He concludes from his work that the trapezoidal section was lest stressed.Spasoje Trifkovic' et al[12] worked on the stress state in the crane hook using exact and approximate methods. Stresses are calculated in various regions of the crane hook material by assuming crane hook as a straight beam and curved beam. Torres et al [13] worked on the causes which led to failure of the crane hook in service. This study shows the simulation of the thermal history of the hook and explains the standards governing the manufacturing.

III. 3D MODELING

From the data exists, the component is modelled in 3D software. The Generated CAD model is shown

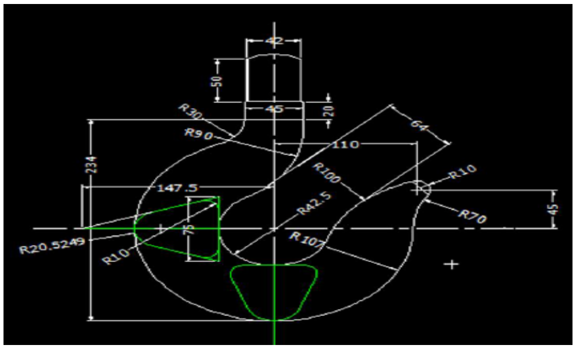


Fig 1-Detailed drawing of crane hook

CAD Model

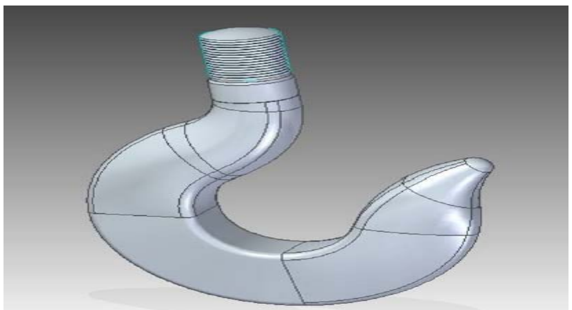


Fig 2–Finite element Model

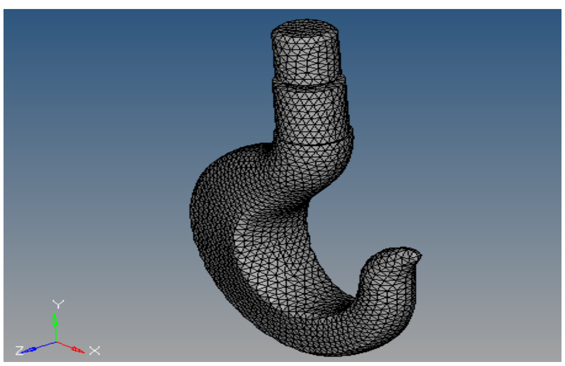


Fig 3– Meshed model

IV FINITE ELEMENT ANALYSIS

There are three methods to solve any engineering problem.

1. Analytical Method
2. Numerical Method
3. Experimental Method

Analytical Method

An analytical solution is a mathematical expression that gives the values of the desired unknown quantity at any location in the body; as a consequence it is valid for infinite number of location in the body. Analytical method is a classic approach which gives accurate results.

But this method is best suitable for simple problems like find the deflection of cantilever, simply supported beams etc and also find stresses and strains etc, by using ready-mate equations. But it consume more time as compare to Numerical Method.

Numerical Method

The use of numerical methods enables the engineer to expand his ability to solve practical design problems. It is not possible to obtain analytical mathematical solutions for many engineering problems. For problems involving complex materials properties and boundary conditions, the engineer's prefer to numerical methods that provide approximate, but acceptable solutions. Numerical method is a mathematical representation which gave approximate results.

Experimental Method

Experimental method is an actual measurement method. It physically test the prototype under varies condition. Thus it gives 100% accurate results. But engineers can't prefer because it require expensive set up and more time consuming method as compare with analytical method and numerical method.

Procedure for Finite Element Analysis

Certain steps in formulating a finite element analysis of a physical problem are common to all such analyses, whether structural, heat transfer, fluid flow, or some other problem. The steps are described as follows:

1. Pre-processing
2. Processing or Solution
3. Post processing

Finite Element Analysis for Design Engineer

The FEA offers many important advantages to the design engineer:-

1. Easily applied to complex, irregular-shaped objects and having complex boundary conditions.
2. Applicable to problems like steady-state, time dependent etc.
3. Applicable to linear and nonlinear problems.
4. One method can solve a wide variety of problems, including problems in solid mechanics, fluid mechanics, chemical reactions, electromagnetic, biomechanics and heat transfer etc.

5. The FEA can be coupled to CAD programs to facilitate solid modeling and mesh generation.

V ANSYS INTRODUCTION

Dr. John Swanson founded ANSYS, Inc. in 1970 with vision to commercialize the concept of computer-simulated engineering, establishing himself as one of the pioneers of finite element analysis (FEA). ANSYS Inc. supports the ongoing development of innovative technology and delivers flexible, enterprise-wide engineering systems that enable companies to solve the full range of analysis problem, maximizing their existing investments in software and hardware.

ANSYS has evolved into multipurpose design analysis software program, recognized around the world for its many capabilities. Today, the program is extremely powerful and easy to use. Each release hosts new and enhanced capabilities that make the program more flexible, more usable, and faster. In this way, ANSYS helps engineers met the pressures and demands of the modern product development environment.

The ANSYS program is a flexible, robust design analysis and optimization package. The software operates on major computers and operating systems, from PCs to workstations to supercomputers. ANSYS features file computability throughout the family of products and across all platforms. ANSYS design data access enables user to import computer-aided design models into ANSYS, eliminating repeat work. This ensures enterprise-wide, flexible engineering solution for all ANSYS user.

ANSYS finite element analysis software enables engineers to perform the following tasks.

1. Build computer models or transfer CAD models of structures, products, and components systems.
2. Apply operating loads or other design performance conditions.
3. Study physical responses such as, stress levels, temperature distributions or the impact of electromagnetic fields.
4. Optimize a design early in the development process to reduce production costs.
5. Do prototype testing in environments where it otherwise would be undesirable or impossible.

The ANSYS program has a comprehensive graphical user interface (GUI) that give easy interactive access to program functions, Commands, documentation and reference material. A system helps users navigate through the ANSYS program. Users can input data using the mouse, keyboard or the combination of both.

VI PROCEDURE FOR ANSYS ANALYSIS

Analysis in ANSYS can be either linear or non-linear. The procedure for any analysis consists of these main steps.
Pre-Processor Phase (Build the model)
Solution Phase (Obtain the solution)
Post-Processor Phase (Review the results)
Material used

Steels are alloys of iron and carbon, widely used in construction and other applications because of their high tensile strengths and low costs. Carbon, other elements, and inclusions within iron act as hardening agents that prevent the movement of dislocations that otherwise occur in the crystal lattices of iron atoms.

The carbon in typical steel alloys may contribute up to 2.1% of its weight. Varying the amount of alloying elements, their formation in the steel either as solute elements, or as precipitated phases, retards the movement of those dislocations that make iron comparatively ductile and weak, or thus controls qualities such as the hardness, ductility, and tensile strength of the resulting steel. Steel's strength compared to pure iron is only possible at the expense of ductility, of which iron has an excess.

Although steel had been produced in bloomer furnaces for thousands of years, steel's use expanded extensively after more efficient production methods were devised in the 17th century for blister steel and then crucible steel. With the invention of the processing the mid-19th century, a new era of mass-produced steel began. This was followed by Siemens-Martin process and then Gilchrist-Thomas process that refined the quality of steel. With their introductions, mild steel replaced wrought iron.

Further refinements in the process, such as basic oxygen steelmaking (BOS), largely replaced earlier methods by further lowering the cost of production and increasing the quality of the metal. Today, steel is one of the most common materials in the world, with more than 1.3 billion tons being produced annually. It is a major component in buildings, infrastructure, tools, ships, automobiles, machines, appliances, and weapons. Modern steel is generally identified by various grades defined by assorted standards organizations.

Calculations & Boundary conditions. Static analysis

We have to perform static analysis to extract stresses and displacements

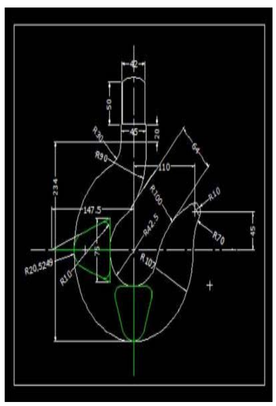
We apply 2000 kgs

So W = mg = 2000 x 9.81 kg m/s2

= 19620 kg m/s2

= 19620 N

$$H = 0.93 \times C = 0.93 \times = 80\ mm$$
$$M = 0.6 \times C = 0.6 \times 8.5 = 51\ mm$$
$$= \frac{\frac{M \times H}{2}}{\left[\left(\frac{M_0 \times r_0}{H}\right)\log\frac{r_0}{r_1} - M\right]}$$
$$= \frac{\frac{51 \times 80}{2}}{\left[\left(\frac{51 \times 122.5}{80}\right)\log\frac{122.5}{42.5} - 51\right]}$$
$$r_n = 70.1899\ mm$$
$$(\sigma_t)\ total = (\sigma_{t)ri} + \sigma_{t)}$$
$$= 86.0457 + 12.4571$$
$$= 98.5028\ N/mm^2$$



Used to determine displacements, Stresses, Strain, Deformation etc. under static loading conditions, comparing two materials as shown below with a load of 19620 N

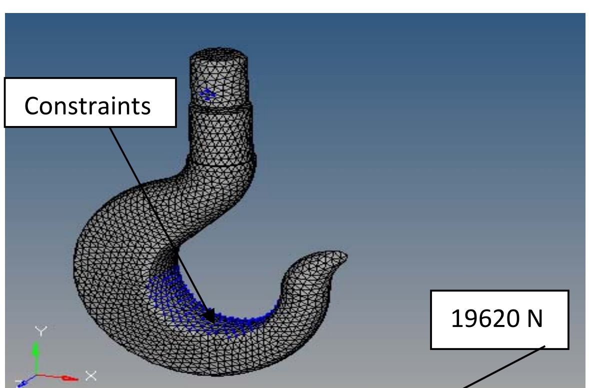


Fig 4–Meshed model

Assumptions

Element: PSOLID to assign solid elements for 3D meshing

Material: Forged steel

Mesh Element type: Tetra element

Mesh Type: Solid Mesh

Load applied was 19620 N

Rigid body element (RBE): Rigid body element can be used on bolt locations to make a rigid region. It does not require a solid bolt, it use a line element and RBE elements to represent the stud and bolt respectively.

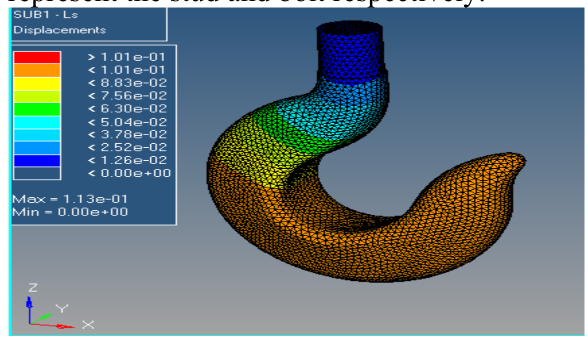


Fig 5- Displacement in Forged steel:0.101mm

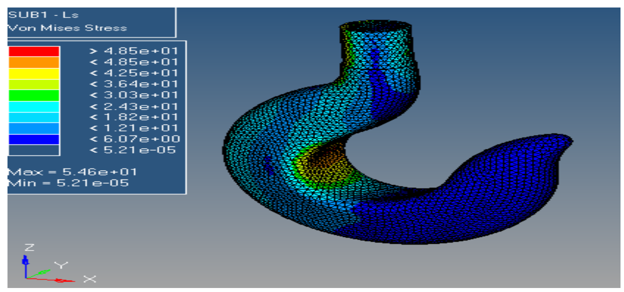


Fig 6-Von misses stresses in Forged steel: 48.5 N/mm2

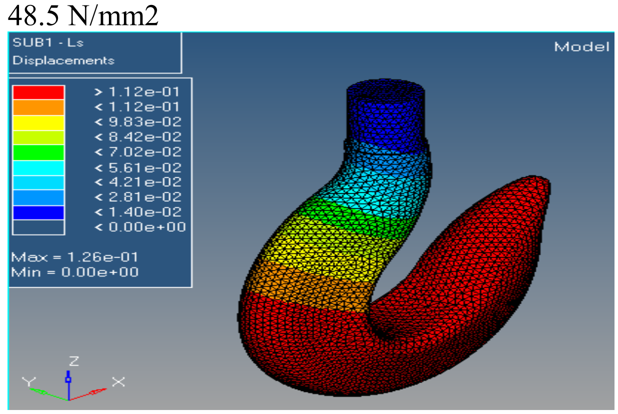


Fig 7- Displacement in Wrought iron: 0.112mm

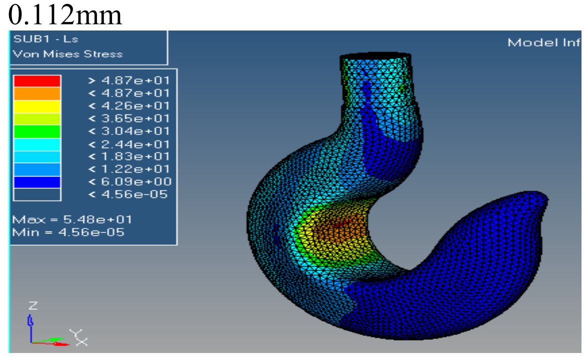


Fig 8-Von misses stresses in Wrought iron: 48.7N/mm2.

Table 1 – Results

Material	Deformations mm	Stresses N/mm2
Forged steel	0.101	48.5
Wrought iron	0.112	48.7

CONCLUSION

In design of Crane hooks FEA tool can be effectively used. Typically it helps the designer to understand behavior of pressure vessel. Thus, among the viable materials used for making hookswhichwere analyzed in this work, it was concluded that wrought iron and forged Steel are more suitable for making crane hooks as they have higher capacity to withstand loading, because results variations is very less.

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] 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" in International journal of soft computing and Engineering, Volume-2, Issue-1, March 2012, ISSN: 2231-2307.

[3] Nishant soni, "crane-hook shapes optimization and thermal analysis using finite element too" in international journal of advanced and innovation research", ISSN: 2278-7844.

[4] 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.

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

[6] 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.

[7] 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.

[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.

[9] Santosh Sahu, Ritesh Dewangan, Manas Patnaik, Narendra Yadav,"Study of Crane Hook Having Trapezoidal Section by Finite Element Method& Design Experiments" in International Journal of Modern Engineering Research, vol. 2.Issue-4,JulyAug 2012,pp-2779-2781.

[10]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.

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

[12] Spasoje Trifkovic' et al, "Stress analysis of crane hook using FEM", INFOTEH-JAHORINA Vol. 10, Ref. C-2, p. 244- 248, March 2011.

[13] Y. Torres, J.M. Gallardo, J. Dominguez, F.J. Jimenez E, "Brittle fractures of a crane hook", Engineering Failure Analysis 17 (2010) 38-47.