JOURNAL OF CRITICAL REVIEWS
ISSN- 2394-5125 VOL 7, ISSUE 15, 2020

DESIGN AND OPTIMIZATION OF CRANE HOOK

GEDDADA DIVAKAR[1], CH.SIVA RAMAKRISHNA[2], SCV.RAMANAMURTYNAIDU[3] K.TARANIKANTH[4], YARRAPRAGADA.K.S.S.RAO[5]

[1,2,4] Dept of Mechanical Engineering, Vignan's Institute of Information Technology, Visakhapatnam, India
[3]Kallam Harinadh Reddy Institute of Technology, Guntur, India
[5]Aditya College of Engineering, Surampalem, India

divakar314@gmail.com[1], srkchadaram@gmail.com[2], scvrmn@gmail.com[3], taranikanth1986@gmail.com[4], subbaraoyerrapragada@gmail.com[5]

ABSTRACT

Crane hook have a very important role in many sectors of various industries like steel industries, ship building, constuction field etc., for lifting loads and carry them. Many types of hooks having different sizes and ratings are used according to the specified objectives. Crane hooks are highly liable components and are allways 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. In this research the design of the hook is done by analytical method and analysis is done using different matrials like wrought iron, low alloy steel (4340) and high strength low alloy steel (HSLA). The modelleing and analysis of crane hook is done in CATIA and ANSYS software used to find out the stresses induced in it. This result helps us for determining of stress in existing model. By predicting the stress concentration area and stress value failure stress and optimized can be obtained.

KEYWORDS: Crane hook, Stresses, Different materials, Modelling and analysis softwares and Optimization

1. INTRODUCTION

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. The commonly used crane hooks with trapezoidal, circular, rectangular and triangular cross section. Crane hooks are the components which are generally used to elevate the heavy load in industries and constructional sites. Crane hooks are always subjected to failure due to accumulation of large amount of stresses which can eventually lead to its failure. If a crack is developed in the crane hook, mainly at stress concentration areas, it can cause fracture of the hook and lead to serious accidents. So, it must be designed and manufactured to deliver maximum performance without failure.

2. LITERATURE REVIEW

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 the 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 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. 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.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. Pradyumnakeshri maharana. [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 least 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. **2.1 summary of literature:** The aim of the literature is to reducing the failures of hooks the estimation of stresses, their magnitudes and possible locations are very important. From the stress analysis, they have observed the cross section of max stress area. If the induced area on the inner side of the crane hook at the portion of maximum stress is broadened then the bending stresses will get minimized. The stress concentration is used to evaluate the strength and durability of structures and engineering machine elements. The lifting hooks design rules require using different newly developed materials to avoid the failure based on simulation parameters

3. METHODOLOGY

The solid model of crane hook is made using CATIA software and it is imported to ANSYS software where the stresses of crane hook is determined. ANSYS has evolved into Multipurpose design analysis software program, recognized around the world for its many 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.**3.1The Procedure For Using ANSYS :**The created 3D models are imported in to the ANSYS WORKBENCH by means of .IGES format. This is a general format used for exchanging of CATIA model data into another software's. The work setup is placed under static analysis .In engineering data book of ANSYS WORKBENCH four types of material properties are added namely wrought iron, low alloy steel and High strength low alloy steel.ANSYS consists of three mains steps for simulation of model. 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 other problem. **3.1.1Pre-Processing :** In this progression, the finite element mesh for the designed model is produced and boundary conditions, material properties, and loads are applied to the composed model.**3.1.2.Solution :** In this progression, the solutions for the problems for the given loads and boundary conditions. The outcomes, are Von Mises stress, displacements, strain, thermal impacts etc., are acquired in this progression.**3.1.3.Post-Processing :**In this progression, the results are pictured as contours, deformed shapes, and plots. This progression helps in the investigating, confirmation and approval of results.

3.1 Hook Design

The Crane-hook is modelled in CATIA Part design. Fig.1 shows the front view of the crane hook.The model is meshed with finite elements as shown in fig.2.

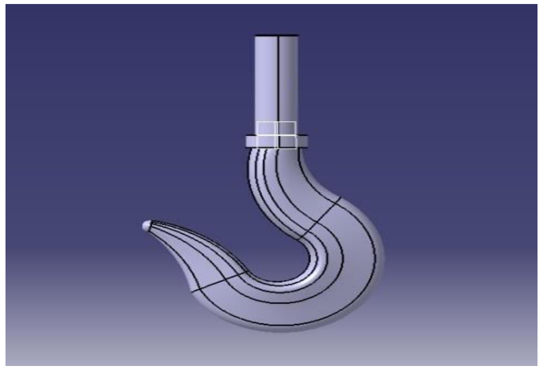

Fig.1 Front view of the crane hook.

3.1.2 **Materials Used :**The materials used in the study of crane hook are wrought iron, low alloy steel and high strength low alloy steel which are given in the below Table.1.

Table.1.1 Material proprетés				
S.No	**Property**	**Wrought Iron**	**Low Alloy Steel**	**High Strength Low Alloy Steel**
1	Yield strength (Pa)	3.44E+08	1.7E+09	3.78E+08
2	Tensile strength (Pa)	5.48E+08	1.17E+09	4.57E+08
3	Young's modulus (Pa)	1.71E+11	2.09E+11	2.1E+11
4	Poisson 's ratio	0.275	0.29	0.3
5	Mass density (kg/m^3)	7130	7850	7850
6	Shear modulus (Pa)	6.7059E+10	8.1008E+10	8.0769E+10

The volumetric properties of the materials for the hook modelled using are shown in Table 2.

Table 2. Volumetric Properties			
S.No	**Materials**	**Mass (kg)**	**Volume (mm^3)**
1	Wrought iron	17.067	2.3937E+006
2	Low alloy steel	18.79	2.3937E+006
3	High strength low alloy steel	18.79	2.3937 E+006

3.2 Meshing

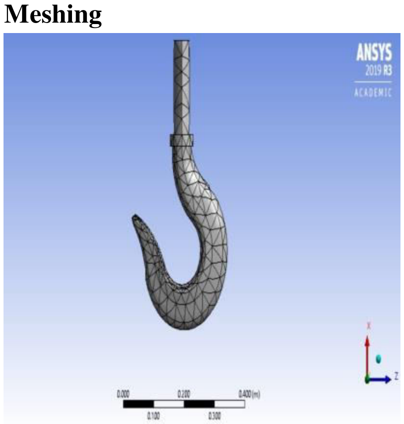

Fig.2 Meshing

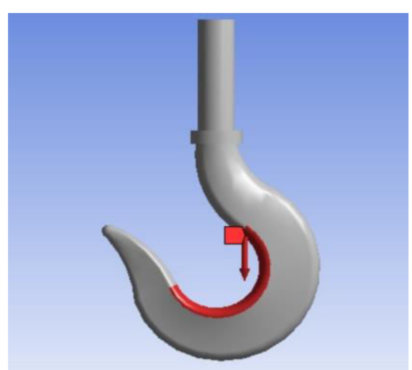

Fig.3 Boundary conditions

Meshing plays an important role in evaluating of results. In this case tetrahedral method of meshing is carried for the imported CAD model and fine type of meshing with Advanced Curvature option is used for them meshing. **3.3.Boundary conditions and load applied :**The one end of the hook is fixed with the help of fixed support option and at the other end load is applied on bunch of node at the lower centre of the hook of load (10 KN) in downward direction as shown in Fig.3.**Solutions :**In solution column three types of results will be needed to evaluate the project namely equivalent stress, equivalent strain and total deformation. These three parameters are developed in solution column and are generated. The solutions are generated and displayed graphically.Equivalent Stress,strain and total deformation for Wrought Iron, Low Alloy Steel and High Strength Low Alloy Steel are shown in Fig.4 to Fig.12.

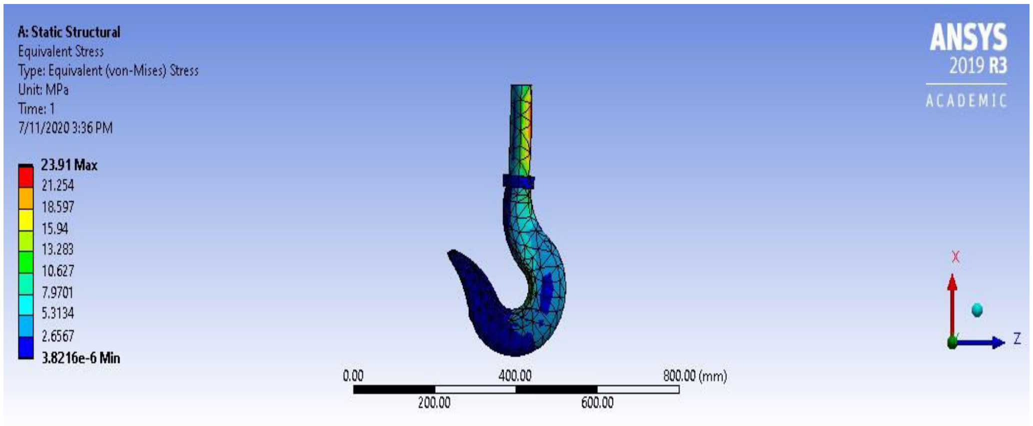

Fig. 4 Equivalent stress for Wrought Iron material

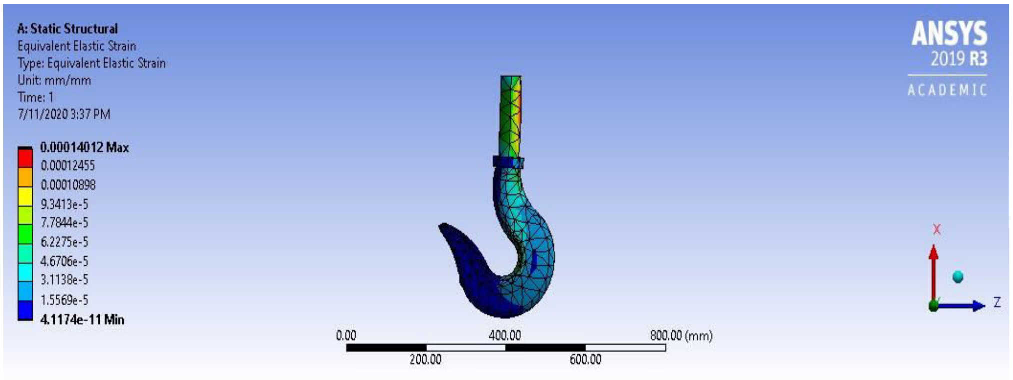

Fig.5 Equivalent strain for Wrought Iron material

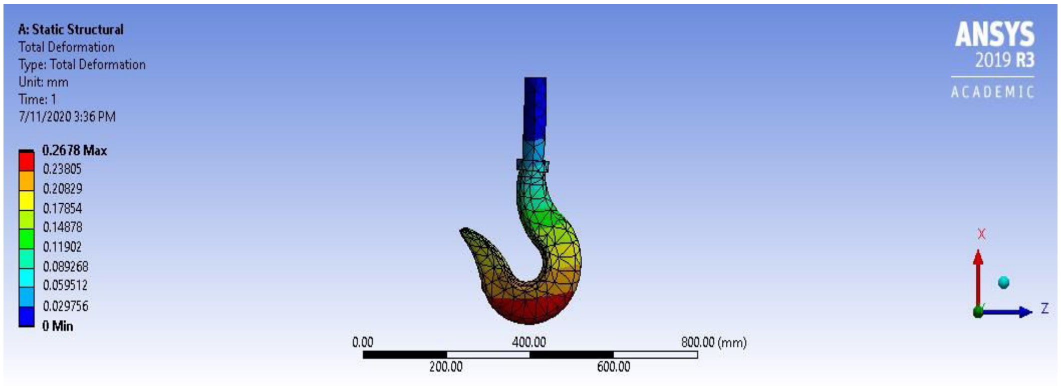

Fig. 6 Total déformation for Wrought Iron material

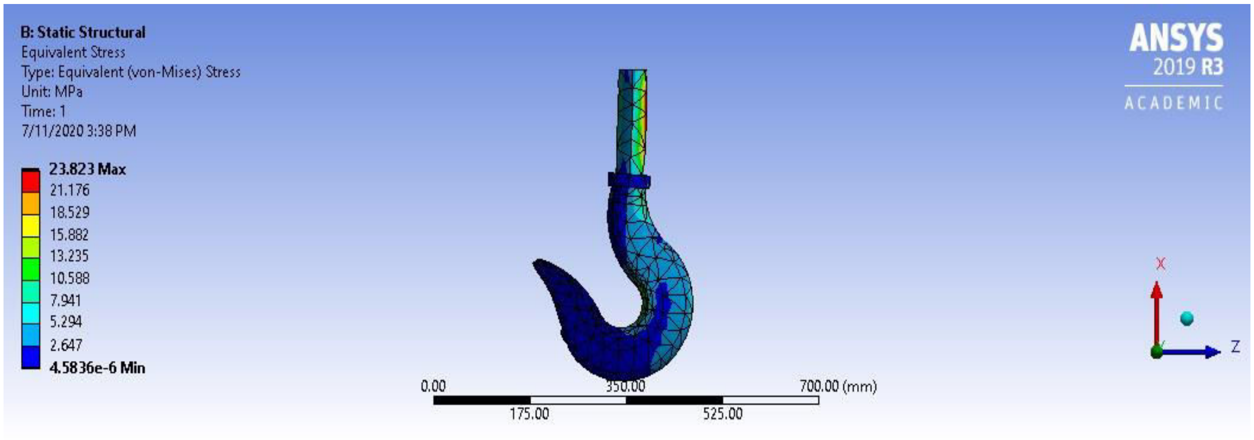

Fig.7. Equivalent stress for low alloy steel

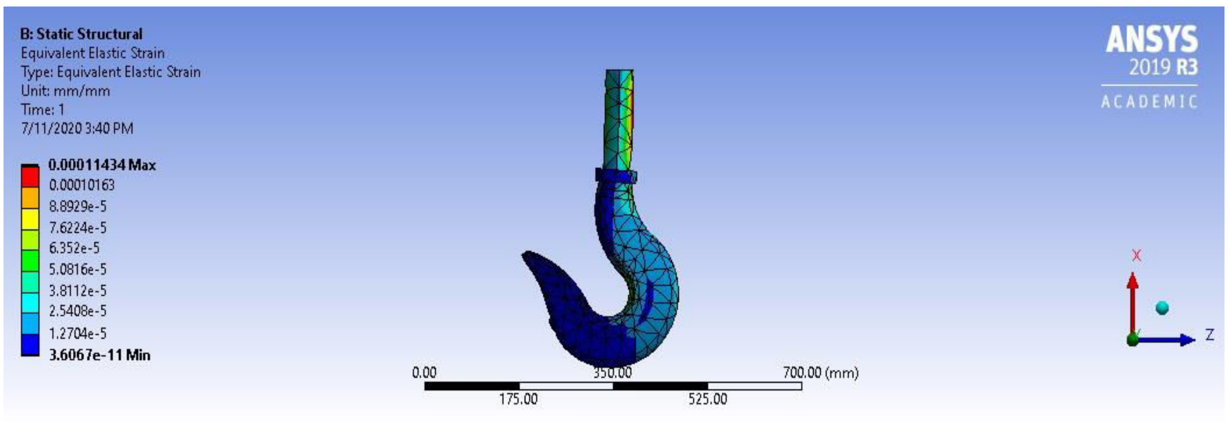

Fig.8. Equivalent strain for low alloy steel

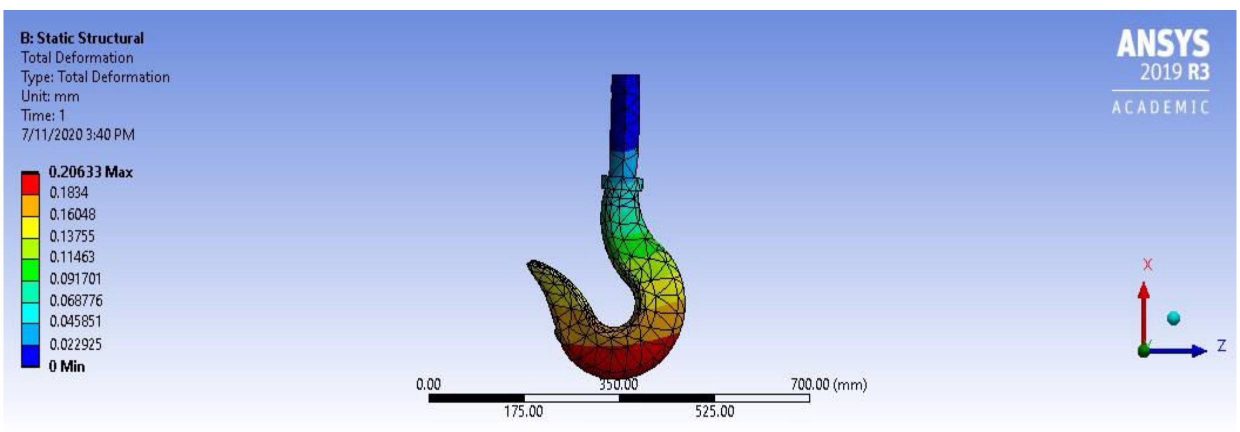

Fig.9. Total déformation for low alloy steel

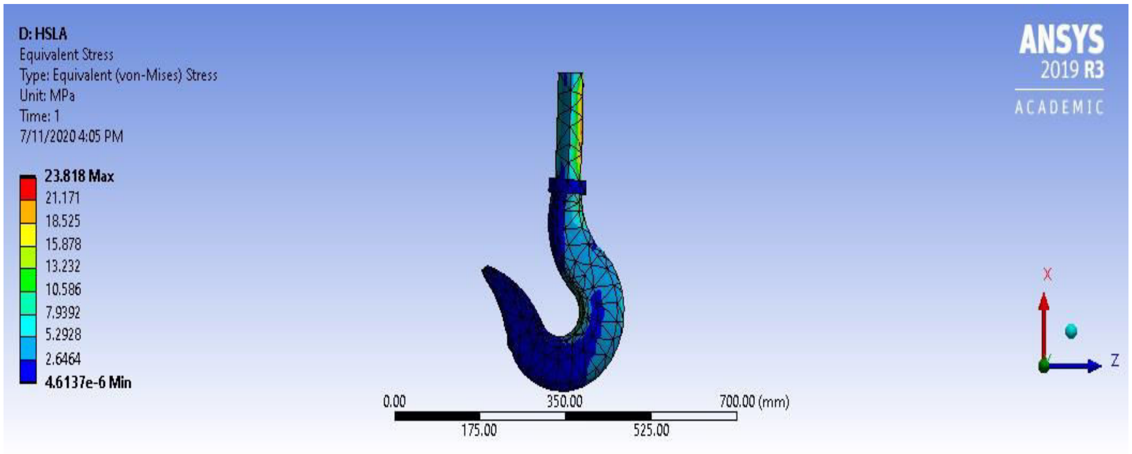

Fig.10. Equivalent stress for high strength low alloy steel

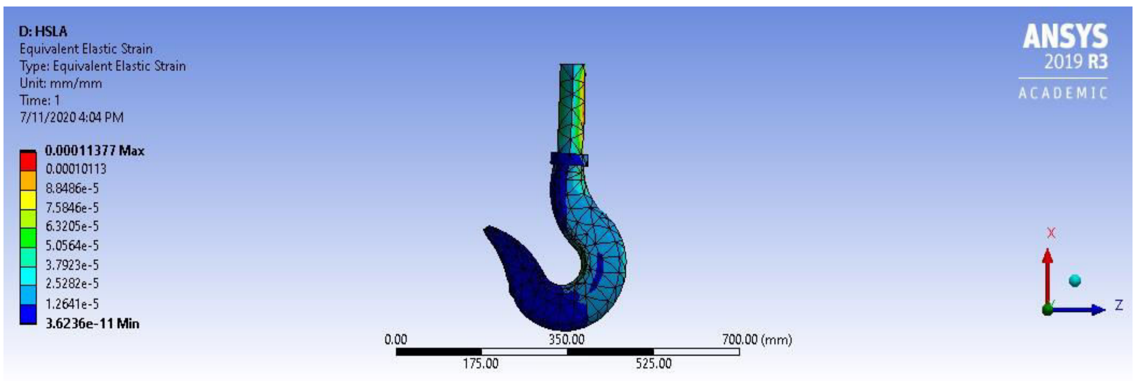

Fig.11Equivalent stress of high strength low alloy steel

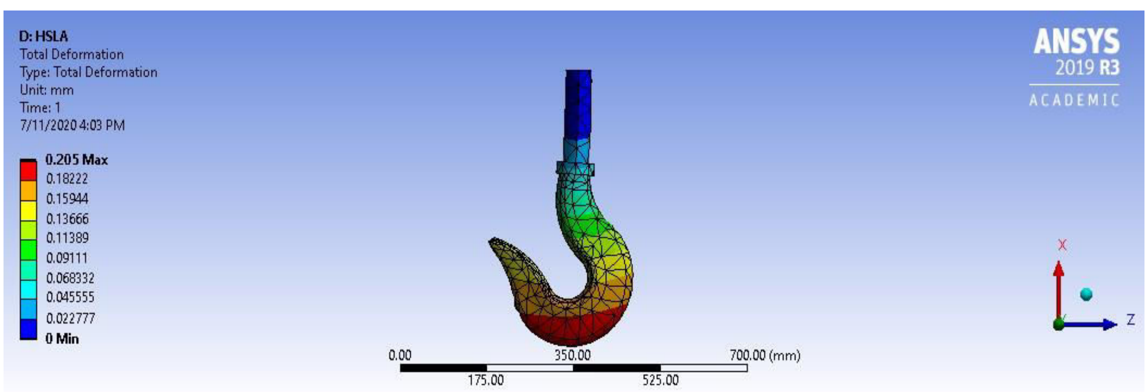

Fig.12. Total deformation of high strength low alloy steel

4. RESULTS AND DESCUSSION

The results of equivalent (von-mises) stress analysis, Equivalent strain analysis and total deformation are evolved from FEM (ANSYS WORKBENCH) software by importing the 3D CATIA model to analyse the models with different materials such as wrought iron, low alloy steel and high strength low alloy steel. The comparision of results are tabulated in Table.3.

Table.3 Comparison of Results				
S.No	Material	Equivalent Stress (MPa)	Equivalent Strain	Total Déformation (mm)
1	Wrought Iron	23.91	0.00014012	0.2678
2	Low Alloy Steel	23.823	0.00011434	0.20633
3	High Strength Low Alloy Steel	23.818	0.00011377	0.205

From the above table equivalent stress, equivalent strain and total deformation are maximu in Wrought iron and minimum in High strength low alloy steel.

5. CONCLUSIONS & FUTURE SCOPE OF WORK

1. The static structural analysis has been carried out to estimate the maximum stress, strain and total deformation in crane hook model using ANSYS software.This analys is concluded that the material which ishaving less deformation will be more stable.The material High strength low alloy steel with less deformation and stress, Hence it is optimized material material for making of crane hook.

6. 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.