iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 7 Issue: IV Month of publication: April 2019

DOI: https://doi.org/10.22214/ijraset.2019.4656

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IV, Apr 2019- Available at www.ijraset.com

Design and Dynamic Analysis of Crane Hook with Niobium Micro Alloying Steel

Manoj Hirani[1], Prof. **Ronak Shah**[2]
[1]P G Student M.E. (MACHINE DESIGN), [2]Assistant Professor, Mechanical Engineering Department, Kalol Institute of Technology and Research, Gujarat Technological University, Gujarat, India.

Abstract: The crane hook mostly subjected to failure due to accumulation of large amount of stresses which can eventually lead to its failure. Failure of a crane hook mainly depends on three major factors i.e. dimension, material, overload. The design of crane hook has been carried out. To minimize the failure of crane hook, the stress occurred in it must be studied Generally, the design of the hook is done for the different materials like SAE-AISI 1040, ASMT GRADE 60, Forged steel, wrought iron, Aluminium alloy, High strength low alloy steel, and Crome steel but I want to use the material of crane hook as a Niobium Micro alloying Steel. And analyse the all parameters of other materials with Niobium Micro alloying Steel. Pro-E Wildfire 5.0 software is used for modelling the crane hook 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, the hook working life increase and reduce the failure stress
Keywords: Crane Hook, Design, FEA, Niobium Micro alloying Steel, ANSYS software

I. INTRODUCTION

Crane hooks are the components which are generally used to lift the heavy load in industries and constructional work. Recently, excavators having a crane-hook are widely used in construction works site. One reason is that such an excavator is convenient since they can perform the conventional digging tasks as well as the suspension works. Another reason is that there are work sites where the crane trucks for suspension work are not available because of the narrowness of the site. In general, an excavator has superior manoeuvrability than a crane truck. Very few people have already worked on the optimization of crane hook. Generally material type and cross section area and radius are design parameter that affects the weight of crane hook. SAE-AISI 1040, ASMT GRADE 60, Forged steel, wrought iron, Aluminium alloy, High strength low alloy steel, and Crome steel are generally used as manufacturing material for crane hook. The behaviour of mechanical properties of different steel grades at elevated temperatures should be well known to understand the behaviour of steel and composite structures at fire. Quite commonly simplified material models are used to estimate e.g. the structural fire resistance of steel structures. In more advanced methods, for example in finite element or finite strip analyses, it is important to use accurate material data to obtain reliable results.

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 cause fatigue failure of the crane hook, but the load cycle frequency is very low. 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. In ductile fracture, the crack propagates continuously and is more easily detectable and hence preferred over brittle fracture. In brittle fracture, there is sudden propagation of the crack and the hook fails suddenly. This type of fracture is very dangerous as it is difficult to detect. Strain aging embrittlement due to continuous loading and unloading changes the microstructure. Bending stresses combined with tensile stresses, weakening of hook due to wear, plastic deformation due to overloading, and excessive thermal stresses are some of the other reasons for failure. Hence continuous use of crane hooks may increase the magnitude of these stresses and ultimately result in failure of the hook.

II. LITERATURE REVIEW

A. S. Shanmugam, R. Thridandapani, D. Misra, T. Mannering, D. Panda, S. Jansto, Some Physical Metallurgy Aspects of Niobium- and Vanadium-Micro alloyed steels for Structural Beams

The broad experience on Nb-metallurgy in flat products can be extended to structural and engineering steel grades. Many users have benefited from the current favourable economics of introducing Nb into long products on low as well as medium carbon steels. The reheating furnaces and appropriate heating and combustion practices are vital to successfully executing the substitution of Nb for other microalloys. Rising requirements for fatigue strength, toughness and weldability can be met by reducing the C-content. The resulting decrease in strength is compensated by addition of small amount of Nb.

B. G.E.V.Ratnakumar, B. Jitendra Kumar, KalapalaPrasad, "Design and Stress Analysis of Various Cross Section of Hook",

This project is carried out to study the stress variation in crane hooks for different cross sections such as circular and square and for different radii of curvature as well, experimentally and theoretically. Experimentally, the loads are obtained for different crane hooks for 5mm elongation on UTM (Universal Testing Machine).

And then the stresses induced in the crane hooks against the loads obtained from experimentation are also calculated theoretically using curved beam theory. Then the different crane hooks are modeled in Pro-E 2.0 and then analysis is done for modeled hooks using ANSYS 12.1 to find the stresses induced in the hooks. Then the stresses evaluated from curved beam theory and ANSYS12.1 is compared and conclusions are made.

C. Amandeep Singh, VinodRohilla, "Optimization and Fatigue Analysis of a Crane Hook Using Finite Element Method"

Stress analysis plays an important role in the design of structures like crane hook under loading conditions. Structure failure of crane hook occurs because of the stress induced due to repetitive loading and unloading conditions. In this study, solid modeling of crane hook having trapezoidal cross-section referring to one of its existing design is done using SOLIDWORKS. Further, analyses are carried out in ANSYS Workbench and n-Code Design Life.

The lengths of two parallel sides of the cross-section of crane hook are varied and different candidates are obtained for loading capacity of 30 tons on the basis of Mass, total Displacement and Von-Mises stress. This is done to reduce weight and balance economy.

The structure strength is an important signal to response the load bearing capability of the elevating equipment. Fatigue damage is the initiation of a crack due to fluctuating loading.

It is caused due to stress levels which are insufficient to cause damage in a single application. It is highly responsible and important component being used for industrial works. It is a hoisting fixture manufactured and designed to engage a link of a chain or ring or the pin of a cable socket or shackle and safety guidelines must be followed. In the present work, study of different candidates of crane hook of trapezoidal cross section based on design parameters are carried out and weight optimization is carried out by varying the design parameters. The present study shows the deformation and fatigue life contour plots of crane hook using Ansys Workbench and Ansysn Code Design Life.

D. Omkar P. Bhatkar, Arun V. Javir, Shinde Ajay Ashok, Sawant Subodh Sumant, UkardeSurajEknath, KavathankarKunalShivram, "Finite Element Analysis of Crane Hook and Optimization Using Taguchi Approach"

A Crane hook is a device for grabbing and lifting loads of heavy duty by mean of a device such as crane hoist. A crane hook is usually equipped with a safety latch to prevent the disengagement of the lifting rope to which the load is being attach. The CAD model of a lifting crane hook is initially prepared from the Analytical Design.

In the present report crane hook of trapezoidal section is modeled using UG-NX 8.0, further, importing the CAD model into ANSYS 15.0 we estimate until 20 tones i.e. equivalent to 200000N load to be gradually applied, eventually we observed the von-misses stress developed in the model by FEM method these results von-misses stresses are being compared with Taguchi L9 orthogonal array for specific results.

Thus, these results would lead us to determine various stresses and deflections in the designed CAD model. In order to reach the most optimum model from different materials of hook would be tested and the most efficient material would be selected. Crane hooks are one of the important components used in industries to carry heavy duty loads; basically, crane hooks are designed with pulley elongated by rope or a chain. 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 and must follow the health and safety guidelines. Thus, such a crucial component in an industry must be fabricated and designed in a way so as to deliver maximum performance without breakdown.

E. A. Gopichand, Ms. R.V.S.Lakshmi, Mr. B. Maheshkrishna, "Optimization Of Design Parameters For Crane Hook Using Taguchi Method"

In the present work optimization of design parameters is carried out using Taguchi method, total three parameters are considered with mixed levels and L16 orthogonal array is generated. The optimum combination of input parameters for minimum Von-mises stresses are determined. From the Taguchi analysis the optimum combination of input parameters for von misses' stress were determined using SN ration plots, the optimum combination is TRIANGULAR cross section, Castiron material and 200mmRADIUS. The first model simulation values are compared with analytical values and the error is 9.26%which is acceptable. In this paper mathematical calculations are done by varying cross sections triangular, trapezoidal, and rectangular with constant area of cross sections for two different materials of steel, cast-iron analysis are evaluated. PRO/E design is prepared, for different cross sections by preparing the trajectory with the parameters taken from the design data book. In this project result of mathematical calculations is compared with the ANSYS with constant area of cross section, modifying the type of material optimization of better results were done in Taguchi Method.

F. Rashmi Uddanwadiker,Department Mechanical Engineering, Visvesvaraya National Institute of Technology, Nagpur, India "Stress Analysis of Crane Hook and Validation by Photo-Elasticity"

To study the stress pattern of crane hook in its loaded condition, a solid model of crane hook is prepared with the help of CMM and CAD software. Real time pattern of stress concentration in 3D model of crane hook is obtained. The stress distribution pattern is verified for its correctness on an acrylic model of crane hook using Diffused light Polari scope set up. By predicting the stress concentration area, the shape of the crane is modified to increase its working life and reduce the failure rates. In this paper to study the stress pattern of crane hook in its loaded condition, a solid model of crane hook is prepared with the help of CMM and CAD software. Real time pattern of stress concentration in 3D model of crane hook is obtained. The stress distribution pattern is verified for its correctness on an acrylic model of crane hook using Diffused light Polariscope set up. By predicting the stress concentration area, the shape of the crane is modified to increase its working life and reduce the failure rates. Max value obtained analytically=12.35 N/mm2 while value obtained from ANSYS = 13.372 N/mm2. The results are in close harmony with a small percentage error = (13.372 – 12.35)/12.35 = 8.26% Possible reasons for variation might be the due to assumption that 1) Loading considered as point loading in analytical calculation while it is taken on a bunch of nodes in ANSYS. 2) Cross sectional area is assumed to be trapezoidal and 3) Plane sections remain plane after deformation.

G. Nutan Joshi1, Dr. K. B. Waghulde, "Finite Element Analysis and Optimization of Crane Hook"

In the present work, the geometry of the crane hook is modelled and finite element analysis (FEA) is applied on the model. The results of static structure analysis are obtained from the FEA software ANSYS for different dimensions. From above project the FEA and experimental results are validated. The error in FEA result and Experimental result is 2.7%. The stresses and deformation are in permissible limit in above design. In future different cross sections are to be used and altered. It can be studied for different type of materials also. Further it is advisable to conduct photo elasticity test for the crane hook under investigation in order to get better insight for stress concentration. 3D model of a crane hook will be drawn in CATIA V5 R19, Meshing will be carried out in Hyper mesh, and ANSYS will be used for post-processing. FE Analysis will be carried out on the existing crane hook. Based on those FEA results topological changes will be made to reduce weight and stress concentration. Various parameters of hook will be changed and analysed for optimum design. If necessary different cross sections of hook such as circular section, T-section, rectangular section etc. will be analysed to find better results FEA results of the existing crane hook will be compared with the FEA results of the optimized crane hook. After getting satisfied results a prototype will be fabricated and tested. The two results i.e. FEA and experimental results will be validated.

III.RESEARCH GAP

Out of all this cross-section, trapezoidal cross section material can be changed with newly develop material like niobium microalloying steel which may be a new alternate of forged steel in crank hook. Finite element analysis, of a crane hook or a lifting hook which find widespread applications in all industries, construction sites etc. is carried out. A CAD model of crane hook will be generated in Pro-E software, and that file is then imported in ANSYS workbench.

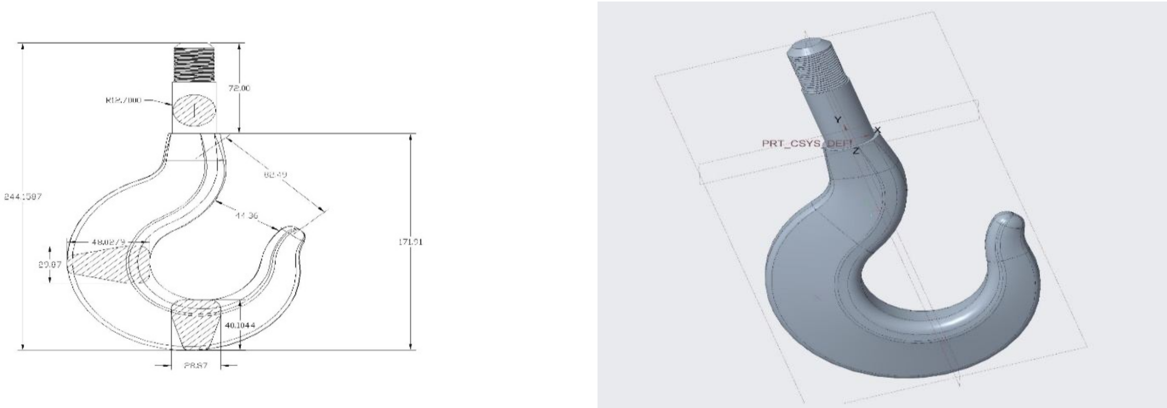
The crane hook is fixed at the top portion with the help of clamping device and then on the bottom curved portion load is applied that produce stress in crane hook. In the present work design optimization of crane hook is carried out the using ANSYS software

IV.OBJECTIVE

The objective of this work is to investigate the property of niobium microalloying steel.To compare the design parameters of niobium microalloying steel with the other materials like forged steel, wrought iron, SAE – AISI 1040, ASMT GRADE 60, structural steel, High strength low alloy steel and Aluminium alloys with different cross-section.A virtual model of lifting hook similar to actual sample is created using Pro-E software and then model was imported to ANSYS software for Finite element stress analysis and the result of stress analysis are cross checked with that of Winkler-Bach formula for curved beams.The objective of this work is to identify the causes that led to a failure of the crane hook in service. The causes of failure are stress concentration, material of hook, manufacturing process of crane hook, wear, plastic deformation due to loading, and continuous use of crane hook may accelerate the failure of crane hook

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IV, Apr 2019- Available at www.ijraset.com

V. DESIGN



Crane Hook Geometry and model

A. *Bed Diameter*

$$a_1 = x\sqrt{p}$$

$= 11\sqrt{19.62} = 48.72$mm
≈ 49 mm (approx)
Where, P = load, KN
X=constant ranging between 11 to 15 for economic design, x should be as minimum as possible.

B. *Throat of Hook (a_2)*

$a_2 = 0.9a_1$
$=0.9\times49=44.1$mm
≈44 mm

C. *Depth of cross-section Area (h_1)*

$h_1 = 10\sqrt{p} + \frac{a_1}{10}$
$=10\sqrt{19.62} + \frac{49}{10}$
=49.19mm
≈49mm
Where,
P=Load, KN
a_1=Bed diameter, mm

D. *Width of cross-section (b_1)*

$b_1 = 0.6h_1$
$= 0.6\times49.19=29.51$mm
≈30mm (approx)

E. *Diameter of cross-section (d_1)*

$d_1 = 0.5h_1$
$=0.5\times49$
=24.5mm

F. *Determine the Length (e_2)*

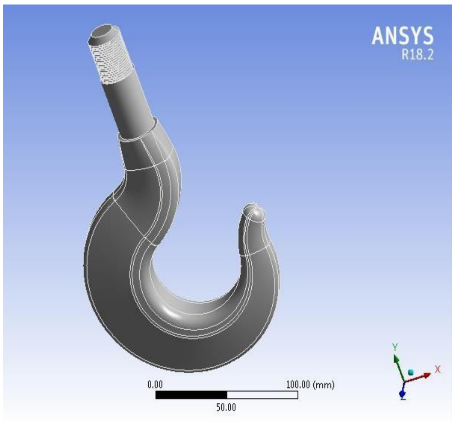
$e_2 = 2.75a_1$
$=2.75\times48.72=131.54$mm
≈132 mm (approx)

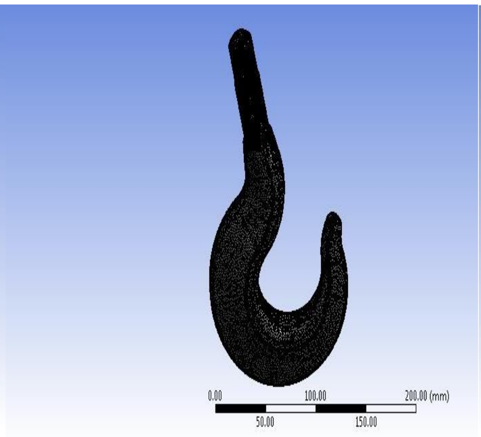
International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IV, Apr 2019- Available at www.ijraset.com

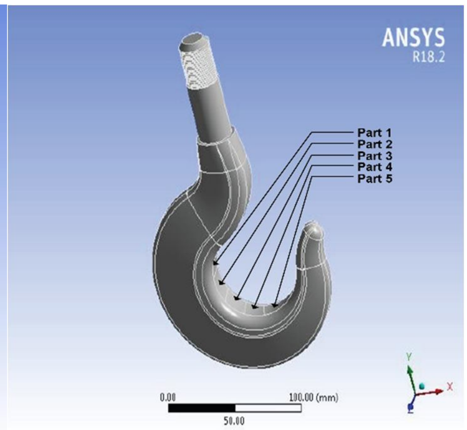
VI. DYNAMIC ANALYSIS

Dimensions are taken through reverse engineering. Dimensions for crane hook were measured from site. This dimensions taken from the actual model of crane hook were used for 3D modelling of crane hook. Below are some of the images taken at site. A rough hand sketch was drawn showing all the dimensions of crane hook. Dimensions are required for calculating of boundary conditions. Hence its CAD model is necessary. CAD model then is made by the commands in Pro-E software.

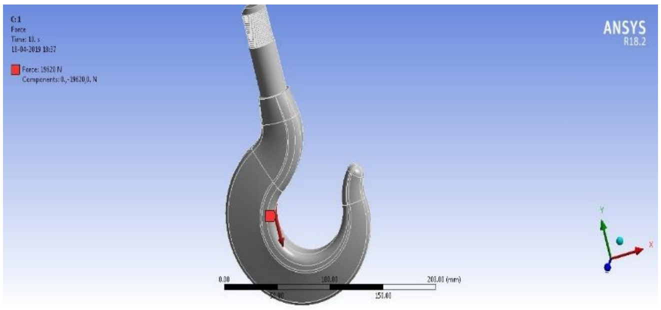
Nodes	233565
Elements	143437
Element Size	1.0 mm

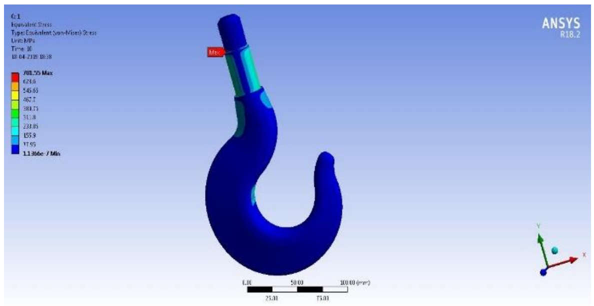


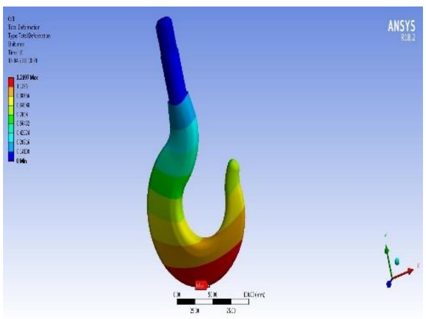


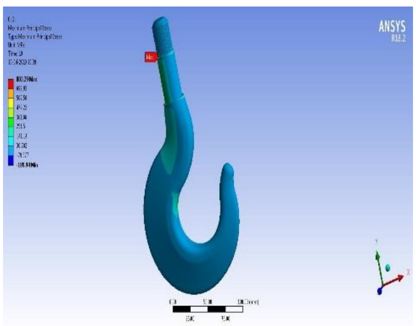


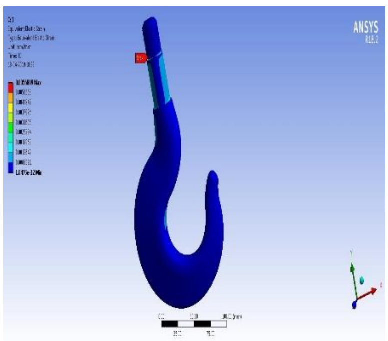
1) For Part 1





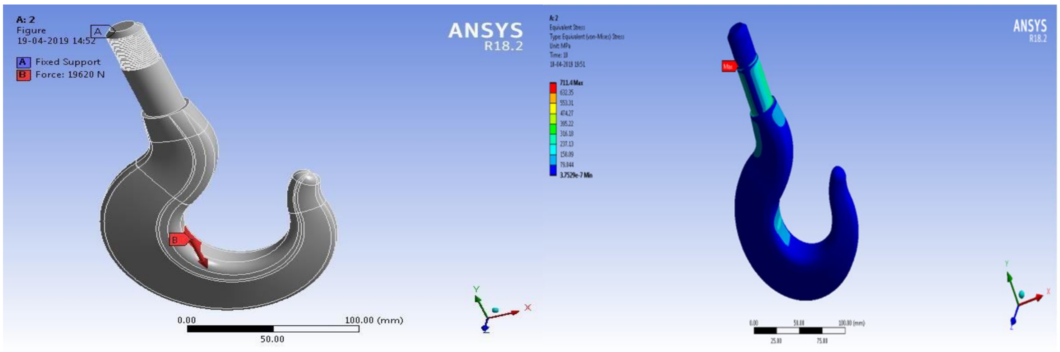


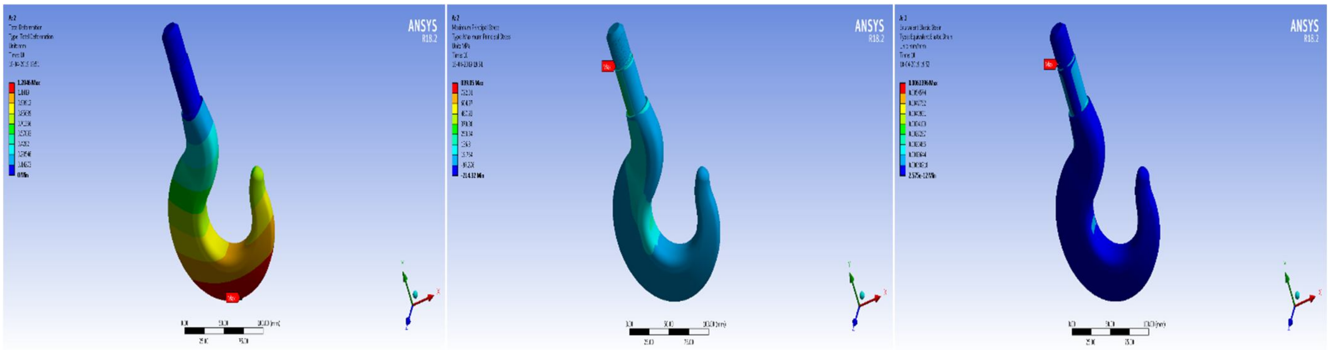




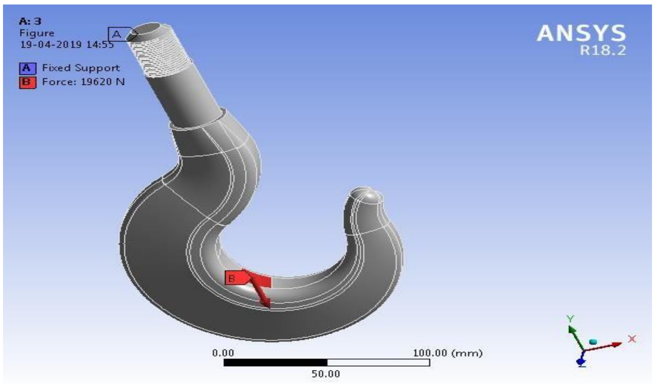
International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IV, Apr 2019- Available at www.ijraset.com

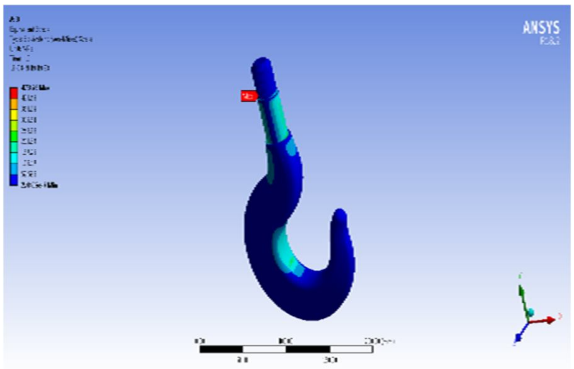
2) For Part 2

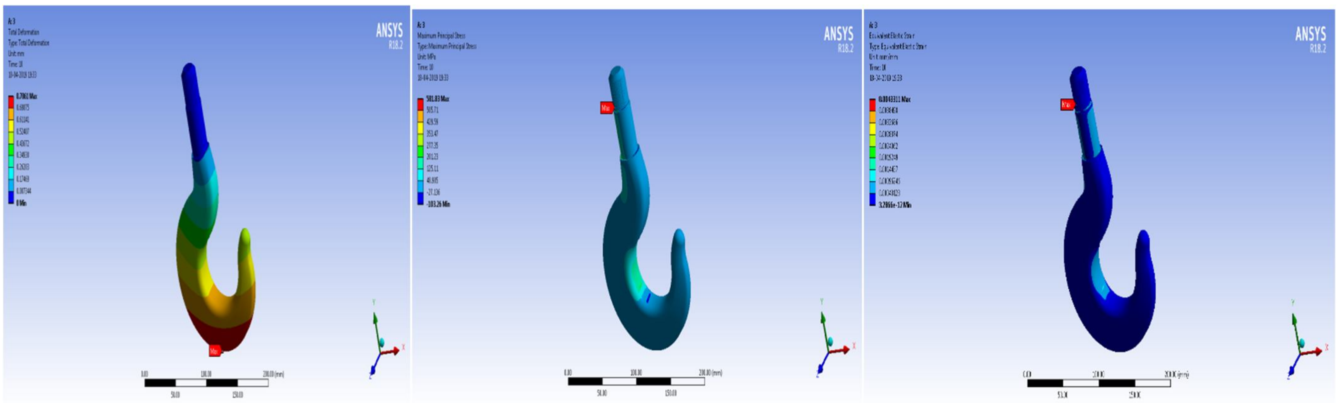




3) For Part 3

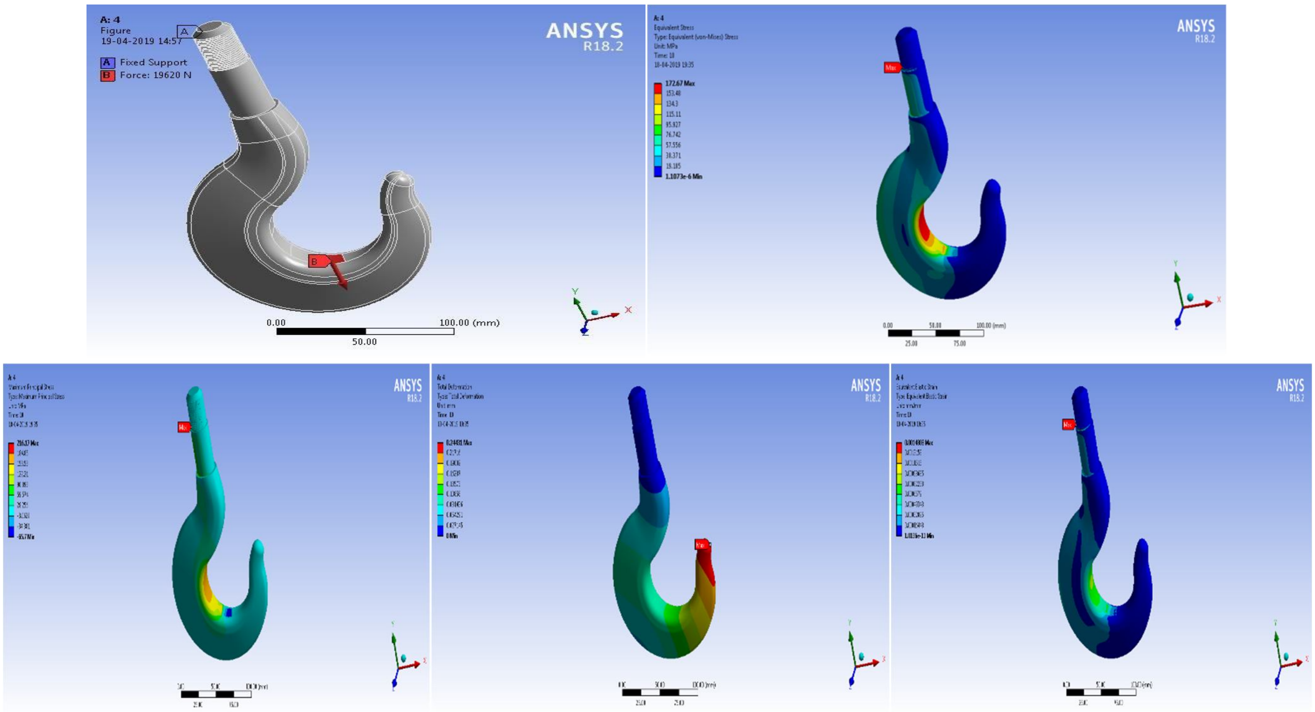




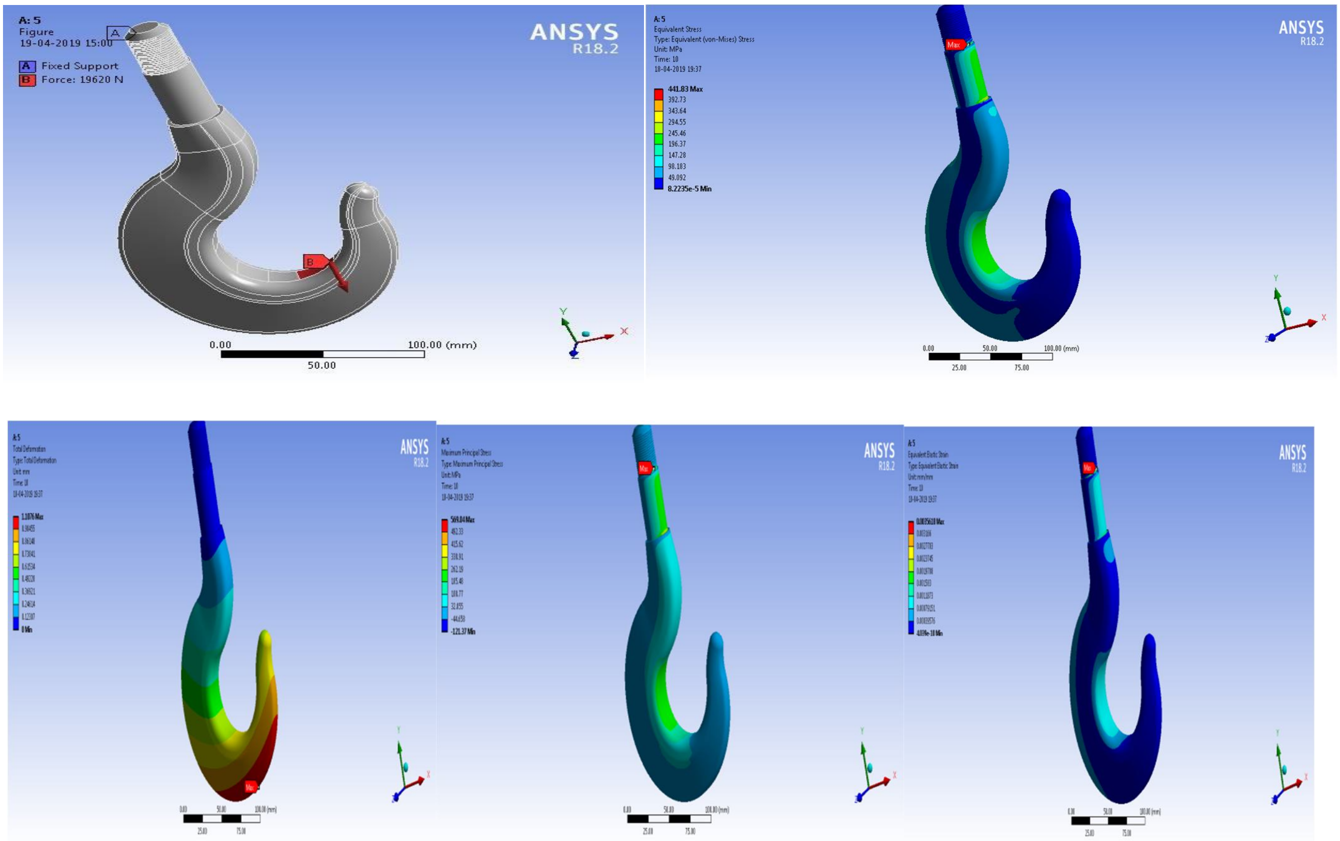


International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.177
Volume 7 Issue IV, Apr 2019- Available at www.ijraset.com

4) For Part 4



5) For Part 5



VII. EXPERIMENTAL RESULTS

	FORCE (N)	MAXIMUM EQUIVALENT (VON-MISES) STRESS (MPa)	MAXUMUM TOTAL DEFORMATION (mm)	MAXIMUM PRINCIPAL STRESS (MPa)	EQUIVALENT ELASTIC STRAIN (mm/mm)
PART 1	19620 N	701.55	1.2697	803.29	0.00569
PART 2	19620 N	711.4	1.2846	839.05	0.00614
PART 3	19620 N	473.26	0.7861	581.83	0.00433
PART 4	**19620 N**	**172.67**	**0.2443**	**216.17**	**0.00148**
PART 5	19620 N	441.83	1.1076	569.04	0.00356

VIII. CONCLUSION

In the present work, the geometry of the crane hook is modelled and finite element analysis (FEA) is applied on the model. The results of static structure analysis are obtained from the FEA software ANSYS. Out of all part from 1 to 5, part 4 is safest for the require load conditions.

From above project the FEA and experimental results are validated. The stresses and deformation are in permissible limit in above design. In future different cross sections are to be used and altered. It can be studied for different type of materials also. Further it is advisable to conduct photo elasticity test for the crane hook under investigation in order to get better insight for stress concentration.

REFERENCES

[1] S. Shanmugam, R. Thridandapani, D. Misra, T. Mannering, D. Panda, S. Jansto, Some Physical Metallurgy Aspects of Niobium- and Vanadium-Micro alloyed steels for Structural Beams, MS&T 2005, Pittsburgh, PA, September 25 –29, 2017

[2] "Design and Stress Analysis 0f Various Cross Section of Hook" G.E.V. Ratnakumar, International Journal of Innovations in Engineering and Technology (IJIET) Volume 4 Issue 4 December 2014.

[3] Amandeep Singh, VinodRohilla, "Optimization and Fatigue Analysis of a Crane Hook Using Finite Element Method", International Journal of Recent advances in Mechanical Engineering, Vol.4, No.4, November 2015. 4403

[4] Omkar P. Bhatkar, Arun V. Javir, Shinde Ajay Ashok, SawantSubodhSumant, UkardeSurajEknath, KavathankarKunalShivram, "Finite Element Analysis of Crane Hook And Optimization Using Taguchi Approach", Journal of The International Association of Advanced Technology and Science, ISSN-3347-4482, Vol. 16 | MARCH 2015

[5] 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.

[6] RashmiUddanwadiker, Department Mechanical Engineering, Visvesvaraya National Institute of Technology, Nagpur, India "Stress Analysis of Crane Hook and Validation by Photo-Elasticity" August 26, 2016

[7] Nutan Joshi1, Dr. K. B. Waghulde, "Finite Element Analysis and Optimization of Crane Hook" Vol. 6, Issue 7, July 2017

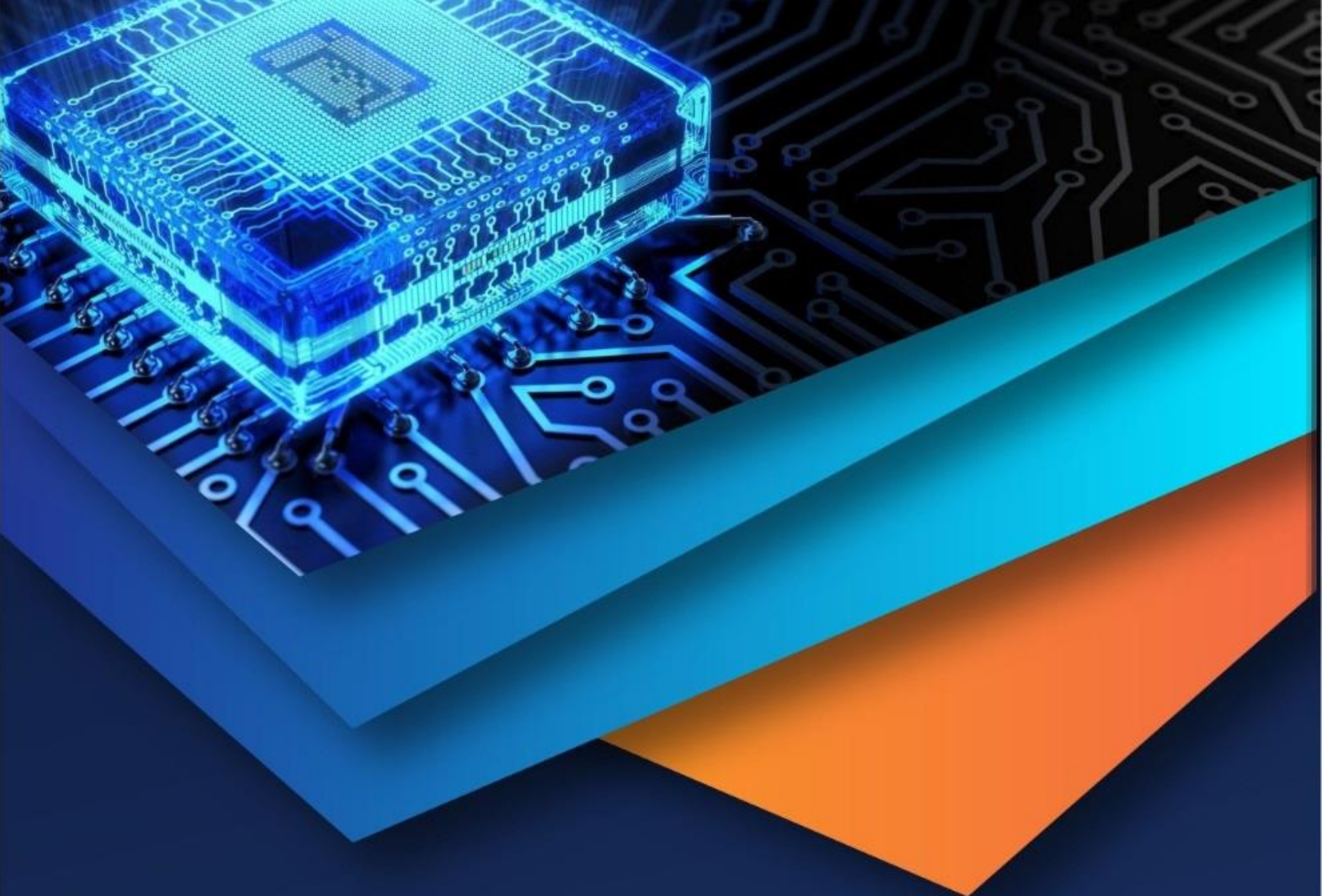
[8] Advanced Mechanics of Materials", by A P Bores i and O M Sidebottam, John Wiley and Sons, 4th Ed. 1985, ISBN: 0-471-84323-7.

[9] DIN 15 401 Part 1, Lifting hooks for lifting appliances; single hooks; Unmachined parts Deutsche Norm, 1982.

[10] "Strength of Materials", by Sadhu Singh, 9th Ed . 2008, ISBN No: 81-7409-048-7.

[11] http://www.steel-tube-steel-pipe.com/Prcoducts/StE-355-Youngs-modulus.html

[12] DIN 15 400, Lifting hooks; materials, mechanical properties, lifting capacity and stresses Deutsche Norm, 1990.



10.22214/IJRASET

INDEX COPERNICUS 45.98

ISRA JIF IMPACT FACTOR: 7.129

TOGETHER WE REACH THE GOAL IMPACT FACTOR: 7.429

ISI

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY