iJRASET
International Journal For Research in
Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 12 **Issue:** XI **Month of publication:** November 2024

DOI: https://doi.org/10.22214/ijraset.2024.65625

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue XI Nov 2024- Available at www.ijraset.com

FEM Analysis of Crane Hooks with Different Cross Section

Valentin Mereuta
Faculty of Engineering, University "Dunarea de Jos", Galati, Romania

Abstract: In this paper the 3D model of the crane hook was realized using Autodesk Inventor Professional 2025 and finite element analysis, was performed using Ansys Workbench, in order to determine the state of stress, deformation and safety factor, by applying restrictions and loads conditions. Two materials were chosen, for crane hook, in accordance with the specialized literature. The materials taken for static analysis was Stainless Steel AISI 1020 and 34CrMo4. The FEM analysis of a crane hook provides critical information for evaluating the design's structural integrity, identifying potential problem areas, and ensuring that the hook can safely carry the loads it is intended to handle.
Keywords: Crane hook, Solid modeling, FEM analysis, Autodesk Inventor, Ansys Workbench

I. INTRODUCTION

Crane hooks are used to move large loads from one location to another especially in the construction industry. It has been observed that a wrong crane load can lead to accidents and additional loads in the critical section of the system [1]. Therefore, it is imperative to carry out a comprehensive investigation of the underlying causes of these errors in order to implement preventive measures in order to avoid accidents. The factors that contribute to the appearance of some unwanted effects, during the use of the crane, are the inadequate design of the components, the improper operation of the crane or its poor maintenance [2]. One of the components that are affected by these factors is the crane hook, its compromised reliability results in unforeseen incidents within the material handling system [3]. Therefore, the design of this device must comply with safety rules so that the crane can be operated safely and maintained easily by a properly trained operator [4]. An evaluation of the stresses in the crane hook is justified to reduce the risk of accidents [5]. Single forged hooks (0.8-160 t), double forged hooks (1-160 t) and lamellar hooks (10-200 t) are used for suspending loads. A crane hook is usually equipped with a safety catch to prevent the cable or chain to which the load is attached from becoming disengaged. The hooks are subjected to repeated stresses, eventually leading to their failure due to fatigue [6]. The tension in the hook must be reduced and this can be achieved by the shape of the hook. The cross section of the hook influences the optimization of shape with weight. The fatigue life of the hook can be extended by reducing the stresses in the hook. The crane hook can have different cross-sections: rectangular; square; round; elongated [6].

In order to reduce crane hook failures, the determination of stresses, their evolution and the areas of maximum values are very important. Because the geometry of the hook is complicated, the analytical calculation is complex and then the use of FEM analysis can be used in the study of the evolution of the stresses in the hook. A change in hook thickness was observed to reduce stresses. The design can be modified by increasing the thickness value on the inner bend so that the chances of failure are considerably reduced. The calculations for the analyzed hook are made by determining the maximum values of the stresses in the dangerous sections, followed by their comparison with the admissible values. Hadiwidodo et al., analyzed a double crane hook using curved beam theory and finite element analysis. The theoretically obtained maximum stress was 73.195 MPa and the Von Mises stress measured by the finite element method was 77.159 MPa. It was found that the error between the results of theoretical and numerical calculations was 4.3% [7].

Desai and Zeytinoglu, using the Solidworks program, aimed to optimize the cross section of the crane hook by comparing three different profiles: square, circular and trapezoidal. The study showed that the trapezoidal cross-section of the hook showed superior performance in terms of peak load compared to circular and square sections [8].

M. Amareswari Reddy et al. [9] studied the load capacity of the crane hook by changing the cross sections. The selected sections were the I section and the T section. He kept the area constant while changing the sizes of two different sections.

Chetan N. Benkar. et al. [10] worked on the optimization of the crane hook by making a solid model and analyzing it using the dedicated FEM analysis software, ANSYS 14. He determined the stress evolution for various cross-section topologies such as rectangular, triangular, trapezoidal and circular, keeping the area constant and found that the area of the rectangular cross-section determines a minimum level of stresses and strains.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue XI Nov 2024- Available at www.ijraset.com

Rashmi Uddanwadiker.et.al. [11] analyzed the differences obtained for the state of stresses and strains in the case of the analytical calculation and the results obtained from the FEM analysis. It was found that the maximum stress is reduced by up to 17% if the thickness of the inner bend is reduced by 3 mm.

Santosh Sahu. et al. [12] made a model of a crane hook with a trapezoidal section using CATIA V5R20 and determined the evolution of the Von Misses stresses after the application of the 2-ton load Apeksha K Patel. et al. [13] worked on beam weight reduction and trapezoidal cross-section hook optimization using ANSYS V12. Torres et al [14] worked on the causes that led to crane hook failure in operation through its thermal analysis

II. CRANE HOOKS DESIGN

Solid modelling of the crane hooks was done using Autodesk Inventor, version 2025, with the literature data, the solid model of the crane hooks is shown in Fig. 1. According to the load apply for the model, the dangerous sections and dimensions are sketched in Fig. 2. In section B-B load applied produces a stress composed of traction, shear and bending, stresses caused by axial force, shear force and bending moment.

Fig. 1 3D model of the crane hook

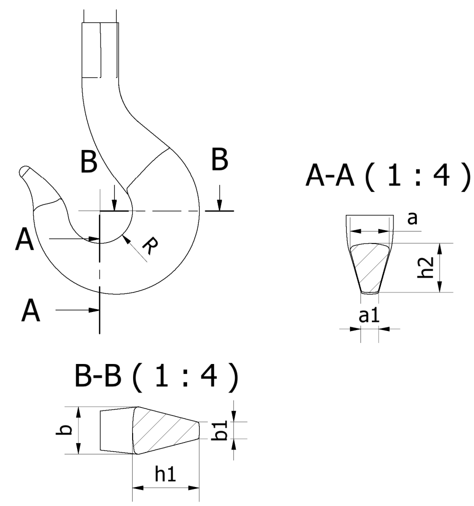

Fig. 2 Dimensions of dangerous sections of the crane hook

TABLE I

DIMENSION OF THE DANGEROUS SECTION

Parameters	*a*	*a1*	*b*	*b1*	*h1*	*h2*	*R*
Dimensions (mm)	36.5	16.5	44	15.5	62	45.5	30

III.STATIC ANALYSIS

In order to carry out this project, the model of crane hook is designed in Autodesk Inventor and then exported to ANSYS in STEP format for the further analysis.

A. Choosing Material

The paper compares the values obtained for two materials used in the construction of the crane hook: Stainless Steel AISI 1020 and 34CrMo4. Materials and their properties are shown in Table 1.

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue XI Nov 2024- Available at www.ijraset.com

TABLE II
MATERIAL PROPERTIES

Parameters	*Stainless Steel AISI 1020*	*34CrMo4*
Young's Modulus [MPa]	205000	210000
Poisson's Ratio	0.29	0.3
Shear Modulus [MPa]	80000	80769
Mass Density [Kg/m^3]	7870	7800
Tensile Strength [MPa]	420	1200
Yield Strength [MPa]	350	900

B. The Restrictions and Load Condition

The hook is fixed at the top end in all the three possible X, Y, and Z direction and the load of 49034 N is applied at the nodes of inner curvature, as show in Fig. 3. The result of stress and deformation obtained for different cross section of crane hook is analysed.

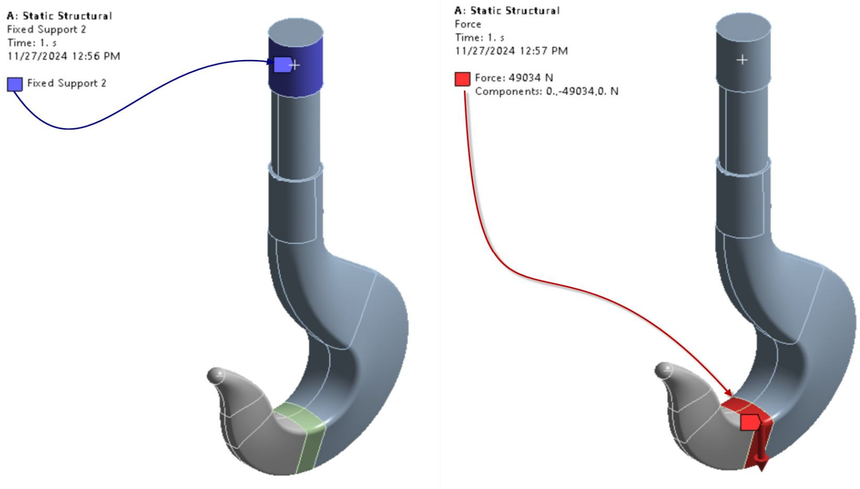

Fig. 3 Restrictions and load condition

C. Generate Meshing

To generate the mesh, the tetrahedrons method was used, element size was 10 mm and the model being meshed into 5322 elements and 9077 nodes, Fig. 4.

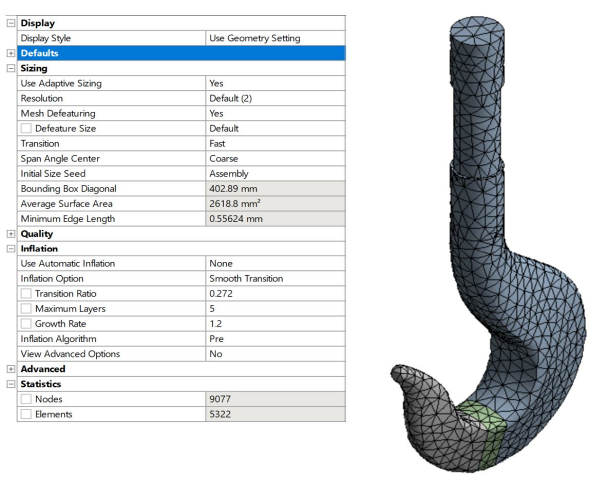

Fig. 4 Crane hook meshing

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.538
Volume 12 Issue XI Nov 2024- Available at www.ijraset.com

D. Static Analysis Results

Next, the simulation was run and the results of the static analysis were obtained, presented in the following figures. Figure 5 shows the Von Mises Stresses values obtained in the case of 34CrMo4 crane hook, it can be seen that their maximum value was 274.28 MPa and Figure 6 shows the Von Mises Stresses values obtained in the case of Stainless Steel AISI 1020 crane hook, maximum value was 273.78 MPa, both below the material's yield point.

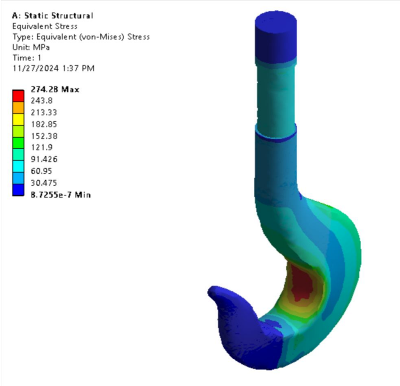

Fig. 5 Von Mises Stress – *34CrMo4*

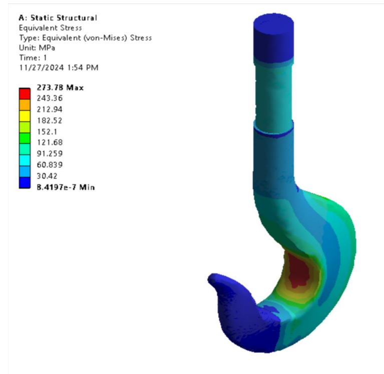

Fig. 6 Von Mises Stress – *Stainless Steel AISI 1020*

Figure 7, 8 shows the results regarding of the maximum deformations of the crane hook. A maximum deformation, 0.0941 mm, was recorded for 34CrMo4 crane hook and a maximum deformation 0.096 mm in the case of Stainless Steel AISI 1020 crane hook.

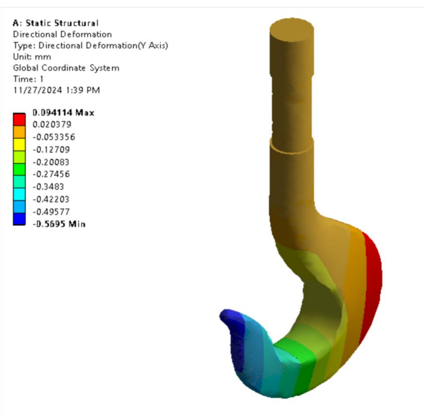

Fig. 7 Deformation – *34CrMo4*

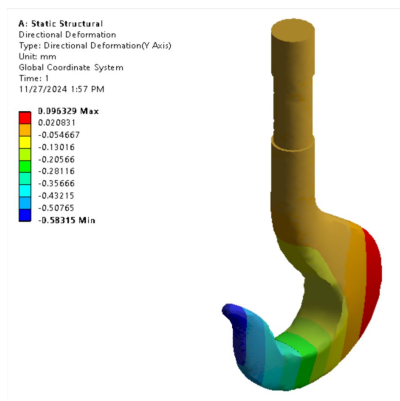

Fig. 8 Deformation – *Stainless Steel AISI 1020*

The minimum value of the safety factor was 3.28, for 34CrMo4 crane hook, Fig. 9 and 1.27 for Stainless Steel AISI 1020 crane hook, Fig. 10.

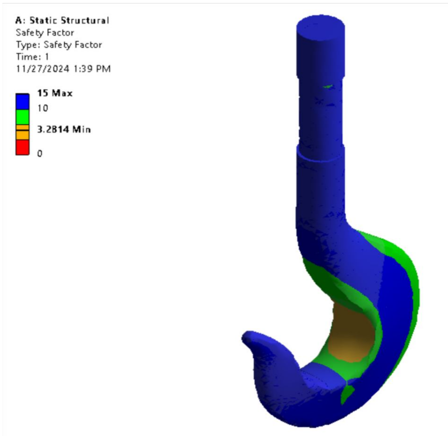

Fig. 9 Safety factor – *34CrMo4*

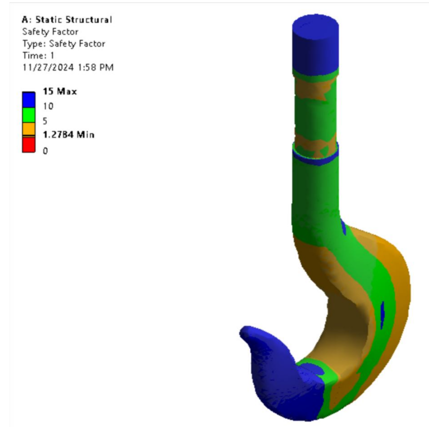

Fig. 10 Safety factor – *Stainless Steel AISI 1020*

IV. CONCLUSION

The static analysis of crane hook made from Stainless Steel AISI 1020 and 34CrMo4 reveals significant insights into their performance and suitability for high-stress environments.

FEM analysis confirms that the chosen materials for the crane hook is appropriate for the expected loading conditions. High-stress areas was found at the points where the load is applied.

Values of maximum deformation was checked to ensure they do not exceed acceptable limits, avoiding excessive bending or twisting that could compromise safety of crane hook.

Safety factor calculations was used to determine if the current design of crane hook has an adequate margin of safety. Safety factor for crane hook was in range from 1.27 to 3.28, depending on material used in crane hook FEM analysis.

Areas with high stress concentrations was highlighted near the hook's attachment zone, suggesting that redesign of crane hook leads to a more uniform distribution of stresses

Recommendations for design improvements might include modifying geometry, increasing the cross-section in high-stress areas, or using higher strength materials.

REFERENCES

[1] K. Kishore, V. Sanjay Gujre, S. Choudhary, A. Sanjay Gujre, M. Vishwakarma, T. Thirumurgan, M. Choudhury, M. Adhikary, A. Kumar, "Failure analysis of a 24 T crane hook using multi-disciplinary approach", Eng. Fail. Anal. 115 104666, https://doi.org/10.1016/j.engfailanal.2020.104666, September 2020.

[2] G. Mukhopadhyay, "Failures of construction equipment and accessories: metallurgical root-cause analysis", Handbook of Materials Failure Analysis, 83–106, https://doi.org/10.1016/b978-0-08-101928-3.00005-7, 2018.

[3] Y.A. Onur, "Investigation of the effect of the sling angle and size on the reliability of lifting hooks", Simulation 94 (10), 931–942, https://doi.org/10.1177/0037549717744646, 11 December 2017.

[4] American Society of Mechanical Engineers, Hooks Safety Standard for Cableways, Cranes, Derricks, Hoists, Hooks, Jacks and Slings, vol. 10, ASME Standard B30, 2009.

[5] R. Uddanwadiker, "Stress analysis of crane hook and validation by photo-elasticity", Engineering 3 (9) 935–941, https://doi.org/10.4236/ eng.2011.39115, 2011.

[6] A. Sahare, K. Gangrade, "Analysis of Deformation and Fatique Life of Crane Hook", International Journal of Research Publication and Reviews, Vol 3, no 12, pp 2256-2259, December 2022.

[7] Y.S. Hadiwidodo, R.G. Permana, Handayanu, "Strength analysis of crane hook structure with trapezoidal cross-section based on rigging configuration on the upper deck lifting process", IOP Conf. Ser. Earth Environ. Sci. 1166 (1) 012033, https://doi.org/10.1088/1755-1315/1166/1/012033 May 2023.

[8] N. Desai, N. Zeytinoglu, "Design and optimization of the geometric properties of a crane hook", World J. Eng. Technol. 4 (3) 391–397, https://doi.org/ 10.4236/wjet.2016.43038, 2016.

[9] 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.

[10] 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.

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

[12] 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.

[13] 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.

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