MATEC Web of Conferences **402**, 01004 (2024) https://doi.org/10.1051/matecconf/202440201004
The 4th ICETIR 2022

Effect of Load Variations on The Static Stress and Fatigue Life Estimation of Crane Hook Using Finite Element Method

Lasinta Ari Nendra Wibawa[1*]

[1]Directorate of Laboratory Management, Research Facilities, and Science and Technology Park, National Research and Innovation Agency (BRIN), Indonesia

Abstract. The crane hook is one of the critical components of a crane that functions to lift objects from one place to another. This equipment is often used for the assembly process of rocket motors. This study aims to examine the effect of load variations on the static stress and fatigue life estimation of Crane Hook. The analysis was carried out using the finite element method with the help of the Ansys Workbench software. Loads of crane hook were varied 5, 6, 7, 8, 9, and 10 Ton. The type of loading is full-reserved, the type of analysis is stress life, and the theory used is the Gerber mean stress theory. The material used is Al 6061-T6. Static stress simulation results show that the crane hook can withstand dynamic loads of up to 10 Tons because it has a safety factor of more than 2, to be exact 2.03. The fatigue life simulation results show that the hook crane fails to reach a fatigue life of 1 million cycles at a load of 10 Tons.

1 Introduction

A crane is a type of heavy equipment extensively employed across various industries such as transportation, construction, shipping, manufacturing, warehousing, and aerospace. Cranes are often used to move heavy materials from one place to another. Several types of cranes include tower cranes, mobile cranes, crawler cranes, hydraulic cranes, and hoist cranes. The selection of the type of crane to be used is generally based on considerations of field conditions, project type, mobility, building structure, and ease of operation both during installation and at the time of dismantling.

Currently, the crane components at the BRIN Garut office still use steel, both the crane structure and the crane hook. Steel components are typically coated with paint to reduce corrosion impact. However, this method proves less effective at BRIN Garut due to its high corrosion rate, being situated near Cilauteureun beach. This situation poses challenges for maintenance, requiring frequent repainting. Moreover, not all equipment receives budget allocations for annual upkeep.

One of the important components of the crane is the crane hook. It is used to lift loads with the aid of tools such as hoists or chains or wire ropes [1]. Crane hooks used in hoists and various types of cranes have a major role in lifting heavy loads in many sectors such as the heavy equipment industry, oil drilling equipment, and automotive. Crane hooks with circular, trapezoidal, rectangular, and triangular cross-section models are the most frequently used.

There are many studies on crane stress analysis, especially crane hooks using Ansys [2–4]. Ansys is a software commonly used for static analysis of machine components using the FEM [5–12]. Therefore, this study uses Ansys to analyze the static stress and fatigue of the crane hook. Fatigue analysis is necessary because the majority of hoist or crane component failures occur due to fatigue, such as crane runway girders [13], crane beams [14], crane drive shaft [15], and wire rope [16,17].

2 Material and Methods

2.1. Material

Material selection is part of the manufacturing process of a product design that aims to find materials that have properties that match the required requirements. Al 6061-T6 was selected as the material for the crane hook. One of the advantages of Al 6061-T6 material is its medium yield strength, light density, and corrosion resistance. Table 1 shows the characteristics of Al 6061-T6.

Table 1. The characteristics of Al 6061-T6.

Properties	Details
Material	Al 6061-T6
Density	2.7 g/cm^3
Yield strength	276 MPa
Tensile strength	310 MPa
Elastic modulus	68.9 GPa
Poisson’s ratio	0.33

* Corresponding author: lasinta.ari.nendra.wibawa@brin.go.id

2.2. Finite element method

One widely adopted numerical method is the finite element method (FEM) [18]. Finite element analysis (FEA) is a numerical mathematical technique used to assess the strength and structural behavior of engineering components. It involves dividing objects into mesh shapes, allowing for detailed analysis of each design and product [19]. FEA is highly favored due to its versatility in modeling various numerical problems, independent of geometry, boundary conditions, and loading [20].

Figure 1 illustrates a crane hook design with slight modifications from previous research (dimensions in mm). Figure 2 details the boundary conditions for the finite element analysis, showing the fixed support on the left and the loading condition on the right for the crane hook.

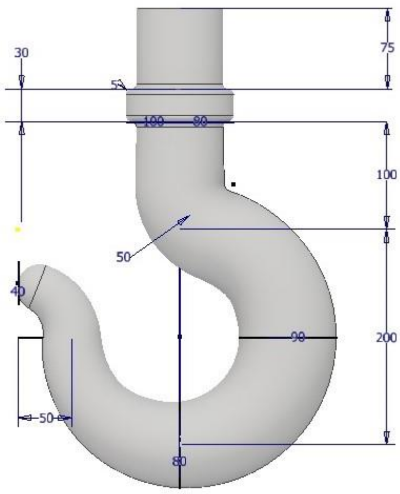
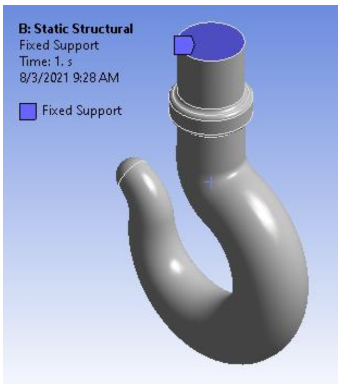
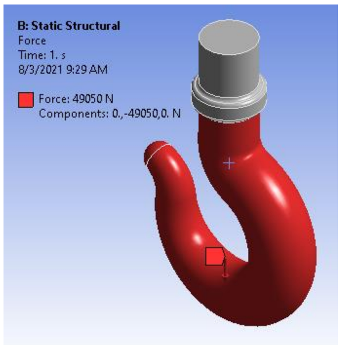


Fig. 1. Crane hook design with slight modification from previous research (in mm) [21].



(a)



(b)

Fig. 2. The fixed support (a) and loading condition (b) of crane hook.

Table 2 shows the finite element analysis assumptions and their parameters using Ansys Workbench.

Table 2. The static stress and fatigue life estimation parameters.

Parameters	**Value**
Crane hook load variation	5, 6, 7, 8, 9, 10 Ton
Size of element	3 mm
Node count	76489
Element count	43890
Factor of safety	Based on the yield strength of Al 6061-T6
Type of loading	Fully-reserved
Type of analysis	Stress life
Mean stress theory	Gerber

3 Results and Discussion

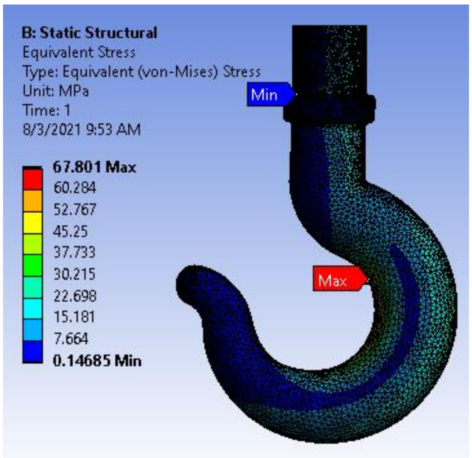
3.1 Static Stress

Static stress analysis is the simplest type of analysis. Static stress analysis with linear material models should only be used in cases where all applied loads are static and all material strains are expected to be within the linear elastic range. The analysis is sufficient to carry out a study of stress, strain, displacement, shear and axial forces resulting from static loading where the load is known and the peak stress time is evident. Static force is assumed to be constant for an indefinite period of time, while the resulting strain, motion, and deformation are small. The condition of the material used will not deform beyond its elastic limit and the dynamic effects resulting from loading are not significant.

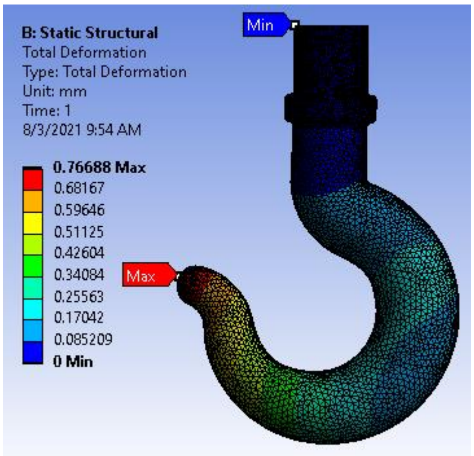
The most common static failure theory uses the von Mises criterion. Von Mises stress is described as the "uniaxial tensile stress that would generate the same distortion energy as the combined applied stresses." The von Mises criterion is a failure theory based on the maximum distortion energy, represented by the von Mises stress, which determines whether a material will fail [22]. Essentially, failure occurs when the von Mises stress exceeds the yield strength of the material.

Crane hook simulation using Ansys Workbench has been carried out with various loadings. The simulation results indicate that at a loading of 5 tons, the maximum von Mises stress is 67.801 MPa (Fig. 3a), the maximum deformation is 0.77 mm (Fig. 3b), and the minimum safety factor is 4.06 (Fig. 3c). The maximum von Mises stress recorded in all variations remains below the yield stress of Aluminum 6061, which is 276 MPa. This indicates that the components are capable of safely withstanding variations in load. The smallest and largest maximum von Mises stresses are 67.80 MPa and 135.60 MPa, respectively (Table 3).

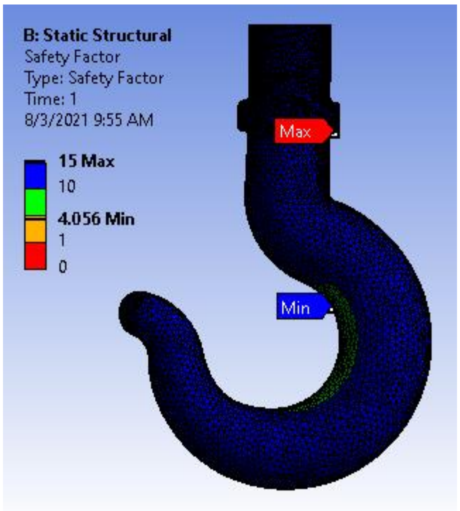
The smallest and largest deformations are 0.77 mm and 1.53 mm (Table 3). That is, this deformation is very small compared to the dimensions of the crane hook component so the component is safe. The simulation results indicate that as the loading increases, the maximum deformation also increases. The smallest and largest safety factors for crane hooks are 2.03 and 4.06 (Table 3). The safety factors are quite safe for crane hook components to withstand static loads.



(a)



(b)



(c)

Fig. 3. The static stress results of crane hook: von Mises stress (a), deformation (b), and factor of safety (c).

Table 3. Summary of the outcomes of the static stress analysis conducted on the crane hook using Ansys Workbench.

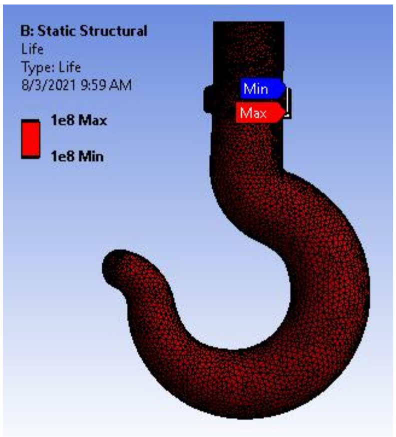
Load (Ton)	Maximum von Mises stress (MPa)	Deformation (mm)	Factor of safety
5	67.80	0.77	4.06
6	81.36	0.92	3.38
7	94.92	1.07	2.90
8	108.48	1.23	2.54
9	122.04	1.38	2.25
10	135.60	1.53	2.03

3.2 Fatigue Life

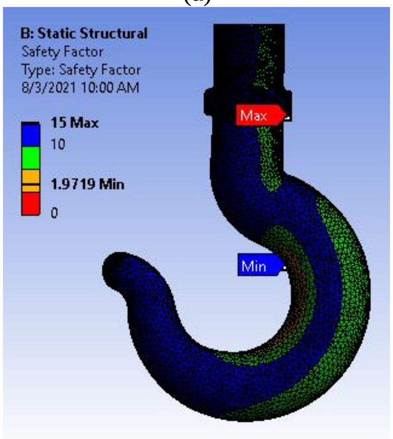
When working, the crane hook often receives high dynamic loads and experiences fatigue failure after components are used for a long time [23]. Therefore, the crane hook must also be analyzed using fatigue life simulation. Fatigue is a type of component failure due to dynamic loads that fluctuate below the yield strength which repeatedly occurs for a long time, while fatigue life is defined as the number of stress and strain cycles that fluctuate from the specific properties that the material will maintain before failure occurs [24].

In general, there are three theories used to predict fatigue life under fluctuating loads, namely the theory of Goodman, Soderberg, and Gerber. In this research, Gerber's theory is used for crane hook fatigue analysis.

Fatigue life simulation and safety factor of crane hook for 5 Tons loading are shown in Fig. 4. The simulation results show that for loading 5 Tons, the fatigue life is 1 x 10^8 cycles with a safety factor of 1.97 (Table 4). The simulation of the fatigue life of the crane hook for 10 Tons loading can reach 9.04 x 10^5 cycles with a safety factor of 0.99 (Table 4). Static stress simulation results show that the crane hook can withstand dynamic loads of up to 10 Tons because it has a safety factor of more than 2, to be exact 2.03. The fatigue life simulation results show that the hook crane fails to reach a fatigue life of 1 million cycles at a load of 10 Tons.



(a)



(b)

Fig. 4. Fatigue life estimation of crane hook: life cycles (a) and factor of safety (b).

Table 4. Fatigue life estimation of crane hook with load variations.

Load (Ton)	Minimum fatigue life (cycles)	Factor of safety
5	1.00×10^8	1.97
6	1.00×10^8	1.64
7	3.02×10^7	1.41
8	6.46×10^6	1.23
9	1.86×10^6	1.10
10	9.04×10^5	0.99

4 Conclusion

Based on the static stress simulation results, the crane hook can withstand dynamic loads of up to 10 tons with a safety factor of 2.03. However, according to the fatigue life simulation results, the crane hook fails to achieve a fatigue life of 1 million cycles at a load of 10 tons.

The author would thank the National Research and Innovation Agency (BRIN) of the Republic of Indonesia, especially the BRIN Garut office, for supporting this research.

References

1. Singh P, Jain O, Yadav P, Nikhil and Sharma S K 2021 Analysing Crane Hook of Different Cross Sections and Different Materials Recent Advances in Mechanical Engineering ed M Muzammil, A Chandra, P K Kankar and H Kumar (Singapore: Springer Singapore) pp 575–80
2. Ji Y, Wang H, Chen H Q, Guo M X and Wu J J 2019 Shape Optimization of Hook for Marine Crane J Phys Conf Ser 1187
3. Ansari O A and Rao P S 2016 Design and analysis of crane hook for load conditions Int J Mech Eng Technol 7 75–9
4. Molnár D, Blatnický M and Dižo J 2021 Design and Strength Analysis of a Crane Hook With a 500 Kg Lifting Capacity Perner's Contacts 16
5. Wibawa L A N and Tuswan 2022 Lightweight Optimization Design of Thin-Walled Cylindrical Rocket Motor Tube Using FEA AIP Conf Proc 2590
6. Godara S S, Brenia V, Soni A K, Singh Shekhawat R and Saxena K K 2022 Design & analysis of connecting rod using ANSYS software Mater Today Proc 56 1896–903
7. Dagar N, Sharma R, Lal Rinawa M, Gupta S, Chaudhary V and Gupta P 2022 Design and analysis of piston using aluminum alloy and composites in Solidworks and Ansys Mater Today Proc 67 784–91
8. Wibawa L A N and Setiadi S 2022 The Effect of Load Variations and Thread Types on the Joint Strength of the Rocket Cap and Tube Using Finite Element Method AIP Conf Proc 2590
9. Alshoaibi A M and Ali Fageehi Y 2022 3D modelling of fatigue crack growth and life predictions using ANSYS Ain Shams Eng J 13 101636
10. Agrawal A P, Ali S and Rathore S 2022 Finite element stress analysis for shape optimization of spur gear using ANSYS Mater Today Proc 64 1147–52
11. Chourasia K and Joladarashi S 2022 Modelling, static and dynamic analysis of offset bearing using ANSYS Mater Today Proc 66 2124–32
12. Wibawa L A N and Tuswan T 2022 Effect of Cylinder Length on The Ratio of Safety Factor and Weight of Rocket Motor Tube Using Thin-Walled Cylinder AIP Conf Proc 2590
13. Rettenmeier P, Roos E, Weihe S and Schuler X 2016 Assessment of mixed mode crack propagation of crane runway girders subjected to cyclic loading Eng Fract Mech 153 11–24

14. Zhao X, Jin N, Liu X and Shi Z 2022 Fatigue failure analysis of steel crane beams with variable-section supports Eng Fail Anal 136 106217
15. Domazet Ž, Lukša F and Bugarin M 2014 Failure of two overhead crane shafts Eng Fail Anal 44 125–35
16. Torkar M and Arzenšek B 2002 Failure of crane wire rope Eng Fail Anal 9 227–33
17. Guerra-Fuentes L, Torres-López M, Hernandez-Rodriguez M A L and Garcia-Sanchez E 2020 Failure analysis of steel wire rope used in overhead crane system Eng Fail Anal 118 104893
18. Wibawa L A N 2021 Effect of Bolt Hole Size on Static Stress and Fatigue Life of UAV Main Landing Gear Using Numerical Simulation J Phys Conf Ser 1811
19. Wibawa L A N, Diharjo K, Raharjo W W and Jihad B H 2020 Stress Analysis of Thick-Walled Cylinder for Rocket Motor Case under Internal Pressure J Adv Res Fluid Mech Therm Sci 70 106–15
20. Ashcroft I A and Mubashar A 2017 Numerical Approach: Finite Element Analysis Handbook of Adhesion Technology ed L F M da Silva, A Öchsner and R D Adams (Cham: Springer International Publishing) pp 1–39
21 Wibawa L A N 2019 Desain dan Analisis Tegangan Crane Hook Model Circular Section Kapasitas 5 Ton Menggunakan Autodesk Inventor 2017 Simetris J Tek Mesin, Elektro dan Ilmu Komput 10 27–32
22 Wibawa L A N, Diharjo K, Raharjo W and Jihad B H 2020 The Effect of Fillet Radius and Length of The Thick-Walled Cylinder on Von Mises Stress and Safety Factor for Rocket Motor Case AIP Conf Proc 2296
23 Kishore K, Sanjay Gujre V, Choudhary S, Sanjay Gujre A, Vishwakarma M, Thirumurgan T, Choudhury M, Adhikary M and Kumar A 2020 Failure analysis of a 24 T crane hook using multi-disciplinary approach Eng Fail Anal 115 104666
24 Wibawa L A N 2021 Effect of Fillet Radius of UAV Main Landing Gear on Static Stress and Fatigue Life using Finite Element Method *J Phys Conf Ser* **1811**