Review Article *Open Access*

Journal of Scientific Engineering Advances

DOI: *doi.org/10.63721/26JSEA0120*

Design and Failure Stress Analysis of Crane Hook by Using Mesh Sensitivity Analysis Approach on Finite Element Method

Zablon Zeleke[1*], Tesfaye Tamiru[1], Belachew Ashuro[1] and Dilnesaw Bogale[2]

[1]Department of Mechanical Engineering, Institute of Technology, Wolaita Sodo University, Sodo, Ethiopia
[2]Faculty of Mechanical Engineering, Arbaminch Institute of Technology, Arbaminch University, Arbaminch, Ethiopia

Citation: Zablon Zeleke, Tesfaye Tamiru, Belachew Ashuro, Dilnesaw Bogale (2026) Design and Failure Stress Analysis of Crane Hook by Using Mesh Sensitivity Analysis Approach on Finite Element Method. J. of Sci Eng Advances 2(1) 1-9. WMJ/JSEA-120

Abstract

In realistic applications, crane hooks are used mostly under high loading conditions and subject-ed to high-stress concentrations which cause failure of this material handling equipment. Failure mainly occurs due to high-stress concentration on the hook at a specific point. Determining these maximum stress concentrations that cause failure and the point where they occur is vital in re-ducing the failure of hooks. Therefore, FEM analyses are useful for obtaining an understanding of stress concentration on hooks that lead to failure. In this study, both analytical and numerical analyses were performed on the hook which is subjected to a vertical load of 50ton (490.5kN) for specifically selected hook material. The maximum stress resulting from analytical calculations is 1470.1MPa, which is safe as compared to the yield strength of hook material. The maximum Von Mises stress, maximum principal strain, and total deformation of hook were determined by nu-merical method. To produce more reliable results numerically on ANSYS Workbench 19.2 mesh sensitivity analysis (MSA) was performed. The meshing size dependence of maximum stress was studied by refinement of meshes. From the convergence study, the maximum failure stress de-termined is 1412.4MPa, which is also safe as compared to the yield strength of the material. Even if there are some variations in the results of analytical and numerical approaches, both re-sults are safe for the loading condition of the crane hook.and governance enhancement.

***Corresponding author:** Zablon Zeleke, Department of Mechanical Engineering, Institute of Technology, Wolaita Sodo University, Sodo, Ethiopia.

Submitted: 24.01.2025 **Accepted:** 28.01.2026 **Published:** 25.02.2026

Keywords: Crane Hook, Convergence Study, Failure, Meshing Size, Stress Concentrations

Introduction

Material handling involves activities of elevating, transporting, conveying, loading and unload-ing, packaging, and sorting materials in industries, construction sites, and other sectors of day-to-day life of humans. This makes material handling both art and science. Material handling is high-ly needed in that it reduces the cost of manual handling and protects the safety of products to in-crease production; therefore, the type and extent of using material handling should be designed carefully. The proper functioning of material handling equipment plays a very important role in the economy and efficient performance of production lines. Inefficient material handling practic-es and using non-improved material handling equipment may cost the enterprise up to going out of business [1,2].

A crane hook is a load-handling attachment used in lifting and lowering loads. The load is usual-ly handled using chain or rope slings attached to hooks. Hooks are used in different sectors like shipbuilding industries, construction and mining industries, logistics, and transport vehicles for lifting, lowering, and carrying loads [3]. During the lifting and lowering of loads, hooks are sub-jected to strain, stress concentration, and deformation. This may cause failure of hooks during operation [4,5]. The failure of the crane hook may cause damage to human life and property, re-duce productivity, result in difficulty of operations, and so on [6,5]. Among many causes of fail-ure of crane hooks; improper design is the most important one. In designing crane hooks, hook geometry, stress, and strain concentrations, and deformation analysis should be performed care-fully. The point of stress and strain concentrations should be determined [7,3].

The cross-section geometry of the crane hook can affect the stress and strain distribution on the hook. This stress and strain concentration may cause the failure of the hook if it is not properly managed and determined. There are four main cross-sections used for crane hooks; rectangular, trapezoidal, triangular, and circular [8,9]. Hooks with trapezoidal cross sections are subjected to low stress concentration and low failure [10-13]. Forged standard hooks are used in vast applica-tions of material handling. These hooks are forged from low-carbon steel and used to lift up to 100 tons tons [2]. Even if single crane hooks are simple to produce and use, their load carrying is not good. Therefore, they are only used most of the time at low-capacity working sites with a load capacity of less than 80 tons [14].

In this paper, a standard hook with trapezoidal geometry was designed analytically and analysis of stress and strain distribution and deformation due to applied load was performed by using ANSYS workbench to determine the point where failure will occur. The FEM analysis on ANSYS is implemented by using Mesh Sensitivity Analysis (MSA) approach to determine the maximum stress, strain, and deformation that causes the failure of the hook. The MSA helps to produce more reliable results and improve effective decision-making in determining the root cause of crane hook failure.

Methods

Trapezoidal Hook Geometry and Analytical Calculations

For trapezoidal cross-section standard forged hooks, dimensions can be determined by using the load (Q) on the hook and safe unit compression pressure (Pall). Figure shows a standard forged trapezoidal cross-section hook [2]. All the numerical values of these dimensions for software modeling are taken from the standard hook catalog [15].

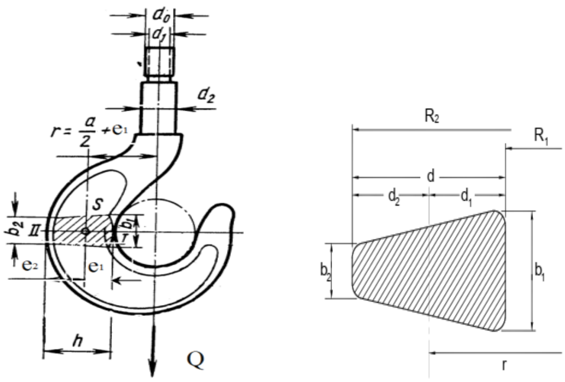


Figure 1: Crane Hook Geometry.

The tensile stress on the shank of the hook can be determined by:

$$\sigma_t = \frac{4Q}{\pi d_2^{\,2}} \le \sigma_{al} \quad (1)$$

The minimum height (H) of the hook nut is determined from permissible pressure in the thread and applied load on the hook and given by:

$$H = \frac{4Qt}{\pi(d_0^{\,2} - d_0^{\,2})P_{all}} \quad (2)$$

The maximum permissible pressure in the thread (Pall) for steel material is 29.419 to 34.323 MPa. The thread of the crane hook nut depends on the lifting capacity in that the lifting capacity is less than 5 tons provided with V-type threads and that of greater than 5 tons are provided with trape-zoidal or buttress threads [2,9].

The unit stress on the saddle of the hook can be determined as:

$$\sigma = \frac{Q}{A} + \frac{M}{Ar} + \frac{M}{Ar}\cdot\frac{1}{X}\cdot\frac{Y}{Y+r} \quad (3)$$

Where: is unit stress for fiber at distance y from the neutral axis [kgf/cm2], Q is the load on the hook [kgf], A is an area of critical cross-section [cm2], r is the radius of curvature of the neu-tral axis at the critical cross-section [cm], X is factor depending on shape of cross-section (trape-zoidal in this case) and the curvature of the beam, Y is distance from fiber to the neutral axis. It is negative if the fiber is between the center of curvature and the neutral axis and positive if the fiber is on the neutral axis, M is the bending moment [kgf. cm]. It is positive if it causes the hook curvature to increase and negative if the curvature decreases. i.e.

$$M = \pm Q(0.5a + e_1)$$

For trapezoidal cross-section, the factor depending on the shape of the cross-section (X) can be determined as:

$$X = -\frac{1}{A}\int \frac{Y}{Y+r}dA \quad (4)$$

$$X = -1 + \frac{r}{A}\left[\left\{b_2 + (b_1 - b_2)R_2/d\right\}\ln\frac{R_2}{R_1} - (b_1 - b_2)\right] (5)$$

Therefore, the maximum tensile stress in inner fiber can be determined for: $M = -Q(0.5a + e_1)$, at radius of

$R = 0.5a + e_1$ $Y = e_1$ and $h = a$ from Eq. (3):

$$\sigma_t = \frac{Q}{A}\cdot\frac{1}{X}\cdot\frac{2e_1}{a} \quad (6)$$

To prevent failure of the hook, the maximum tensile and compressive stresses in the above cal-culations should not exceed 1400.1MPa. This value of stress is determined analytically for 50ton (490.5kN) by substituting all the unknowns from the standard forged single crane hook catalog. The stress is safe for selected material for the hook with yield strength of 1590MPa.

Hook Material Selection

Forged standard crane hooks are made up of low-carbon steel with maximum lifting loads of up to 100 ton. The material selection of the hook is affected by factors like lifting load, machining process, geometry of hook (cross-section of hook), and working environments. In this hook analysis for maximum load lifting capacity, selecting material with high mechanical properties like strength is needed. This is due to knowing safe material for lifting maximum load is im-portant in failure reduction. Table 1 Shows the mechanical properties of the selected material for crane hook analysis [16].

Table 1: Material Properties of Crane Hook.

	Mechanical properties		
AISI No	Tensile strength (MPa)	Yield strength (MPa)	Elongation (%)
4340	1720	1590	10

This a low alloy steel containing nominally 0.4% C, 0.8% Cr, 0.25% Mo and 1.8% Ni composi-tion. It has higher strength and toughness, very good fatigue and wear resistance, and atmos-pheric corrosion resistance

Modeling Crane Hook

The hook with a trapezoidal cross section is modeled by using SOLID WORKS software with standard dimensions [14,17]. The feature of the hook has no constant cross-section, thus modeling of the hook; and trapezoidal cross-section should be performed carefully Figure. 2 shows a standard single-piece crane hook model on SOLID WORKS with a trapezoidal cross-section.

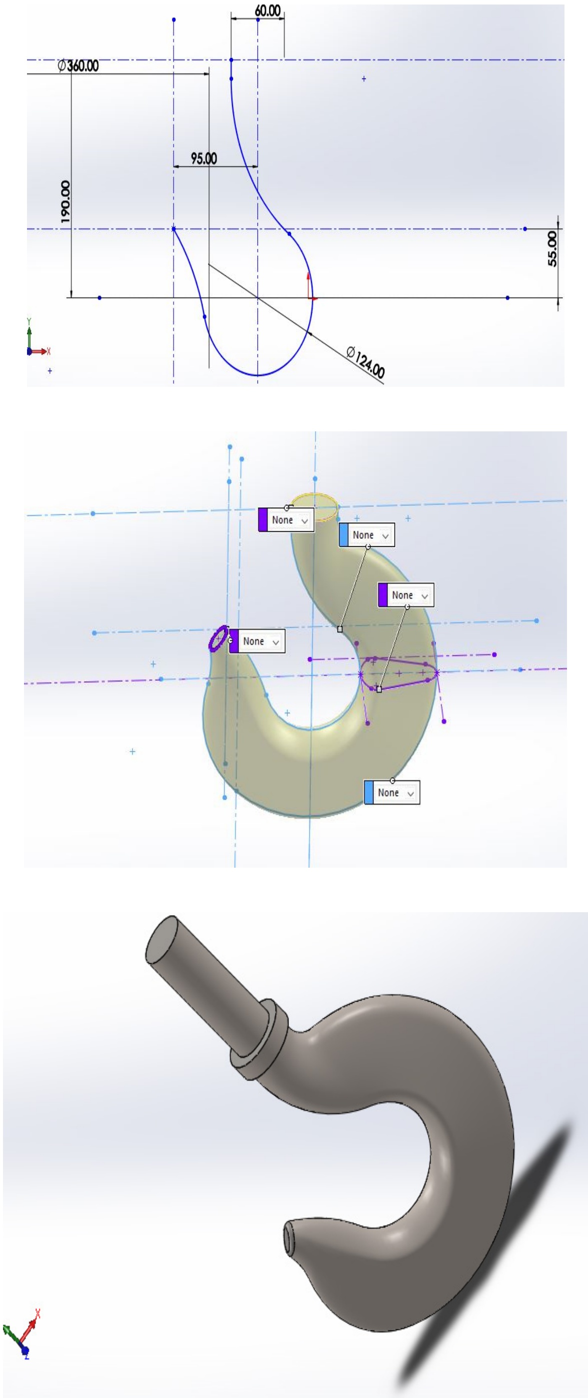


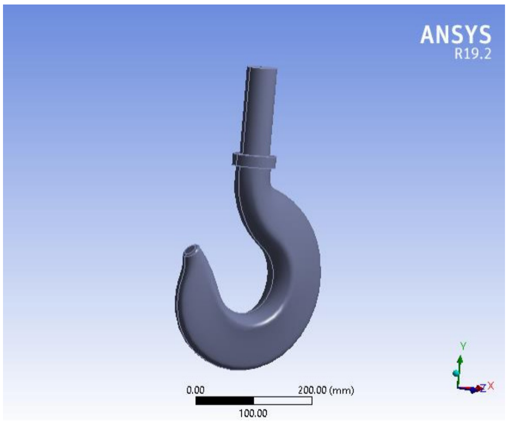
Figure 2: Standard Trapezoidal Cross-Section Crane Hook Model.

FEM analysis

The modeled crane hook is analyzed for maximum stress, strain, and deformation by using FEA on ANSYS 19.2 software. Analyzing maximum stress distribution and determining their location is vital to reduce the failure of crane hooks. The ANSYS 19.2 workbench analysis for the modeled crane hook starts with imputing the required engineering properties of the selected material in Table 1 and the modeled geometry of the crane hook. The geometry of the hook imputed can be meshed, the correct boundary condition and load (force) is applied and finally, the analysis for required parameters can be performed.

In analyzing maximum parameters, MSA is performed to produce more reliable results and im-prove effective decision-making in determining the cause of crane hook failure. Even if single crane hooks are simple to produce and use, their load carrying is not good. Therefore, they are only used most of the time at low-capacity working sites with a load capacity of less than 80 tons [14].

The static analysis has been done with a load of 50 tons (490.5kN), which is the maximum load that the single-piece forged standard crane hook can lift for this specific analysis. The analysis of static parameters; stress, strain, and deformation of the crane hook for this load is performed by using selected material via ANSYS 19.2. In applying boundary conditions, all the displacement and rotation degrees of freedom of the top surface above the neck of the hook have been fixed by fixed-type support. The downward load of 490.5kN is applied in the plane to the hook. Figure. 3 shows the imported geometry, meshing, boundary condition, and force application of the crane hook on the ANSYS 19.2 workbench. In performing mesh sensitivity, four mesh sizes (default, 10mm, 5mm, and 2.5mm) were used to produce more reliable stress results for constant load. Figure 3 shows the mesh sizing of the crane hook for default mesh sizes, 10mm, 5mm, and 2.5mm mesh sizes respectively.



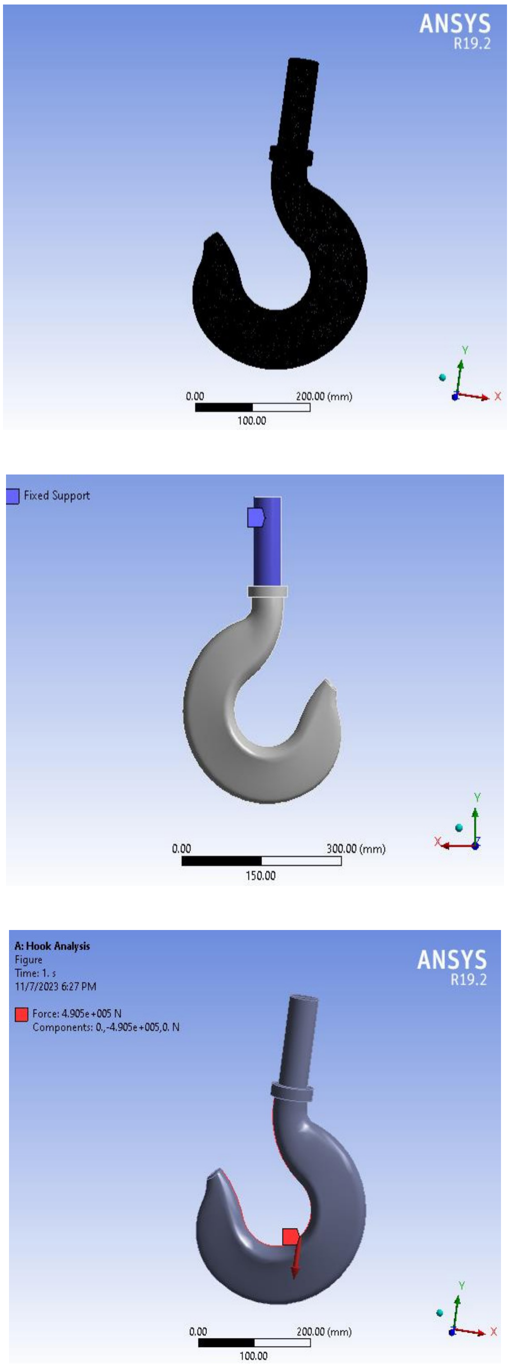


Figure 3: Geometry, Meshing, Boundary Condition, and Load Application.

In performing mesh sensitivity, four mesh sizes (default, 10mm, 5mm, and 2.5mm) were used to produce more reliable stress results for constant load. Figure. 4 shows the mesh sizing of the crane hook for default mesh sizes, 10mm, 5mm, and 2.5mm mesh sizes respectively.

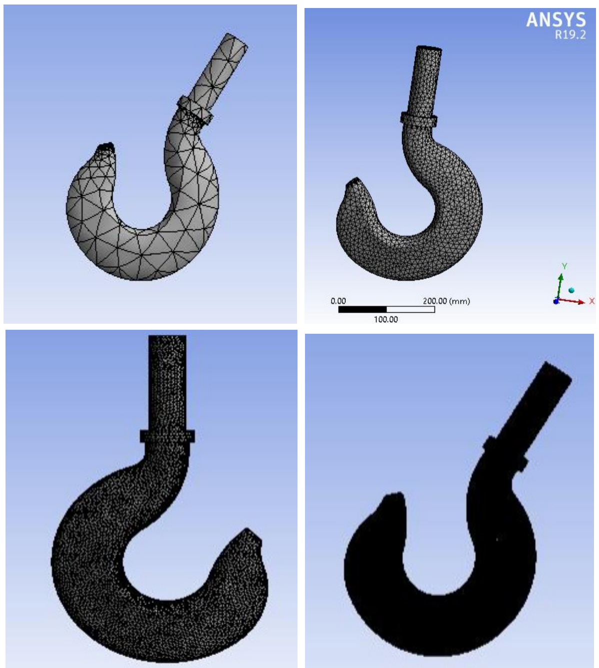


Figure 4: Mesh Refinement.

In mesh sensitivity analysis, mesh convergence and refinement are performed with respective stress plots. This is used to produce the result that is more reliable and in turn, it helps to know the maximum stress result that causes failure to the hook. The meshing size is reduced until the change in stress value is negligible and the point of stress concentration on the hook is identi-fied. By reducing the meshing size as shown in Figure. 4 all the results were recorded. The change in results of stress is negligible after a meshing size of 2.5mm.

Results and Discussion

The failure stress due to the applied load on the crane hook was analyzed by using ANSYS workbench 19.2. To produce reliable results, mesh sensitivity analysis (MSA) was performed until the result converged.

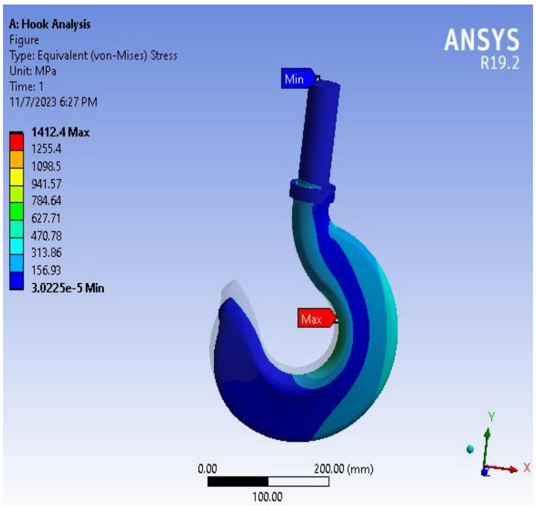
Table 2 shows the effect of meshing sizes on analysis results of stress, strain, and total defor-mation of the hook. As recognized from the results, for MSA of different meshing sizes, there is negligible change in results after 2.5mm meshing size for constant load and similar hook mate-rial. Therefore, the maximum stress, strain, and deformation on the hook assertively determined from this analysis for applied load are 1412.4MPa, 0.0069747mm/mm, and 4.0808mm respec-tively. For these values of stress, strain, and deformation, we have compared them with the ma-terial properties of the hook in Table 1.

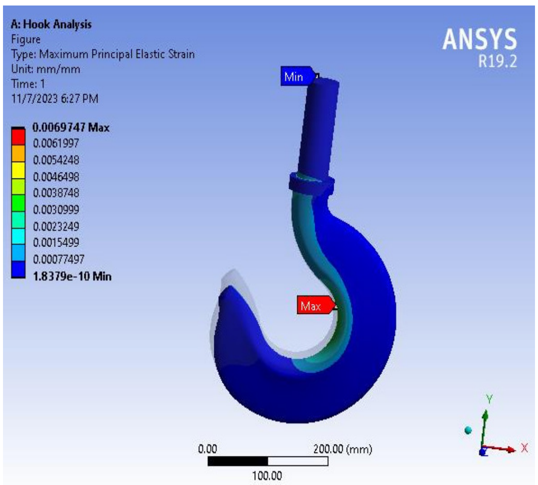
Table 2: Effect of Meshing Size on Analysis Results

Meshing Size	Results			
	Von Mises Stress(MPa)	Max. Strain(mm/mm)	Total Deformation(mm)	Converged
Default	786.92	0.0040236	4.1577	Start
10mm	844.17	0.0042604	4.0515	Not
5mm	913.19	0.0045557	4.0484	Not
2.5mm	1412.4	0.0069747	4.0808	Not
1mm	1412.4	0.0069747	4.0808	Yes

Figure. 5 shows the maximum stress, strain, and total deformation analyzed for the loading condition on the crane hook. To produce reliable results, the MSA was performed with mesh convergence and refinement. The maximum value of stress and strain after MSA shows that the crane hook is safe for the loading condition as compared to the material property in Table 1. The point where maximum stress can occur is also determined and it is near the inner neck of the hook. This shows that the hook may fail at that point if the load greater than the current one is applied. That is the fracture of the hook can occur at a point for stress value greater than 1412.4MPa. The fracture can occur for greater than determined result of stress because, the stress value determined by the analysis is less than the yielding strength of the material selected for hook design.

That is why because the design of this hook is safe. The deformation is maximum to the curved end of the crane hook due to the applied load condition.





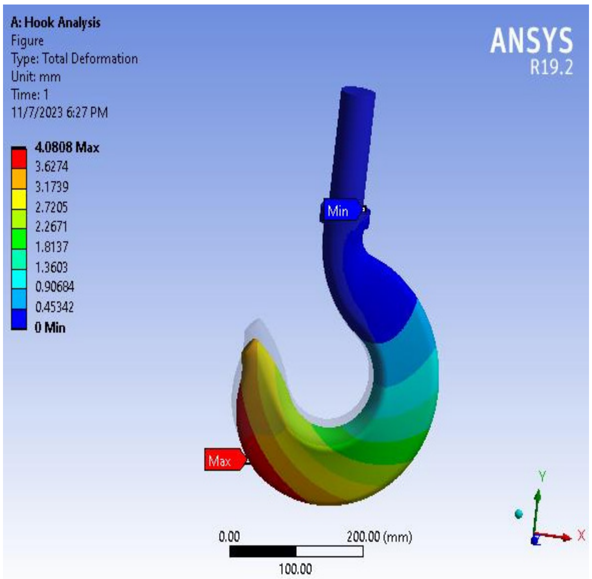


Figure 5: The Maximum Stress, Strain Distribution, And Deformation of the Hook.

Figure 6 illustrates the generated convergence history of maximum stress for various meshing sizes from ANSYS workbench 19.2. The solution number shows that the number of mesh re-finement by changing meshing sizes for default, 10mm, 5mm, 2.5mm and 1mm. The analysis started from the default mesh size until the change in the result of stress was negligible. The convergence graph shows that the maximum stress value for default meshing size is 786.92MPa and the converged final result at 2.5mm meshing size is 1412.4MPa. To produce this, 5% of the allowable change in the result of stress was used and resulted in a high convergent result with -0.1004% change. It was clearly observed, there is a highest change in stress result from meshing size 5mm to 2.5mm due to the refinement of the elements from 9361 to 33918 and nodes from 14844 to 5176 respectively. This is about 63.596% change of the result. Therefore, the maxi-mum converged failure stress value due to the applied load 50ton (490.5kN) is 1412.4MPa. Alt-hough it is a failure stress on the hook, the design of this hook is safe for selected material and applied load. For constant loading and similar hook material, it was considered that there is a great variation in the results of stress that cause failure on the hook. To know the maximum value of stress that cause failure, performing numerical analysis only one or two times without performing the mesh sensitivity is not enough. Therefore, MSA highlights the maximum stress that cause failure on the hook for similar loading condition that can be determined by perform-ing many mesh refinements to converge the results of stress. This is important to reduce the wrong conclusion made on the value of maximum stress result that causes failure by showing correct maximum value of the stress that may occur due to the applied load.

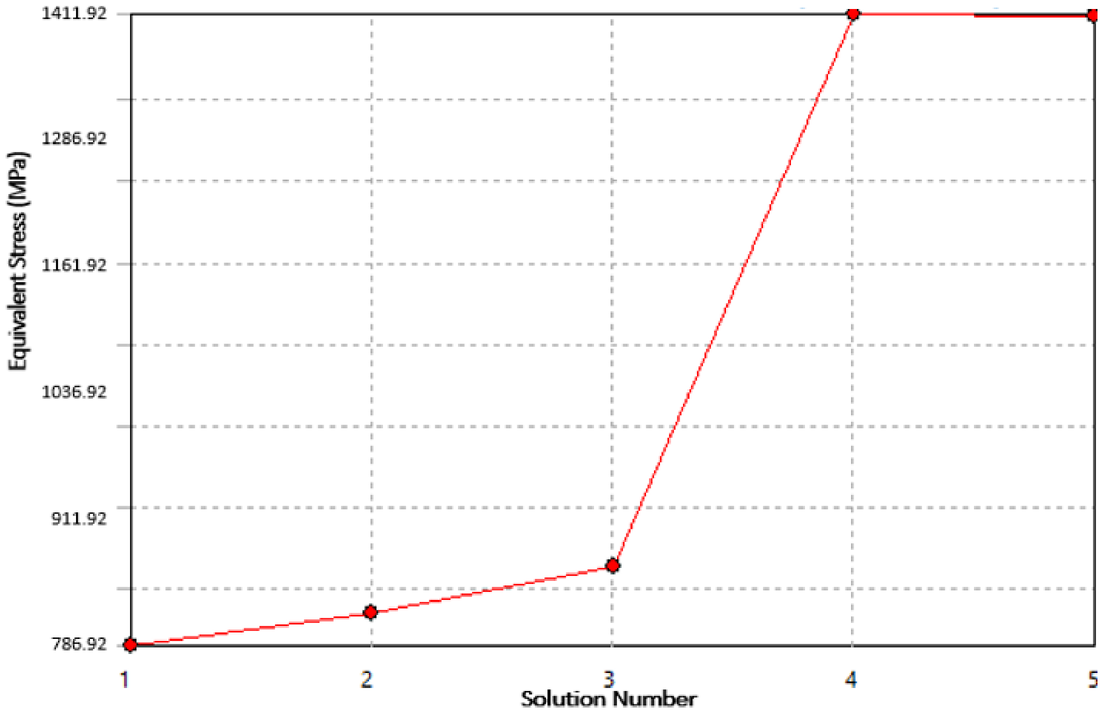


Figure 6: Stress Convergence History Plot.

Conclusions

In an analysis of the failure stress of a crane hook, the maximum stress that may cause failure is determined by both analytical and numerical methods. Using standard dimensions from the crane hook catalog and the permissible load hook can lift, the maximum stress was determined analytically and compared for the safety of the design with the material property. The maxi-mum stress, strain, and deformation of the crane hook were also determined by the numerical method of ANSYS workbench 19.2 by using the mesh sensitivity analysis approach to produce more reliable results. MSA in ANSYS workbench is the best approach to produce reliable re-sults by performing mesh convergence and refinement. The converged result was compared to the material property of the hook and showed the safety of the design. By doing this, MSA ap-proach is noticed as the best way to determine the maximum failure stress and reduces wrong conclusion made on the results of stress. Both analytical and numerical results are safe for ap-plied load with some variations of results within each other.

References

1. S Ray (2008) Introduction to Materials Handling. New Delhi: New Age International https://seatracker.ru/viewtopic.php?t=40318.
2. D Kitaw (2001) Materials Handling Equipment, 1st ed. Addis Ababa,Ethiopia: Addis Aba-ba University.
3. G Bhagyaraj, K Suryaprakash, K S Rao (2017) Crane Hook Design and Analysis. Int. Res. J. Eng. Technol 4: 61-65.
4. R Uddanwadiker (2011) Stress Analysis of Crane Hook and Validation by Photo-Elasticity. Engineering 03: 935-94.
5. Y Torres, JM Gallardo, J Domínguez, F J Jiménez E (2008) Brittle fracture of a crane hook. Eng Fail Anal 17: 38-47.
6. K Kishore (2020) Failure analysis of a 24 T crane hook using multi-disciplinary approach. Eng Fail Anal 115: 104666.
7. I T Teke, M Akbulut, A H Ertas (2021) Topology optimization and fatigue analysis of a lifting hook. Procedia Struct. Integr 33: 75-83.

8. S Sahu, R Dewangan, M Patnaik, N Yadav (2012) Study of Crane Hook Having Trapezoi-dal Section by Finite Element Method & Design of Experiments. Int J Mod Eng Res 2: 2779-2781.
9. S Nageswara, T Hariprasad, P Mallikarjunarao (2014) Structural Analysis of Crane Hook Using the Finite Element Analysis (FEA) Approach. Int J Sci Mod Eng 03-310.
10. S Ghosh, B Pati, R Ghosh, A Palo, R N Barman (2017) Static analysis of crane hooks with different cross sections - A comparative study using Ansys Workbench 16.2. Int J Mech Eng Technol 8: 474-482.
11. Y B Chunkawan, R S Subramaniyam (2017) Static Structural Analysis Of Crane Hook. Int Res J Eng Technol 4: 2265-2274.
12. M M R Zade (2017) Finite Element Analysis and Fatigue Analysis of Crane Hook With Different Materials. Int Res J Eng Technol 4.
13. S K Lakshmana Moorthy, B Prakash (2020) Design and Analysis of Crane Hooks of Dif-ferent Cross Sections Made of Hardened-Tempered Alloy Steel AISI 6150 and AISI 4140. Int J Eng Res 9: 435-439.
14. DGCRANE Crane Hook catalog Xinxiang.
15. Hebezeuge, Lasthaken (1978) Hook Catelog Din.
16. R G Budynas, J K Nisbett (2011) Shigley's Mechanical Engineering Design, Ninth Edit.
17. S S B Roy (2010) Indian Standard, point hooks with shanksup to 160 tonne- specification. 0.