

Analysis of the Effect of Thermal Condition Parameters on the Strength of the Coil Lifting C-Hook Using the Finite Element Method (Case Study at PT. X)

Raja Shadam Ali Imron*, Sigit Pradana, Damora Rhakasywi
Universitas Pembangunan Nasional Veteran Jakarta, Indonesia
Email: 2110311049@mahasiswa.upnvj.ac.id*

ABSTRACT

In the steel industry, Coil Lifting C-Hooks are critical tools used to lift high-temperature wire rods after the rolling process. The main challenge with this equipment involves repeated mechanical loading and exposure to elevated temperatures, both of which significantly affect structural integrity and operational safety. This research aims to analyze the stress, deformation, fatigue life, and critical failure areas of a C-Hook under thermal-mechanical loading. This study employs the Finite Element Method (FEM) simulation in ANSYS to systematically analyze the impact of repeated loading and high-temperature conditions on structural strength parameters, specifically examining thermal conditions at 170°C and 200°C. The simulation methodology integrates steady-state thermal analysis with static structural analysis to comprehensively evaluate the C-Hook's performance under realistic operating conditions. The results show that repeated loading and increased temperature cause increased deformation (maximum 2.4358 mm at 170°C and maximum 2.425 mm at 200°C), von Mises stress (maximum 140.86 MPa at 170°C and maximum 158.53 MPa at 200°C), and reduced fatigue life (minimum 2,286,900 cycles at 170°C and minimum 1,394,500 cycles at 200°C), as well as a reduced safety factor (minimum 1.0954 at 170°C and minimum 0.973 at 200°C), particularly in the C-Hook curved area. This underscores the importance of C-Hook design that accounts for thermal effects. This research contributes to the existing body of knowledge by combining thermal and structural analysis specific to S275N material under realistic steel industry conditions, offering practical insights for preventing premature equipment failure and enhancing workplace safety in high-temperature lifting operations.

KEYWORDS

C – Hook; wire rod; thermal condition; Finite Element Method (FEM).



This work is licensed under a Creative Commons Attribution-ShareAlike 4.0 International

INTRODUCTION

Lifting is a structured work technique for moving loads using tools such as cranes (Chovnyuk, Diachenko, Ivanov, Dichek, & Orel, 2022). This equipment ranges from heavy equipment to small accessories such as ropes, hooks, and chains (Mukhopadhyay, 2018). In the steel industry, one important lifting tool is the Coil Lifting C-Hook, which is used to lift high-temperature steel wire rods after the production process with billets as the main raw material heated to 1050-1100°C. This process is called the reheating furnace (RHF) process (Schmitz et al., 2021).

The Coil Lifting C-Hook is made of high-strength steel and is capable of operating in extreme conditions (Mahanthesh, Girisha, Shreyas Babu, & Shivananda, 2020). This tool is vital in the production process of PT X, which manufactures wire rods. During operation, the C-Hook experiences repeated mechanical loads and high temperatures due to the weight of the wire rod lifted by the C-Hook and direct contact with hot wire rods after the rolling process, which risks material fatigue and performance degradation.

Sudden failure of this component can hamper the entire production process because each process is interconnected. One such failure that has occurred in the field in relation to the C-Hook is cracking, as can be seen in Figure 1 and others. Optimization can be done through calculation or simulation. The input that needs to be known in advance is the parameter causing

the failure, one of which is the distribution of repeated stress, in order to predict stress behavior that meets classification standards (Boland, Li, Salem, Boddeti, & Lajnef, 2022). Therefore, strength and damage analysis is important (Saad-Eldeen, Garbatov, & Soares, 2016). Using the finite element method, parameters such as deformation, stress, service life, and thermal effects can be analyzed to evaluate and optimize the performance of these components.



Figure 1. One Type of Failure: Crack on the Lifting Coil C – Hook
Source: Documentation from PT. X

The occurrence of such failures carries significant operational and safety implications. Sudden, unexpected failure of a C-Hook during lifting operations can result in dropped loads, creating immediate hazards to personnel working in the vicinity and causing substantial equipment damage. Furthermore, even minor structural degradation can necessitate unplanned production shutdowns for equipment inspection and replacement, directly impacting manufacturing efficiency and economic performance (Chukwunweike, Anang, Adeniran, & Dike, 2024). Each process in steel production is interconnected, meaning that disruption in one stage cascades through the entire production line, potentially resulting in considerable financial losses.

Despite the critical importance of C-Hook performance in steel manufacturing, a significant research gap exists in the literature regarding comprehensive thermal-structural analysis of these specific components under realistic operational conditions. While several studies have examined related aspects—including general fatigue behavior of structural steels, stress analysis of crane hooks, and structural evaluation of lifting devices—none have comprehensively integrated thermal and mechanical analysis specifically for C-Hooks operating in the temperature range and loading conditions typical of wire rod handling operations.

Previous research by Aldeeb & Abduelmula (2018) investigated the fatigue characteristics of S275 steel through numerical simulation to establish the material's fatigue limit. Their methodology employed FEM with cyclic loading protocols, yielding a fatigue limit of approximately 195.47 MPa for S275 steel. However, this investigation focused exclusively on the general fatigue properties of the material under ambient conditions, without consideration of elevated temperature effects or application to specific industrial components. The absence of thermal considerations represents a significant limitation, as material properties—including yield strength, elastic modulus, and fatigue resistance—are known to vary substantially with temperature (Bazli & Abolfazli, 2020).

Subsequently, Wunda et al. (2019) conducted FEM simulations examining crane hooks, with emphasis on stress analysis and safety factor determination. Their study utilized different

materials than S275 steel, producing stress values of 59-60 MPa with safety factors exceeding 5. While this research provided valuable insights into structural safety assessment methodologies for lifting hooks, it did not examine thermal loading conditions or the effects of repeated thermal cycles characteristic of steel industry applications. The significantly lower stress values reported suggest substantially different loading conditions or geometric configurations than those encountered in wire rod handling.

More recently, Sabbilli et al. (2021) investigated C-Hooks manufactured from ST52 steel using FEM analysis, reporting maximum stress values of 164 MPa in critical structural regions. This study more closely approximated actual C-Hook applications in industrial contexts; however, it did not incorporate elevated temperature variables or detailed fatigue life analysis (Mailer & Hidemyr, 2021). Given that thermal conditions represent a primary factor affecting both immediate structural performance and long-term durability in steel coil lifting operations, this omission limits the applicability of the findings to actual operating environments.

Additional relevant studies have examined related thermal-structural phenomena in different applications. Marpaung et al. (2021) performed thermal-structural simulations on first-stage blades of geothermal turbines constructed from stainless steel 420, utilizing CATIA V5 and ANSYS software. Their methodology incorporated steady-state thermal analysis followed by static structural analysis to evaluate heat distribution, stress development, and deformation patterns—an approach methodologically similar to the present study. However, their focus on turbine blades involved substantially different geometric configurations, loading patterns, and temperature regimes than C-Hook applications.

Pramudya et al. (2024) examined the structural strength of skylift truck crane baskets manufactured from ASTM A36 steel with hollow profiles (40×40×4 mm and 80×80×5 mm), investigating load variations from 150 kg to 250 kg. Their analysis generated comprehensive data on von-Mises stress, deformation, and safety factors using FEM (Wang, Liu, Wen, & Xie, 2020). Furthermore, Mahanthesh et al. (2020) [10] developed a balanced C-hook model for lifting coils with a 10-ton load capacity using CATIA V5 and ANSYS. They analyzed deformation, von-Mises stress, and modal characteristics across different materials (stainless steel, structural steel, ASTM Grade 60), concluding that ASTM Grade 60 offered superior safety margins. However, neither study incorporated thermal analysis, limiting their applicability to high-temperature lifting operations.

Based on this comprehensive literature review, a clear research gap emerges: no existing study has comprehensively integrated thermal and structural analysis specifically for C-Hooks manufactured from S275N material operating under the combined thermal-mechanical loading conditions typical of wire rod handling in steel manufacturing facilities. This gap represents both a scientific opportunity and a practical necessity, as the steel industry requires evidence-based design guidelines and operational protocols for such critical equipment.

The present study addresses this gap by providing a comprehensive investigation that combines steady-state thermal analysis with static structural analysis of C-Hooks manufactured from S275N steel, utilizing FEM simulation in ANSYS under realistic high-temperature conditions (170°C and 200°C). This research not only analyzes stress distribution and deformation patterns but also evaluates fatigue life and safety factors under thermal loading

conditions, thus providing a more complete and realistic assessment of C-Hook performance in actual steel industry working environments. The novelty of this research lies in: (1) the integration of thermal and mechanical analysis specific to S275N material C-Hooks, (2) the investigation of performance degradation across a realistic temperature range encountered in wire rod handling operations, (3) the comprehensive evaluation of multiple performance parameters including deformation, stress, safety factor, and fatigue life under thermal conditions, and (4) the validation of simulation results against actual field observations and documented failure cases.

This research is driven by the urgent need to enhance occupational safety by preventing catastrophic equipment failure and to address significant economic pressures in steel manufacturing by minimizing unplanned downtime. Furthermore, it responds to the industry's shift towards predictive maintenance and aims to fill a critical knowledge gap, as comprehensive design guidelines for the thermal effects on lifting equipment are currently lacking (Mołęda, Małysiak-Mrozek, Ding, Sunderam, & Mrozek, 2023). To tackle these issues, the study will employ a comprehensive FEM-based thermal-structural analysis of S275N steel C-Hooks to determine how elevated temperatures of 170°C and 200°C specifically impact stress, deformation, safety factors, and fatigue life, thereby identifying critical failure zones and establishing the temperature thresholds for operational risk.

The outcomes of this investigation are expected to deliver substantial theoretical and practical benefits, beginning with a fundamental contribution to the understanding of thermal-mechanical behavior in structural steel and the provision of validated computational models for wider application. Practically, the findings will directly inform the optimization of C-Hook geometry and the development of evidence-based maintenance protocols, ultimately enhancing workplace safety, reducing equipment failures, and improving manufacturing efficiency (Blejwas, 2021). Moreover, the FEM methodology itself will serve as a valuable template for analyzing other types of critical lifting equipment operating under similarly severe conditions (Szabó & Babuška, 2021).

METHOD

This study used a simulation approach based on the finite element method, beginning with direct observation at PT. X to identify problems and collect important data such as material type, lifting load, and post-rolling temperature range. All data were analyzed with the support of relevant literature references to strengthen the research results.

Previous research conducted by Aldeeb & Abdueilmula (2018) [4] focused on fatigue analysis of S275 steel using numerical simulations to determine the material's fatigue limit. The method used was the Finite Element Method (FEM) with cyclic loading, and the results showed that S275 steel has a fatigue limit of approximately 195.47 MPa. However, this study only emphasized the general fatigue aspect of the material, without considering the effects of high temperatures or specific applications in industrial components. Furthermore, a study by Wunda et al. (2019) [5] examined crane hooks using FEM simulations, focusing on stress analysis and safety factors. The material used was different from S275, and the results showed a stress of 59–60 MPa with a safety factor >5. This study provides an overview of the structural safety of the hook, but does not examine the thermal conditions or the effects of repeated heat cycles common in the steel industry. Meanwhile, Sabbilli et al. (2021) [6] studied C-Hooks

made of ST52 using FEM analysis, and the results showed a maximum stress of 164 MPa in the critical area. This study approaches the actual context of C-Hook use in industry, but does not include high temperature variables or detailed fatigue life analysis, even though heat conditions are an important factor in steel coil lifting operations.

There are other similar studies but some with different research objects that are used as references in this study, such as those conducted by Marpaung et al (2021) [7] who conducted thermal-structural simulations on stage 1 blades of geothermal turbines made of stainless steel 420 using CATIA V5 and ANSYS, with steady-state thermal and static structural analysis to evaluate heat distribution, stress, and deformation, in that study also used the same method as this study, namely using the finite element method (FEM) with simulation analysis that includes thermal and structural analysis as carried out in this study. The research of Pramudya et al. (2024) [8] examined the strength of skylift truck crane baskets made of ASTM A36 with hollow profiles of 40x40x4 mm and 80x80x5 mm through a loading variation of 150–250 kg, producing von Mises stress data, deformation, and safety factors using the finite element method. Furthermore, Mahanthesh et al. (2020) [9] modeled a balanced C-hook to lift a coil with a load of 10 tons using CATIA V5 and ANSYS, analyzed deformation, von Mises stress, and mode shapes with different materials (stainless steel, structural steel, ASTM grade 60), and concluded that ASTM grade 60 is safer because it has a higher safety factor value.

Based on several previous studies, this study contributes by combining thermal and structural analysis on C-Hook made of S275 material, using FEM simulation in ANSYS with high temperature conditions (170°C and 200°C). This study not only analyzes stress and deformation, but also evaluates fatigue life and safety factor under thermal conditions, thus providing a more comprehensive and realistic picture of the performance of C-Hook in the steel industry working environment.

This research covers the design process of the C-Hook coil lifting research object created with the help of Solidworks CAD software, with the following specifications and visualizations of the research object:

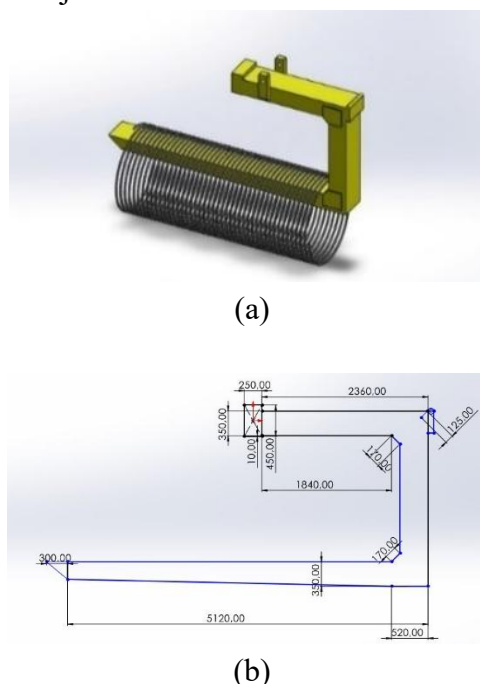


Figure 2. (a) Research Object Design (b) 2D Design of Research Objects

Simulation analysis of the C-Hook coil lifting design in this study uses the finite element method with the help of Ansys Workbench software. This software was chosen because it is capable of solving structural problems from complex designs effectively and quickly using the finite element method. This simulation will use the C-Hook coil lifting design model with the Steady-State Thermal Analysis and Static Structural Analysis methods. The following are the simulation analysis process parameters used in Figure 3.

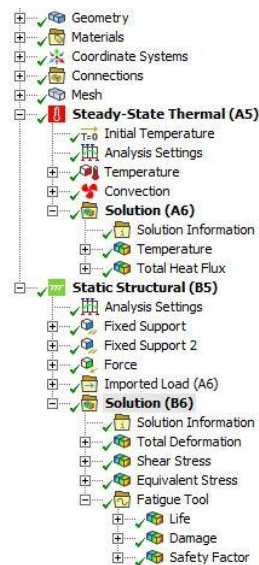
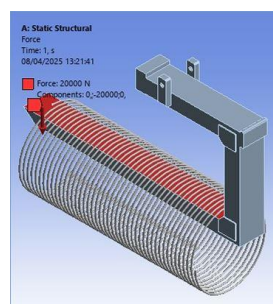
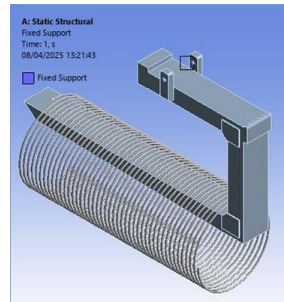


Figure 3. *Solution Tools*

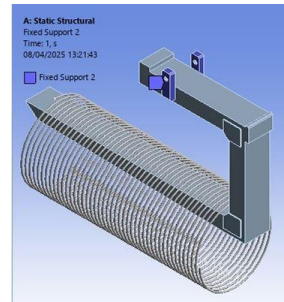
Based on the observation results, the object receives a load of 2 tons (20.000 N) in one work cycle. In the simulation, boundary conditions in the form of fixed support at the top of the C-Hook connected to the monorail were used, with the assumption that no translational or rotational displacement occurred in this area (all degrees of freedom = 0). The visualization of the application of the loading conditions and boundary conditions in the form of fixed support can be seen in Figures 4.



(a)



(b)



(c)

Figure 4. (a) Force on Coil Lifting C-Hook (b) Fixed Support on Coil Lifting C – Hook 1 (c) Fixed Support on Coil Lifting C – Hook 2.

In finite element analysis (FEA), meshing is performed to divide a structure into small elements so that it can be analyzed numerically. In this study, a tetrahedral mesh was used because of its ability to handle complex geometric shapes and provide good results in simulations. The visualization of the meshing application is shown in Figure 5 [10].

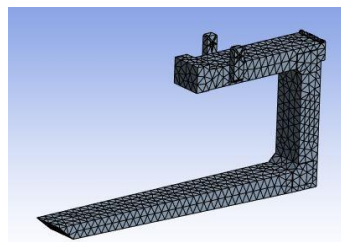


Figure 5. Visualization of Coil Lifting C-Hook Meshing

Analysis of the C-Hook coil lifting research object using S275 N structural steel material. The material properties of S275 N structural steel are used as supporting data parameters in the C-Hook coil lifting strength simulation analysis process [11].

Table 1. Material Properties Structural Steel S275 N

<i>Parameters</i>	<i>Values</i>
<i>Density, ρ (kg/m³)</i>	7850
<i>Young's Modulus, E (MPa)</i>	205,800
<i>Poisson's ratio</i>	0.3
<i>Yield stress, σ_Y (MPa)</i>	235

<i>Ultimate tensile strength (MPa)</i>	400
--	-----

This study uses steady-state thermal and static structural simulations in Ansys Workbench to analyze the effect of high temperatures (170–200°C) on stress and deformation in C-Hooks. Elastic properties are influenced by the elastic modulus of the material. Deformation is a change in the shape of a material due to force, with two main types: elastic (temporary and returns to its original shape) if it is still below the melting point (yield point) and plastic (permanent even if the load is stopped) if it exceeds it [12]. The equations used in this study are as follows [13]:

Strain is calculated using the formula:

$$\varepsilon = \Delta L / L_0 \quad (1)$$

where ΔL is the change in length, and L_0 is the initial length.

The elastic modulus of a material is obtained from the ratio between stress and strain:

$$E = \sigma / e \quad (2)$$

where E is the modulus of elasticity, σ is the stress (in units of Pascal (Pa) or N/m^2), and e is the strain.

The basic equations for thermal analysis and their relationship to stress and strain are explained as follows. Heat transfer analysis, including thermal analysis of materials, can use Fourier's Law. Fourier's Law states that heat flow (f_x) in a material is proportional to the temperature gradient and oppositely directed. This means that heat will flow from areas of high temperature to areas of low temperature. The negative sign indicates that heat transfers in the direction of decreasing temperature gradient.

$$F_x = -k \partial T / \partial x \quad (3)$$

where f_x is heat flux in units of W/m^2 , k is the thermal conductivity of the material ($W/m \cdot K$), and $\partial T / \partial x$ is the temperature gradient (K/m).

Analysis of thermal strain equations in thermal analysis states that changes in the shape of a material due to temperature changes are directly proportional to the temperature changes and the thermal expansion coefficient of the material.

$$\varepsilon_o = \alpha \Delta T \quad (4)$$

where ε_o is thermal strain, which is a change in the shape of a material due to a change in temperature, α is the thermal expansion coefficient, which indicates how much the material expands or contracts due to temperature changes ($1/^\circ C$ atau $1/K$), ΔT is a change in temperature ($T_f - T_i$), which is the difference between the final temperature and the initial temperature (in Kelvin). The greater the value of ΔT or α , the greater the expansion or contraction that occurs.

The total strain equation states that the total strain of a material is the sum of the elastic strain and thermal strain.

$$\varepsilon = \varepsilon_e + \varepsilon_o \quad (5)$$

where ε is the total strain, which is the total amount of strain that occurs in a material, ε_e is elastic strain, which is strain that occurs due to mechanical stress and can return to its original

shape after the load is removed, and ϵ_0 is thermal strain, which is strain that occurs due to changes in temperature.

The stress equation for materials undergoing thermal effects derives from Hooke's Law, which has been modified to take thermal strain into consideration.

$$\sigma = E (\epsilon - \epsilon_0) \quad (6)$$

where σ is the stress on the material (N/m² or Pa (Pascal)), E is the modulus of elasticity (Young's modulus), which indicates the stiffness of the material, ϵ is the total strain, and ϵ_0 is thermal strain, calculated using the formula $\epsilon_0 = \alpha \Delta T$, where α is the thermal expansion coefficient and ΔT is a change in temperature.

The stress equation for materials considering thermal strain comes from Hooke's Law, which has been modified for thermal effects.

$$\sigma = E\epsilon_e = E (\epsilon - \epsilon_0) \quad (7)$$

where σ is the stress on the material (N/m² or Pa (Pascal)), E is the modulus of elasticity (Young's modulus) which indicates the stiffness of the material, ϵ_e is elastic strain, which is strain caused by mechanical stress, ϵ is the total strain, and ϵ_0 is thermal strain, calculated using the formula: $\epsilon_0 = \alpha \Delta T$, where α is the thermal expansion coefficient and ΔT is the temperature change.

In steady-state thermal analysis, temperature, material properties, and boundary conditions are assumed to be constant over time to determine the stable temperature distribution according to Fourier's law of conduction. In this study, heat is transferred through conduction when the C-Hook is in contact with the heating coil, and through convection due to the ambient temperature. Conduction occurs without particle movement in solids, while convection involves particle movement in fluids. The following equation is the general form of the Steady-State Thermal Heat Conduction Equation.

$$k\nabla^2 T = -q_v \quad (8)$$

where k is the thermal conductivity of the material (W/m·K), that is the ability of a material to conduct heat, $\nabla^2 T$ is the Laplacian of temperature (Laplace equation), indicating how quickly the temperature changes with respect to space at a given point. If $q_v > 0$, there is an internal heat source (e.g., frictional heat, chemical reactions, electric current) that causes the temperature to rise in certain areas. If $q_v < 0$, There is no internal heat source, so the formula can be simplified to $\nabla^2 T = 0$ (Laplace equation), which means that the temperature distribution is only affected by boundary conditions. The negative sign indicates that the direction of heat flow follows the temperature gradient (heat flows from hot to cold).

Von-Mises stress is a unique combination of maximum principal stress (σ_1) and minimum principal stress (σ_2) that can be directly compared with the yield strength of a material to predict failure due to yielding. The von Mises criterion is used as a guideline to determine when a material begins to experience plastic deformation, stating that the material begins to undergo plastic deformation when the von Mises stress exceeds the yield stress of the material [14]. This is defined in theory as follows: [15]

$$\sigma_{vm} = \sqrt{\frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{2}} \quad (9)$$

where σ_{vm} is von-Mises stress (Pa), $\sigma_{1,2,3}$ is principal stress (Pa). $\sigma_1 > \sigma_2 > \sigma_3$. If $\sigma_v < \sigma_{yield}$ = safe material. If $\sigma_v \geq \sigma_{yield}$ = material yielding or failure.

The maximum von Mises value increases as the load increases, because stress is directly proportional to the force received. This increase has a direct impact on the safety level of the design. [8]

The safety factor compares the strength of a structure or material to the load it receives or must be greater than the stress applied [14]. Ideally, it should be between 1,5 and 2,0. A value below 1,0 poses a risk of failure, while a value that is too high results in material waste and increased costs. This factor is calculated from the ratio of yield strength to maximum von Mises stress [16].

$$n = (S_y)/\sigma_e \quad (10)$$

where n is safety factor (factor of safety), S_y is yield strength, and σ_e is the maximum von Mises stress analyzed.

The value of this design factor must be large to avoid material failure. Failure that occurs because the material is subjected to repeated stress is called fatigue (material fatigue). The safety factor value is categorized as safe if the value of n is in the range of 1,2 to 4 [17].

RESULT AND DISCUSSION

The C-Hook simulation was conducted with a load of 2 tons and a temperature of 170°C–200°C through steady-state thermal and static structural analysis. A 150 mm mesh was selected because it produced the best skewness results, plus refinement in critical areas to improve accuracy. Simulations without refinement were used to compare the effects of temperature and evaluate the consistency of the results. The simulation results show several main data parameter values as shown in Table 2.

Table. 2 Simulation Parameter Values Without Mesh Refinement

Parameters	Values (200°C)	Valuess (170°C)
Temperature (max)	680,68°C	564,85°C
Heat Flux (max)	0,57511 W/mm ²	0,47241 W/mm ²
Total Deformation (max)	2,451 mm	1,624 mm
Equivalent von – Mises Stress (max)	409,35 MPa	336,1 MPa
Shear Stress (max)	52,059 MPa	42,941 MPa

Simulations without mesh refinement show increased deformation and stress at temperatures of 170°C and 200°C, with stress exceeding the elastic limit of S275N. This indicates that the material is not suitable for full load, so refinement is still necessary for accuracy and safety.

Mesh Convergence Analysis

Mesh convergence analysis was carried out to determine the optimal mesh size by comparing the maximum von Mises stress at temperatures of 200°C and 170°C. A mesh size

of 150 mm was selected because it was in the middle of the tested range, was considered the most representative, and was used in the main simulation.

The following table presents the mesh convergence data from the simulation results, along with an explanation of the trends observed in the mesh convergence graph based on the relationship between mesh element size and maximum von-Mises stress for the two different temperature conditions (200°C and 170°C);

Table. 3 Simulation Results at a Temperature of 200°C

Element Size (mm)	Maximum von - Mises Stress (Mpa)	Skewness (Average)	Error	Nodes	Element
146	152,46	0,30963	-0,000382893 (- 3,83%)	110403	73915
148	164,06	0,30986	0,00034883 (+3,49%)	105852	70592
150	158,53	0,31009	0 (0,00%)	106846	71541
152	135,22	0,30772	-0,001470384 (- 14,70%)	106243	71061
154	168,58	0,30835	0,000633949 (+6,34%)	104760	70116

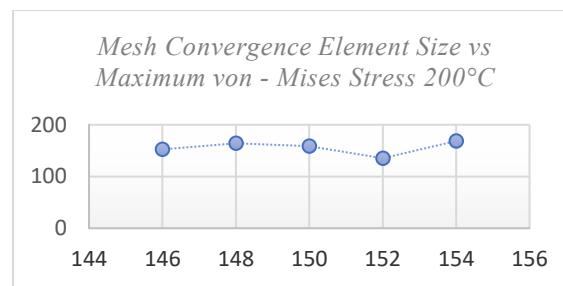


Figure 7. Curved Convergence Mesh Element Size vs Maximum von - Mises Stress 200°C

Mesh convergence analysis at 200°C compares the maximum von Mises stress of mesh sizes 146–154 mm. Stress values range from 152,46 to 168,58 MPa, with the 150 mm mesh producing 158,53 MPa and serving as a reference. The 146 mm and 148 mm meshes have relative errors of -3,83% and +3,49% (within the $\pm 5\%$ limit), while the 152 mm and 154 mm meshes deviate further, at -14,70% and +6,34%, and are therefore not considered convergent. All meshes have a Skewness of 0,307–0,310, indicating good quality. With convergent, representative, and computationally efficient results, the 150 mm mesh is selected as the most optimal. This data is presented in Table 3 and Figure 7 above.

Table 4. Simulation Results at a Temperature of 170°C

Element Size (mm)	Maximum von - Mises Stress (Mpa)	Skewness (Average)	Error	Nodes	Element
146	124,75	0,30963	-0,001143689 (- 11,44%)	110403	73915
148	134,27	0,30986	-0,00046784 (- 4,68%)	105852	70592
150	140,86	0,31009	0 (0,00%)	106846	71541
152	111,02	0,30772	-0,002118415 (- 21,18%)	106243	71061

154	138,37	0,30835	-0,000176771 (-1,77%)	104760	70116
-----	--------	---------	-----------------------	--------	-------

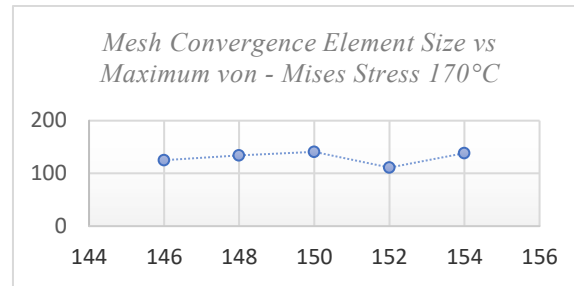


Figure 8. Curved Convergence Mesh Element Size vs Maximum von - Mises Stress 170°C

At 170°C, the mesh convergence trend shows a similar pattern with lower stress. The von Mises stress increases from 124,75 MPa (146 mm mesh) to 140,86 MPa (150 mm mesh), then decreases to 111,02 MPa (152 mm mesh), and rises again to 138,37 MPa (154 mm mesh). Mesh sizes 148 mm and 154 mm have relative errors of -4,68% and -1,77%, respectively, still within the $\pm 5\%$ limit, while mesh sizes 146 mm and 152 mm are not convergent. Therefore, meshes 148 mm, 150 mm, and 154 mm are considered convergent, with mesh 150 mm as the optimal choice due to its stability and central position on the trend graph. The mesh quality is also good, with tetrahedral element skewness between 0,25–0,38, well below the 0,5 limit, making this mesh suitable for accurate and stable simulations. This data is presented in Table 4, Figure 8, and Figure 9.

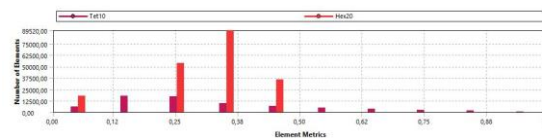


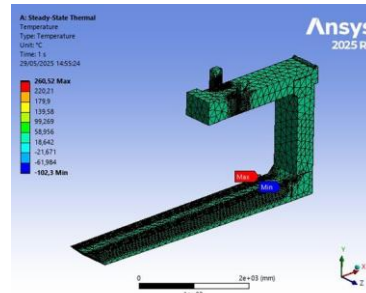
Figure 9. Element Metric Graph Based on Skewness

Steady-State Thermal Analysis Results

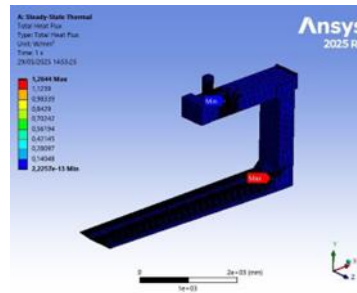
A steady-state thermal simulation was conducted to analyze the temperature distribution resulting from direct contact between the steel coil and the C-hook. The convection temperature on the outer surface was set at 200°C and 170°C, with an ambient temperature (film coefficient) of 32°C and a heat transfer coefficient of 25 W/m²·°C, based on actual conditions from field studies.

a. Temperature Distribution (Steady-State Thermal) 200°C

The simulation shows a maximum temperature of 260,52°C and an average of 31,997°C, with heat concentrated in the C-Hook and coil contact area and then spreading conductively. The maximum heat flux is 1,2644 W/mm², the minimum is $2,23 \times 10^{-13}$ W/mm², and the average is $2,4651 \times 10^{-4}$ W/mm², indicating a uniform heat distribution without extreme differences, consistent with the principles of conduction and real- conditions. The visualization of the simulation results and values can be seen in Figure 10.



(a)

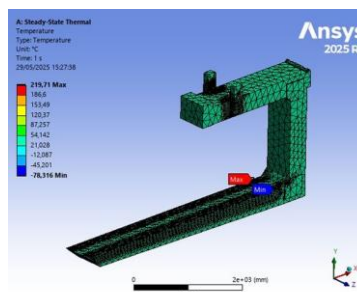


(b)

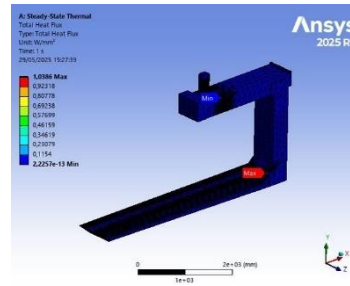
Figure 10. (a) Visualization of Temperature Output Values from Steady-State Thermal Analysis Simulation at a Temperature of 200°C (b) Visualization of Heat Flux Output Values from Steady-State Thermal Analysis Simulation at a Temperature of 200°C.

b. Temperature Distribution (Steady-State Thermal) 170°C

The simulation shows a maximum temperature of 219,71°C and an average of 31,998°C, with heat concentrated in the C-Hook and coil contact areas. The maximum heat flux is 1,0386 W/mm², the minimum is $2,2257 \times 10^{-13}$ W/mm², and the average is $2,0249 \times 10^{-4}$ W/mm². Heat spreads conductively outward and decreases with distance, reflecting real-conditions that are also influenced by convective cooling from the surrounding air. The visualization of the simulation results and values can be seen in Figure 11.



(a)



(b)

Figure 11. (a) Visualization of Temperature Output Values from Steady-State Thermal Analysis Simulation at a Temperature of 170°C (b) Visualization of Heat Flux Output Values from Steady-State Thermal Analysis Simulation at a Temperature of 170°C.

Static Structural Analysis Results

Static analysis was performed with a vertical load of 20,000 N (2 tons) on the hook lifting area. The temperature from the Steady-State Thermal simulation was used as the thermal body load to reflect heat distribution under actual conditions, so that the simulation results were more representative of the actual operating environment.

a. Results of Static Structural Analysis at 200°C

Based on the heat distribution imported from the thermal step analysis, the structural simulation shows the following results:

Total Deformation: maximum (2,425 mm).

The maximum total deformation of 2,425 mm occurred at the end of the C-Hook curve, the point furthest from the support. This value is still within reasonable limits, indicating that the structure remains strong without experiencing damage or excessive flexibility. The visualization of the results and simulation values can be seen in Figure 12.

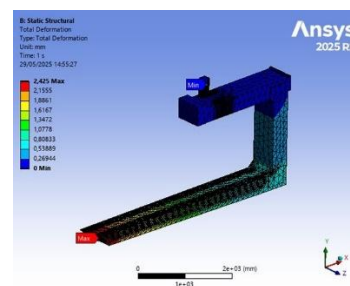


Figure 12. Visualization of Total Deformation Output Values from Static Structural Analysis Simulation at a Temperature of 200°C

Von-Mises Stress (Equivalent Stress): maximum (158,53 MPa).

The simulation shows a maximum von Mises stress of 158,53 MPa on the lower horizontal arm bend due to a combination of vertical loads and thermal expansion. High temperatures cause the material to expand, increasing stress concentration in rigid areas. Although still below the elastic limit of S275 (237 MPa), the value is sufficiently close

and should be considered in structural design. The results and values of the simulation can be seen in Figure 13.

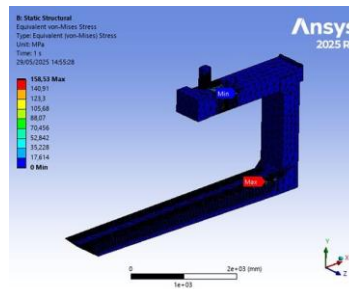


Figure 13. Visualization of von-Mises stress (equivalent stress) output values from static structural analysis simulations at a temperature of 200°C.

Safety Factor: minimum (0,973).

The simulation shows a maximum safety factor of 15 and a minimum of 0,973, indicating a critical point even though the overall structure is still safe. The fatigue life reaches 1.394.500 cycles, indicating a risk of damage if used continuously in high temperature conditions, especially in areas with maximum stress and deformation. The visualization of the results and simulation values can be seen in Figure 14.

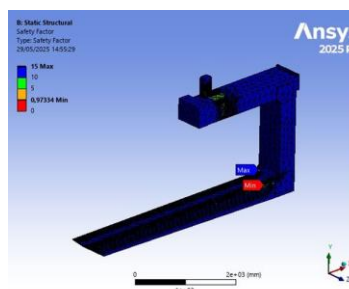


Figure 14. Visualization of Safety Factor Output Values from Static Structural Analysis Simulation Results at a Temperature of 200°C

Fatigue Life: minimum (1.394.500 cycles).

The simulation shows that the C-Hook has a fatigue life of over 1 million cycles, indicating long-term durability. Mesh refinement improves the accuracy of stress and deformation distribution and helps identify critical points, although the safety factor is not yet ideal. The visualization of the results and simulation values can be seen in Figure 15.

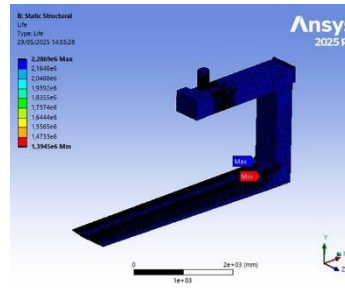


Figure 15. Visualization of Fatigue Life Output Values from Static Structural Analysis Simulation at a Temperature of 200°C

b. Results of Static Structural Analysis at 170°C

Total Deformation: maximum (2,4358 mm).

The simulation results show a maximum deformation of 2,4358 mm and an average of 0,958 mm at the end of the C-Hook curve due to loading and high temperatures. These values are still within safe limits and do not interfere with the hook's function in the short term. The visualization of the results and simulation values can be seen in Figure 16.

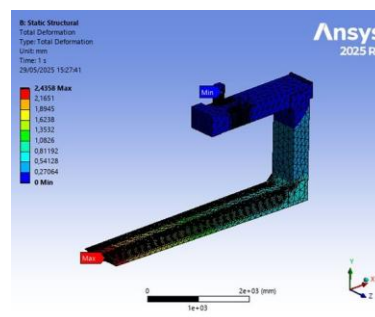


Figure 16. Visualization of Total Deformation Output Values from Static Structural Analysis Simulation at a Temperature of 170°C

Von-Mises Stress (Equivalent Stress): maximum (140,86 MPa).

The simulation shows a maximum stress of 140,86 MPa near the hook connection due to load and thermal expansion. This value is close to the elastic limit of S275, indicating that the structure is still safe but susceptible to fatigue. The highest stress shifts to the top of the hook at a temperature of 170°C, indicating the influence of tensile force and temperature. The visualization of the simulation results and values can be seen in Figure 17.

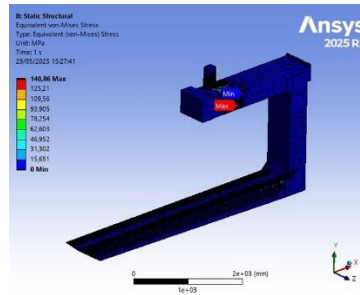


Figure 17. Visualization of von-Mises stress (equivalent stress) output values from static structural analysis simulations at a temperature of 170°C.

Safety Factor: minimum (1,0954).

The simulation shows a minimum safety factor of 1,0954 and an average of 14,07. Although still safe, this minimum value is below the ideal standard ($\geq 1,2-4$), with the risk area located at the point of maximum stress and deformation, indicating potential thermal and mechanical failure. The visualization of the results and simulation values can be seen in Figure 18.

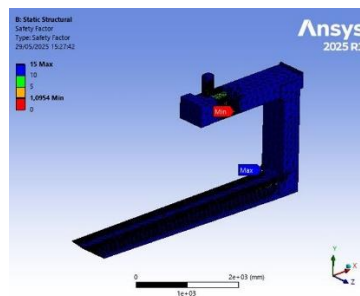


Figure 18. Visualization of Safety Factor Output Values from Static Structural Analysis Simulation Results at a Temperature of 170°C

Fatigue Life: minimum (2.286.900 cycles).

The fatigue life of the C-Hook theoretically reaches 2.286.900 cycles, but it can decrease due to repeated exposure to heat. Critical points appear in certain areas, and mesh refinement improves the accuracy of stress and deformation distribution, although the safety factor is not yet ideal. The visualization of the results and simulation values can be seen in Figure 19.

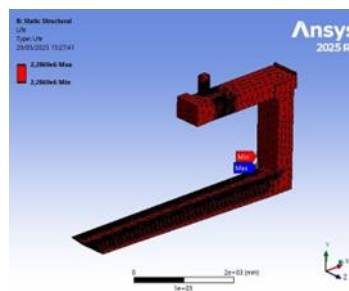


Figure 19. Visualization of Fatigue Life Output Values from Static Structural Analysis Simulation at a Temperature of 170°C

Simulations with refined meshes at temperatures of 170°C and 200°C produced more accurate stresses with smaller errors. However, the low safety factor and fatigue life indicate the need for heat-resistant coatings or more suitable materials. These results correspond to actual damage or conditions, proving the effectiveness of thermal-structural FEM analysis.

Fatigue Life and Safety Factor Analysis

Fatigue life simulation of C-Hooks with loads of 8.000–16.000 N showed a constant life of 2.286.900 cycles at 170°C (infinite life) and a safety factor of 1,1018–1,1135, indicating a safe structure. However, at 200°C, the fatigue life decreases to 1,256,900–1.347.300 cycles, with a safety factor below 1 (0,96731–0,97134), indicating that the structure is no longer statically safe. This proves that high temperatures accelerate damage and reduce resistance to repeated loads. The following are the results of the simulation showing data as presented in Table 5.

Table 5. Variations in Simulation Results at a Temperature of 200°C

No.	Load (N)	Equivalent von – Mises Stress (max) (MPa)	Fatigue Life (cycles)	Safety Factor
1.	40% (8.000 N)	159,52 Mpa	$1,2569 \times 10^6$ cycles	0,96731
2.	50% (10.000 N)	159,35 Mpa	$1,279 \times 10^6$ cycles	0,96832
3.	60% (12.000 N)	159,18 Mpa	$1,3015 \times 10^6$ cycles	0,96933
4.	70% (14.000 N)	159,02 Mpa	$1,3242 \times 10^6$ cycles	0,97033
5.	80% (16.000 N)	158,85 Mpa	$1,3473 \times 10^6$ cycles	0,97134

Therefore, at a temperature of 200°C, an S-N curve can be formed because there is a clear relationship between the stress level (von-Mises stress) and the decrease in fatigue life, as shown in Figure 20.

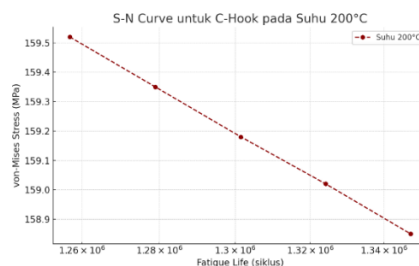


Figure 20. S-N Curve Graph 200°C

The curve shows that an increase in temperature from 170°C to 200°C significantly reduces the fatigue life of the C-Hook. At 170°C, the service life is still good, but at 200°C it begins to decline due to material fatigue. This emphasizes the importance of considering the effects of high temperatures in the design of structures in high temperature environments.

Simulation Results Validation

To ensure that the simulation results have a solid foundation, validation was performed by comparing the simulation results with previous studies on maximum stress, deformation, safety factor, and fatigue life. According to Aldeeb & Abduelmula (2018) [4], the fatigue limit of S275 steel is approximately 195,47 MPa. Simulations at 170°C resulted in stress levels

below this limit with a service life of >2 million cycles (infinite life), but at 200°C, stress increased to 158 MPa and the service life decreased to 1,3 million cycles. Compared to other studies such as Wunda et al. (2019) [5] and Sabbilli et al. (2021) [6], the differences in stress and safety factor are due to variations in temperature, material, and design. These results confirm the significant influence of high temperature and repeated loading on the reliability of C-Hooks in the steel industry.

CONCLUSION

FEM analysis shows that the maximum stress is located in the inner curve of the C-Hook and shifts upward at a temperature of 170°C due to the effects of tensile force and heat. Deformation is still safe but increases due to high temperatures, which also accelerates material fatigue. Although the von Mises stress is still below the yield limit and the safety factor is safe (1,5–2), the service life still decreases if the effect of temperature is ignored. It is recommended to optimize the design in critical areas, use heat-resistant materials, conduct routine inspections, and incorporate thermal insulators or passive cooling systems. Further studies should include dynamic loading and experimental validation.

REFERENCE

- Abidin, Z., & Rama, B. R. (2015). Analisa distribusi tegangan dan defleksi connecting rod sepeda motor 100 cc menggunakan metode elemen hingga. *Jurnal Rekayasa Mesin Universitas Sriwijaya*, 15(1), 30–39.
- Aldeeb, T., & Abdueh, M. (2018). Fatigue strength of S275 mild steel under cyclic loading. *World Academy of Science, Engineering and Technology International Journal of Materials and Metallurgical Engineering*, 12(10), 564–570.
- Bazli, M., & Abolfazli, M. (2020). Mechanical properties of fibre reinforced polymers under elevated temperatures: An overview. *Polymers*, 12(11), 2600.
- Blejwas, C. (2021). Designing non-pneumatic anti-shock trousers.
- Bolandi, H., Li, X., Salem, T., Boddeti, V. N., & Lajnef, N. (2022). Deep learning paradigm for prediction of stress distribution in damaged structural components with stress concentrations. *Advances in Engineering Software*, 173, 103240.
- Chovnyuk, Y. V., Diachenko, L., Ivanov, Y. O., Dichek, N. P., & Orel, O. (2022). Optimisation of dynamic loads of rope systems of lifting mechanisms of bridge cranes during cargo handling. *Scientific Herald of Uzhhorod University. Series "Physics"*, (51), 59–73.
- Chukwunweike, J., Anang, A. N., Adeniran, A. A., & Dike, J. (2024). Enhancing manufacturing efficiency and quality through automation and deep learning: Addressing redundancy, defects, vibration analysis, and material strength optimization. *World Journal of Advanced Research and Reviews*, 23, 1–10.
- Marpaung, F., Harmadi, R., & Guardi, A. (2021). Simulasi thermal-struktur blade stage satu turbin geothermal. *Jurnal Asimetrik: Jurnal Ilmiah Rekayasa & Inovasi*, 35–42.
- Mailer, H., & Hidemyr, E. (2021). Design, manufacturing and structural analysis of a lifting hook.
- Mahanthesh, M. R., Girisha, L., Shreyas Babu, C., & Shivananda, D. C. (2020). Modelling and simulation of below-the-hook lifting device balanced C-hook for load to investigate the static and modal analysis for various grades of steels by numerical method. *Journal of Modeling and Simulation of Materials*, 3(1), 61–69.
- Molęda, M., Małyśiak-Mrozek, B., Ding, W., Sunderam, V., & Mrozek, D. (2023). From corrective to predictive maintenance—A review of maintenance approaches for the

- power industry. *Sensors*, 23(13), 5970.
- Mukhopadhyay, G. (2018). Failures of construction equipment and accessories: Metallurgical root-cause analysis. Dalam *Handbook of Materials Failure Analysis* (hlm. 83–106). Elsevier.
- Pramudya, A. D., Fiveriati, A., Wahid, M. A., Sari, E. N., & Catrawedarma, I. (2024). Analisis kekuatan keranjang skylift truck crane menggunakan metode elemen hingga. *Journal of Scientech Research and Development*, 6(1), 1711–1718.
- Saad-Eldeen, S., Garbatov, Y., & Soares, C. G. (2016). Ultimate strength analysis of highly damaged plates. *Marine Structures*, 45, 63–85.
- Sabbilli, D. N., Sidi, P., & Pramaningtyas, W. E. (2021). Analisis struktur C-hook SWL 30 ton menggunakan metode elemen hingga. Dalam *Proceedings Conference on Design Manufacture Engineering and Its Application* (hlm. 19–23).
- Schmitz, N., Sankowski, L., Kaiser, F., Schwotzer, C., Echterhof, T., & Pfeifer, H. (2021). Towards CO₂-neutral process heat generation for continuous reheating furnaces in steel hot rolling mills—a case study. *Energy*, 224, 120155.
- Szabó, B., & Babuška, I. (2021). Finite element analysis: Method, verification and validation.
- Wang, H., Liu, J., Wen, G., & Xie, Y. M. (2020). The robust fail-safe topological designs based on the von Mises stress. *Finite Elements in Analysis and Design*, 171, 103376.
- Wunda, S., Johannes, A. Z., Pingak, R. K., & Ahab, A. S. (2019). Analisis tegangan, regangan dan deformasi crane hook dari material baja AISI 1045 dan baja ST 37 menggunakan software Elmer. *Jurnal Fisika: Fisika Sains dan Aplikasinya*, 4(2), 131–137.