

Experimental strain evaluation and generative design optimization of a crane hook using finite element analysis

Cihat Sofracı¹, Eren Kalay¹, Adem Candaş^{1*}, Cevat Erdem İmrak¹

¹ Faculty of Mechanical Engineering, Istanbul Technical University, İnönü Cd 65 Beyoğlu, Istanbul, Türkiye

* Corresponding author's e-mail: candas@itu.edu.tr

ABSTRACT

Crane hooks are safety-critical mechanical components used for lifting heavy loads, and their structural integrity directly affects operational reliability. The conventional crane hook designs based on analytical approaches may lead to conservative geometries with excessive material usage. This study investigated the generative design optimization of a crane hook modeled in accordance with the DIN 15401 standard. Experimental strain measurements were conducted at the critical cross-section of the hook under axial loading conditions. The measured strain responses were compared with finite element analysis results and used to improve the definition of the numerical material parameters. After establishing the finite element model, design constraints, preserved regions, obstacle regions, and boundary conditions were defined for the generative design process. The final optimized crane hook achieved an approximate weight reduction of 39.4% compared with the classical design while maintaining stresses below the material yield strength. The results show that experimental strain evaluation combined with finite element analysis can provide a useful basis for assessing and guiding generative design optimization of crane hook geometries.

Keywords: crane hook, experimental strain measurement, generative design, finite element analysis.

INTRODUCTION

Crane hooks are lifting and handling components with complex geometries that ensure the secure holding of loads to be lifted or transported. They have a wide range of applications in sectors such as construction, transportation, manufacturing, mining, and machinery, as well as in daily load transportation. Therefore, designing and selecting the most suitable and safest load-holding element is crucial, taking into account factors such as the weight of the load, the lifting distance, and the service life. Like other material handling equipment, crane hooks also have essential design requirements and application prerequisites. Loads should be held and released quickly and easily. The holding operation should be performed with as few personnel as possible. Loads should be held securely against breakage and load slippage. The load holding elements should not damage the

material being transported [1]. To improve the efficiency of lifting machines, crane hooks should be as lightweight and strong as possible. Hooks are commonly used load-holding elements in both small- and large-scale lifting applications and are generally classified according to their shape [2]. Hook blocks are classified as long-shaft and short-shaft according to the type of their shafts. The hooks used in lifting machines can be divided into single-end simple hooks, double-end hooks, and lamellar hooks [1]. This article focused specifically on single-end simple hooks.

Due to the complex geometry of crane hooks, conventional analytical approaches have often led to overly conservative designs. The geometry of a hook, which is bound by design constraints, can be redesigned using new methodological approaches. It is possible to manufacture hooks with similar structures, which have been used in almost every field for years, at a more affordable cost by using

new analytical possibilities, maintaining design criteria, and taking into account developing manufacturing methods. Studies in the literature generally have focused on the optimization studies aiming at reducing the stress values of the hook under the same load or saving material while maintaining a similar stress. The research aimed at reducing stress values generally involves changes in cross-sectional geometry, material selection, or certain dimensions of the hook geometry. Some studies focused on various cross-sectional geometries, such as circular and rectangular sections [2], trapezoidal sections [3], T-sections [4,5], and a combination of these sections [6]. The study concluded that the trapezoidal hook made of wrought iron showed the least stress and highest fatigue resistance under load [7].

Previous studies have demonstrated the effectiveness of combining finite element analysis with analytical approaches for lifting system components and the hook stress evaluation [8]. In particular, a 40 kN lifting hook designed according to DIN 15400 [9] and DIN 15401 [10] was modeled and analyzed using FEM, and the numerical results were validated through curved beam theory [11]. A comprehensive optimization study was presented in which different crane hook cross-sectional geometries were evaluated using finite element analysis and classical curved beam theory. The results indicated that a trapezoidal cross-sectional profile provides superior performance in terms of von Mises stress and factor of safety under identical boundary conditions [12]. Analytical formulations based on Winkler's theory have been proposed for curved beam cross-sections to achieve balanced stress distributions under combined bending and axial loading, and validated by three-dimensional finite element analyses of a hook geometry [13]. Topology optimization and lightweight design of crane hooks using 3D modeling tools and additive manufacturing have been investigated, demonstrating that significant weight reduction can be achieved while maintaining structural integrity within material strength limits [14]. The failure mechanisms of crane hooks have been investigated using a multidisciplinary approach combining metallurgical examination, analytical calculations, and finite element analysis. The results highlight that fatigue failure typically initiates at stress concentration regions, such as threaded roots or surface irregularities, and that finite element simulations accurately predict the critical high-stress zones associated with failure

[15]. Analytical expressions [16] for circumferential stress concentration factors in lifting hooks were developed by fitting finite element analysis results for shallow notches. Based on Neuber-type formulations, the proposed simplified equations enable rapid estimation of stress concentrations without resorting to finite element modeling, while also addressing stress triaxiality effects and the associated reduction in material ductility at notch regions [17]. The mechanical response of crane hooks under payload was investigated through theoretical, numerical, and experimental approaches by combining curved beam theory, approximate analytical methods, finite element analysis, and strain gauge measurements. A good agreement was reported between analytical predictions, finite element results, and experimental data obtained from strain gauge measurements on a DIN 15401-compliant crane hook [18]. Alternative approaches based on advanced constitutive modeling and isogeometric analysis have also been reported for strongly curved beams, highlighting the limitations of simplified formulations commonly used in numerical analyses [19]. A three-step optimal design methodology was developed for 2-ton forged steel lifting hooks based on ISO 7597:2013, combining finite element analysis and weight optimization to improve performance, resulting in a 27.5% reduction in material cost and a 42.16% increase in safety compared to the initial design [20]. Using topology optimization and finite element analysis, the study achieved a lighter hook design with a 5% weight reduction while maintaining equivalent strength and fatigue life, demonstrating the efficiency of geometry optimization in structural improvement [21].

When the studies available in the literature are examined, it is observed that crane hook analyses are predominantly based on analytical formulations, numerical approaches [22] – particularly the finite element method – and, to a limited extent, experimental investigations. With the increasing accessibility and widespread use of numerical simulations in recent years, experimental data acquisition has remained relatively scarce. More specifically, for crane hook geometries, the application of strain gauges on both the inner and outer surfaces along mutually perpendicular directions to obtain real-time strain data has not, to the authors' knowledge, been reported in the open literature. In this context, simultaneous strain measurements collected from four orthogonal axes on the inner and outer surfaces of

a crane hook have not been addressed, which constitutes a clear gap in existing research. To bridge this gap, the present study obtained experimental strain measurements from both the inner and outer curved surfaces of a crane hook under loading conditions. The finite element analysis results were compared with the experimentally measured strain data to evaluate the numerical model. Based on the resulting finite element model, a generative design optimization study was then conducted to investigate the potential for weight reduction while maintaining stresses below the material yield strength.

METHODOLOGY

This section first presents the geometrical details of the crane hook considered in this study. Then, the mesh structure and boundary conditions used in the finite element analysis were defined. Finally, the experimental setup was introduced.

Geometrical details and materials specifications

The single-ended simple crane hook used in this study was modeled according to the dimensions specified in the DIN 15401 standard. Hook number NR025 was selected owing to its suitable dimensions for experimental testing. The selected hook type is defined as RSN, corresponding to a drop-forged hook (S), an unmachined part (R), and a hook with a nose (N). The nose protruding from the shank was not included in the model, since its effect on the stress distribution was considered negligible. Figure 1 shows the critical section (A-B) defined in the DIN 15401 standard. Three-dimensional geometry and main geometrical details are provided in the technical drawing. Section naming and position details follow the relevant standard.

According to the standard, the designation of a type RSN single hook No. 025 of strength class P is “Single hook DIN 15401 – RSN 025 – P”. The material was defined in accordance with DIN 15400. According to the standard, for hook number 025 and shank diameters d_1 in the range of 14–36 mm, the specified material for class P is StE 355. The material properties of StE 355 steel are presented in Table 1

The upper yield strength, R_{eH} or 0.2% proof stress, $R_{p0.2}$, in N/mm² is 315 MPa. These are

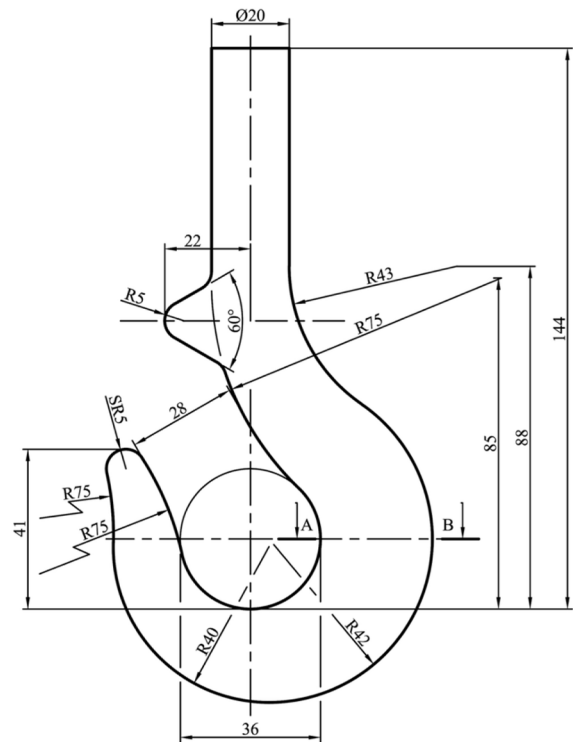


Figure 1. Geometrical details and technical drawing of the crane hook NR025 according to DIN 15401

defined as minimum requirements for the basic series of fine grain structural steel as in DIN 17 102 and DIN 17 103. The material properties of structural steel were defined with a Young's modulus of 200 GPa and a Poisson's ratio of 0.3. These values were initially adopted as default material parameters. Subsequently, an inverse calibration procedure was carried out to identify the effective elastic parameters that minimized the discrepancy between the experimentally measured and numerically predicted strain responses. These calibrated parameters were adopted only for the numerical model used in this study and should not be interpreted as intrinsic material properties.

Finite element analysis setup

The mesh structure of the analysis model was created in the Structural Analysis module of Ansys Workbench 19.2 software. An element size of 1 mm was selected, corresponding to 511,562 nodes and 147,625 elements (Figure 2). A hexahedral-dominant mesh structure was preferred to maintain mesh quality and numerical accuracy in the curved regions of the hook. The refined mesh was selected to capture the localized stress and strain gradients in the curved and experimentally

Table 1. Material properties of the crane hook according to DIN 15400

Hook number	Class	Shank diameter (mm)	Yield strength (MPa)
025	P	20	315

monitored regions, consistent with related numerical studies [23–25]. Although a coarser mesh would reduce the computational cost, it could smooth the local peak values required for the numerical–experimental comparison; nevertheless, the solution time remained manageable for the present static analyses. Since the applied loads generated stresses below the yield strength of the material, the large deformation option was disabled to reduce analysis time.

In the numerical model, the shaft section of the part was not directly considered. Instead, the upper shaft region was fixed along a height of 100 mm to ensure consistency with the experimental setup (Figure 3). For the defined section, zero displacement was applied in all degrees of freedom.

The hook model was loaded from the inner side of the curved surface along the axis passing through the center of the shaft to avoid generating any additional bending moment. The load was applied in 10 steps, increasing linearly up to 10 kN to simulate the tensile test conditions. According to Table 3 of DIN 15400, the DIN 15401 RSN 025 hook with strength class P and drive group 1Bm (M3) has a rated lifting capacity of 630 kg, equivalent to approximately 6.18 kN. The maximum experimental force of 10 kN therefore corresponds to approximately 161.8% of the rated load. This force was selected as a conservative

rounded value slightly above 1.5 times the rated load (9.27 kN). At 10 kN, the maximum von Mises stress was approximately 269 MPa, which remained below the material yield strength of 315 MPa; therefore, the measurements were conducted within the elastic regime.

Although the original load-bearing contact surface and bending radius were preserved during optimization, the effects of flexible wire ropes or webbing slings, friction, nonuniform contact pressure, and cyclic wear were not investigated; therefore, future work should include nonlinear hook–sling contact analyses and corresponding experimental tests under representative operating conditions.

Experimental setup

Strain gauges were employed to measure the surface strains developed on the hook under tensile loading. The gauges operate based on the changes in electrical resistance resulting from structural deformation. Each strain gauge

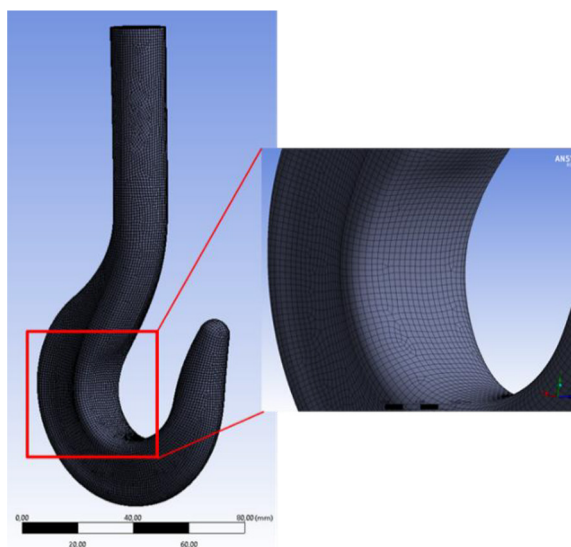

Figure 2. Mesh structure and detail view

Figure 3. Boundary conditions on the hook geometry

consists of a thin metallic foil bonded to the specimen surface using a suitable adhesive and carefully aligned with the selected measurement directions. When the material is subjected to loading, the strain gauge deforms simultaneously with the surface, leading to a change in electrical resistance. To precisely measure this resistance variation and convert it into meaningful strain data, a Wheatstone bridge circuit is required. The wiring configuration on the test specimen must be arranged according to the type of Wheatstone bridge employed. Depending on the measurement objective, Wheatstone bridge circuits are classified as quarter-, half-, or full-bridge configurations. The bridge type was selected based on the capabilities of the data acquisition system and the desired output signal characteristics [26].

Prior to bonding the strain gauges, it is essential that the metal surface be free of contaminants, such as dust, debris, or machining residues. Additionally, any surface coating or paint layer must be removed to ensure accurate strain transfer from the base material to the sensor. After surface preparation, the strain gauges were bonded using an adhesive compatible with the substrate material and securely fixed in place. In the present study, the available measurement area on the outer surface of the hook curvature was relatively limited; therefore, a rosette-type strain gauge was selected. For the hook specimen, the paint coating on the designated sensor locations – namely the inner and outer curved surfaces – was removed by sanding prior to sensor installation (Figure 4a, b). The strain gauges were bonded to the predefined locations using a CC-36 adhesive, which is recommended for the applications within a temperature range of -30°C to 120°C (Figure 4c).

The adhesive ensured direct contact between the strain gauge and the base material, allowing strain transfer from the hook surface to the sensor without the influence of the removed coating layer. Rosette-type strain gauges capable of measuring biaxial strain at a single location were employed. The strain gauge had a nominal resistance of $350.9\ \Omega$ and a gauge length of 2 mm. The strain gauges with higher resistance values are commonly preferred in low-current, high-sensitivity measurement systems. A Kyowa KFGS-2-350 biaxial stacked rosette strain gauge with a $0^{\circ}/90^{\circ}$ configuration was used. The strain gauge layout marked on the hook is shown in Figure 5. The strain gauge positioned on the inner surface of the hook and aligned parallel to the shaft axis was defined as SG_{IA} , representing the inner axial strain gauge. The strain gauge positioned on the inner surface perpendicular to the shaft axis was defined as SG_{IT} , representing the inner transverse strain gauge. Similarly, the gauges bonded to the outer surface of the hook were denoted as SG_{OA} , representing outer axial strain gauge for the gauge aligned parallel to the shaft axis, and SG_{OT} , representing outer transverse strain gauge for the one oriented perpendicular to the shaft axis.

An M24 lifting eye bolt manufactured from C15 steel in accordance with the DIN 580 standard was prepared to mount the hook specimen to the grips of the tensile testing machine (Figure 6a). The tensile testing and data acquisition procedures were conducted at the Mechanical Testing Laboratory of the Integrated Manufacturing Technologies Research and Application Center at Sabancı University. Tensile tests were performed using an Instron 8803 servo-hydraulic testing machine (Figure 6b). The threaded portion of the lifting eye bolt was machined into a

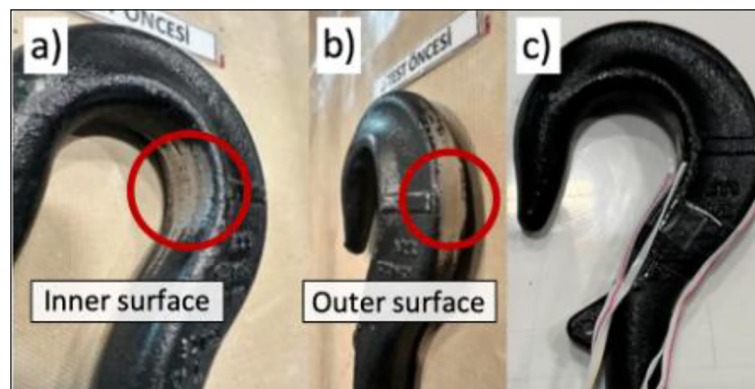


Figure 4. a) Polished inner, b) outer surface of the hook, and c) Installed strain gauges

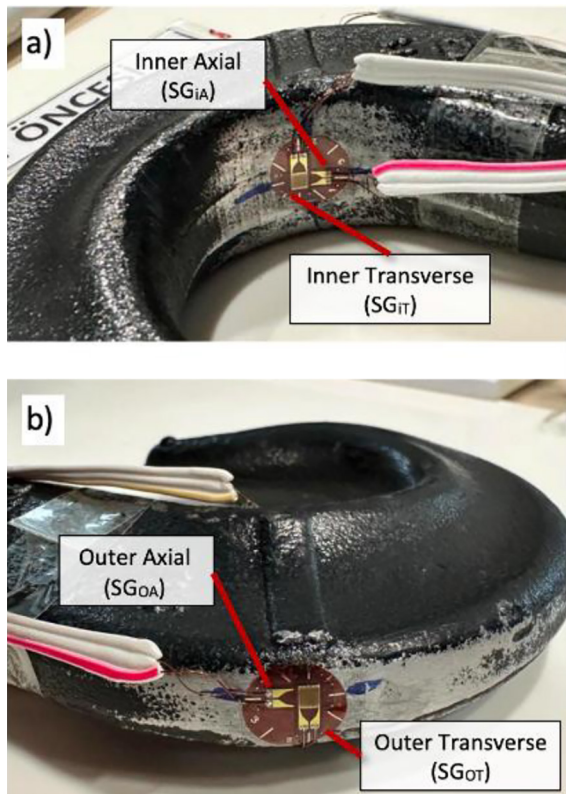


Figure 5. a) Rosette-type strain gauge placement on the inner surface and (b) on the outer surface

smooth shaft with a diameter of 18 mm to ensure proper alignment and secure clamping within the testing machine grips.

The lifting eye bolt and the hook specimen were mounted concentrically in the grips of the testing machine, and the data acquisition connections were then completed. The Wheatstone bridge circuits were configured automatically within the data acquisition system. Each strain gauge had two output terminals; however, three wires were used for each sensor: one for the positive signal, one for the negative signal, and one reference wire. The reference wire was employed to improve measurement stability and reduce the influence of temperature variations.

Data acquisition and processing were carried out using the Instron Bluehill software provided by the tensile testing machine manufacturer. All tests were conducted at a relative humidity of 49.6% and a room temperature of 22.8 °C. To ensure accurate strain measurement, the tensile loading rate was set to 1 mm/min. The applied tensile force was increased incrementally from 0.5 kN up to 10 kN, while the strain values at the designated locations on the specimen were recorded continuously.

NUMERICAL AND EXPERIMENTAL RESULTS

This section presents the numerical and experimental results obtained for the conventional crane hook. First, the stress distribution obtained from finite element analysis was discussed. Then, the experimentally measured strain values were presented and compared with the numerical results. Finally, the material parameters used in the finite element model were evaluated based on the experimental strain measurements.

Stress distribution from FEA

The maximum von Mises stress in the hook was observed on the inner surface of the curved region, reaching 269.4 MPa (Figure 7). This region experiences the highest stress due to the combined effect of bending and axial tensile loading. Consequently, this location represents the most critical region of the hook and corresponds to one of the locations where strain gauges were placed.

As shown in Figure 8, the examined cross-section A-B was taken through the critical curved region of the hook. Along this cross-section, the stress distribution varies due to the bending-dominated response of the curved hook geometry. The outer surface of the section is predominantly subjected to compressive stresses, reaching a von Mises equivalent stress of approximately 116 MPa. In contrast, the regions closer to the mid-surface show a noticeable reduction in stress levels due to the transition between tensile and compressive stress states. As a result, the equivalent stress in these areas decreases to values as low as 31 MPa. The highest stress levels are concentrated near the inner surface of the section, where the von Mises equivalent stress reaches approximately 269 MPa.

Experimental strain measurements

The experimentally measured strain values are presented in Figure 9. Figure 9a illustrates the axial and transverse strain data obtained from the inner strain gauges as the applied load was gradually increased from 0 to 10 kN. As expected from the bending-dominated behavior of the hook, positive axial strain values were observed on the inner surface, indicating tensile deformation, while the transverse strain values were negative,

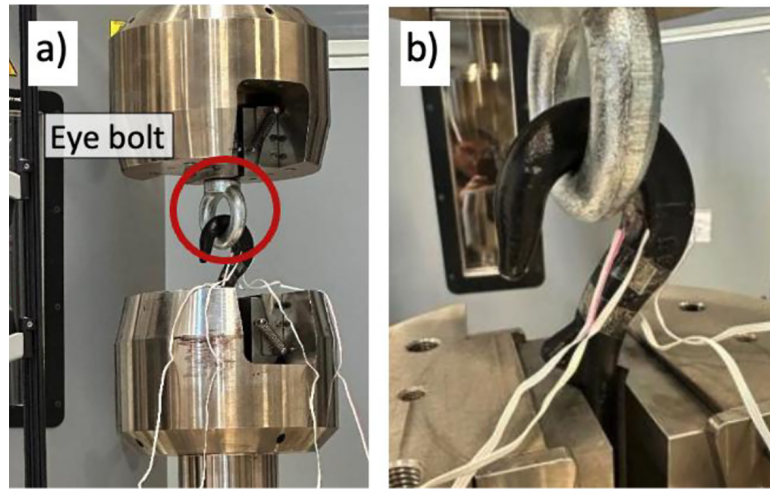


Figure 6. a) General view of the experimental setup and b) close-up view of the hook–eye bolt and lower grip connection

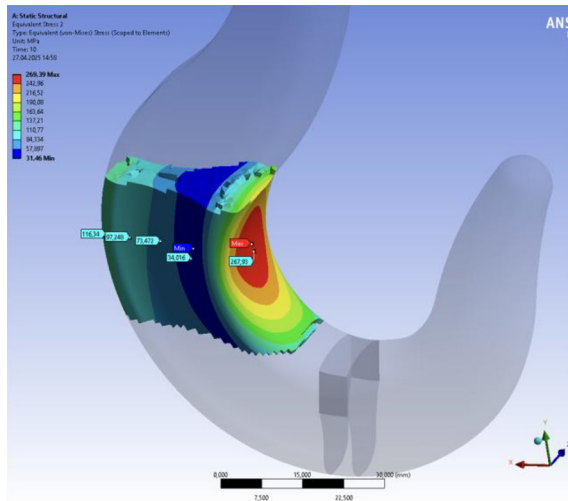


Figure 7. Stress distribution in the critical section

corresponding to the compressive strain induced by the Poisson effect. Similarly, Figure 9b presents the axial and transverse strain measurements obtained from the strain gauges located on the outer surface of the hook. Due to the compressive stress state in the axial direction on the outer surface, negative axial strain values were recorded. In contrast, positive transverse strain values were measured, which can be attributed to the lateral expansion associated with axial compression. These results are consistent with the expected stress–strain distribution of a curved structural member subjected to tensile loading.

The ratio between the transverse strain measured perpendicular to the applied load and the axial strain measured parallel to the applied load was evaluated from the experimental data, as

shown in Figure 10. An apparent Poisson's ratio, calculated as $\nu = -\epsilon_T/\epsilon_A$, was evaluated as 0.328 using the inner strain gauge measurements. When the same ratio was computed using the outer strain gauge measurements, the apparent Poisson's ratio was found to be $\nu = 0.339$. The slight discrepancy between the Poisson's ratio values obtained from the inner and outer strain gauges can be attributed to several factors, including local stress gradients arising from the curved geometry of the hook, non-uniform strain distribution through the cross-section under bending, and minor variations in strain gauge positioning and bonding quality. Additionally, the combined effects of bending and axial loading may lead to localized deviations from ideal uniaxial stress conditions, resulting in small differences in

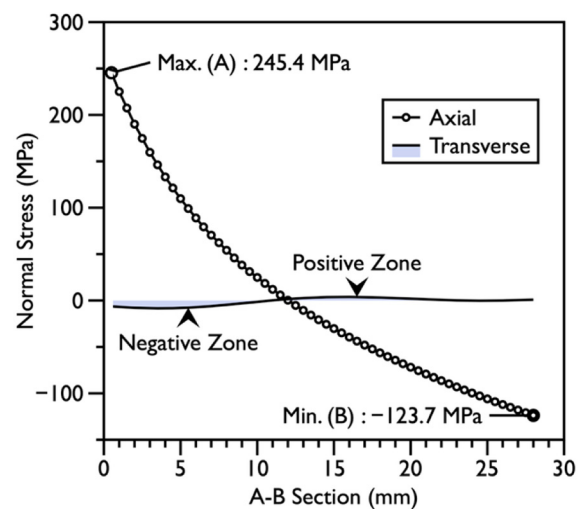


Figure 8. Axial and transverse stresses in the critical section

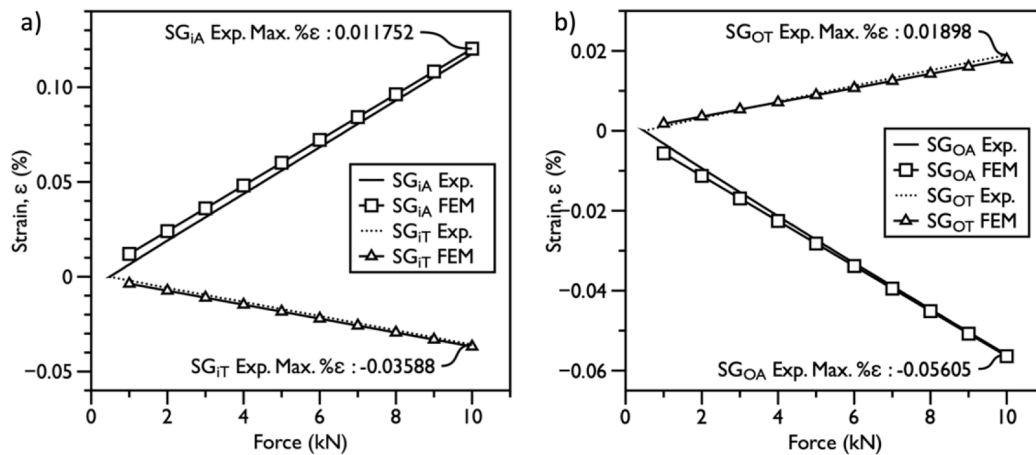


Figure 9. Strain comparison of experimental and finite element analysis for a) Inner strain gauge and b) Outer strain gauge

the experimentally evaluated Poisson's ratio. Such differences may arise from the non-uniform stress state as well as the measurement uncertainties associated with transverse strain, gauge alignment, angular positioning, and low-load offsets [27]. The variation of the evaluated Poisson's ratio as a function of the applied force for both the inner and outer strain gauge measurements is shown in Figure 10. At low load levels, pronounced fluctuations are observed in the Poisson's ratio values for both gauge locations. This behavior can be attributed to initial seating effects, strain gauge stabilization, and signal noise dominance when the strain magnitudes are relatively small.

As the applied force increases, the Poisson's ratio values gradually converge and exhibit a more stable trend. The inner gauge data stabilizes around a higher Poisson's ratio value, whereas the outer gauge data converges to a slightly lower level. This consistent separation between the inner and outer measurements is primarily associated with the non-uniform strain distribution induced by the combined bending and axial loading of the curved hook geometry. The inner surface experiences predominantly tensile axial strains, while the outer surface is subjected to compressive axial strains, leading to different transverse strain responses and consequently distinct Poisson's ratio evaluations. Moreover, local stress gradients across the hook cross-section and deviations from ideal uniaxial stress conditions contribute to the observed discrepancy between the inner and outer Poisson's ratio values. Despite these differences, both datasets remain close to the typical Poisson's ratio range reported for structural steel, indicating the consistency of the experimental measurements.

To improve the agreement between the experimental strain measurements and the numerical results, a series of finite element analyses were conducted by systematically varying the Young's modulus and Poisson's ratio. This parametric approach was adopted to reduce the difference between the experimentally measured strain responses and those predicted by the finite element model. The parametric study was carried out using the Taguchi design of experiments [28] methodology implemented in the Minitab 16 software. A two-factor, three-level experimental design was employed, and the simulations were organized according to an L9 orthogonal array. The signal-to-noise ratio was defined using the "smaller-is-better" criterion, where the response variable was selected as the mean absolute difference between experimental and numerical strain values for each simulation case. The Poisson's ratio levels were selected within the range of 0.30 to 0.34, based on the values experimentally estimated from the strain measurements, which exhibited fluctuations within this interval. The influence of both material parameters on the average strain deviation was subsequently evaluated, enabling the identification of their relative contributions to the model accuracy. The analysis revealed that the minimum average deviation between the experimental and numerical results was obtained when the Young's modulus was set to 220 GPa and the Poisson's ratio to 0.32. This combination was therefore selected as the most suitable material parameter set for the finite element model, providing the lowest average deviation from the experimental observations (Figure 9).

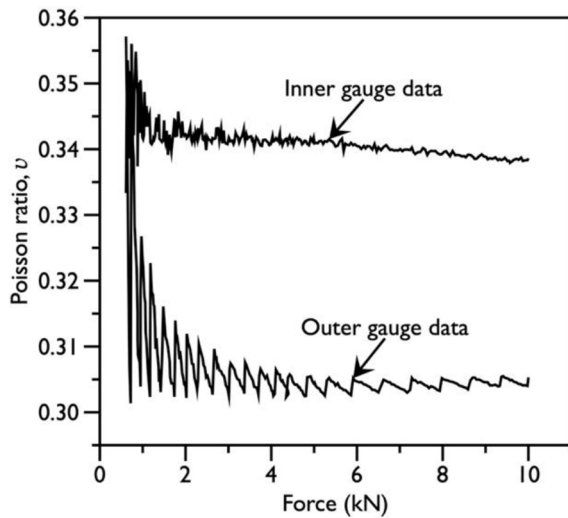


Figure 10. Poisson's ratio–force relationship derived from inner and outer strain gauge data

Initially, the finite element simulations were performed using default material properties commonly adopted for structural steel, namely a Young's modulus of 200 GPa and a Poisson's ratio of 0.30. The strain response obtained under these assumptions was compared with the experimental measurements, and noticeable inconsistencies were observed. Following this parameter refinement, the finite element model employing the updated material properties showed improved correspondence with the experimentally measured strain trends, particularly in terms of the axial strain evolution under increasing load. Obtaining complete agreement between experimental and numerical results is practically unattainable due to several inherent factors. First, the hook geometry in the numerical model was defined using nominal dimensions in accordance with the DIN 15401 standard, whereas the manufactured specimen may include dimensional deviations. Second, the hook was produced using a forging process [29], which may result in material heterogeneity and local variations in mechanical properties. Third, the finite element model inherently relies on mathematical idealizations and simplifying assumptions, which introduce unavoidable differences compared to real structural behavior. Despite these limitations, the comparison results show acceptable agreement for the purposes of the present numerical assessment. The linear relationship observed between strain and applied load in the experimental data indicates that the loading conditions remained within the elastic limits. This finding is consistent with previous finite

element-based crane hook studies, in which the structural safety was evaluated by comparing the maximum von Mises equivalent stress with the material yield strength [30]. Under these conditions, the finite element model, combined with the refined material parameters, provided a suitable basis for the subsequent application of the generative design methodology.

Using the experimentally measured axial and transverse strain values, the corresponding stress components on the inner and outer surfaces of the hook were calculated assuming a linear elastic, isotropic material behavior. The Young's modulus and Poisson's ratio were taken as 220 GPa and 0.32, respectively, based on the refined material parameters obtained from the Taguchi analysis. A plane stress condition was assumed, which is appropriate for surface strain measurements. Under plane stress conditions, the axial and transverse normal stresses can be expressed in terms of the measured strains as:

$$\sigma_A = \frac{E}{1 - \nu^2} (\epsilon_A + \nu \epsilon_T) \quad (1)$$

and

$$\sigma_T = \frac{E}{1 - \nu^2} (\epsilon_T + \nu \epsilon_A) \quad (2)$$

where: σ_A and σ_T are the axial and transverse normal stresses, ϵ_A and ϵ_T are the measured axial and transverse strains, E is the Young's modulus, and ν is the Poisson's ratio. The von Mises equivalent stress was then calculated using:

$$\sigma_{Eq.} = \sqrt{\sigma_A^2 + \sigma_T^2 - \sigma_A \sigma_T} \quad (3)$$

For the outer surface, the measured strain values were $\epsilon_A = -0.05605\%$ and $\epsilon_T = 0.01898\%$. Substituting these values into the constitutive equations yields: $\sigma_A = -122.5$ MPa and $\sigma_T = 2.6$ MPa. The corresponding von Mises stress on the outer surface is: $\sigma_{Eq.} = 123.8$ MPa. The negative axial stress confirms that the outer surface of the hook is predominantly subjected to compressive loading, while the transverse stress remains relatively small.

For the inner surface, the measured strain values were $\epsilon_A = 0.11752\%$ and $\epsilon_T = -0.03588\%$. Using the same formulation, the calculated stresses are $\sigma_A = 259.9$ MPa and $\sigma_T = 4.2$ MPa. The resulting von Mises equivalent stress on the inner

surface is $\sigma_{Eq} = 257.8$ MPa. The experimental and numerical axial and transverse strains are compared in detail in Figure 9, while the corresponding stress components and von Mises equivalent stresses are summarized side by side in Table 2.

The differences between the experimentally derived and numerical axial stresses are approximately 1.0% on the outer surface and 5.6% on the inner surface. The corresponding differences in the von Mises equivalent stresses are approximately 6.3% and 4.5%, respectively, indicating reasonable agreement between the experimental and numerical results. The inner surface is therefore subjected to significantly higher tensile stresses, which is consistent with the bending-dominated loading condition of the hook geometry. The stress calculations indicate that the maximum von Mises stress occurs on the inner surface of the hook, where tensile axial strains dominate. The obtained von Mises stress levels remain below the material yield strength of structural steel (315 MPa), confirming that the applied loading conditions are within the elastic regime. These findings are consistent with the finite element analysis results and support the use of the refined material parameters in the numerical model.

GENERATIVE DESIGN APPROACH

Following the experimental measurements and finite element analyses presented in the previous sections, a generative design study was conducted to investigate the potential for weight reduction and geometric optimization of the crane hook while preserving structural safety. The refined material properties and boundary conditions used in the finite element model were transferred to the generative design environment to ensure consistency between the numerical, experimental, and optimization stages of the study. The generative design workflow was implemented using Autodesk Fusion 360 (2021 version), which provides an integrated cloud-based solution for geometry generation and preliminary structural

evaluation. In this stage, the objective was to reduce mass while maintaining a continuous load path within the hook geometry under the applied loading conditions.

Geometry definition and space partitioning

The baseline geometry of the DIN 15401 NR025 crane hook was imported into Fusion 360 as the reference model. In order to guide the generative design algorithm, the model was divided into three distinct regions: obstacle geometry (red area), preserve geometries (green area), and starting geometry (yellow area). These regions were generated as separate bodies using surface partitioning tools within the CAD environment (Figure 11).

Obstacle geometries define the volumes where material addition is prohibited. In this study, an offset volume of the inner curved surface was defined as an obstacle geometry to ensure that the load-bearing contact surface of the hook remained unchanged. Preserve geometries represent the regions that must remain intact throughout the optimization process; these included the inner curved surface and the shaft regions intended for threaded or guided connections. The starting geometry defines the volume in which the algorithm is allowed to remove or redistribute material. The selection of this region was informed by the finite element results obtained earlier, with particular emphasis on including the zones exhibiting relatively low stress levels under the applied load.

Boundary conditions and parameters

A static load of 630 kg, corresponding to the rated load capacity of the hook, was applied along the shaft axis. The upper shaft region was fully constrained in all degrees of freedom to replicate the experimental and numerical setup. A safety factor of 1.5 was prescribed to ensure structural reliability. Symmetry constraints were imposed with respect to the mid-plane of the hook to maintain manufacturability

Table 2. Experimental and numerical simulation stress comparison

Surface	σ_A (MPa) Exp. / Num.	σ_T (MPa) Exp. / Num.	σ_{Eq} (MPa) Exp. / Num.
Outer	-122.5 / -123.7	2.6 / 0.9	123.8 / 116.0
Inner	259.9 / 245.4	4.2 / -5.8	257.8 / 269.4

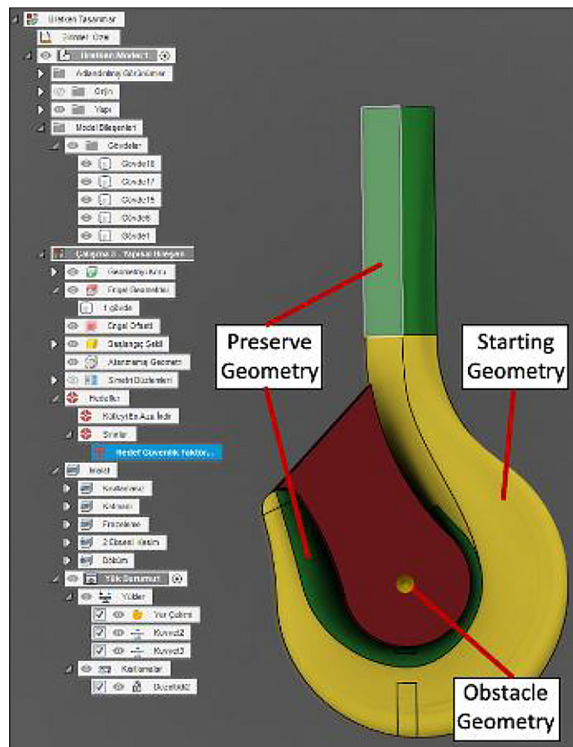


Figure 11. Three regions of the body

and consistency with the original geometry. No manufacturing method was explicitly enforced; therefore, the software was allowed to propose the designs suitable for additive manufacturing, machining, casting, or unrestricted fabrication. Material properties were defined based on the refined parameters obtained from the experimental–numerical comparison. The Young’s modulus and Poisson’s ratio were set to 220 GPa and 0.32, respectively, while the yield strength was specified as 315 MPa in accordance with the DIN 15400 standard. These properties were actively used by the algorithm during both geometry generation and internal stress evaluations.

Generative design alternatives and comparative evaluation

The generative design process resulted in five alternative geometries. All alternatives are presented in Figure 12. Due to their similar structural characteristics and stress responses, the first four alternatives were evaluated collectively, while the fifth alternative was discussed separately as a refined design.

The first alternative (Figure 12a) exhibits material removal across the thickness just above the critical A–B cross-section. This region corresponds to a low-stress zone identified in the finite element results, where compressive stresses induced by bending partially counteract normal tensile stresses. Fusion 360 suggested additive manufacturing for this geometry due to internal voids, with an estimated mass of 371.5 g, corresponding to approximately 42% weight reduction compared to the original design. The reported maximum von Mises stress was approximately 210 MPa. The second alternative (Figure 12b) was generated under unrestricted manufacturing conditions. In this design, the width near the outer surface of the A–B cross-section was reduced by approximately 40%, and the preserved inner surface was connected to the modified region through a wavy transition. This alternative achieved the highest weight reduction, approximately 46.5%, with an estimated mass of 342.9 g. Fusion 360 again reported a maximum von Mises stress of 210 MPa. A detailed finite element analysis was performed on the second alternative to check the stress level predicted by the generative design environment. The ANSYS Workbench structural analysis revealed a maximum von Mises stress of 281.67 MPa, which is approximately 33% higher than the value reported by Fusion 360 but still

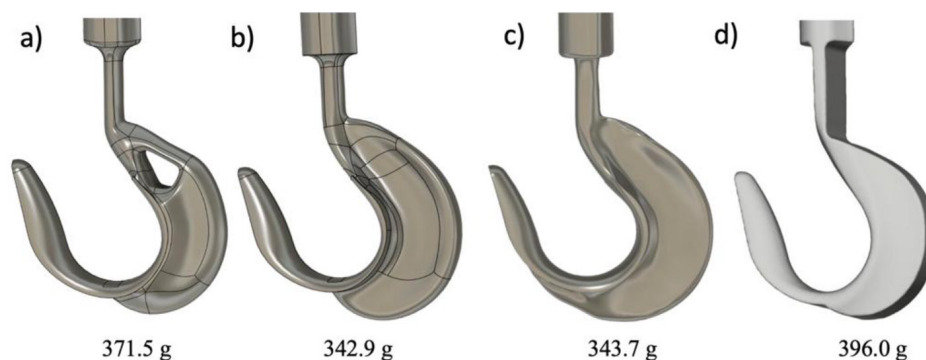


Figure 12. Alternative geometries obtained from the generative design process: (a) additive manufacturing, (b) unrestricted manufacturing, (c) die casting, and (d) two-axis machining

below the material yield strength of 315 MPa. Fusion 360 automatically generated the mesh during the generative design procedure, and its explicit element type as well as numerical mesh size were not available. This represents a limitation of the study and may have contributed to the difference between the Fusion 360 and ANSYS stress results. Therefore, the Fusion 360 value was treated as a preliminary estimate. This discrepancy emphasizes the necessity of checking generative design results using detailed finite element analyses for safety-critical components. The third alternative (Figure 12c) is geometrically similar to the second but was optimized for die casting. Feeders were incorporated to facilitate molten metal flow during casting. The estimated mass was 343.7 g, corresponding to a weight reduction of approximately 46.3%. As with the previous alternatives, Fusion 360 reported a maximum von Mises stress of approximately 210 MPa. The fourth alternative (Figure 12d) differs more significantly from the previous designs, as it was optimized for two-axis machining. In this configuration, the material reduction from the inner to the outer surface was implemented in a more linear manner, and the shaft geometry was adapted accordingly. The estimated mass was 396 g, corresponding to a weight reduction of approximately 38.1%, the lowest among the first four alternatives. The second alternative was selected as the basis for further revision because it provided the highest weight reduction (approximately 46.5%) and was generated without imposing a specific manufacturing process. In addition, its maximum von Mises stress obtained from the detailed ANSYS analysis remained below the material yield strength. Thus, it offered the most promising balance between mass reduction and static structural performance, although the abrupt transition near the free end required further geometric refinement.

During the evaluation of these alternatives, it was observed that the generative design algorithm removed all material between the free end of the hook and the cross-section passing through the central axis of the hook shaft, leading to relatively sharp geometric transitions. To mitigate the potential stress concentration effects and account for uncertainties in operational loading, the preserve geometry was revised (Figure 13).

Following this revision, a fifth alternative was generated (Figure 14). This design exhibits a smoother transition between the free end and the

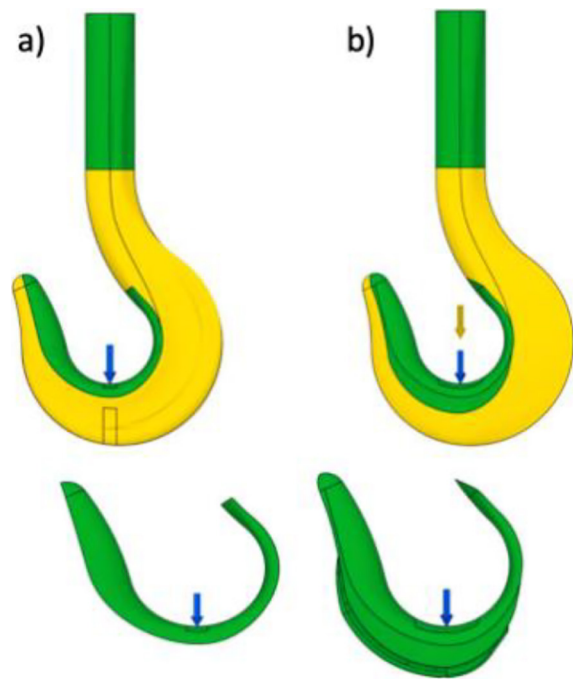


Figure 13. Revised preserve geometry

cross-section passing through the central axis of the hook shaft and can be considered a reinforced version of the second alternative.

The estimated mass was 387.4 g, corresponding to a weight reduction of approximately 39.4%. The manufacturing type remained unrestricted, the safety factor was 1.5, and Fusion 360 reported a preliminary maximum von Mises stress of 210 MPa during the generative design stage. The safety factor of 1.5 was adopted as a preliminary static optimization constraint, corresponding to a target von Mises stress of 210 MPa. It was not intended as a universal standard safety factor for lifting hooks. Since the revised geometry was mainly intended to improve geometric continuity near the free end and the section through to the central shaft axis, the final design was evaluated in terms of mass reduction, manufacturability, and detailed finite element stress response. Moreover, the geometry remained symmetric and free of severe undercut surfaces, which makes it more suitable for conventional manufacturing processes after minor draft angle and fillet modifications. A detailed finite element analysis of the final geometry was also performed in ANSYS Workbench to check the stress response of the selected design candidate. The maximum axial stress along the A–B section of the final geometry was approximately 277 MPa, which is close to the corresponding value of the original geometry,



Figure 14. Final geometry obtained from generative design

approximately 265 MPa. This result indicates that the final geometry provides a comparable stress response in the critical section while achieving approximately 39.4% weight reduction.

From a manufacturability point of view, the final geometry is more favorable than the first four alternatives because it avoids abrupt material removal between the free end of the hook and the section through to the central axis. However, the wavy transitions generated by the algorithm may still require local smoothing, fillet addition, and draft angle modification before manufacturing. These regions should be carefully examined, since small local stress concentrators may occur at sharp transitions, internal corners, or sudden thickness changes. Therefore, the proposed geometry should be considered as a structurally promising design candidate, rather than a directly manufacturable final product. Detailed finite element checks remain necessary, since the stress values reported by the generative design environment should be treated as preliminary design-stage estimates.

Discussion of generative design limitations

Although generative design provided effective guidance for material redistribution and weight reduction, certain limitations should be emphasized. The stress values obtained from the generative design environment are based on simplified numerical assumptions and therefore require checking through a detailed finite element analysis. In addition, the optimization results were found to be highly sensitive to the definition of preserve and obstacle regions; without proper constraints, the algorithm may produce

geometries with abrupt transitions, as observed in Figure 12a-d, which may not be suitable for safety-critical applications. Furthermore, fatigue behavior and long-term service conditions are not explicitly considered within the generative design framework. Manufacturing constraints also remain a challenge, since some optimized geometries implicitly favor additive manufacturing. Therefore, generative design should be regarded as a conceptual optimization tool that should be supported by detailed finite element analyses, manufacturability assessment, and experimental evaluation before practical use in safety-critical lifting components.

Generative design should be regarded as a constrained design-exploration method, rather than a stand-alone structural qualification procedure. Buonamici et al. [31] demonstrated that the alternatives generated by Autodesk Fusion 360 depend strongly on the prescribed loads, objectives, design space, preserve regions, and manufacturing settings. Generative design also differs from conventional topology optimization by exploring multiple solutions that may simultaneously account for structural and manufacturing requirements [32]. This sensitivity was observed in the present study: modifying the preserve geometry eliminated abrupt transitions, whereas different manufacturing constraints produced substantially different geometries and mass reductions. Previous manufacturing-oriented optimization studies have emphasized that minimum feature size, tool accessibility, casting and ejection requirements, build orientation, overhangs, and support structures should be incorporated during optimization, rather than addressed solely through post-processing [33,34]. These

considerations are particularly important for safety-critical components. Complex optimized geometries may also reduce inspectability, while their functional qualification generally requires process-specific material data, non-destructive inspection, independent numerical verification, and physical testing [35,36]. Therefore, the present optimized hook should be regarded as a preliminary design candidate requiring fatigue, contact, manufacturing, and experimental qualification before industrial implementation.

CONCLUSIONS

This study investigated the integration of experimental strain measurements, finite element analysis, and generative design optimization for improving crane hook designs. Starting from a conventional hook geometry modeled in accordance with the DIN 15401 standard, experimental strain measurements were used to evaluate the strain response of the hook and refine the material parameters adopted in the numerical model. The subsequent generative design process, carried out under defined constraints, preserved regions, and boundary conditions, produced alternative geometries aimed at reducing mass while maintaining stresses below the material yield strength. The final optimized crane hook achieved a weight reduction of approximately 39.4% relative to the classical design, while the detailed finite element check showed a comparable axial stress level along the critical A–B section. Moreover, the final geometry provided smoother material continuity than the initial generative design alternatives, reducing the risk of abrupt geometric transitions. These results indicate the potential of generative design for developing lightweight crane hook geometries when supported by detailed finite element assessment and manufacturability considerations. Overall, the findings suggest that combining experimental strain evaluation with finite element analysis and generative design can provide a useful workflow for the structural assessment and redesign of safety-critical lifting components. The proposed approach can also be extended to other load-bearing mechanical elements, contributing to material efficiency and structural performance in industrial lifting applications. Because the optimization was based solely on static strength criteria, the fatigue life

of the mass-reduced design remains unverified; therefore, future work should assess its durability under representative variable-amplitude loading using validated material fatigue data and, where possible, experimental testing. Although the optimized design provides a 39.4% reduction in material usage, its economic feasibility remains dependent on the selected manufacturing process, production volume, tooling requirements, and post-processing costs; therefore, a manufacturing-specific break-even analysis should be conducted before industrial implementation.

Acknowledgments

This work was supported by Scientific Research Projects Department of Istanbul Technical University. Project Number: MGA-2023-44600.

REFERENCES

- Öztepe H. Material Handling Technology: Lifting and Conveying Machinery (in Turkish). İstanbul: Eğitim Yayınları Web Ofset Tesisleri; 1999.
- Desai N, Zeytinoglu N. Design and optimization of the geometric properties of a crane hook. *World Journal of Engineering and Technology* 2016;4:391–7. <https://doi.org/10.4236/wjet.2016.43038>
- Dandavate K, Wagh R, Kale P, Pingale A. Design and analysis of crane hook with trapezoidal cross-section: A study on static structural and modal analysis. *Journal of Mines, Metals and Fuels* 2025;V:807–14. <https://doi.org/10.18311/jmmf/2025/47426>
- Pavlović G, Savković M, Zdravković N, Marković G, Stanojković J. Analysis and optimization of T-cross section of crane hook considered as a curved beam. *IMK-14 - Istrazivanje i Razvoj* 2018;24:51–8. <https://doi.org/10.5937/IMK1802051P>
- Krishnaveni MNV, Reddy MA, Roy MR. Static analysis of crane hook with t-section using Ansys. *International Journal of Engineering Trends and Technology* 2015;25:53–8. <https://doi.org/10.14445/22315381/IJETT-V25P209>
- Shrestha B, Bhandari A, Poudel S, Rao MrK V. Crane hook analysis for different cross-section using ANSYS. *International Journal of Advances in Scientific Research and Engineering* 2020;5:67–73. <https://doi.org/10.31695/ijasre.2019.33644>
- Zade MR. Finite element analysis and fatigue analysis of crane hook with different materials. *International Research Journal of Engineering and Technology (IRJET)* 2017;4:1429–36.

8. Düzenli S, Güney T, Soyaslan M. Structural analysis of a rope slewing system for loads with a variable center of gravity. *European Mechanical Science* 2025;9:78–86. <https://doi.org/10.26701/EMS.1629942>
9. DIN 15400:1990-06, Lasthaken für Hebezeuge; Mechanische Eigenschaften, Werkstoffe, Tragfähigkeiten und vorhandene Spannungen. Berlin: 1990. <https://doi.org/10.31030/2350245>
10. DIN 15401-2:1983-09, Lasthaken für Hebezeuge; Einfachhaken; Fertigteile mit Gewindenschaft. Berlin: 1983. <https://doi.org/10.31030/1155040>
11. Onur YA. Computer aided lifting hook modeling and stress analysis. *Erzincan Üniversitesi Fen Bilimleri Enstitüsü Dergisi* 2018;11:231–6. <https://doi.org/10.18185/erzifbed.371751>
12. Dipu MNH, Apu MH, Chowdhury PP. Identification of the effective crane hook's cross-section by incorporating finite element method and programming language. *Heliyon* 2024;10:e29918. <https://doi.org/10.1016/J.HELİYON.2024.E29918>
13. Dragoni E. Designing the cross-section of curved beams for equal magnitude of peak bending stresses. *J Strain Anal Eng Des* 2001;36:473–9. <https://doi.org/10.1243/0309324011514638>
14. Hiep VD, Quynh NX, Tung TT. A topology optimization design of a crane hook. *Results in Engineering* 2024;23:102492. <https://doi.org/10.1016/J.RINENG.2024.102492>
15. Kishore K, Sanjay Gujre V, Choudhary S, Sanjay Gujre A, Vishwakarma M, Thirumurgan T, et al. Failure analysis of a 24 T crane hook using multi-disciplinary approach. *Eng Fail Anal* 2020;115:104666. <https://doi.org/10.1016/j.engfailanal.2020.104666>
16. Sahu N, Yadav N. Design and stress analysis of various cross section of hook. *International Journal of Modern Engineering Research (IJMER)* 2013;3:2187–9.
17. Narvydas E, Puodžiuniene N. Circumferential stress concentration factors at the asymmetric shallow notches of the lifting hooks of trapezoidal cross-section. *Mechanika* 2012;18:152–7. <https://doi.org/10.5755/j01.mech.18.2.1574>
18. Onur YA, Özdemir A. Theoretical and experimental investigation of stress distribution in a crane hook. *Materials Testing* 2019;61:573–8. <https://doi.org/10.3139/120.111357>
19. Cazzani A, Malagù M, Turco E, Stochino F. Constitutive models for strongly curved beams in the frame of isogeometric analysis. *Mathematics and Mechanics of Solids* 2016;21:182–209. <https://doi.org/10.1177/1081286515577043>
20. Manee-Ngam A, Saisirirat P, Suwankan P. Hook design loading by the optimization method with weighted factors rating method. *Energy Procedia* 2017;138:337–42. <https://doi.org/10.1016/j.egypro.2017.10.132>
21. Teke IT, Akbulut M, Ertas AH. Topology optimization and fatigue analysis of a lifting hook. *Procedia Structural Integrity* 2021;33:75–83. <https://doi.org/10.1016/j.prostr.2021.10.011>
22. Singh A, Rohilla V. Optimization and fatigue analysis of a crane hook using finite element method. *International Journal of Recent Advances in Mechanical Engineering* 2015;4:31–43. <https://doi.org/10.14810/ijmech.2015.4403>
23. Hacıoglu A, Candas A, Baykara C. Large deflections of functionally graded nonlinearly elastic cantilever beams. *J Eng Mater Technol* 2023;145:1–7. <https://doi.org/10.1115/1.4056034>
24. Topal T, Candaş A, Kalay E, İmrak CE. Thermo-mechanical behavior of IN718 bolted joints in aerospace propulsion at elevated temperatures. *Scientific Journal of Silesian University of Technology Series Transport* 2026;131:207–28. <https://doi.org/10.20858/sjsutst.2026.131.13>
25. Karakoca MR, Candaş A, İmrak CE. Topology optimization-based analysis and redesign of a belt conveyor bracket. *Gazi University Journal of Science Part C: Design and Technology* 2026;14:666–76. <https://doi.org/10.29109/gujsc.1858575>
26. Candaş A, Kurt S, Gerdemeli İ, Kayaoğlu E. An experimental stress analysis of a jib crane. *Key Eng Mater* 2014;572:173–6. <https://doi.org/10.4028/www.scientific.net/KEM.572.173>
27. Gabauer W. UNCERT COP 16: 2000 Manual of Codes of Practice for the Determination of Uncertainties in Mechanical Tests on Metallic Materials The Determination of Uncertainties of Poisson's Ratio (from a Tension Test). 2000.
28. Kwon KS, Rong-Ming L. Robust finite element model updating using Taguchi method. *J Sound Vib* 2005;280:77–99. <https://doi.org/10.1016/J.JSV.2003.12.013>
29. Chandran V, Manvel Raj T, Lenin N, Subramanian A. Mechanical investigation of crane hook using AISI 4340 alloy steel with various proportions of vanadium. *Journal of Automation and Automobile Engineering* 2019;4:1–6.
30. Sahu S, Dewangan R, Patnaik M, Yadav N. Study of crane hook having trapezoidal section by finite element method & design of experiments. *International Journal of Modern Engineering Research (IJMER)* 2012;2:2779–81.
31. Buonamici F, Carfagni M, Furferi R, Volpe Y, Governi L. Generative design: An explorative study. *Comput Aided Des Appl* 2020;18:144–55. <https://doi.org/10.14733/cadaps.2021.144-155>
32. Srivastava J, Kawakami H. Systematic review of difference between topology optimization and generative

- design. IFAC-PapersOnLine 2023;56:6561–8. <https://doi.org/10.1016/j.ifacol.2023.10.307>
33. Liu J, Ma Y. A survey of manufacturing oriented topology optimization methods. *Advances in Engineering Software* 2016;100:161–75. <https://doi.org/10.1016/j.advengsoft.2016.07.017>
34. Thompson MK, Moroni G, Vaneker T, Fadel G, Campbell RI, Gibson I, et al. Design for additive manufacturing: Trends, opportunities, considerations, and constraints. *CIRP Annals* 2016;65:737–60. <https://doi.org/10.1016/J.CIRP.2016.05.004>
35. Mahan T, Katch L, Arguelles AP, Menold J. Design for inspectability: A framework to increase inspectability of additive manufacturing parts for pulse-echo ultrasonic inspection methods. *Journal of Mechanical Design* 2022;144. <https://doi.org/10.1115/1.4053812>
36. Biehler M, Mock R, Kode S, Mehmood M, Bhardwaj P, Shi J. AUDIT: Functional qualification in additive manufacturing via physical and digital twins. *J Manuf Sci Eng* 2024;146. <https://doi.org/10.1115/1.4063655>