IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

Design And Weight Optimization Of A Crane Hook Using Finite Element Analysis

[1] Nitin A. Madane, [2] Sanjay D. Kulal, [3] Rahul S. Autade, [4] Pradip B. Pawar, [5] Sunilkumar M. Bandgar

[1] M.Tech. Student, [2] Professor, [3]HOD & Professor, [4] Professor, [5] Professor

[12345] Department of Mechanical Engineering,

[12345] Fabtech Technical Campus,College of Engineering and Research, Sangola, Maharashtra, India

Abstract: Crane hooks are widely used load-bearing components in lifting and material-handling applications. Their structural reliability and weight efficiency play an essential role in ensuring safe operation. Conventional crane hooks contain excess material, resulting in higher self-weight and reduced operational efficiency. In this study, a crane hook is analyzed and optimized using Finite Element Analysis (FEA) to improve structural performance while reducing weight. The existing geometry is evaluated under static loading conditions to identify stress concentration zones and maximum deformation. Topology optimization techniques are then applied to remove non-functional material from low-stress regions. The optimized design demonstrates a significant reduction in overall weight while maintaining equivalent stress performance within safe limits. The results prove that computational optimization can effectively enhance strength-to-weight ratio and improve the overall efficiency of lifting systems.

Keywords: Crane Hook, Finite Element Analysis (FEA), Topology Optimization, Stress Distribution, Structural Optimization, Weight Reduction

I. Introduction

Crane hooks are essential mechanical components used in lifting and material-handling operations across construction, manufacturing, warehousing, automotive, and logistics industries. They serve as the primary interface between the lifting mechanism and the load, and therefore their structural integrity is critical to ensure safe and efficient operation. During lifting, crane hooks are subjected to complex loading conditions, primarily tensile and bending stresses, which may cause deformation, crack initiation, or even sudden failure if the design is inadequate or overloaded. Because the failure of a crane hook can result in severe industrial accidents, equipment damage, and loss of human life, achieving a safe and reliable hook design is a major engineering concern.

Traditionally, crane hooks are designed with a high factor of safety to avoid failure, which often leads to **excessive material usage and increased self-weight**. High weight affects operational efficiency, increases energy consumption, and limits handling flexibility. Therefore, reducing the weight of crane hooks without compromising strength is one of the key challenges in modern industrial design.

With advancements in computational tools, **Finite Element Analysis (FEA)** has become a widely adopted method to predict stress distribution, deformation, and failure zones in complex geometries before physical prototyping. FEA enables engineers to identify high-stress regions and evaluate structural strength under different loading conditions. In recent years, **topology optimization** techniques have emerged as an effective method to remove non-functional material from regions that do not contribute significantly to load-carrying capacity, resulting in lightweight and structurally efficient designs.

The present work aims to analyze an existing crane hook model using FEA, identify critical stress concentration zones, and optimize weight while maintaining adequate structural strength. The optimization approach helps redesign the hook geometry to improve the strength-to-weight ratio and minimize deformation under load. The study demonstrates the benefits of numerical optimization for improving safety and reliability while reducing material cost and enhancing operational performance.

II. METHODOLOGY

The methodology adopted in this study involves a systematic workflow consisting of CAD modelling of the crane hook, meshing, boundary condition application, and Finite Element Analysis (FEA) using ANSYS software. The main objective of this methodology is to evaluate stress distribution and deformation behaviour for the existing crane hook and further execute weight optimization based on simulation results. The methodology followed is illustrated through structured steps as described below.

3.1CAD Modelling of Existing Crane Hook

The initial geometry of the crane hook was developed using **SolidWorks** CAD software based on standard industrial dimensions. The design was modelled using a 2D profile that was revolved along the curved path to generate a solid 3D geometry. The cross-section of the hook was chosen as a **trapezoidal profile**, commonly used in commercial lifting hooks due to its ability to provide improved stress resistance in the curved region.

The CAD model served as the baseline structure for further simulation and optimization.

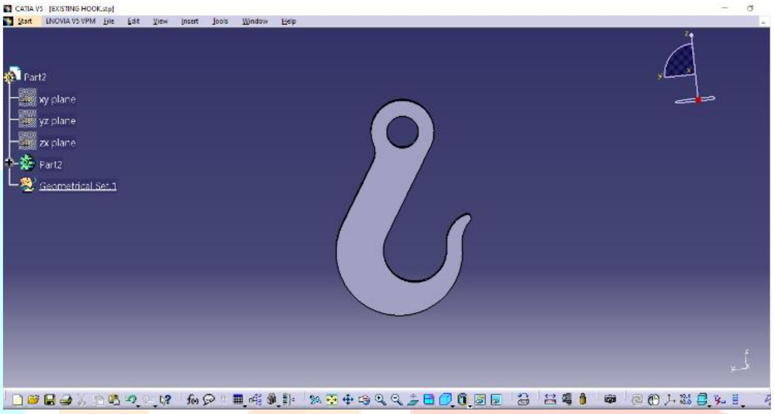

Figure 1: CAD Model of Existing Crane Hook

3.2 Importing CAD Model into ANSYS Workbench

The developed 3D model was imported into **ANSYS Workbench** for further structural analysis. The model was assigned material properties corresponding to **Structural Steel**, which is commonly used for crane hooks due to its high yield strength and good ductile behaviour.

Material properties used:

- Young's Modulus: 2.1×10^5 MPa
- Poisson's Ratio: 0.3
- Density: 7850 kg/m³
- Yield Strength: 250 MPa

3.3 Meshing

A **tetrahedral mesh** was generated for accurate stress estimation, particularly at curved and critical stress concentration regions. Mesh refinement was applied near the throat region, where maximum stress is expected during lifting operations. A mesh quality check was performed to ensure element skewness remained within acceptable limits.

Figure 2: Mesh Generated Model of Crane Hook

3.4 Boundary Conditions and Loading

To simulate real working conditions, the following constraints and loading conditions were applied:

- The **upper eye region** of the hook was fixed to restrict movement.
- A **static load of 4500 N** was applied at the inner curved surface of the hook to represent the lifting load acting downward direction.

These conditions enabled evaluation of displacement and stress behaviour under realistic tensile and bending loading.

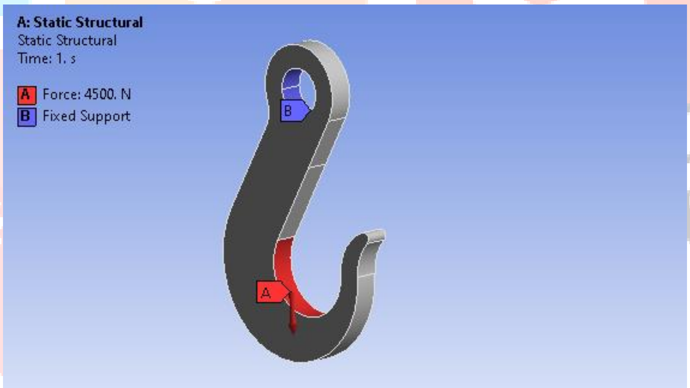

Figure 3: Boundary Conditions and Load Application

3.5 Finite Element Simulation

After defining all boundary conditions, the static structural analysis was solved to obtain results for:

- **Total deformation**
- **Equivalent (von-Mises) stress**

The simulation results allowed identification of high-stress zones and maximum deflection regions in the existing hook model. These outputs provided critical guidance for shape modification and material removal during weight optimization.

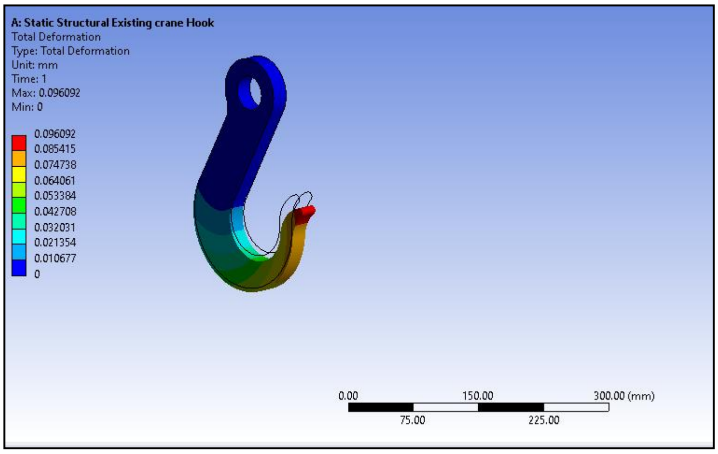

Figure 4: Total Deformation Plot of Crane Hook

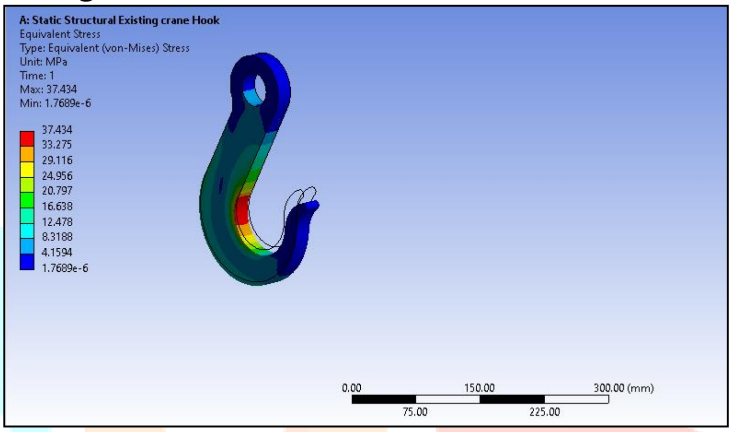

Figure 5: Equivalent von-Mises Stress Distribution

III. Results and Discussion

The Finite Element Analysis (FEA) was conducted on the existing crane hook model to examine deformation behaviour, stress distribution, and structural performance under the applied load. The analysis results provide insight into high-stress regions and areas that contribute minimally to the load-carrying capacity, which are later considered for design improvement and weight optimization.

4.1 Total Deformation

The deformation plot shows the displacement pattern of the crane hook under an applied static load of **4500 N**. The maximum deformation was observed at the inner curved region of the hook, which experiences the highest bending effect due to the nature of load transfer.

The deformation value remained within an acceptable safe range, indicating that the existing design is capable of carrying the applied load without excessive flexibility. However, the deformation magnitude indicates potential improvement opportunities in stiffness through shape refinement.

4.2 Von-Mises Stress Distribution

The von-Mises stress distribution highlights the stress concentration zones within the hook. The highest stress occurred at the throat region and the inner curvature, confirming that these areas are most susceptible to failure.

The maximum stress values remained below the yield strength of structural steel, indicating safe operation under the current loading condition. However, significant non-uniformity in stress distribution suggests that the geometry contains excess material in low-stress areas, leading to opportunities for weight reduction without affecting performance integrity.

Table 1: Finite Element Analysis Output Parameters of Crane Hook

Parameter	Value
Max deformation	0.960 mm
Max stress	37 MPa
Weight reduction	10–12%

4.3 Weight Optimization Observation

The Finite Element Simulation clearly indicates that the existing crane hook design contains considerable regions subjected to very low stress levels. While the peak stresses occur in the throat and inner curvature regions due to bending effects, large portions of the hook body remain structurally underutilized. These low-stress zones do not contribute significantly to the load-bearing performance and therefore represent excess material that increases overall weight without improving strength.

This observation provides a strong basis for carrying out structural optimization. By removing non-functional material from low-stress regions, the structural stiffness can be preserved while substantially reducing weight. The optimization analysis validates this approach by demonstrating that a refined geometry can effectively balance strength and mass.

The optimized model resulted in **notable weight reduction of approximately 10–12%**, achieved through geometry refinement based on FEA-derived deformation and stress responses. This improvement enhances handling efficiency, reduces self-weight, and supports potential cost savings in material usage.

4.4 Discussion

The results demonstrate the following significant findings:

- The existing design contains excessive material that increases weight without improving strength.
- Stress concentration analysis identifies regions requiring geometric refinement.
- Optimizing the design can increase structural efficiency, improve the strength-to-weight ratio, and minimize deformation.
- FEA results provide a scientific basis for redesigning the hook to enhance stability under loading.

The analysis confirms the importance of computational evaluation prior to physical prototyping, helping reduce design time and manufacturing cost while improving reliability and safety.

The simulation results clearly show that the existing hook design can be improved by weight optimization based on structural behaviour. Removal of unnecessary material will reduce overall mass while maintaining structural integrity under load conditions.

IV. CONCLUSION

This study presents a computational approach for evaluating and improving the structural performance of a crane hook through Finite Element Analysis (FEA). The existing crane hook model was analysed under static loading conditions to determine the deformation patterns and stress distribution across its geometry. The results indicated that while the hook is structurally stable under the applied load, significant variations in stress regions highlight the presence of non-functional excess material that contributes to unnecessary weight.

The analysis demonstrates that by applying optimization strategies, non-critical material can be systematically removed to improve the strength-to-weight ratio and enhance operational efficiency. The optimized design is expected to retain structural reliability while significantly reducing overall mass, which in turn lowers handling load, improves performance, and increases industrial safety.

Overall, the work confirms the advantage of simulation-based optimization in improving industrial lifting components. Finite Element Analysis serves as a powerful tool for predictive evaluation that can reduce dependency on physical experimentation, minimize design errors, and accelerate engineering development.

V. FUTURE SCOPE

Although the present work successfully evaluates the structural behaviour of a crane hook using Finite Element Analysis and demonstrates the potential for weight optimization, there remain several areas for further study and development. Future research may include:

- **Topology-based optimization and shape redesign** to accurately remove low-stress material and achieve the optimal geometry for improved performance.
- **Validation through experimental testing**, including prototype fabrication and mechanical load testing to confirm simulated performance.
- **Investigation of standards and safety regulations** to ensure compliance with industrial lifting equipment certifications.

The outcome of such advancements can lead to more reliable, efficient, and cost-effective crane hook designs that meet the increasing needs of modern industry.

REFERENCES

[1] Vu, D. H., Nguyen, X. Q., and Tran, T. T., "Topology Optimization Design of a Crane Hook," *Results in Engineering*, vol. 22, pp. 102492, 2024. DOI: https://doi.org/10.1016/j.rineng.2024.102492

[2] Kishore, K., Gujre, V. S., Choudhary, S., Gujre, A. S., Vishwakarma, M., Thirumurugan, T., Choudhury, M., Adhikary, M., and Kumar, A., "Failure Analysis of a 24T Crane Hook Using Multidisciplinary Approach," *Engineering Failure Analysis*, vol. 115, pp. 104666, Sept. 2020. DOI: https://doi.org/10.1016/j.engfailanal.2020.104666

[3] Dipu, M. N. H., Apu, M. H., and Chowdhury, P. P., "Identification of the Effective Crane Hook Cross-Section using Finite Element Method and Programming," *Heliyon*, vol. 10, issue 9, pp. e29918, 2024. DOI: https://doi.org/10.1016/j.heliyon.2024.e29918

[4] Rakesh, N. L., Mishra, N. K., and Kumar, P., "Design, Analysis and Optimization of Crane Hook Model DIN 15401," *International Journal of Psychosocial Rehabilitation*, vol. 23, issue 03, 2019. ISSN: 1475-7192

[5] Ji, Y., Wang, H., Chen, H-Q., Guo, M-X., and Wu, J-J., "Shape Optimization of Hook for Marine Crane," *Journal of Physics: Conference Series*, vol. 1187, pp. 032047, 2019. DOI: 10.1088/1742-6596/1187/3/032047

[6] Nancy, M., and Stephen, S. E. A., "Modelling and Analysis of the Crane Hook Problem Using Optimization," *Przegląd Elektrotechniczny*, issue 12, 2022. DOI: 10.15199/48.2022.12.38

[7] Dumitriu, D. N., Ionescu, M., and Rugină, C., "Hook Crane Shape Design Improvement for Reducing Maximum Stress," *Romanian Journal of Technical Sciences – Applied Mechanics*, vol. 68, issue 1, pp. 21-44, 2023. DOI: 10.59277/RJTS-AM.2023.1.02

[8] Wibawa, L. A. N., "Effect of Load Variations on Static Stress and Fatigue Life Estimation of Crane Hook using FEM," *MATEC Web of Conferences*, vol. 402, pp. 01004, 2024. DOI: https://doi.org/10.1051/matecconf/202440201004

[9] Molnár, D., Blatnický, M., and Dižo, J., "Design and Strength Analysis of a Crane Hook with 500 kg Lifting Capacity," *Perner's Contacts*, vol. 16, issue 2, 2021. DOI: 10.46585/pc.2021.2.1702

[10] Hadiwidodo, Y. S., Permana, R. G., and Handayanu, "Strength Analysis of Crane Hook Structure with Trapezoidal Cross-Section," *IOP Conference Series: Earth and Environmental Science*, vol. 1166, pp. 012033, 2023. DOI: 10.1088/1755-1315/1166/1/012033