

Static Structural Performance Evaluation of a Workshop Hoist Using ANSYS Mechanical

Tilak Bohra¹, Dipak P Kharat², Santosh R. Shekokar³, Yugesh A. Kharche⁴,
Nitin A Kharche⁵, Sachin M. Bhole⁶

¹P.G. student, Department of Mechanical Engineering, Padm. Dr. V. B. Kolte College of Engineering,
Malkapur, Maharashtra, India

^{2,3,4,5,6}Department of Mechanical Engineering, Padm. Dr. V. B. Kolte College of Engineering, Malkapur,
Maharashtra, India

Abstract:

Material handling equipment is crucial in manufacturing industries, automobile workshops, maintenance facilities, warehouses, and assembly units. The advent of Computer Aided Engineering (CAE) tools has significantly enhanced mechanical design and analysis. Finite Element Analysis (FEA) allows engineers to predict structural behavior under various loads before manufacturing, reducing development costs, minimizing design errors, and improving reliability. In this study, a detailed modeling and structural analysis of a workshop hoist were performed using ANSYS Workbench 19.2. The three-dimensional hoist model, imported in IGES format, consists of 27 structural components made of structural steel. The hoist has approximate dimensions of 1887.7 mm in length, 1651 mm in height, and 940.21 mm in width, with a total mass of 214.72 kg. Material properties assigned included a Young's modulus of 200 GPa, Poisson's ratio of 0.3, yield strength of 250 MPa, and ultimate tensile strength of 460 MPa.

For structural analysis, fixed support boundary conditions were applied to the base frame, while a downward force of 4500 N was applied at the lifting point to simulate operational conditions. Static structural analysis determined total deformation, equivalent elastic strain, maximum principal strain, maximum shear strain, equivalent von-Mises stress, and maximum shear stress. The maximum deformation recorded was 16.278 mm, with maximum equivalent elastic strain and principal strain of 1.8636×10^{-3} mm/mm and 1.8528×10^{-3} mm/mm, respectively. The maximum shear strain was 2.5653×10^{-3} mm/mm. Stress analysis revealed a maximum von-Mises stress of 371.05 MPa and maximum shear stress of 197.33 MPa. The highest stress concentrations occurred near the boom linkage and load-carrying joint regions, where bending moments and load transfer forces are greatest, highlighting critical areas for design optimization.

Keywords: Workshop Hoist, Finite Element Analysis, ANSYS Workbench, Structural Analysis, Von-Mises Stress, Total Deformation, CAE, Structural Steel, Material Handling Equipment.

Introduction

2.1 Introduction to Material Handling Systems

Material handling is one of the most important activities in modern industries. It involves the movement, storage, protection, and control of materials throughout the manufacturing, warehousing, distribution, consumption, and disposal processes. Efficient material handling systems increase productivity, reduce labor requirements, improve safety, and minimize operational costs.

In manufacturing industries, raw materials, semi-finished products, and finished goods must be transported from one location to another during production processes. Manual handling of heavy loads is difficult, time-consuming, and potentially dangerous for workers. Therefore, mechanical lifting and transportation devices are widely employed to improve efficiency and ensure worker safety.

Material handling equipment includes cranes, conveyors, forklifts, hoists, elevators, pallet trucks, stackers, and automated guided vehicles (AGVs). Among these devices, workshop hoists are particularly important because they provide a simple and effective solution for lifting heavy loads within limited working spaces. Workshop hoists are extensively used in maintenance workshops, automobile service centers, machine shops, warehouses, assembly lines, and fabrication industries.

The primary objective of any material handling system is to transport loads safely while minimizing energy consumption, equipment wear, and operating costs. Proper design and analysis of lifting equipment are therefore essential for ensuring reliable performance under various loading conditions.

2.2 Workshop Hoist

A workshop hoist is a mechanical lifting device designed to raise, lower, and position heavy objects using hydraulic, pneumatic, electric, or manual power sources. Workshop hoists are commonly used for lifting engines, machine components, industrial equipment, molds, dies, pumps, compressors, and other heavy machinery parts. The hoist analyzed in this project is a hydraulic workshop hoist consisting of a base frame, vertical support structure, boom arm, hydraulic cylinder, wheels, and lifting hook. The boom arm provides the required reach, while the hydraulic cylinder generates lifting force. The structural frame supports the entire load during lifting operations.

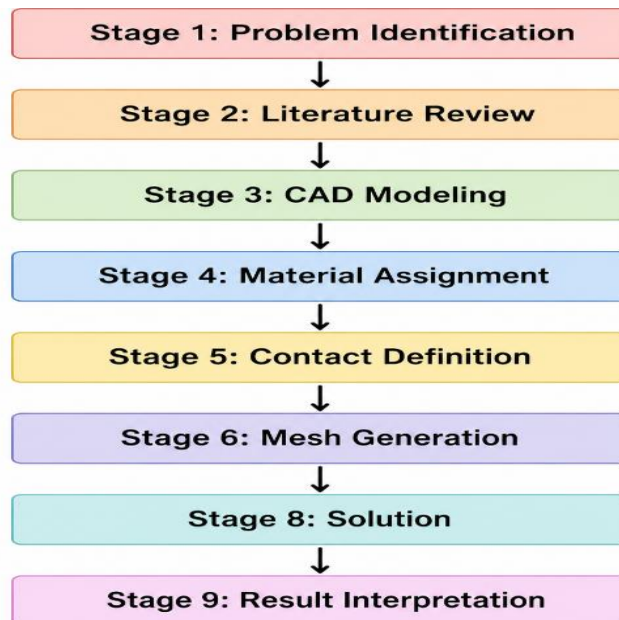


Figure 2.1: Overall Project Workflow Diagram



Figure 2.2 – Workshop Hoist Assembly Model

2.3 Working Principle of Workshop Hoist

The operation of a workshop hoist is based on hydraulic power transmission. When hydraulic pressure is applied inside the cylinder, the piston extends outward. This extension generates a lifting force that rotates the boom arm about its pivot point.

As the boom arm rises, the load attached to the hook is lifted. The support frame transfers the load safely to the base structure and eventually to the ground through the wheels and support members.

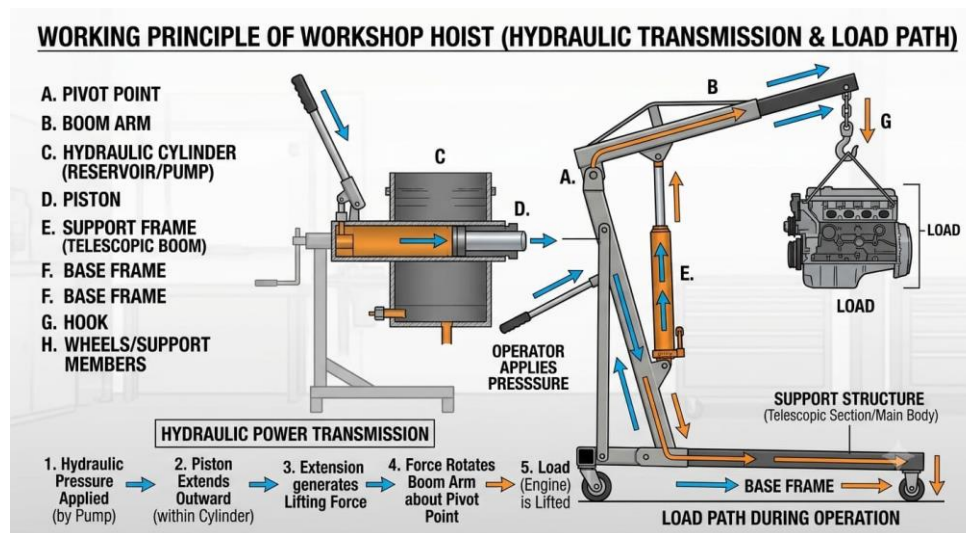


Figure 2.3 – Material Handling System Diagram

2.4 Need for Structural Analysis

Structural analysis is a vital step in the design of lifting equipment such as workshop hoists. It helps ensure that the structure can safely carry applied loads without excessive deformation, yielding, fatigue failure, joint failure, or instability. By analyzing stresses, deflections, and load distribution before manufacturing, engineers can improve safety, reliability, and overall performance of the hoist during operation.

Literature Review

3.1 Introduction

The purpose of the literature review is to identify existing methods, research gaps, analysis techniques, and opportunities for further investigation. In the present study, literature related to workshop hoists, crane structures, lifting mechanisms, finite element analysis, structural optimization, stress analysis, and Computer Aided Engineering (CAE) applications has been reviewed.

The development of Computer Aided Engineering software has transformed the design process of lifting equipment. Modern simulation tools such as ANSYS, Abaqus, HyperMesh, and SolidWorks Simulation allow engineers to predict structural behavior accurately before manufacturing. Researchers have widely used these tools for evaluating stresses, strains, deformations, fatigue life, and structural stability of cranes and hoists.

3.2 Review of literature

Several studies have highlighted the importance of finite element analysis (FEA) in the design and optimization of lifting equipment. Maharana (2012) developed a complete finite element model of an Electric Overhead Travelling (EOT) crane hoisting mechanism and demonstrated that FEA accurately predicts structural behavior, with maximum stresses occurring near load transfer regions, and allows design modifications to improve safety. Similarly, Chauhan and Bhatt (2012) investigated stress distribution in crane structures, showing that structural optimization through FEA reduces stress concentrations, enhances fatigue life, and identifies critical failure locations, which is directly relevant to workshop hoist design.

Desai and Zeytinoglu (2016) focused on optimizing crane hook geometry, reporting that different cross-sectional profiles under identical loads reduced maximum stress, improved material utilization, and decreased structural weight without compromising strength. Mamta Zade (2017) analyzed fatigue performance of crane hooks under cyclic loading and found that fatigue cracks initiate at stress concentration zones, while proper material selection and design significantly improve durability. Bhatkar et al. (2015) applied Taguchi optimization to crane hook design, achieving reduced stress concentrations, improved structural efficiency, and weight reduction, demonstrating the applicability of optimization techniques to hoist components.

Kumhar (2015) and Devaraj (2015) also confirmed that FEA reliably predicts stress and deformation in crane hooks, emphasizing the importance of material selection and cross-sectional design for improved safety and performance. More recent research by Nguyen et al. (2023) simulated dynamic load-bearing conditions of double girder cranes, highlighting that dynamic loads generate higher stresses than static loads, and load position significantly influences stress distribution, suggesting potential extensions of hoist analysis to dynamic conditions. Pavlovic (2018) compared different crane hook cross sections, finding that circular profiles and geometry optimization reduce peak stresses and enhance structural efficiency. Finally, Khan et al. (2017) evaluated ramshorn hook configurations using CAE tools, showing improved load distribution, reduced stress concentration, and increased structural reliability. Collectively, these studies demonstrate the effectiveness of FEA for evaluating lifting mechanisms, optimizing geometry, enhancing safety, and guiding material selection, providing a strong foundation for the structural analysis and design improvement of workshop hoists.

Table No. 1- Literature Review Summary

Sr. No	Author	Year	Analysis Tool	Research Area
1.	Maharana	2012	ANSYS	Hoisting Mechanism
2.	Chauhan	2012	FEA	Crane Structure
3.	Desai	2016	ANSYS	Hook Optimization
4.	Zade	2017	FEA	Fatigue Analysis
5.	Bhatkar	2015	ANSYS	Optimization
6.	Kumhar	2015	ANSYS	Hook Design
7.	Devaraj	2015	Workbench	Material Selection
8.	Nguyen	2023	Simulation	Dynamic Analysis
9.	Pavlovic	2018	FEM	Cross Sections
10.	Khan	2017	CAE	Ramshorn Hook

3.3 Research Gap Identification

- Most studies focus on crane hooks rather than complete workshop hoist assemblies.
- Limited research has been performed on integrated hoist structures including boom arm, support frame, and hydraulic lifting mechanism.
- Many studies consider only simplified models instead of full assembly analysis.
- Experimental validation is often costly and time-consuming, creating a need for accurate virtual simulations.

- Few researchers have investigated complete workshop hoists under realistic loading conditions using modern CAE tools.

Work And Research Methodology

4.1 Introduction

Material handling equipment such as workshop hoists are extensively used in industries for lifting and transporting heavy loads. These lifting systems are subjected to various mechanical forces during operation. The safety and efficiency of a workshop hoist depend upon its structural strength, stiffness, and ability to withstand operational loads without failure.

With increasing industrial requirements, hoists are expected to lift heavier loads while maintaining structural reliability and operational safety. Failure of any structural component may lead to equipment damage, production loss, and serious accidents. Therefore, proper structural analysis is essential before manufacturing.

Traditional design approaches mainly depend on theoretical calculations and prototype testing. However, these methods are time-consuming and expensive. Modern engineering practice utilizes Computer Aided Engineering (CAE) tools and Finite Element Analysis (FEA) to evaluate structural performance virtually before fabrication.

The present work focuses on modeling and static structural analysis of a workshop hoist using ANSYS Workbench 19.2. The study investigates stress distribution, deformation, strain behavior, and structural integrity under working load conditions.

4.2 Problem Definition

A workshop hoist experiences different types of loads during lifting operations. The suspended load creates bending moments, tensile forces, compressive forces, and shear stresses in various structural members such as the boom arm, support column, base frame, and joints.

The major challenges associated with workshop hoists include:

- **Excessive Deformation:** - Large deformation may affect lifting accuracy and reduce operational stability.
- **High Stress Concentration:** - Stress concentration generally occurs near welded joints, boom connections, pivot locations, and hook attachment points.
- **Structural Failure:** - If stresses exceed material yield strength, permanent deformation or structural failure may occur.
- **Fatigue Damage:** - Repeated lifting operations may cause fatigue crack initiation and propagation in critical regions.
- **Increased Manufacturing Cost:** - Without proper analysis, multiple prototypes may be required, increasing development time and cost.

4.3 Objectives of the Project

- To develop a three-dimensional CAD model of the workshop hoist assembly.
- To import the geometry into ANSYS Workbench for finite element analysis.
- To assign structural steel material properties to all components.
- To generate an appropriate finite element mesh for accurate simulation.
- To define contact conditions between structural members.
- To apply realistic boundary conditions and loading conditions.

4.4 Research Methodology

The research methodology adopted for this investigation consisted of several systematic stages to evaluate the structural performance of the workshop hoist. Initially, the structural behavior of the hoist under lifting load conditions was identified as the main research problem. A detailed literature review on hoists, cranes, finite element analysis, and structural optimization was carried out to establish the research framework.

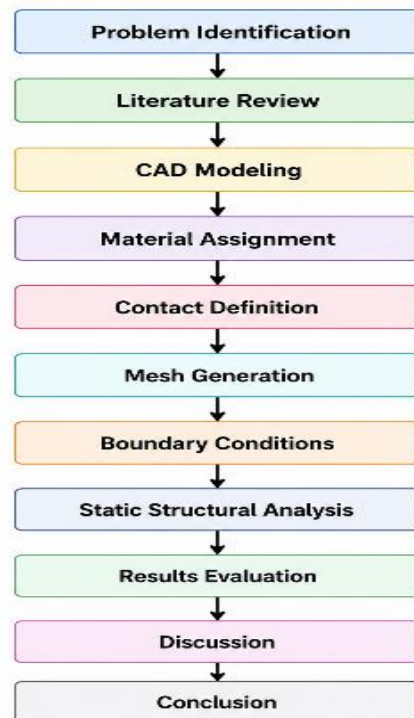


Figure 4.1: Research Methodology Flowchart

A three-dimensional CAD model of the workshop hoist assembly was then prepared and imported into ANSYS Mechanical APDL using IGES format. The assembly contained 27 bodies with a total mass of 214.72 kg. Structural steel material properties such as density, Young's modulus, Poisson's ratio, yield strength, and ultimate strength were assigned to the model. Automatic bonded contacts were generated between interacting components, resulting in 42 contact regions. The geometry was discretized into finite elements with a mesh containing 60,793 nodes and 29,195 elements. Fixed support boundary conditions were applied at the base frame, while a downward load of 4500 N was applied at the lifting point. Static structural analysis was then performed using the ANSYS solver to obtain results including total deformation, equivalent elastic strain, maximum principal strain, maximum shear strain, equivalent stress, and maximum shear stress. Finally, the simulation results were interpreted to identify critical stress regions, maximum deformation locations, structural safety, and possible design improvements.

Table 4.2: Material Properties

Property	Value
Density	7850 kg/m ³
Young's Modulus	200 GPa
Poisson Ratio	0.30
Yield Strength	250 MPa
Ultimate Strength	460 MPa

Design Calculations And Mathematical Modeling

5.1 Introduction

The design of a workshop hoist requires careful evaluation of loads, stresses, bending moments, and structural stability. Before performing finite element analysis, theoretical calculations are carried out to estimate the forces acting on the structure and verify the adequacy of the design.

Mathematical modeling provides a simplified representation of the actual hoist system and helps in understanding load transfer mechanisms. These calculations also serve as a basis for validating numerical results obtained through ANSYS Workbench.

The present chapter presents load calculations, force analysis, bending moment calculations, stress calculations, deformation equations, and factor of safety evaluation for the workshop hoist.

5.2 Design Data

The workshop hoist considered in the present investigation has the following specifications:

Table 5.1 Design Parameters

Parameter	Value
Hoist Capacity	450 kg
Applied Load	4500 N
Material	Structural Steel
Young's Modulus	200 GPa
Yield Strength	250 MPa
Ultimate Strength	460 MPa
Total Mass of Hoist	214.72 kg

5.3 Load Calculation

The first step is to determine the load acting on the hoist structure.

The relationship between force and mass is: $W = m \times g$

Where:

(W) = Weight (N)

(m) = Mass (kg)

(g) = Acceleration due to gravity (9.81 m/s^2)

Substituting values: $W = 450 \times 9.81$ $W = 4414.5 \text{ N}$

The theoretical load is therefore: $W = 4414.5 \text{ N}$

For design safety purposes, ANSYS analysis uses: $W = 4500 \text{ N}$

Results And Discussion

6.1 Results

The primary objective of finite element analysis is to evaluate the structural behavior of the workshop hoist under the applied loading condition. After defining geometry, material properties, mesh, contact regions, boundary conditions, and loading parameters, the model was solved using the ANSYS Mechanical APDL solver.

The finite element analysis generated detailed information regarding deformation, strain, and stress distribution throughout the hoist structure. These results help identify critical locations, evaluate structural integrity, and determine whether the design can safely withstand the applied load.

The present chapter discusses the following output parameters:

1. Total Deformation
2. Equivalent Elastic Strain.
3. Maximum Principal Elastic Strain
4. Maximum Shear Elastic Strain
5. Equivalent (Von-Mises) Stress
6. Maximum Shear Stress
7. All results are discussed in detail along with engineering interpretation.

6.2 Total Deformation Analysis

Total deformation represents the overall displacement experienced by the structure due to the applied load. It is one of the most important indicators of structural stiffness.

The ANSYS analysis shows that deformation gradually increases from the fixed support towards the free end of the boom arm.

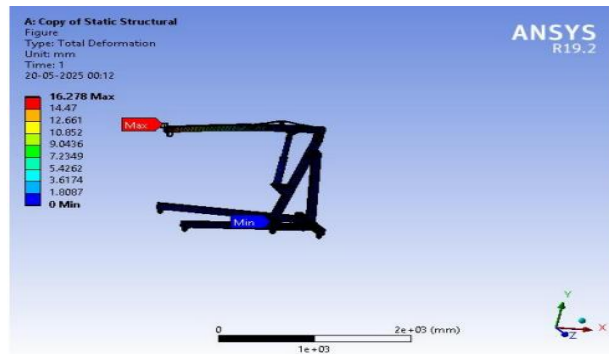


Figure 6.1 Total Deformation Contour

Table 6.1 Total Deformation Results

Results						
Object Name	Total Deformation	Equivalent Elastic Strain	Maximum Principal Elastic Strain	Maximum Shear Elastic Strain	Equivalent Stress	Maximum Shear Stress
Minimum	0. mm	0. mm/mm	-3.5645e-006 mm/mm	0.mm/mm	0. MPa	Minimum
Maximum	16.278 mm	1.8636e-003 mm/mm	1.8528e-003 mm/mm	2.5653e-003 mm/mm	371.05 MPa	197.33 MPa
Average	0.63615 mm	6.8342e-005 mm/mm	4.0744e-005 mm/mm	7.9498e-005 mm/mm	11.531 MPa	6.1153 MPa

The minimum deformation occurs at the fixed support because all translational movements are restricted. The maximum deformation of 16.278 mm occurs at the free end of the boom arm where the external load is applied. This behavior is expected because the boom behaves similarly to a cantilever beam subjected to a concentrated load.

9.3 Equivalent Elastic Strain Analysis

Equivalent elastic strain represents the intensity of deformation occurring within the material. Higher strain values indicate regions where material deformation is more significant.

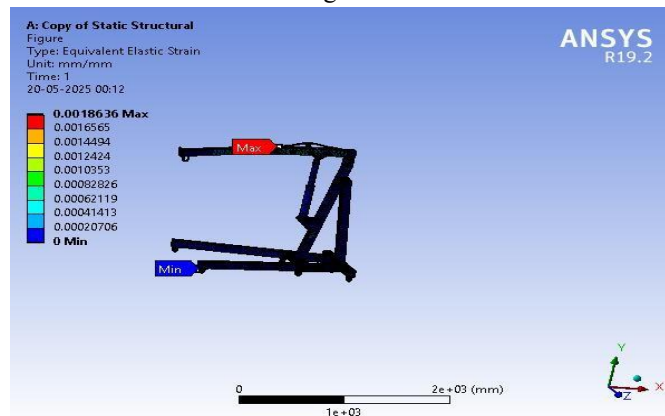


Figure 6.2 Equivalent Elastic Strain

The highest strain values occur near highly stressed regions of the boom and joint connections. The maximum equivalent strain: 1.8636×10^{-3} indicates elastic deformation within the structural steel material. The strain distribution pattern closely follows the stress distribution pattern, validating the finite element results.

9.4 Maximum Principal Elastic Strain Analysis

Principal strain represents the maximum tensile strain experienced by the structure. This parameter is important because tensile strains are directly related to crack initiation and failure mechanisms.

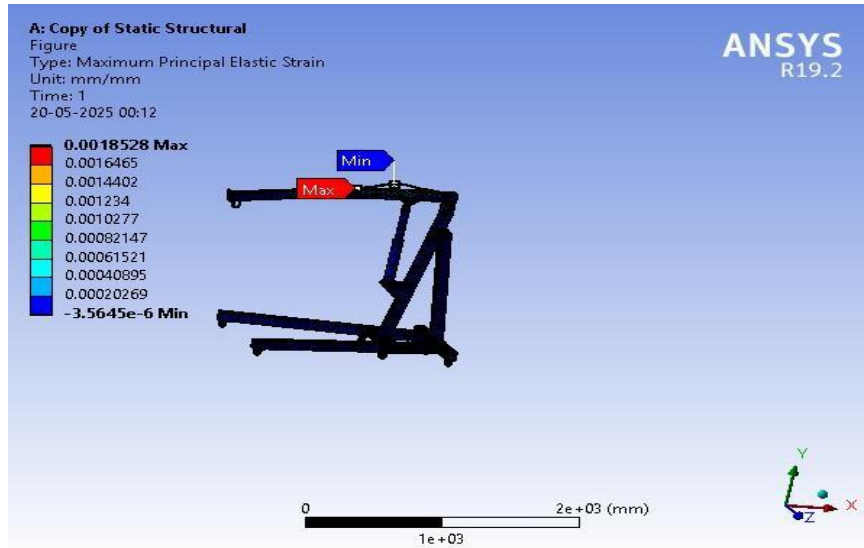


Figure 6.3 Maximum Principal Elastic Strain

The maximum principal strain value is: 1.8528×10^{-3} which occurs near the boom connection and pivot regions. These locations experience high tensile loading due to bending moments generated during lifting operations. The strain values remain within the elastic range of structural steel, indicating that no large-scale plastic deformation occurs throughout most of the structure.

9.5 Maximum Shear Elastic Strain Analysis

Shear strain measures angular distortion occurring within the material. This parameter is useful for evaluating regions subjected to significant load transfer.

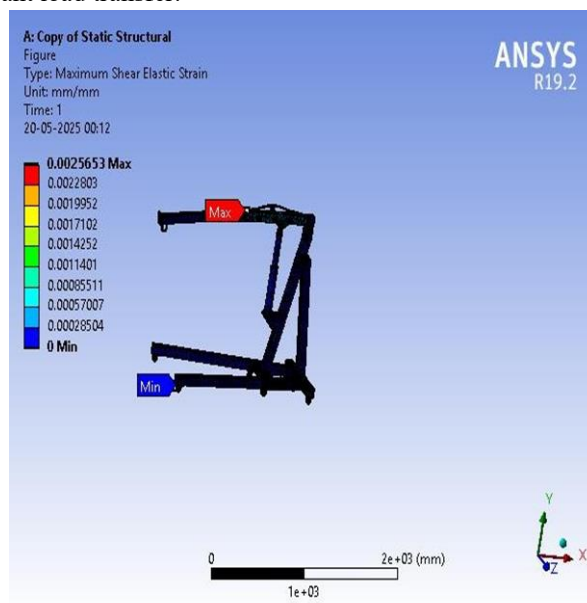


Figure 6.4 Maximum Shear Elastic Strain

The maximum shear strain: 2.5653×10^{-3} is observed in highly loaded joint regions where force transmission occurs between structural members. The results indicate: Effective load transfer between components., No excessive shear deformation. And Adequate structural rigidity.

9.6 Equivalent (Von-Mises) Stress Analysis

Equivalent stress, commonly known as Von-Mises stress, is the most important parameter for evaluating yielding of ductile materials. According to Von-Mises theory, yielding begins when equivalent stress exceeds material yield strength.

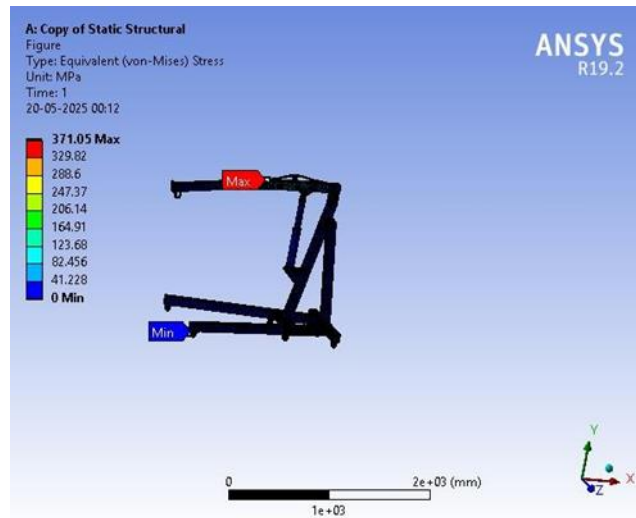


Figure 9.5 Equivalent Stress Distribution

The analysis predicts a maximum equivalent stress of: 371.05 MPa This stress is concentrated near: Boom arm connection, Pivot region and Load transfer joints The material yield strength is: 250 MPa Since: $371.05 > 250$ localized yielding may occur in these critical regions. Possible reasons include: Sharp geometric transitions, High bending moment concentration and Local stress intensification Therefore, these regions should be strengthened through: Increased section thickness, Additional stiffeners and Improved joint design.

9.7 Maximum Shear Stress Analysis

Maximum shear stress is important for evaluating potential shear failure.

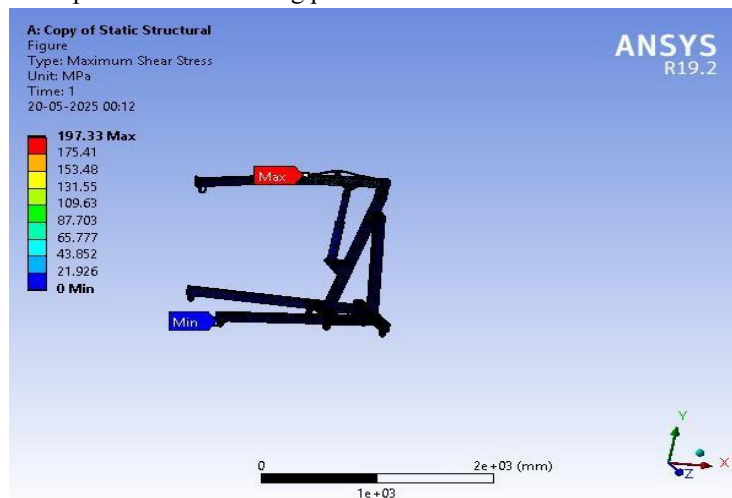


Figure 9.6 Maximum Shear Stress Distribution

The maximum shear stress obtained is:197.33 MPa This stress occurs near highly loaded structural joints. The results indicate significant load transfer through: Pivot connections, Boom linkage and Support interfaces Proper welding quality and adequate joint design are therefore essential for long-term structural reliability.

Conclusions And Future Scope

Conclusions

Based on the finite element analysis performed using ANSYS Workbench 19.2, it can be concluded that the workshop hoist structure was successfully modelled and analyzed using a three-dimensional finite element approach. The imported geometry consisted of 27 structural bodies with a total mass of 214.72 kg. A sufficiently refined finite element mesh containing 60,793 nodes and 29,195 elements was generated to ensure accurate structural analysis. A downward load of 4500 N was applied while fixing the base structure to simulate actual operating conditions. The analysis results showed that the maximum total deformation was 16.278 mm, while the maximum equivalent elastic strain, principal strain, and shear strain were 1.8636×10^{-3} , 1.8528×10^{-3} and 2.5653×10^{-3} respectively. The maximum equivalent (Von-Mises) stress developed in the structure was 371.05 Mpa, whereas the maximum shear stress observed was 197.33 Mpa. It was found that the highest stress concentrations occurred near the boom joints and pivot regions where load transfer takes place. The study demonstrated that finite element analysis is highly effective for predicting structural behavior, identifying critical stress regions, and suggesting areas requiring design improvement. Furthermore, the methodology adopted in this work can be effectively utilized for future optimization and redesign of workshop hoists.

Future Scope

The present work provides a strong foundation for further research and development in the field of workshop hoist structural analysis. Future studies may include fatigue analysis to evaluate structural life under repeated loading conditions, dynamic analysis to investigate inertia and impact effects during lifting operations, and modal analysis to determine natural frequencies and vibration behavior. Further research can also focus on buckling analysis for stability evaluation under compressive loads and topology optimization for reducing structural weight while maintaining sufficient strength. The use of lightweight composite materials may also be explored as an alternative to conventional structural steel. In addition, experimental validation through physical testing can be carried out to verify numerical simulation results. Multi-objective optimization techniques may also be adopted to simultaneously optimize weight, strength, and overall manufacturing cost.

References

1. Naresh Chauhan and P. M. Bhatt, "Improving the Durability of E.O.T. Crane Structure by Finite Element Analysis and Optimization of Hook Material," *Procedia Engineering*, Vol. 38, pp. 837–842, 2012.
2. Pradyumna Keshari Maharana, "Computer Aided Analysis and Design of Hoisting Mechanism of an EOT Crane," M.Tech Thesis, National Institute of Technology Rourkela, India, 2012.
3. Z. Domazet, F. Lukša and M. Bugarin, "Failure of Two Overhead Crane Shafts," *Engineering Failure Analysis*, Vol. 44, pp. 125–134, 2014.
4. Mamta R. Zade, "Finite Element Analysis and Fatigue Analysis of Crane Hook with Different Materials," *International Research Journal of Engineering and Technology*, Vol. 4, Issue 1, pp. 1429–1436, 2017.
5. Niranjana Desai and Nuri Zeytinoglu, "Design and Optimization of the Geometric Properties of a Crane Hook," *World Journal of Engineering and Technology*, Vol. 4, pp. 391–397, 2016.
6. Omkar P. Bhatkar, Arun V. Javir, Shinde A. A., Sawant S. S., Ukarde S. E., and Kavathankar K. S., "Finite Element Analysis of Crane Hook and Optimization Using Taguchi Approach," *Journal of Advance Research in Mechanical and Civil Engineering*, Vol. 2, No. 1, pp. 63–67, 2015.
7. Vikky Kumhar, "Design and Analysis of Crane Hook Trapezoidal Cross Section Using Finite Element Analysis," *VSRD International Journal of Mechanical, Civil, Automobile and Production Engineering*, Vol. 5, No. 5, pp. 89–94, 2015.

8. Abhijit Devaraj, "Design of a Crane Hook of Different Materials and Stress Analysis Using ANSYS Workbench," *International Journal for Research in Applied Science and Engineering Technology*, Vol. 3, Issue 7, pp. 310–314, 2015.
9. Sunil J. Tiwari and Vivek Padole, "Design, Analysis and Material Optimization of Laminated Crane Hook: A Review," *International Journal of Analytical, Experimental and Finite Element Analysis*, Vol. 2, No. 3, pp. 109–111, 2015.
10. Hong Tien Nguyen, Tuan-Linh Nguyen, Van Trong Thai and Tan Huynh Nhu, "Simulation of Dynamic Load-Bearing State of Double Girder Crane," *International Journal of Research in Engineering and Science*, Vol. 11, Issue 5, pp. 820–824, 2023.
11. Goran Pavlovic, "Comparative Analysis and Optimization of Different Cross Sections of Crane Hook Subject to Stresses According to Winkler-Bach Theory," *Mechanical Engineering in the 21st Century Conference Proceedings*, 2018.
12. Nausad Khan, Gian Bhushan and Pankaj Chandna, "Design and Stress Analysis of Ramshorn Hook with Different Cross Section Using CAE Tools," *International Journal of Engineering Technology Science and Research*, Vol. 4, Issue 1, pp. 1–8, 2017.
13. Rajendra Parmanik, "Design of Hoisting Arrangement of EOT Crane," Technical Publication, 2008.
14. American Society of Mechanical Engineers (ASME), *ASME B30.2 – Overhead and Gantry Cranes*, New York, USA, 2005.
15. Bureau of Indian Standards, *IS 3177:1999 – Code of Practice for Electric Overhead Travelling Cranes and Gantry Cranes*, New Delhi, India.
16. Bureau of Indian Standards, *IS 807:2006 – Design, Erection and Testing (Structural Portion) of Cranes and Hoists*, New Delhi, India.
17. Michele Ambrosino et al., "Modeling and Control of 5-DoF Boom Crane," *IEEE/Control Systems Research*, 2021.
18. Michele Ambrosino and Emanuele Garone, "Modelling and Control of a Knuckle Boom Crane," *Control Engineering Research*, 2021.
19. P. A. Ospina-Henao and Framsol Lopez-Suspes, "Dynamic Analysis and PID Control of a Gantry Crane," *International Journal of Mechanical Systems*, 2017.
20. Kalpesh R. Salunkhe, "Finite Element Analysis and Structural Optimization of EOT Crane Components," *International Advanced Research Journal in Science, Engineering and Technology*, 2016