A B

Deepali Shinde Paper

ashik

Document Details

Submission ID

trn:oid:::3618:123467408

Submission Date

Dec 2, 2025, 6:08 PM GMT+5:30

Download Date

Dec 2, 2025, 6:09 PM GMT+5:30

File Name

Deepali Shinde Paper.docx

File Size

3.8 MB

7 Pages

2,247 Words

12,866 Characters

6% Overall Similarity

The combined total of all matches, including overlapping sources, for each database.

Filtered from the Report

▸ Bibliography

Match Groups

9 Not Cited or Quoted 6%
Matches with neither in-text citation nor quotation marks

0 Missing Quotations 0%
Matches that are still very similar to source material

0 Missing Citation 0%
Matches that have quotation marks, but no in-text citation

0 Cited and Quoted 0%
Matches with in-text citation present, but no quotation marks

Top Sources

6% 🌐 Internet sources

2% 📖 Publications

3% 👤 Submitted works (Student Papers)

Integrity Flags

0 Integrity Flags for Review

Our system's algorithms look deeply at a document for any inconsistencies that would set it apart from a normal submission. If we notice something strange, we flag it for you to review.

A Flag is not necessarily an indicator of a problem. However, we'd recommend you focus your attention there for further review.

Match Groups

🔲 **9** Not Cited or Quoted 6%
Matches with neither in-text citation nor quotation marks

💬 **0** Missing Quotations 0%
Matches that are still very similar to source material

📄 **0** Missing Citation 0%
Matches that have quotation marks, but no in-text citation

◈ **0** Cited and Quoted 0%
Matches with in-text citation present, but no quotation marks

Top Sources

6% 🌐 Internet sources

2% 📖 Publications

3% 👤 Submitted works (Student Papers)

Top Sources

The sources with the highest number of matches within the submission. Overlapping sources will not be displayed.

1	Internet		
ijritcc.org			**3%**

2	Internet		
naac.sveri.ac.in			**2%**

3	Publication		
P. A. Dhawale, B. P. Ronge Name. "Prediction of weld strength by parametric opti...			**<1%**

4	Student papers		
Southampton Solent University on 2023-04-24			**<1%**

5	Internet		
ewh.ieee.org			**<1%**

6	Student papers		
The Robert Gordon University on 2008-04-09			**<1%**

Topology Optimization of Jib Crain using Finite Element Method

Deepali Shinde[1]
Student, Department of Mechanical Engineering, SVERI's College of Engineering, Pandharpur, Maharashtra, India, deepalibs8@gmail.com

Sandeep S. Wangikar[2]
Department of Mechanical Engineering, SVERI's College of Engineering, Pandharpur, Maharashtra, India, sswangikar@coe.sveri.ac.in

Digambar Kashid[3]
Department of Mechanical Engineering, SVERI's College of Engineering, Pandharpur, Maharashtra, India,dtkashid@coe.sveri.ac.in

ABSTRACT: This paper explains the methodology that has been created to be used in structural optimization and how it is applied in the engineering design processes. The study is based on a swing arm component that is unilateral. The major purpose of this was to accomplish a mass reduction in an already existent steel girder arm, which is a component of a 500 kg capacity overhead crane, by the use of topology optimization techniques. Its cost-effectiveness and performance can only be improved with this kind of weight loss.

This paper explains the methodology that has been created to be used in structural optimization and how it is applied in the engineering design processes. The study is based on a swing arm component that is unilateral. The primary objective was to use topology optimization approaches in order to reduce the mass of an existing steel girder arm. This arm is a part of an overhead crane with a 500 kg capacity. Improving its efficiency and performance need such a reduction in weight.

Keywords: Rebar, Crain, Optimization, Finite Element Analysis, Ansys, Modal, and Static Analysis.

1. INTRODUCTION

The overhead cranes are required in the field of the logistics, construction, and manufacturing industries to lift, handle and transport heavy materials in the field of modern industrial environment. These systems play the role of ensuring the continued efficiency of operations, amongst the most essential structural elements is the crane girders. Crane girders being the primary load-bearing element should be designed in such a way that they meet the high standards of performance and safety to ensure structural integrity and overall functionality optimization.

The optimization techniques and Finite Element Analysis (FEA) as they pertain to girders used in overhead cranes are covered extensively in this study. Gaining a deeper understanding of how crane girders behave structurally under varying loads and then coming up with optimization ways to make them better is the key aim. The combination of FEA and optimization techniques is proposed to give this research an opportunity to enhance the structural efficiency and at the same time reduce the material consumption and costs.

Using finite element analysis (FEA) to optimize various crane designs has been studied in previous research studies [1-5], but it has been shown that this practice enables the significant decrease of weight, and at the same time, the stress and deformation should not exceed acceptable values. Crane design optimization using the Optimization Toolbox of MATLAB has also been effective at design optimization [6], usually by specifying an objective function to reduce mass, with constraints on allowable stress and deformation. Such nonlinear programming constraints are usually handled directly using the gradient projection method. Moreover, the urgency of optimal steel structures has been discussed because specific cranes have no standard national design regulations, and parametric optimization models, which

combine the use of FEA and stability theory, are presented. Previous research has used finite element analysis (FEA) to validate analytical results, and it has optimized crane girders by incorporating buckling assessment along with the cumulative impacts of bending and shear forces.History and Relevant Projects.

2. FINITE ELEMENT ANALYSIS MODEL

A 3D CAD model of the crane provided by the manufacturer acted as the foundation for the analysis. Dassault System's CATIA 3D modeling program was used to generate the CAD model. Current plans call for a 500-kilogram lifting capability from a single-girder overhead crane that is structurally stabilized. Only the girder structure was included in the Finite Element (FE) study; the static support system was left out. The crane girder is the primary structural element supporting the lifting motor. It is supported at one end by vertical members that permit movement along the supporting structure. Stiffeners are integrated into the end support members to enhance structural strength and stability.
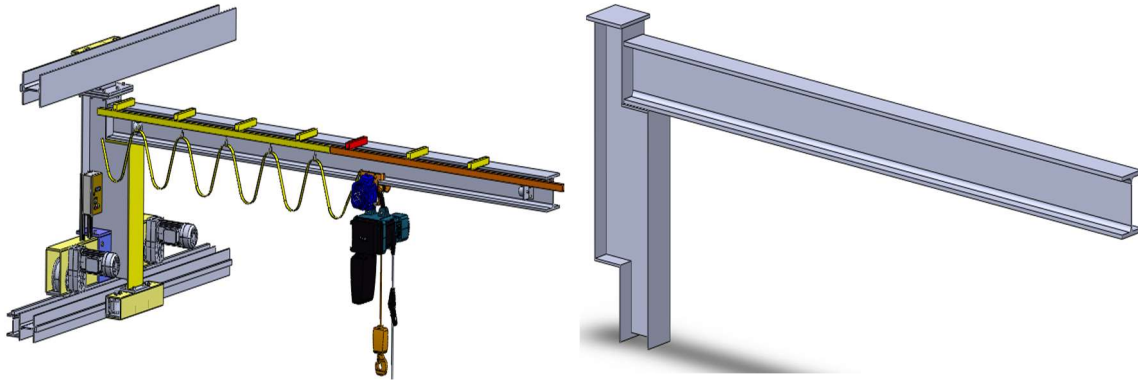

Figure 1. CAD Design of the Girder Structure

Table 1. you can see the simulated steel's mechanical characteristics.

Table 1. Steel Analysis Material Properties

Property	Value
Density (kg/mm3)	7860
Modulus of Elasticity E (GPa)	200
Ultimate Strength (MPa)	450
Yield Strength (MPa)	250

To address changing impacts, the pressures or weights applied to the crane components must be adjusted using load factors based on the crane or hoist classification, according to IS:807-1976 standards.

In the case of a 500-kilogram loading crane:

- Static Load (N): $500 \times 9.81 = 9810$ N1000 \times 9.81 = 4950 N

- Equivalent Dynamic Load (N): Impact Factor (1.5) $\times 500 \times 9.81 = 7360$ N

Impact factor ensures that the structural analysis adequately considers the additional forces resulting from motion, inertia, and impact effects.

3. FINITE ELEMENT MESH

ANSYS Workbench 16.0 was employed to generate the finite element mesh model. The process involved importing the swing arm's CAD geometry from CATIA into ANSYS, followed by geometry cleanup. A 3D hex-dominant mesh was used to discretize the solid components, which optimizes element quality for accurate simulation. Figure 2. illustrates the meshed model.
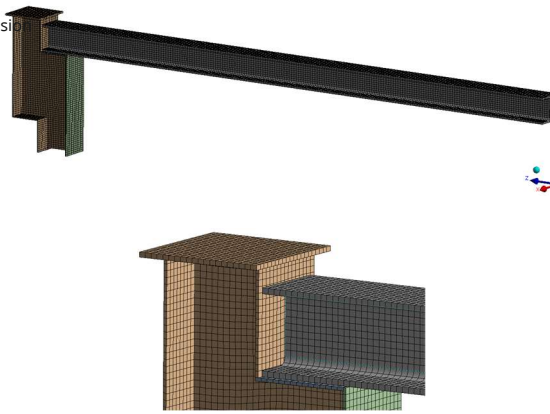
Figure 2. Finite Element Mesh using 3D Hex Element

4. LOADS AND BOUNDARY CONDITIONS

Loads

In ANSYS, distributed loads are applied to simulate forces acting over a surface or along a line, as opposed to a single point 43. This approach is standard for modelling pressure and other uniformly distributed forces44. The correct definition of distributed loads ensures an accurate representation of structural behaviour under real-world conditions 45. Figure 3. illustrates the application of the distributed load on the girder structure46. The applied force is 7360 N.
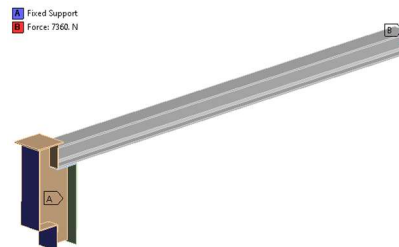

Figure 3. Load Applied on Girder

Boundary Conditions

Boundary Conditions a Fixed Support boundary condition was used in ANSYS Workbench to model a fully constrained or immovable region. This is also known as a Fixed Displacement or Zero Displacement condition. Applying fixed support involves constraining the degrees of freedom (DOF). Typically, all three translational DOFs (X, Y, and Z) are restricted, as seen in Figure 3 and Figure 4. Rotational constraints may also be applied based on the specific problem requirements.
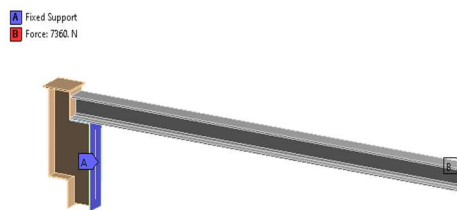

Figure 4. Fixed Support Constraint

5. RESULTS OF BASELINE ANALYSIS

The highest possible load, 7360 N, which stands for the maximum dynamic load, was used in the static study. Under this equivalent load, the girder's stress and displacement levels were evaluated using the static structural analysis. The study revealed that the girder reached a maximum stress of 200 MPa, which is significantly below the yield strength of steel, which is 250 MPa. No deflection was more than 17 mm. and display the outcomes of the deflection and stress analyses.
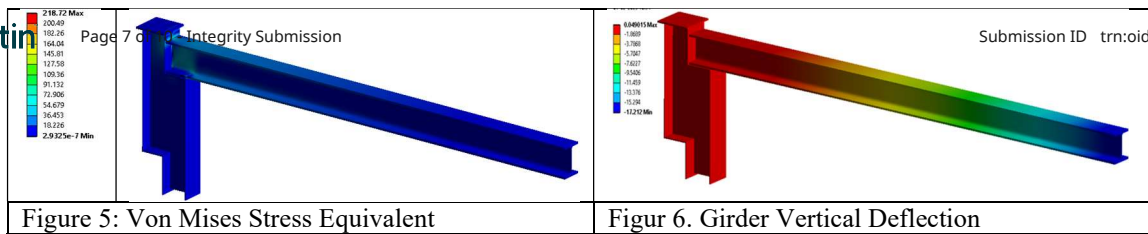
| Figure 5: Von Mises Stress Equivalent | Figur 6. Girder Vertical Deflection |

Allowable Deflection for Girder

The International Standard for Overhead Crane Girders (IS800:2007) specifies the maximum allowed deflection. The maximum permissible deflection for a crane with a capacity of up to fifty tons is determined by,

$$\delta = \frac{\text{Beam Span}}{250} = \frac{5775}{250} = 23.1 \text{ mm}$$

There is room for improvement in the present design since the maximum deviation of 17 mm is lower than the permissible deflection of 23.1 mm, according to the preliminary data. Consequently, in an effort to conserve weight and money, a topology optimization model was established with the goal of reducing material from the girder section. The modal analysis check had also been considered to be necessary before the topology optimization.

6. OPTIMIZATION SET UP

To optimize the design of a static analysis, the extreme conditions of load are necessary to be taken into consideration to make the design resistant to the worst-case scenario. The objective formulation was developed in terms of defining the objective function and the constraints, and responses. The optimization process was based on the results of the last run of the base model.

Design and Non-Design Space

There were two categories in the swing arm's finite element mesh model: designable and non-designable before the optimization analysis was carried out as shown in Figure 8.

- In order to produce the best possible structure, optimization might alter the distribution of materials within the design area.

- The non-design space includes critical regions such as load application points and contact interfaces, which must remain unchanged to preserve functional integrity.
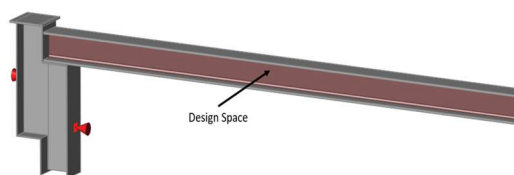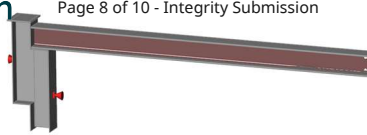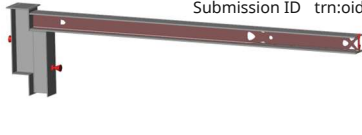

Figure 8. Design Space

The optimization process was executed with the Altair Inspire tool, following the generation of responses for overall mass and displacement.Three distinct analyses were conducted with target mass reductions of 10%, 15%, and 20%. Figure 9 suggested where material could be removed from the design space. After considering material continuity, manufacturing ease, and handling, the optimal shape was recreated in CAD software. The rebuilt optimized model was then subjected to a static analysis to ensure that the stress and displacement levels were within acceptable ranges.

First Optimal Setup—10% Mass Decrease Goal	2 Optimal Setup—15% Mass Decrease Goal	Target Reduction of 20% Mass in Optimized Configuration 3

Figure 9: Geometries Ideal for Varying Target Masses

Findings from Static Analysis on Optimal Geometry

In order to determine the equivalent Von Mises stress and deformation, static analysis was conducted on the three optimized geometries (Optimized Configurations).

Similar Von Mises Stress - Optimal Setup -1	Optimal Setup for Equivalent Von Mises Stress-2	Von Mises Equivalent Stress - Optimal Setup -3

Figure 10 shows the optimized geometries with equivalent von Mises stress.

Figure 10 shows that the stress levels are within the allowed range. Two of the optimized configurations, with a combined stress of 220 MPa, show the highest levels of performance.

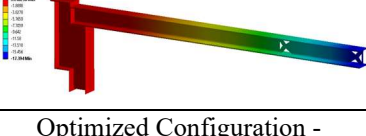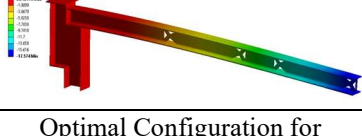In Figure 11, we can see the girder deformation for optimum shapes.

Optimized Configuration - Deformation -1	Optimized Configuration - Deformation -2	Optimal Configuration for Deformation -3

Figure 11. Optimal Geometries Deflecting

Figure 11 shows that all Optimized Configurations have deformation below the permitted deflection of 22.1 mm. In the third optimized configuration, the maximum deflection is 17.57 mm.

Table 3. Comparison Summary

Girder Model	Mass Kg	Mass Saved	% saving	Stress MPa	Deflection mm
Original Model	547	--	--	200	17.2
Optimized Configuration_1	500	47	8.59%	219	17.33
Optimized Configuration_2	475	72	14.40%	220	17.42
Optimized Configuration_3	440	107	22.53%	220	17.57

Figure 12. shows the graphical representation of original geometries and optimized geometries.
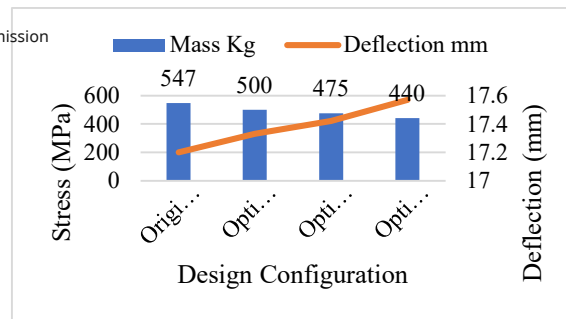
Figure 12. Graphical representation of Results in original and optimized geometries.

CONCLUSION

In order to optimize the weight of mechanical components utilizing finite element tools, this research effectively devised a strategy. Important findings include the following:

- Weight Reduction: A significant decrease in weight was experienced in all the three Optimized Configurations. With OptimizedConfiguration 1, the weight reduction was 8.59% and OptimizedConfiguration 2 was 14.4 and OptimizedConfiguration 3 was 22.53% which is a substantial reduction in weight..
- Stress Analysis: An increase in stress was observed in all optimized models[92].
  - Optimized Configuration 1 showed a 9% increase (from 20 MPa to 219 MPa).
  - Optimized Configuration 2 showed a 10% increase (from 200 MPa to 220 MPa).
  - Optimized Configuration 3 showed a 10% increase (from 200 MPa to 220 MPa).
  - Importantly, the values of stress observed do not exceed the limits of steel material at all.
- Deformation Analysis: Deformation was slightly higher in the optimum Optimized Configurations.

  - Optimized Configuration 1 increased by 0.8%, Optimized Configuration 2 increased by 1.25 percent and Optimized Configuration 3 increased by 2.5 percent.
  - Even though the deformation was greatest in Optimized Configuration 3 which is 17.57mm, this has been comfortably under the allowable deformation of 22.1mm..

REFERENCES

[1] Zuberi, Rehan H., Long Kai, and Zuo Zhengxing. "Design optimization of EOT crane bridge." Eng Opt (2008): 192-201, International Conference on Engineering Optimization.

[2] Patel, Harshil, Dhruv Upadhyay, and Divyang Patel. "Design optimization of box girder in gantry crane using finite element analysis software." International Research Journal of Engineering and Technology 7.08 (2020): 1906-1917.

[3] Mohamed, Hassan S., and Sabreen A. Abdelwahab. "Double-girder overhead crane optimum design using weighted decision matrix and Finite Element Analysis." Journal of The Institution of Engineers (India): Series C 102 (2021): 283-304.

[4] Alkin, C., C. E. Imrak, and H. Kocabas. "Solid modeling and finite element analysis of an overhead crane bridge." Acta Polytechnica 45.3 (2005).

[5] Suratkar, Abhinay, Vishal Shukla, and Dr KS Zakiuddin. "Design optimization of overhead EOT crane box girder using finite element analysis." Int. J. Eng. Res. Tech 2.7 (2013): 720-724.

[6] Pavlovic, G. V., et al. "Design optimization of the rectangular box section of the double beam bridge crane using Matlab optimization toolbox." Triennial International Conference Heavy Machinery-HM. 2017.

[7] Shijun, Song, et al. "APDL language-based optimization of tower crane." 2010 International Conference on Digital Manufacturing & Automation. Vol. 1. IEEE, 2010.

[8] Marzuki, Mohammad Al Bukhari, Mohammad Firdaus Mohammed Azmi, and Rafidah Laili Jaswadi. "Design optimization of automotive component through numerical investigation for additive manufacturing." J. Built Environ., Technol. Eng. 6 (2019): 19-26.

[9] Song, Shi Jun, Lei Li, Ji Yong Wang, and Lian Yu Song. "An Optimization and its Realization of Mast Crane Based on APDL." Advanced Materials Research 199 (2011): 1414-1418.

[10] Kumar, Ashutosh, Vidyasagar R. Bajaj, and Rachayya R. Arakerimath. "Design, Analysis and optimization of crane Girder via Buckling analysis.", International Research Journal of Engineering and Technology (2016).

[11] Zdravković, Nebojša, Mile Savković, Goran Marković, Dragan Pršić, and Goran Pavlović. "Numerical solution for the deflection of the column-mounted jib crane structure." Mechanics Transport Communications-Academic journal, 1825, no. 2019/3 (2019).

[12] Shan, N., Li, K.L., Tang, W.X., Zhao, X.P. and Liu, X.J., 2016. Finite Element Analysis for the Jib System of the MQ2535 Crane. In Proceedings of the 6th International Asia Conference on Industrial

Engineering and Management Innovation: Core Theory and Applications of Industrial Engineering (volume 1) (pp. 783-790). Atlantis Press.