International Journal on Recent and Innovation Trends in Computing and Communication
ISSN: 2321-8169 Volume: 11 Issue: 11
***Article Received:** 10 August 2023 **Revised:** 20 October 2023 **Accepted:** 16 December 2023*

Design Optimization of Single Overhead Crain Girder using Finite Element Analysis

Rohan S. Navale, Prasad P. Kulkarni, Sameer D. Katekar*

Department of Mechanical Engineering, SKN Sinhgad College of Engineering, Pandharpur-413304, Maharashtra, India.

**Email: rohan.s.navale@gmail.com*, sameer.katekar@sknscoe.ac.in

Abstract: This study aims to develop strategies in the field of structural optimization and implement these strategies into the design processes. The single-sided swing arm has been selected as the subject of this study. The primary objective of this project is to reduce the mass of an existing steel girder arm, a component of a 1-ton capacity overhead crane, through the application of topology optimization techniques. This weight reduction is essential to enhance cost-effectiveness and operational performance. To achieve this goal, the Finite Element software Ansys, equipped with multiple modules, has been utilized. Ansys is responsible for preparing the finite element model and conducting modal and static analyses. Topology optimization is performed using Inspire. Static analysis is carried out, considering the effective dynamic load on the crane girder in accordance with IS codes. The optimization method applied in this study has successfully resulted in an 18.97% reduction in the mass of the existing girder.

Keywords: Crain Girder, Optimization, FEA, Ansys, Modal analysis, Static analysis.

I. INTRODUCTION

The modern industrial landscape relies heavily on overhead cranes to facilitate the lifting, movement, and transportation of heavy loads within manufacturing, construction, and logistics sectors. Overhead crane systems are integral to the efficient functioning of various industries, and their structural components, particularly the crane girders, must be designed to meet stringent performance and safety criteria. The crane girder, as the primary load-bearing element of an overhead crane, plays a pivotal role in ensuring the structural integrity and overall functionality of the crane system.

This paper presents a comprehensive exploration of the Finite Element Analysis (FEA) and optimization techniques applied to overhead crane girders. The primary focus of this research is to enhance the understanding of the structural behaviour of crane girders under diverse loading conditions and to develop strategies for optimizing their design. The combination of FEA and optimization methodologies offers an effective means of achieving the twin objectives of structural efficiency and cost-effectiveness.

Many authors [1-5] have demonstrated the optimization of different types of crane design with FEA simulation approach. It shows the FEA simulation helps to reduce weight by limiting the stress and deformation. MATLAB optimization toolbox also proves helpful in design optimization [6]. It uses objective function to minimize the mass while constraints include as permissible stress and deformation, adopting the gradient projection method, which directly deals with nonlinear programming constraints. Shi et. al [9] discussed the lack of national design standards for mast cranes and the need for optimal design of their steel structure. It proposes a parametric optimization model based on Ansys OPT, combined with FEA and stability theory, to achieve rapid optimization design of mast cranes. Kumar et. al [10] demonstrated the design and optimization of crane girders by considering buckling analysis and the summation of bending and shear loads. The analytical process is further validated using FEA.

II. FINITE ELEMENT ANALYSIS MODEL

A 3D CAD model of the crane is taken from the manufacturer. CAD model is prepared using CATIA, a 3D modelling tool from Dassault System. Figure 1, shows the model of the crane.

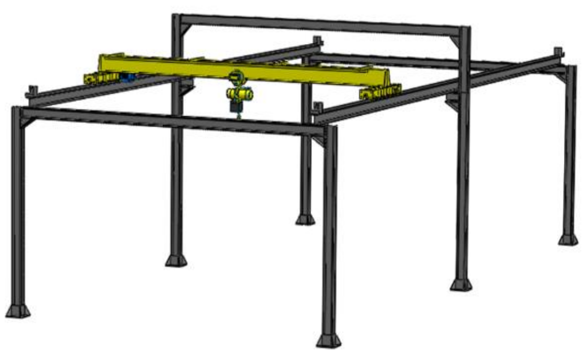


Fig. 1. 3D Model of crane

Crane specifications: Span - 5.25m, capacity - 1000 kg. For the FE analysis only, crane girder and its support (Refer Fig. **2**) is considered as it is main load bearing component of the system.

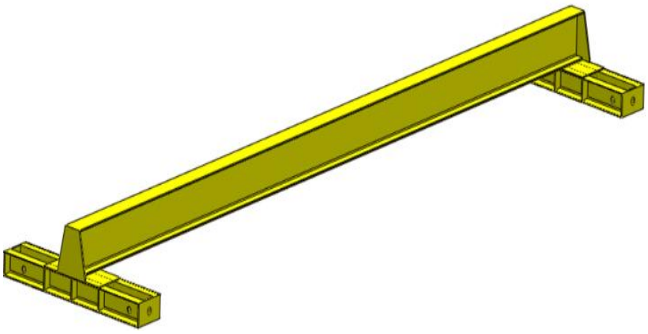


Fig. 2. Crane girder

A. *Material Property, Loads and Boundary Conditions*

Table **1** shows the mechanical properties of the steel used in simulation.

Table 1. Material Properties of Steel

Sr. No.	Property	Value
1	Density (kg/mm3)	7860
2	Modulus of Elasticity E (GPa)	200
3	Ultimate Strength (MPa)	450
4	Yield Strength (MPa)	250

In order to make allowance for the dynamic effects, the forces or loads acting upon cranes or any portion thereof shall be multiplied by the relevant factors according to the classification of the crane or hoist. These load factors are taken from IS:807-1976.

Rated Load on crane = 1000 kg
Static Load (N) = 1000 x 9.81
= 9810 N
Equivalent Dynamic Load * Impact Factor,
= (1.5)*1000*9.81
= 14715 N

B. *Mesh Model*

Ansys Workbench 16.0 was used for creation of finite element mesh model. To generate the mesh for the component, first of all CAD geometry of the swing arm was imported from CATIA into Ansys Workbench 16.0 and geometry clean-up was performed to prepare the model for meshing. 3-D mesh with hex dominant elements was employed to mesh all the solids. Fig. **3** shows the mesh model of the girder structure.

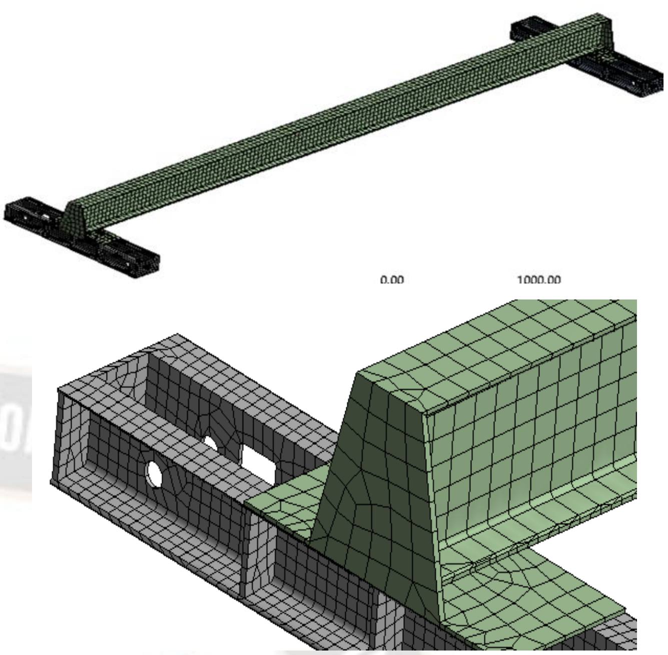


Fig. 3. FE Mesh of the Model

C. *Analysis Settings*

In ANSYS, a widely used Finite Element Analysis (FEA) software, applying a distributed load involves specifying a load that is distributed over a surface or line on a finite element model. This is commonly done when you want to simulate loads like pressure, thermal loads, or forces that act over an area rather than at a single point. The Fig. **4** shows the distribute load applied on girder structure and fixed support on the bottom surface of the supports.

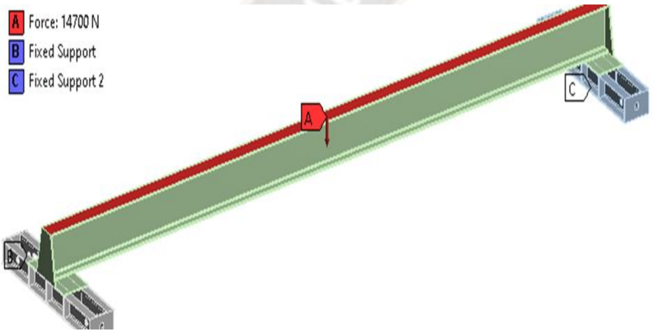


Fig. 4. Loads and Constraint

Typically, all the three translational degrees of freedom (X,Yand Z) are constrained for a fully fixed support.

III. RESULTS

The maximum load considered for static analysis is 14715 N which represents maximum dynamic load. The static structural analysis is performed for checking stress and displacement level in the girder for effect of equivalent load as shown in Fig. **5** and **6**. The maximum stress observed in girder is 38.61 MPa which is less than the yield strength of steel (220-250MPa) and maximum deflection of 0.89 mm. The maximum deflection for the overhead crane girders is specified in IS800:2007. The deflection allowed for crane capacity up to 50 ton is given as,

$$\begin{aligned} \Delta &= (\text{Beam Span})/1000 \\ &= 5250/1000 \\ &= 5.25\text{mm} \end{aligned}$$

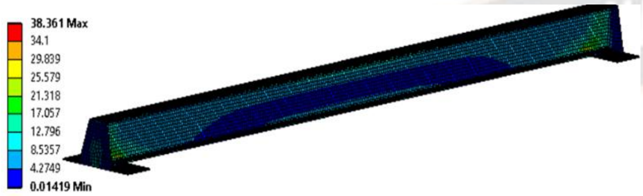


Fig. 5. Equivalent Von Mises Stress in Girder

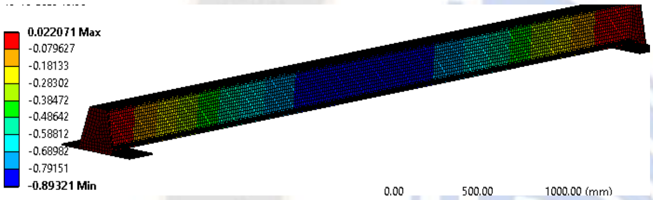


Fig. 6. Deflection in Girder

By looking into the results, there is scope for optimizing current design. A topology optimization model is set up to reduce the material from girder section so that girder weight can be reduced and cost can be saved. Form FE Analysis, it can be seen all optimized variants are safe and satisfy the acceptance criteria. Comparison of all optimized variants against original design are summarized in Table **2**.

OPTIMIZATION MODEL

While performing optimization for static analysis, extreme load conditions must be considered. The optimization formulation was done by setting objective function, constraints and responses as follows and results of the base (final) iteration were used for the optimization purpose.

A. DESIGN AND NON DESIGN SPACE

Before carrying out an optimization analysis, the finite element mesh model of the swing arm was divided into two parts i.e. designable and non-designable part of the domain as shown in Fig. **7**. The design space represents the volume in which the material distribution was allowed to change during the optimization. Non-design space contains the load acting points and contact points, so volume in that space was not allowed to change.

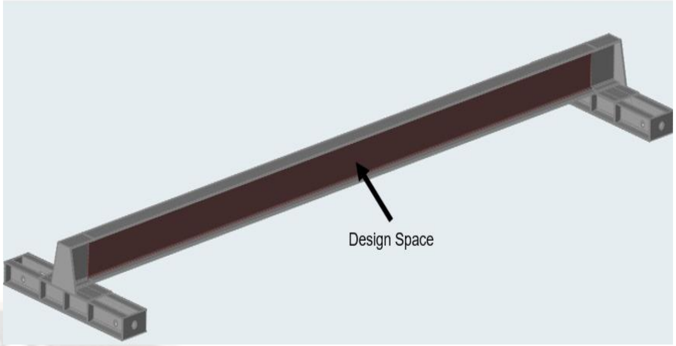
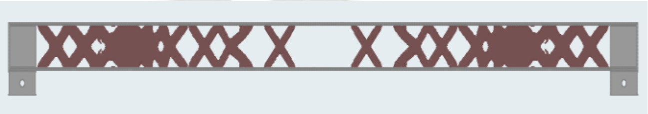


Fig. 7. Design Space for Topology Optimization.

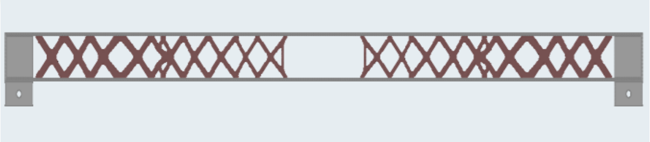
Responses for total mass and displacement were created. The optimization was carried out in Altair Inspire tool. The three different analyzed were carried out to reduce the mass in the design space by 5%, 15% and 20%.

From Fig. **8**, software suggested the which material from design space can be eliminated. Based on these results we had to take the decision which part of the component should be eliminated and which was not. The optimized geometry (optimized model) is exported in stl format and reconstructed similar to optimized geometry result using CAD software. While creating the geometry material continuity, ease of manufacturing and handling were considered. Again, static analysis was carried out on this optimized model to check whether the stresses and displacement were within limit or not.

Variant 1 – Target Reduction 5% Mass



Variant 2 – Target Reduction 15% Mass



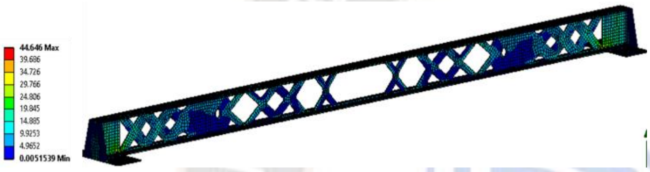
Variant 3 – Target Reduction 20% Mass

Fig. 8. Optimized Geometries for different Target Mass

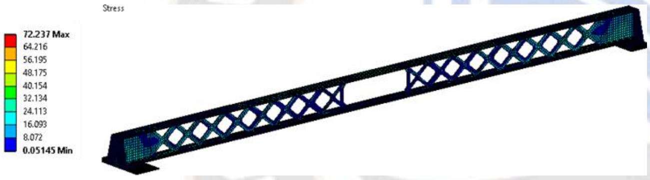
B. STATIC ANALYSIS RESULTS ON OPTIMIZED GEOMETRY

The static analysis was performed on above optimized geometry. The equivalent von mises stress and deformation is calculated for optimized geometries. The Fig. **9** shows the results of static analysis on optimized geometries.

Variant 1. Equivalent Von Mises Stress



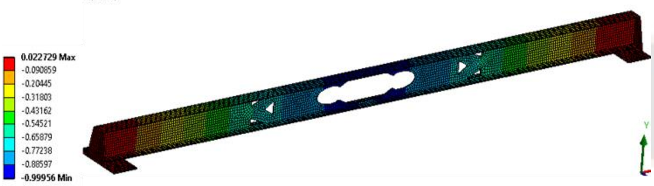
Variant 2. Equivalent Von Mises Stress



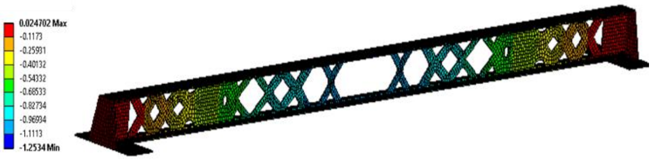
Variant 3. Equivalent Von Mises Stress –

Fig. 9. Equivalent Von Mises Stress in Optimized Geometries

The stress values are within the limit. The maximum stress is observed in variant-3, with value equal to 72.28 MPa. The Fig. **10** shows the deformation in girder for optimized geometries.



Deformation - Variant -1



Deformation - Variant -2

Deformation - Variant -3

Fig. 10. Deflection in Optimized Geometries

It can be seen that deformation in all variants are below allowable deflection of 5.25 mm. The maximum deflection is observed in variant-3, with value equal to 1.5mm. Form FE analysis can be seen all optimized variants are safe and satisfy the acceptance criteria. Comparison of all optimized variants against original design are summarized in Table **2**.

Table. 2. Summary of comparison

Girder Model	**Mass** Kg	**Percentage saving**	**Stress in Girder** MPa	**Deformation of Girder** mm
Original Model	203	--	38.36	0.88
Variant_1	191.7	5.53%	40.23	0.99
Variant_2	174.5	14.04%	44.65	1.2
Variant_3	164.5	18.97%	72.25	1.5

IV. CONCLUSION

This study has established a procedure for weight optimization of mechanical components using finite element tool. The following conclusions can be drawn from the current work presented in this report.

1. In FE analysis, it is observed a notable reduction in weight across the three variants. Specifically, we recorded a 5.53% weight reduction in variant-1, a 14.04% reduction in variant-2, and a substantial 18.97% reduction in variant-3.
2. In all optimized models, an increase in stress is observed. This increase is measured at 4.87% (from 38.36 MPa to 40.23 MPa) in variant-1, 16.4% (from 38.36 MPa to 44.65 MPa) in variant-2, and a significant 88.35% rise (from 38.36 MPa to 72.25 MPa) in variant-3. It's important to note that all these stress values remain well within the acceptable limits for steel.
3. The deformation in variant-1, variant-2, and variant-3 increased by 12.5%, 36.36%, and 70.45%, respectively. Notably, even with the highest deformation observed in variant-3 at 1.5 mm, it remains well below the permissible deformation limit of 5.25 mm.

REFERENCES

[1] Zuberi, Rehan H., Long Kai, and Zuo Zhengxing, Design optimization of EOT crane bridge. Eng. Opt., **2008**, 192-201.

[2] Patel, Harshil, Dhruv Upadhyay, and Divyang Patel, Design optimization of box girder in gantry crane using finite element analysis software. Int. Research J. Engineering and Technology, **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. J. 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**, 50-56.

[5] Suratkar, Abhinay, Vishal Shukla, and 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 Int. Conf. Heavy Machinery-HM. **2017**.

[7] Shijun, Song, et al., APDL language-based optimization of tower crane. Int. Conf. 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. Advan. 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. Int. Research J. 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 (3), **2019**.

[12] Shan, N., Li, K.L., Tang, W.X., Zhao, X.P. and Liu, X.J., Finite Element Analysis for the Jib System of the MQ2535 Crane. In Proceed. 6th Int. Asia Conf. on Industrial Engineering and Management Innovation: Core Theory and Applications of Industrial Engineering, Atlantis Press, Vol. 1, **2016**, 783-790.