Design Optimization of Box Girder in Gantry Crane using Finite Element Analysis Software

Harshil Patel [1], Dhruv Upadhyay [1], Divyang Patel [1]

[1]Scholars-Dept. of Mechanical Engineering, G H Patel College of Engineering & Technology, Anand, Gujarat, India

---***---

ABSTRACT: *Gantry crane is one of the most necessary material handling equipment for major industries. But in recent years less attention has been thrown towards changes in design of Heavy Gantry Crane Bridges and its optimization. Since, Gantry cranes are critical material handling equipment, manufacturers takes less attention towards the changes in design for safety and security purpose. But, now the cost of structural steels continuously increasing, optimization of design becomes necessary. Gantry crane manufacturers are required to give better cost effective products to industry users. The objective of this paper is to optimize design and analyze the 550 ton of capacity Gantry crane with welded box section of Girder. The design optimization is made by altering the dimensions of gantry crane girder sections and also the position of crab on the Girder by keeping other parameters constant. This approach includes comparison of the existing analytical results to that of the data obtained by finite element analysis and simulation software. The Main focus of the work is to modify the existing design with reduced cross section dimensions of girder of gantry crane to minimize the material utilization and cost. Simultaneously, an improved design should provide more safe design. While stimulating Finite Element Methodology, we have taken into consideration Shear Stress, Total Deformation, Maximum Principal Stress and Minimum Principal Stress for design optimization. Optimization includes change in design parameters such as size and thickness of plates. Optimal girder so designed is efficient in respect of design technique and verified as cost-effective due to 8.39% weight reduction from existing design.*

Key Words: Design Optimization, Finite Element Analysis, Double Girder, Gantry Crane

1. INTRODUCTION

Cranes are frequently used for different industrial applications and are essential part of any mechanical industry. Some of most commonly used industrial cranes are Gantry crane, Electric overhead travel(EOT) crane, Foundry cranes for foundry industries, Container handling cranes and Tower cranes The overhead gantry crane type is widely used to serve medium and heavy duty jobs, like a repair shop, buildings service, in a machine shop or also in construction of bridges. But for any type of industry crane with lightweight and high capacity is an ideal choice. To reach such requirement, a customized box section of crane girder is a motivating optimization research.

Gantry Cranes are widely used in the industries in order to assist the manufacturing process. A gantry crane is lifting equipment with single or double girder configuration supported by one leg or two legs. It moves on wheels or along a track or rail system. It is commonly used for loading or unloading ranging from small loads to heavy and bulky loads items both indoors or outdoors. It consists of various parts out of which major structural part is girder which is the longest and heaviest part of the crane. It forms the bridge between the two gantry girders on which the trolley moves. Trolley is the moving part of crane which moves on the girder.

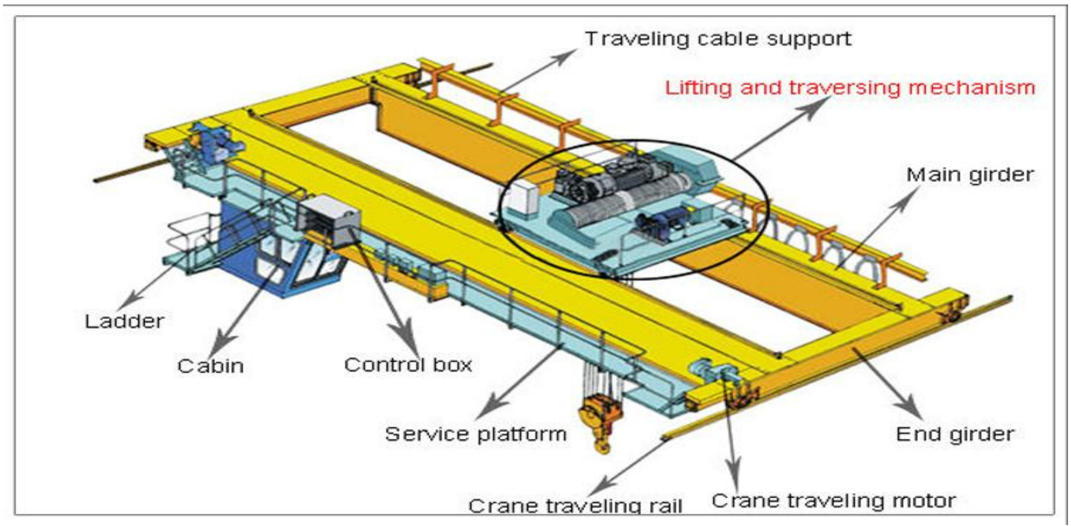


Figure 1: Double Girder Crane Layout

In the figure no. 1 shown above yellow part represents the structure (Girder). The blue part is known as the trolley which consists of crab mechanism which moves on the girder to lift the loads. The crane has three motions-Hoisting, cross travel and long travel. All these motions impose load on the structure. The whole crane is running on the gantry girder which again creates one source for developing the loads on the crane structure. Three prime movers along with power resource are required to provide the required crane motions. As discussed we have to target to reduce the crane weight so one such possible source for weight reduction is crane girder. In the next point crane girder is introduced.

1.1 Crane Girder

Mainly crane girder is an assembly of steel plates, rails and angles with steel plates forming the flanges and web of the girder while the angles acts as stiffeners for girder. The web is supported internally by diaphragms usually steel plates together all forming box section. Sectional view of girder is shown below:

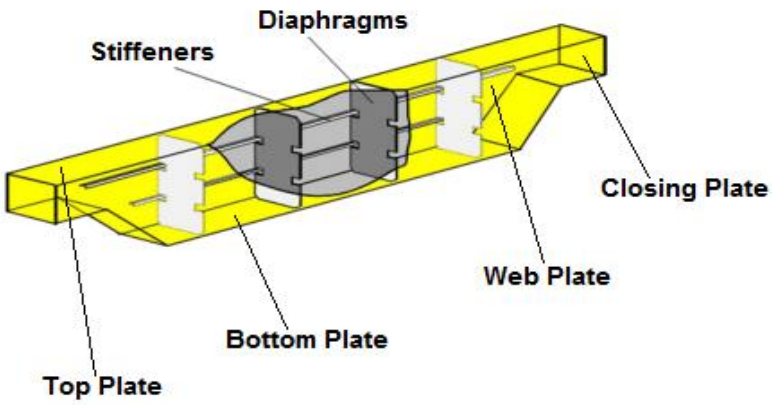


Figure 2: Crane Girder

The box section of girder is fabricated by welding the steel plates with each other and then after completion of box section rail is welded on top flange of the girder to facilitate the trolley movement. It should be noted that, it is not necessary to have the girder of box section only. It may be of I beam shape, tapered box section or another sections. The selection of girder section mainly depends on the load carrying capacity of the crane. Usually for small capacity cranes I section with I-beam is preferred and those cranes having high capacity loading are required to have box section to satisfy high load carrying capacity. Fabrication work for girder manufacturing is simple as it has wide allowable tolerance so it don't require high skilled workers. As box section is usually having high sectional properties and also has high torsional strength property making it more reliable for high capacity crane girders whereas due to low lateral rigidity observed in I beam section, box section are ideal choice for heavy duty girders.

2. LITERATURE REVIEW

N. Raghu Prasad, Jeeoot Singh [1], in this paper author investigates into the Buckling of the plates. Modal analysis using Finite Element Method (FEM) is used to determine natural frequencies and mode shapes. These plates are simply supported ends. Various methods of meshing were used to get optimized results.

Indian Standard (807-2006) [2], this standard describe design of structural portion for cranes, hoists, specifies permissible stress and other details of the design. In order to ensure economy in design in reliability in operation .To deal with the subject conventionally, cranes have been broadly classified into eight categories based on their nature of duty and number of hours in service per year. It is procedure or manufactures responsibility to ensure the correct classification.

Abhinay Suratkar and Vishal Shukal [3], in this paper author has done Three Dimensional Modeling and Finite Element Analysis of EOT Crane and made a comparison between the analytical calculations and FE analysis. As a result study they have proposed the design optimization method for Overhead crane.

Patil P. and Nirav K. [4], in "Design and analysis of major components of 120T capacity of EOT crane" analyzed various components of crane like wheels, pulleys, rope drum and girder. They have done the manual calculations using Indian standards and on the basis of these calculations 3D modeling and analysis has been carried out. For modeling they have used Creo software and ANSYS as analysis software.

Rudenko N. [5], in the book of Material Handling Equipment briefed the structure of overhead travelling crane. The structure of an overhead travelling crane with a plate girder is composed of two main longitudinal girders assembled with the two end carriages which accommodate the travelling wheels. The main factors in the solution of plate girders are the safe unit bending stress and the permissible girder deflection. The vertical loads on the girders are dead weights and the force exerted by the wheel of the trolley carrying the maximum load.

Design Optimization of Overhead EOT Crane Box Girder Using Finite Element Analysis Abhinay Suratkar et al. (2013) [6], in this paper the design optimization of double box girder has been done and a comparative study of results of finite element analysis of a crane with 10 ton capacity and 12 m span length has been conducted. The crane design was modelled with solids; material, Loads and boundary conditions were applied to solid model. Finite Element meshes were generated from the solid model. After a comparison of the finite element analyses, and the conventional calculations, the analysis was found to give the most realistic results. As a result of this study, a design optimization for an overhead crane box girder has been done. In this paper, the comparison between the analytical calculations and the finite element analysis results were investigated. We have reduced the overall mass of the girder by 29%.

3. OBJECTIVES OF OPTIMIZATION

The main aim of our research is to reduce the gap between the values of allowable stress (considering factor of safety) and the values of stress obtained by FE Simulation and Analysis. For our purpose, we worked on the existing design of 550 ton gantry crane and also carried out FE simulation and analysis on the same. This will lead us directly to,

- Optimized design of the girder
- Reduces the Gantry girder size and its weight
- Small Reduction in the motive power for movement of crane
- Reduced power consumption leads to reduction in the overall running cost from existing version of crane

4. ANALYTICAL APPROACH

As it is known that any software design validation is considered null without the analytical proof & any analytical proof has the standard as its base. The analytical study was made on the girders of the gantry cranes taking into account the following considerations:

1. The girder is supposed to be a simply supported beam having fixed ends.
2. The critical loads acting on the girder are: Two pair of point loads of 1668000 N & 1392000 N along with the uniformly distributed load of 830404.32 N and a gravity force acting on the top flange of the girder.

3. Other loads acting on the girder (if any) are neglected as their values are too small.

So while preparing the research content, the standards from IS-807_2006, IS-3177_2006, CMMA #70, ASME B30.2_2005 were adopted. The material selected for this design was IS 2062 E 350B while since crane is involved with the human application referring to IS-807_2006, Factor of Safety (FoS) is 1.5. Hence the permissible principal stress is 219.92 MPa.

The results for existing design from the performed analytical calculations are shown in table below:

Output	Value
Total Deformation	20.82 mm
Maximum Principal Stress	155.43 MPa
Minimum Principal Stress	155.43 MPa
Shear Stress	39.62 MPa

Table 1: Analytical Results for Existing Design

Since it is known that analytical equations take many assumptions, there is a permissible variation in values of the stresses and deformation.

Output	Analytical Value	ANSYS Value	Variation
Total Deformation	20.82 mm	20.226mm	2.85%
Principal Stress	155.43 MPa	145.29MPa	6.52%
Shear Stress	39.62 MPa	39.35MPa	0.68%

Table 2: Comparison between Analytical Results and ANSYS Results for Existing Design

5. FINITE ELEMENT ANALYSIS

Finite element analysis (FEA) is a computerised method for predicting how a product reacts to real-world forces, vibration, heat, fluid flow and other physical effects. Finite element analysis shows whether a product will break, wear out or work the way it was designed. It is called analysis, but in the product development process, it is used to predict what's going to happen when the product is used.

Finite element analysis works by breaking down a real object/structure into a large number (thousands to hundreds of thousands) of finite elements, such as little cubes called as meshing. Mathematical equations help predict the behaviour of each element. A computer then adds up all the individual behaviours to predict the behaviour of the actual object/structure.

Finite element analysis helps predict the behaviour of products affected by many physical effects, including: mechanical stresses, mechanical vibration, fatigue, motion, heat transfer, fluid flow, electrostatics and many more.

The finite element software can analyse any geometry, and solves both stresses and displacements with respect to the known applied loads. In this study finite element meshing and analysis, is carried out by means of the ANSYS 2018.

6. 3-D MODELLING AND IMPLEMENTING FINITE ELEMENT ANALYSIS

A 3-D model is a digital representation of the geometry of an existing or envisioned physical object. Designers may specify points, curves, and surfaces, and stitch them together to define complete representation of the structure with its boundaries of the object. Alternatively, they may select models of simple shapes, such as blocks or cylinders, specify their dimensions, position, and orientation, and combine them using assembly constraints, union, intersection or difference operators. In finite element analysis, the 3-D model prepared is called in the analysis software interface and then meshing is done followed by assigning various type of end supports to boundary of model and then all the forces are applied according to their nature and value. Finally, analysis is carried out by the software and results are generated. Based on the results further action on the model is carried out from initial stage by changing dimension of the 3D model and all the procedure of meshing, choosing supports, applying loads, generating results are carried out again until satisfactory results. Our workflow for finite element analysis using software is shown in Figure no. 3.

First, the double girder of crane is modelled as a solid. Solid modelling of overhead double girder crane has been done as per the technical specifications. The solid model as shown in Figure no. 4. For getting the results from stress analysis, the following task were performed as follows, first assign the material for the each part of the box girder, sets the safety factor as the yield strength, the maximum permissible yield strength value has been set to 219.92 N/mm2. Maximum allowable deflection as per standard is 37.78 mm. I.S. 2062 E 350B material has applied to girder parts. After assigning the material, the boundary conditions have been set as fixed constraint. Contact condition of box girder set to bonded (welded) has been set. Two pair of point loads of 1668000 N & 1392000 N is applied on the top flange of the girder along with the uniformly distributed load of 830404.32 N and a gravity force. Average element size is 500 mm. Later, a mesh is created. The number of nodes were created is 109776 and the elements were 42008. In this study, a tetrahedral type element is used. The solid meshed model as shown in Figure no. 5.

550 ton double girder double crab gantry crane specifications	
Main Hoist Capacity	550 t
LT Wheel load	55 t
Crane Span	34000 mm
Crane Capacity + BPB Wt	480000 kg
Duty Factor	1.11
Impact Factor	1.15
Hoist Speed	0.02 m/sec
Travel Speed	0.33 m/sec
Wt. of Crab	60000 kg
Wt. of Crane	500000 kg
No. of Driven Wheels	4 Nos.
Distance between Crab wheels	500 cm
No. of CT Wheels	8 Nos.
Material	E350 or ST 52
Min. distance of hook from L.H.S	500 cm
C.T Rail Height	15.00 cm
No. of wheels of L.T.	32 Nos.
Dead Load (Self Wt. of Platform)	110 kg/m
Point Load (Wt. of Lt mechanism, Control Panel and accessories)	3000 kg
Wt. of Crab	60000 kg
Lifted Load	480000 kg

Table 3: Specification Sheet for 550 ton Gantry Crane

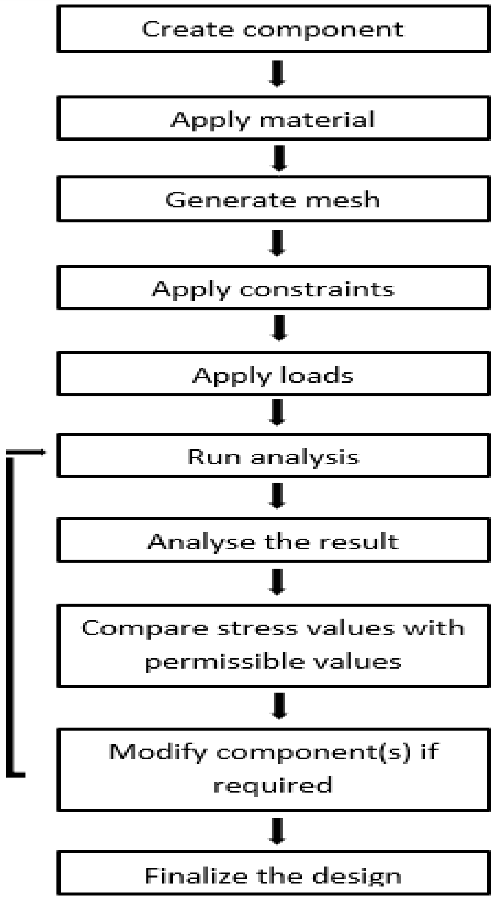


Figure 3: Flow chart for analysis process

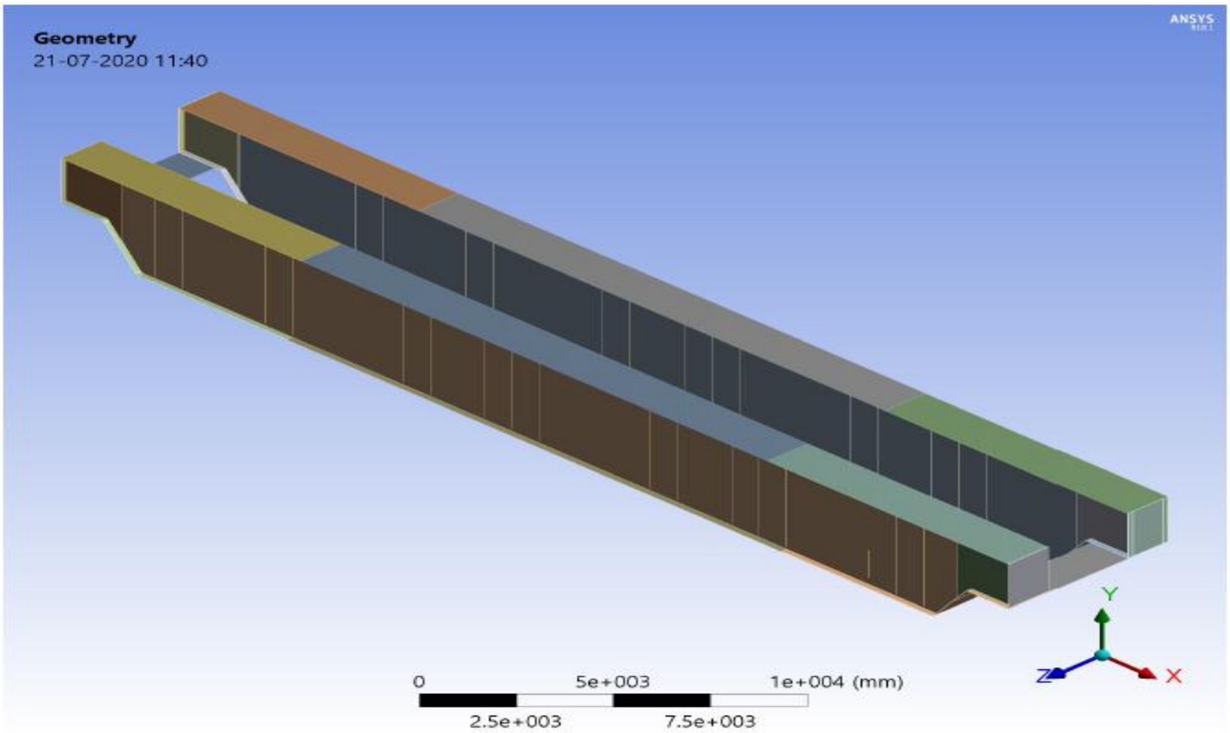


Figure 4: Solid Model of Double Girder

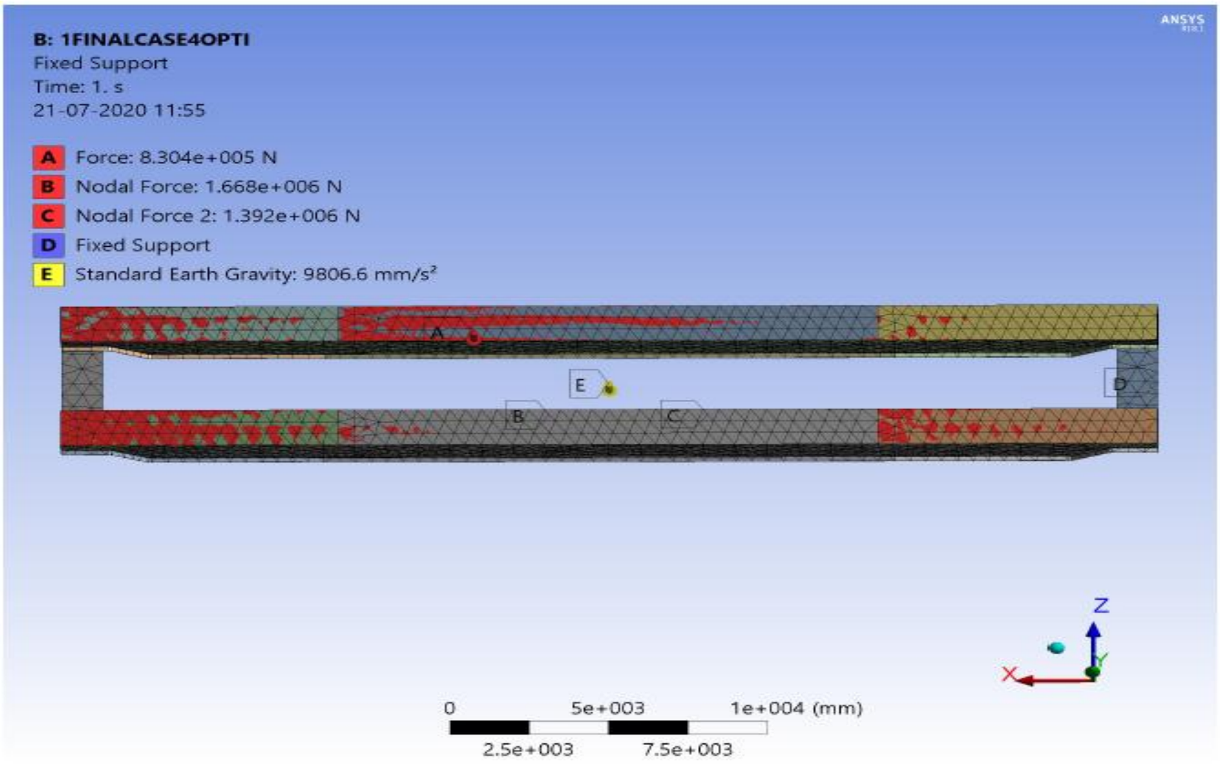


Figure 5: Solid Meshed Model with Tetrahedral Element

7. DESIGN OPTIMIZATION PROCEDURE

Girder acts as beam and provides primary support for any type of industrial crane and also is responsible for transferring the load down to the foundation. Also, girder is heaviest part of any crane hence, for optimization of the above mentioned

550 tons Gantry crane changes were made in dimensions of the box girder based on results obtained from analysis software. The optimization of Gantry crane girder is carried out in two stages.

STAGE 1:

In this stage, after studying and analysing various capacity cranes, the load transfer pattern is as via Hook-rope-rope pulley-crab mechanism-crab wheels-rail-girder (girder top plate-girder web plates)-leg and finally to foundation. Also, it is obvious that the stresses generated will be maximum when the position of the crab mechanism which travels on rails having transverse motion is at the centre of the girder span. It was observed that with the crab at the centre and load transfer pattern, the thickness of the web plates can be altered by changing the position of rails on top plate of both the girder. Hence, the position of the rails was changed and on analysis the position keeping it at all possible positions at centre, at outer side and at inner side of top plate maximum stresses generated are obtained when the rail is at extreme inner side of the top plate of the girder. So, the thickness of the outer web plate is reduced and on analysing the resultant stresses were in allowable range.

STAGE 2:

In this stage, difference of the values of allowable stresses (considering factor of safety) and resultant stresses is lessen. Usually for such large capacity gantry crane an E-room is provided for maintenance and operating purpose and so it becomes mandatory to have some minimum height of the web plate throughout the span of the girder. For such large height of web plate, on analysing the result large difference in numerical values observed between allowable and resultant stress values. Based on the above observation the position of the E room is shifted outside one of the girder end and platform is provided outside of the girder for maintenance purposes and the height of the web plate is reduced by definite intervals of about 50 mm and each step is analysed for all type of failure and value of the stresses generated are in allowable limit.

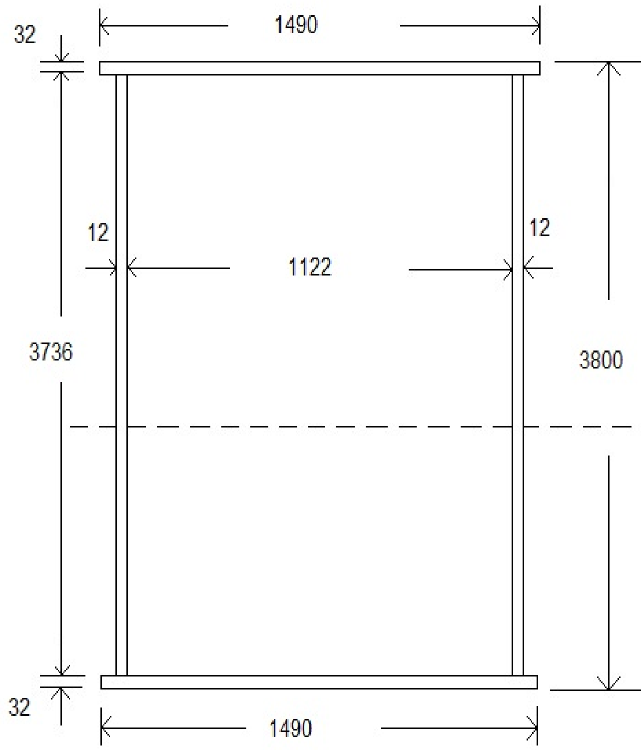


Figure 6: Cut Section of Girder as per Existing Design

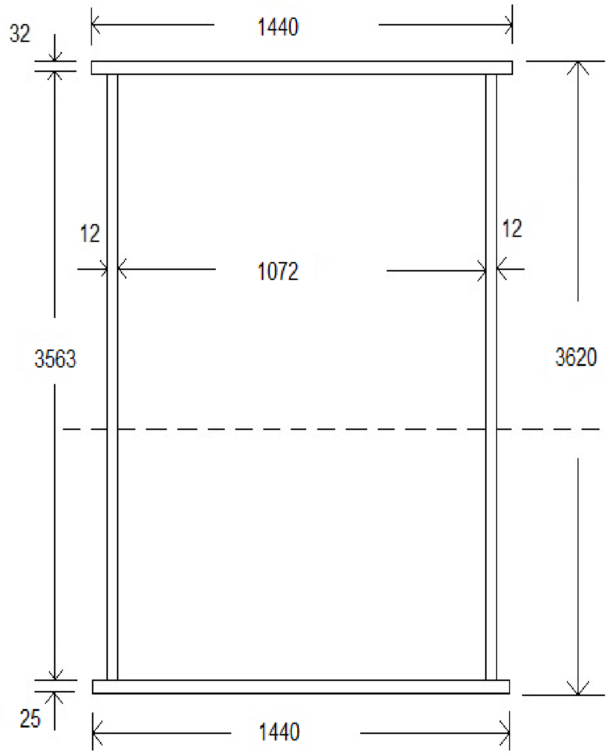


Figure 7: Cut Section of Girder after Optimization

8. RESULTS OF FINITE ELEMENT ANALYSIS

A four-node tetrahedral element was used for finite element analysis, using the girder solid model generated by means of ANSYS 2018 version. The maximum bending moment is occurring at the midspan of the girder. Young's Modulus (E) is 200 GPa and the Poisson Ratio is 0.3 for finite element analysis. The maximum principal stress of the complete box girder is

152.43 N/mm^2 shown in Figure no. 8. It is clearly seen from the stress diagram that the maximum principal stress is developed at the centre of the bottom plate of girder. The minimum principal stress of the complete box girder is 165.49 N/mm^2 shown in Figure no. 9. It is shown in the stress diagram that the minimum principal stress is developed at the centre of the top plate of girder. The shear stress of the complete box girder is 42.67 N/mm^2 which is shown in Figure no. 10. It is observed from the stress diagram that the shear stress seems to be maximum at the end section of the girder. The displacement of the modelled overhead crane girder was obtained from Finite Element Analysis, and is occurring at the mid span of the girder, illustrated in Figure no. 11. The value of maximum displacement of the girder is about 19.333 mm.

For the above solid model of 550 tons crane girder, meshing of four-node tetrahedral element was done for finite element analysis using the 2018 ANSYS version.

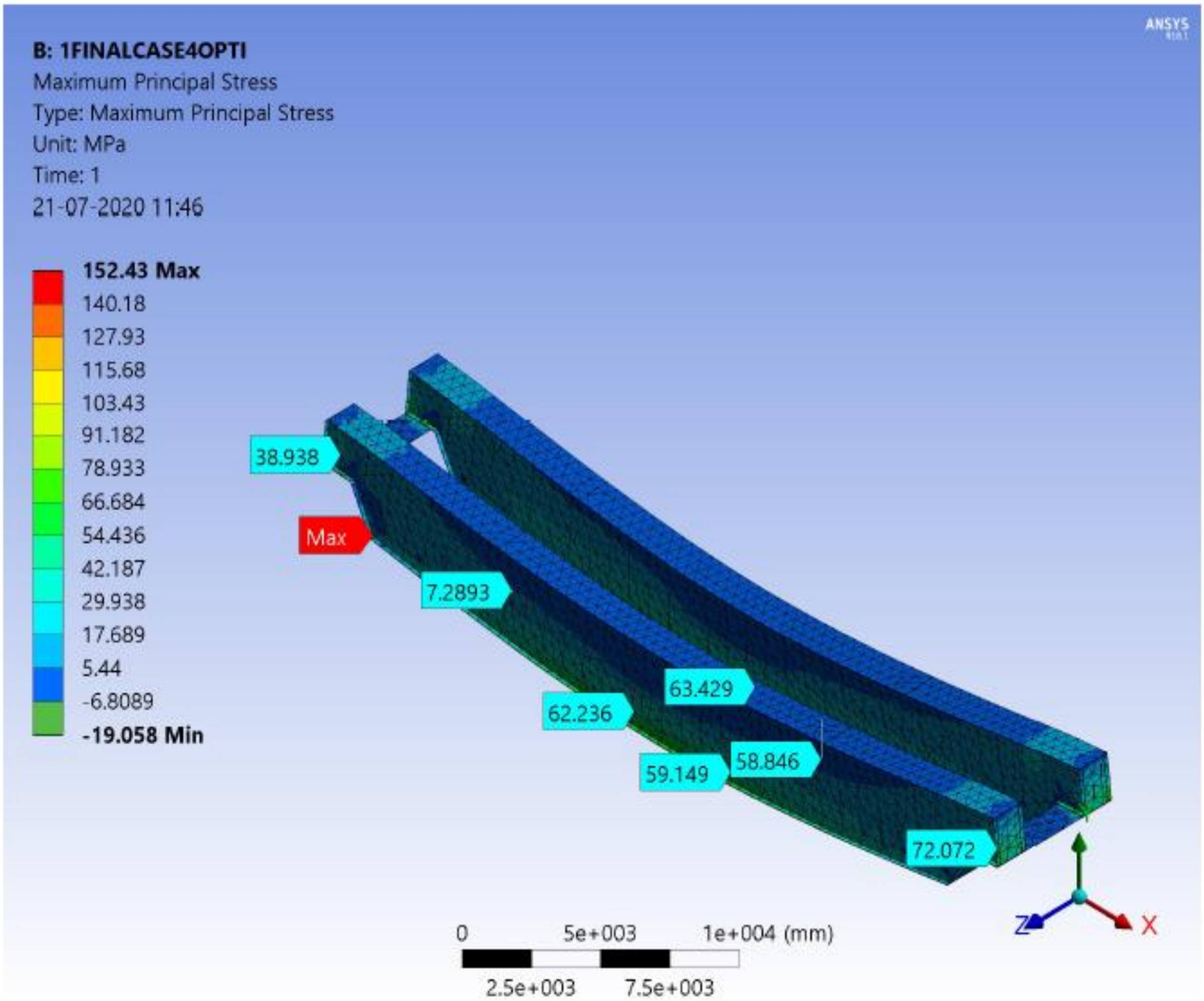


Figure 8: Maximum Principal Stress

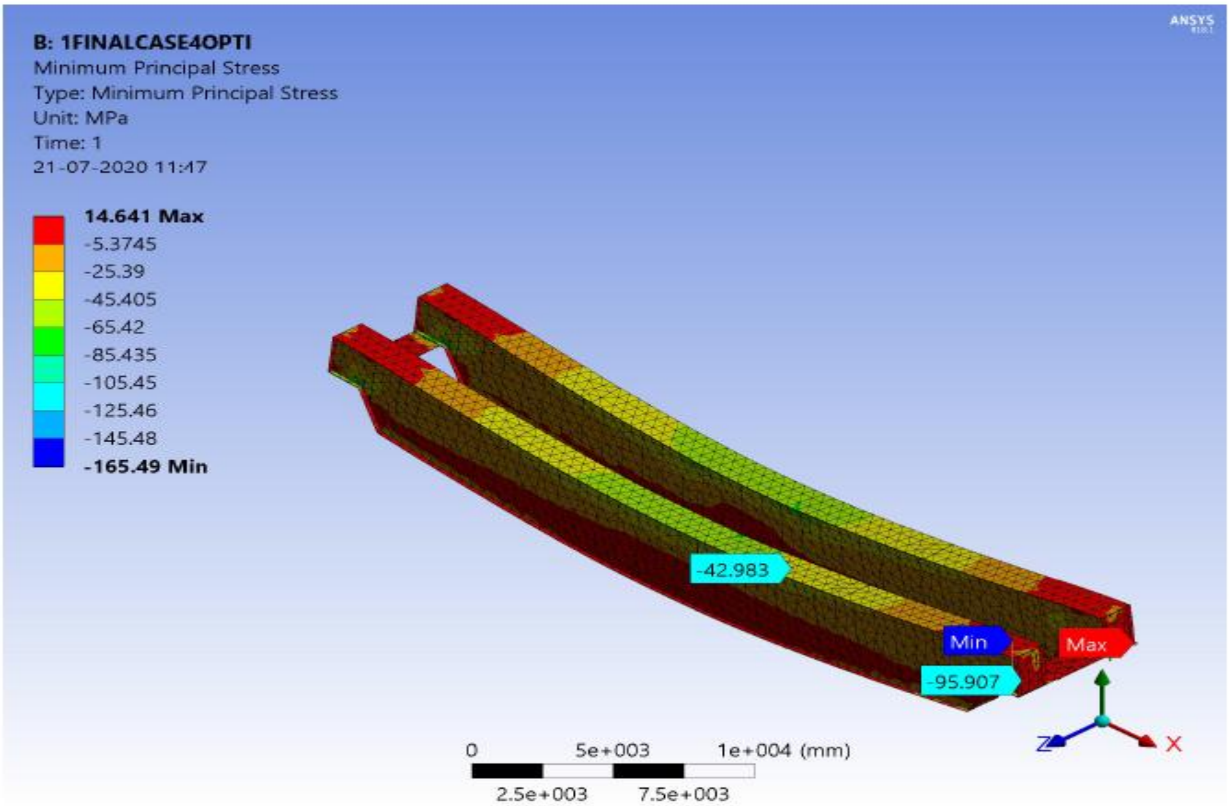


Figure 9: Minimum Principal Stress

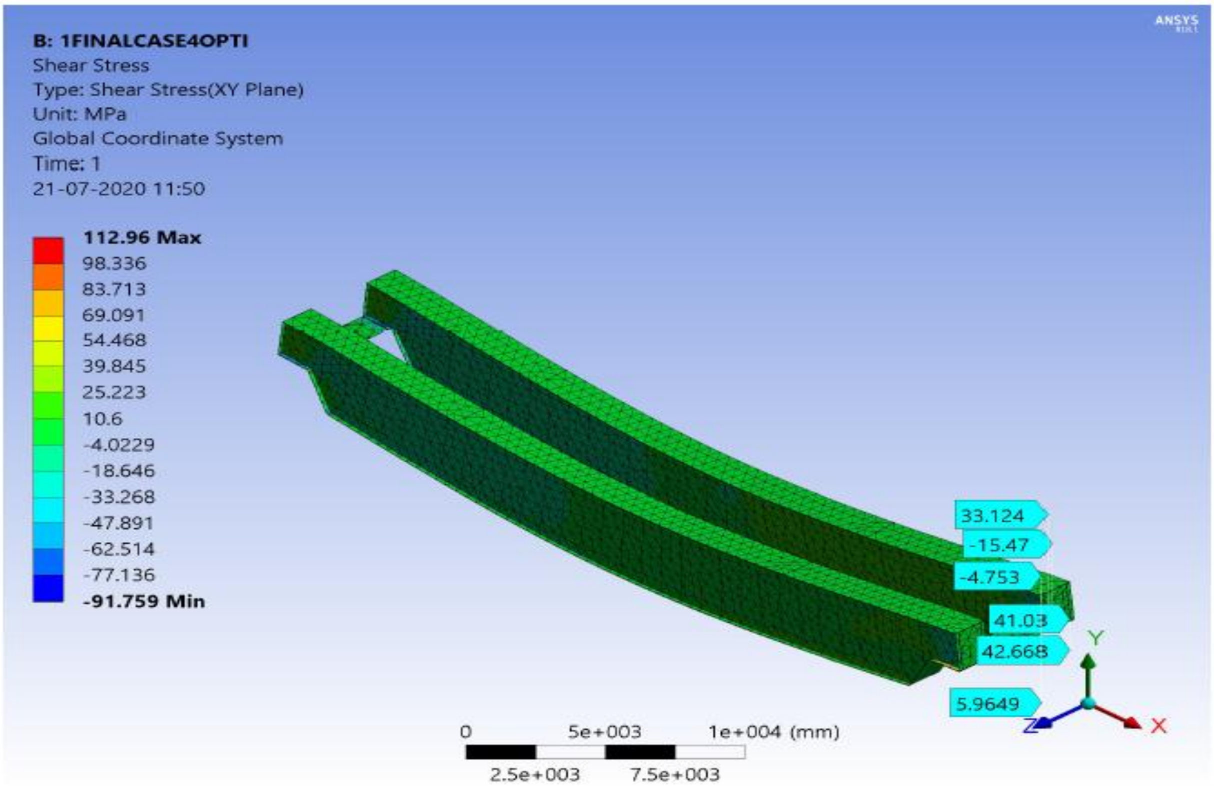


Figure 10: Shear Stress

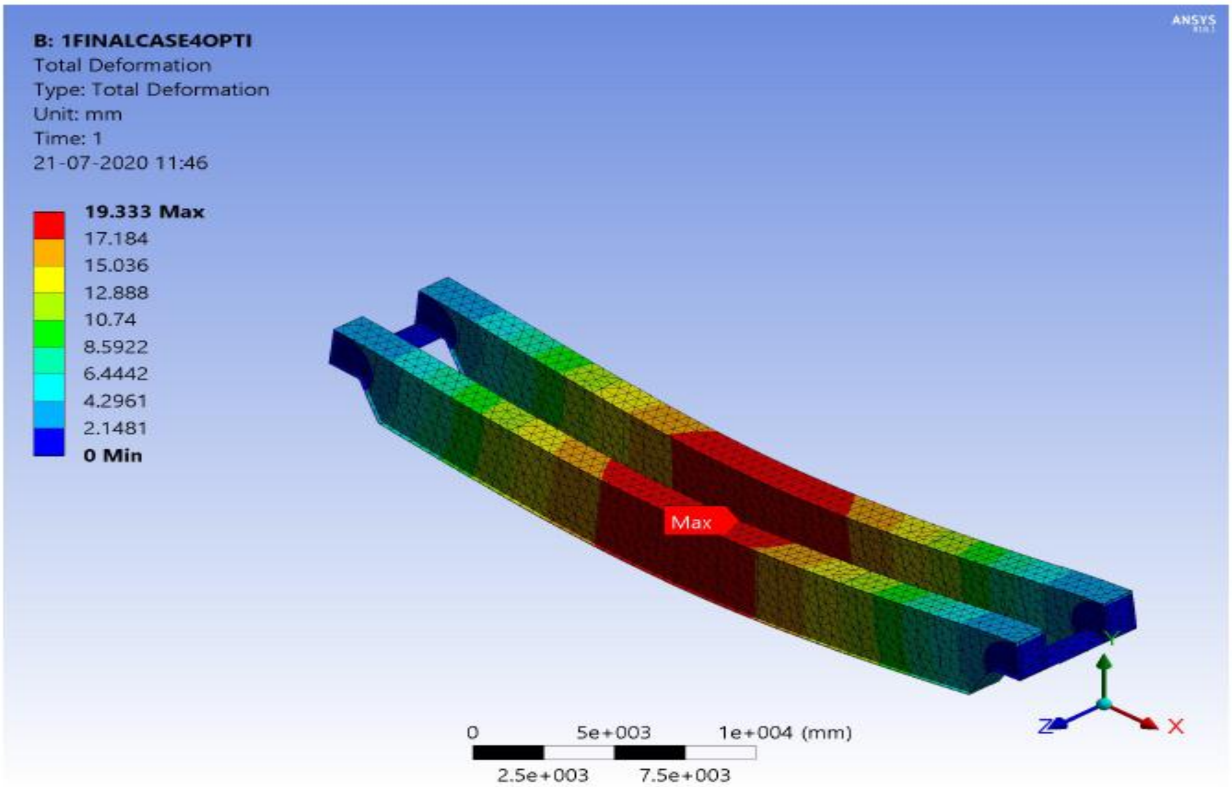


Figure 11: Total Deformation

Output	Value	Subjected Section
Total Deformation	19.333 mm	Centre of girder
Maximum Principal Stress	152.43 MPa	Centre of the bottom plate of girder
Minimum Principal Stress	165.49 MPa	Centre of the top plate of girder
Shear Stress	42.67 MPa	End section of the girder

Table 4: Results

9. CONCLUSION

In this paper, there is a comparison shown between the results from existing design and the results from optimized design using finite element analysis of solid model girder as shown in table no. 5. From above comparison between the allowable parameters of Indian Standard codes which includes maximum principal stress, minimum principal stress, shear stress and deformation. From the results of finite element analysis of optimized re-designed box girder, it is clearly seen that the stress & displacement which are obtained from the Finite Element Analysis are within the allowable limit of the Indian standard codes. Hence, from the above results and comparison it is quite clear that the optimization of 550 tons Box type Double Girder Gantry Crane has been achieved without compromising the required parameters and also its overall strength and rigidity. Here, there is reduction of 8.3893% in overall mass of girder which is equivalent to 5.24 tons of structure material. This results in reduction of overall cost of crane and the power consumption of crane.

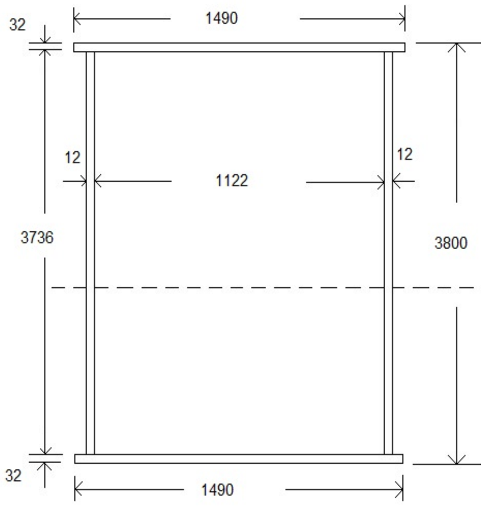
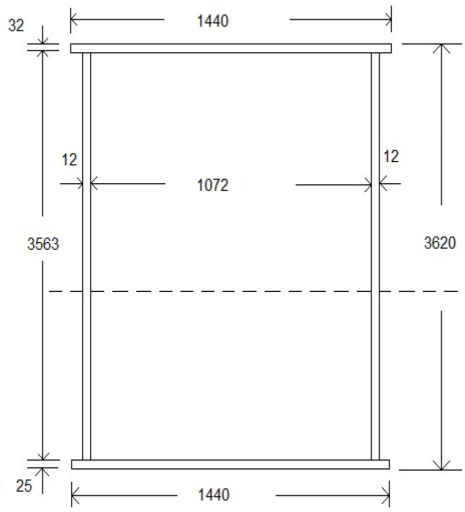
COMPARISON CHART		
	Results of Existing Design (ANSYS Results)	Results of Optimized Design (ANSYS Results)
Girder Cut Section		
Weight, Kg	62460	57220
Ixx, mm^4	44277616.6	35323241.1
y, mm	1900	1810
Zxx, mm^4	233040.0874	184420
Maximum Principal Stress, MPa	110.76	152.43
Minimum Principal Stress, MPa	145.29	165.49
Shear Stress, MPa	39.35	42.67
Total Deformation, mm	20.226	19.333

Table 5: Comparison Chart

ACKNOWLEDGEMENT

This work is carried out with Anupam Industries Ltd, Vithal Udyognagar, Anand. We are thankful to Mr. Mokshesh Shah, Senior Design Manager & Mr. P. Jha, Senior Vice President for their full cooperation and constant encouragement during this work. We are also thankful to our project college guide Dr. Hemant Thakkar, G. H. Patel College of Engineering and Technology, Anand, Gujarat for his continuous support and guidance.

REFERENCES

1. Prasad N Raghu, Singh Jeeoot, Buckling analysis of rectangular plates with cut-out and partial edge Compression, IARJSET, 2015, pp 126-129.

2. Indian Standard Design, Erection and testing (Structural portion) of cranes and hoists code of practice (Second revision).IS 807:2006.

3. Abhinay Suratkar, Vishal Shukla “3-D Modelling and Finite Element Analysis of EOT Crane”- International Journal of Mechanical and Production Engineering, ISSN: 2320-2092, Volume-1, Issue-2.

4. Patil P. and Nirav K, "Design and analysis of major components of 120 Tones capacity of EOT crane", IJEDR 2014, ISSN: 2321-9939, volume-2, issue 2.

5. N. Rudenko, Material Handling Equipment, 2nd edition, Envee Publishers, New Delhi.

6. Abhinay Suratkar, Vishal Shukla, Dr. K. S. Zakiuddin, "Design Optimization of Overhead EOT Crane Box Girder Using Finite Element Analysis", International Journal of Engineering Research & Technology (IJERT), Vol. 2 Issue 7, July - 2013

7. Electrically Operated Travelling Crane Design as per IS-3177:1999 (reaffirmed: 2006) by Bureau of Indian Standard.

8. Design, Erection and Testing of Cranes and Hoists Code of Practice as per IS-807:2006, by Bureau of Indian Standard.

9. Overhead and Gantry Cranes (Top Running Bridge, Single or Multiple Girder, Top Running Trolley Hoist) "ASME B30.2-2005 (Revision of ASME B30.2-2001)".

10. MITREV Rosen" Mathematical modelling of translational motion of rail-guided cart with suspended payload" Zhejiang University SCIENCE a ISSN 1673-565X (Print)-2007.

11. Design Optimization of EOT Crane Bridge, Rehan H Zuberi, Dr. Long Kai 2, Prof. Zuo Zhengxing3, and EngOpt 2008 - International Conference on Engineering Optimization, Rio de Janeiro, Brazil, and 1 - 05 June 2008.

12. Farkas, J. Jarmai, K. Multiobjective optimal design of welded box beams Microcomputers in Civil Engineering. V 10 n 4 Jul1995. P 249- 255.

13. Gibczynska, Teresa. Sularz, Stanislaw. Optimization of shape of the box girders. Archiwum Budowy Maszyn, V 34 n 2 1987 p 153-171.

14. P. Hanjun, X. Xiaopeng, L.G.,Y. X., & P. Haining, "Analysis for Dynamic Characteristics in Load-lifting system of the Crane", Procedia Engineering, 16(2011) 586-593.

15. W. X., Chena B.O, Z. Dan, & L. Jian, "The Research on Optimal Design of Large Metallurgical Crane", Procedia Engineering, 24 (2011) 783- 787.

16. Kulshresta, A. & Prashant, A., "FEA Analysis for Customized Overhead Gantry ", International Journal of Emerging Research in Management & Technology, Vol.4 (3), 2015, pp. 15–18.

17. Patel, P.R., Patel, B.J., & Patel, V.K., "Design, Analysis and Optimization of Overhead Crane Girder", International Journal of Advance Engineering and Research Development, Vol.1 (6), 2014, pp. 1–10.

18. Patel, P., & Kamdar, N., "Design and Analysis of Major Component of 120 Tones Capacity of EOT Crane", International Journal of Engineering Development and Research, Vol.2 (2), 2014, pp. 2150– 2154.

19. Patel, P.R., & Patel, V.K., "A Review on Structural Analysis of Overhead Crane Girder Using FEA Technique", International Journal of Engineering Science and Innovative Technology, Vol.2 (4), 2013, pp. 2008–2011.

20. Zhou, C., Wan, J., Chen, X., & Shuai, F., "Structure Improvement and Fatigue Analysis of Single-Girder Crane Main-Beam", Advanced Materials Research, Vol.479–481, 2012, pp. 2099–2103.

21. Sankar, V.V.A. Vijayan, P.D., & Ashraf, I., "Reducing the Structural Mass of a Real-World Double Girder Overhead Crane", International Journal of Advances in Engineering & Technology, Vol.8 (2), 2015, pp. 150–162.