International Journal of Advance Engineering and Research Development (IJAERD)
Volume 1,Issue 6,June 2014, e-ISSN: 2348 - 4470 , print-ISSN:2348-6406

Design, Analysis and Optimization of Overhead Crane Girder

Pratik R Patel[1], Bhargav J Patel[2], Vipul K Patel[3]

[1]*M.E student in Mechanical Department, S K Patel Engineering College, Visnagar*
[2]*Assistant Professor in Mechanical Department, K J Faculty of Engineering, Vadasma*
[3]*Assistant Professor in Mechanical Department, S K Patel Engineering College, Visnagar*

Abstract: In the current material handling equipment the life of the overhead crane as well as the cost of the material handling equipment is too important to stay in the competitive market of the industries. The cost of the material handling equipment is depend on the weight of the material. The performance of the material handling equipment will be done by the working on the optimization of the overhead crane used in the industries. In this project, Design, Analysis and Optimization of the overhead bridge crane girder for the 75 ton capacity is carried out by changing the dimensions of section of overhead crane girder. In Design we are using 750 x 450I section for girder and calculate stress (98.251 N/mm^2) ,deformation(9.8844 mm), safe factor(0.87734),and weight(26976 Kg).In the Analysis of girder analyze of von misses stress (98.251 N/mm^2) ,deformation(9.8844 mm), safe factor(0.87734).
From analysis and analytically we found weight of girder is higher so that optimization is required for the optimize weight of girder.In the optimization, we have selected different sizes (740 x 440, 730 x 430, 720 x 420, 710 x 410, 700 x 400**)** of I section of girder. We have found out deformation, stress and weight of girder for different section. Fromthat, Optimum section (700 x 400) is found out. For Optimum section, Value for von misses stress is 117.61 $/mm^2$. Value for Deformation is 12.228 mm and Value for Weight is 24455.

Keywords — Overhead, FEA analysis, von misses stress

I. INTRODUCTION OF MATERIAL HANDLING

Expressed in simple language, Material handling is loading, moving and unloading of materials. To do it safely and economically, different types of tackles, gadgets and equipments are used, when the material handling is referred to as mechanical handling of materials. Since primitive men discovered the use of wheels and levers, they have been moving materials mechanically. Any human activity involving materials need material handling. However, in the field of engineering and technology, the term material handling is used with reference to industrial activity. In any industry, be it big or small, involving manufacturing or construction type work, materials have to be handled as raw materials, intermediate goods or finished products from the point of receipt and storage of raw materials, through production processes and up to finished goods storage and dispatch points.
Material handling as such is not a production process and hence do not add to the value of the product. It also costs money; therefore it should be eliminated or at least reduced as much as possible. However, the important point in favors of material handling is that it helps production. Depending on the weight, volume and throughput of materials, mechanical handling of materials may become unavoidable. In many cases, mechanical handling reduces the cost of manual handling of materials, where such material handling is highly desirable. All these facts indicate that the type and extent of use of material handling should be carefully designed to suit the application and which becomes cost effective.
Material handling uses different equipment and mechanisms called Material Handling Equipment. Though in one of the definitions, processing operations and inspection have been specifically excluded from scope of material handling operations, it is worth mentioning that in specific cases

processing or inspection of materials may be accomplished simultaneously with handling activity. [1]

II. LITERATURE REVIEW

AbhinaySuratkar, Vishal Shukla, Dr. K. S. Zakiuddinhas done optimization of double box girder 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. In the initial phase of the study, conventional design calculations proposed by Indian Standard Rules were performed. The crane design was modelled with solids, Loads and boundary conditions were applied to solid model. Assign material to the 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 table. From the above comparison between the allowable parameters of Indian Standard codes and the results of finite element analysis of re-designed box girder, it is clearly seen that the maximum stress & displacement which is obtained from the Finite Element Analysis are within the allowable limit of the Indian standard codes. The safety factor is on higher side against the Indian standard codes. Thus from the above results, we can state that the design of EOT crane box girder has been achieved without compromising the strength and rigidity. They have reduced the overall mass of the girder by 29%. As the overall mass of the girder has reduced, the initial cost for the structural building, civil work and electrical consumption for the crane has also reduced. [2]

Table No. 2.1 Comparison

Values of Finite Element Analysis for Re-Designed EOT Crane Box Girder			
Sr. No	**Description**	**Allowable Parameters as Per IS:3177 & IS:807**	**Results From FEA**
1	Maximum Stress	166Mpa	110Mpa
2	Minimum Safety Factor	1.5	1.96
3	Maximum Displacement in Y-Direction	16mm	3.13mm

AbhinaySuratkar, Vishal Shukla had considered design of an overhead crane with a double box girder has been investigated and a study of a crane with 10 ton capacity and 12 m span length has been conducted. With regard to this, the creation of 3-D models for researching and analyzing the behavior of an overhead crane, becomes the main goal of the present working the initial phase of the study, conventional design calculations proposed by Indian Standard Rules were performed. The crane design was modeled with solids Loads and boundary conditions were applied to solid model. Assign material to the 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 method for an overhead crane is proposed. A four-node tetrahedral element was used for finite element analysis, using the girder solid model generated by means of Inventor software 2012. The maximum bending moment is occurring at the mid-span of the girder. In this study, the comparison between the analytical calculations and the finite element analysis results were investigated. In order to show how the finite element analysis of overhead crane bridge can be done. The maximum stress value is 53 N/mm2 for a four-node tetrahedral element. Taking into account the safety factor, which is obtained from Finite Element Analysis are on higher side against the conventional standards as well as the stress values obtained from the finite element analysis is very much low than the allowable stress. The maximum displacement is on lower side against the allowable deflection. Thus from the above study, the design optimization of EOT crane Girder Bridge shall be proposed. [3]

CameliaBretoteanPinca et.al used shell type elements with three or four nodes per element in order to find out the best sizes for resistance structure in tension and deformation state. Here cosmos software was used for analyzing the tension and deformation state of the resistance structure of an overhead crane bridge. This was performed in iron and steel department of continuous casting. The maximum equivalent tension calculated according to the theory of the specific form modifying energy (the theory of von misses). They had distributed the tensions within the resistance structure of the crane bridge more appropriately. All features which causes tensions and deformations in resistance structure were described mathematically by differential equations. In this way like evaluation of stress state and pointing out the critical areas we can increase solidity and bearing capacity of the strength structure for the rolling bridge. After analyzing the stress fields, they were realized by the fact that there are two critical areas that we have to take into consideration.

1. The connection between the longitudinal beam I and the right-end beam.
2. The area of the right-end beam near the global axis system [4].

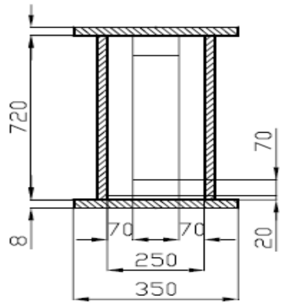

Fig - 2.1 The cross section of the resistance elements

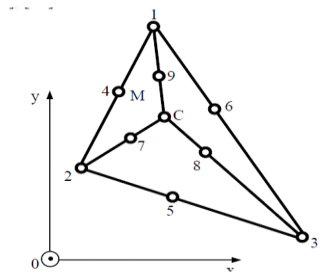

Fig - 2.2 Evolved finite element of shell type thin plate triangular finite element

Ismail Gerdemeliet.al.had developing new techniques of finite element method and here they had used new techniques rather than using old finite element methods. All calculations of elements related rubber tired container stacking crane were done and then it was modeled. In addition of this, they stress and deformation analysis of crane bridge girder and buckling analysis of the crane legs were performed.Ansys workbench was also used for finite element method and modeling was done on Autodesk INVENTOR 2010 program. Comparison of calculations regarding Stress, deformation and buckling analysis were done by author.There is no significant difference between the analysis and calculation result for the stresses and deformations. Therefore analysis result can be taken into consideration.The main aim of this work is to achieve best finite element methods than conventional methods for getting advantage of new methods. Result shows that stress values remain under the yield strength of the steel which was used for Crane Bridge and legs. They concluded that it is new method but it gives better result than conventional method. [5]

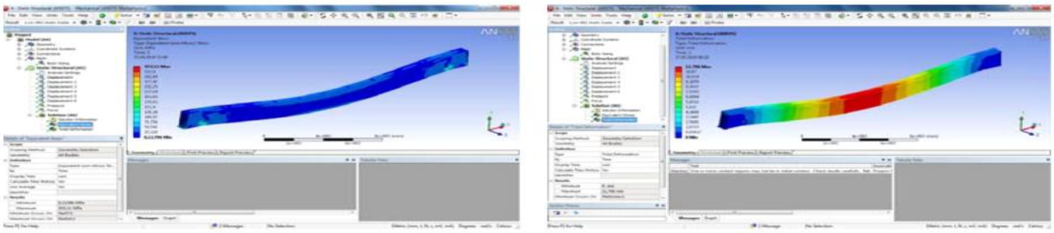

Fig- 2.3 Stress and Deformation analysis result of the bridge girder

M. Euler, U. KuhlmannstudiedaboutMulti axial fatigue problem is created due to wheel load and block rail fastened by longitudinal fillet welds. At the point of wheel load application the top region of the runway girder is subjected to a stress field comprising local stress components induced by the concentrated load. here author had tested two crane runway girders under proportional multi axial loading, i.e. under flexural bending and stationary wheel load, both simultaneously pulsating. The paper presents the fatigue evaluation of these tests using different local concepts. Crane runways are subjected to a complex multi axial state of stress due to geometric and metallurgic notch effects and the introduction of concentrated loads. The amplitudes of the stress components do not occur simultaneously resulting in an alternating direction of the shear stress that has been recognized to control the crack initiation. [6]

III. DESIGN OF GIRDER

3.1.1 Specifications of Overhead Crane

Table No. 3.1 Specification

Material	IS2062 E250 (FE 410W) B
Class of Crane	M5
Safe Working Load (Te) SWL (W_1)	75 Ton
Span	21 meter
Distance of Main Hook From Wheel X	3.0 meter
Self-Weight of End Carriage	1500 Kg.
Impact Factor	1.32
Duty Factor	1.06
Lifting Height of the crane (H)	15 meter

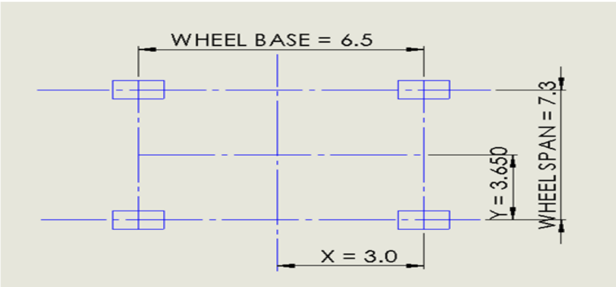

Fig 3.1 Space diagram for wheel base.

3.1.2 Stresses and material used for the design of the crane

Material used for the Crane column and Crane Beam is FE410. The property of the FE410 are mentioned as under:

Table No. 3.2 Material Properties S[2]

1	Ultimate tensile stress	σ_{ut}	410 N/mm^2
2	Yield stress	σ_y	250 N/mm^2
3	YoungsModulas	E	2 x 10^5 N/mm^2

According to application area of the Bridge crane, it should be sustain the wind load as well as the actual condition load, so the allowable stress for designing the Bridge crane are calculated as under:

$$\sigma_a = \sigma_{ut}/(C_{df} \times C_{bf} \times C_{sf})$$

Table No. 3.3 Factors for finding Allowable Stress

σ_{ut}	Ultimate tensile stress for the Bridge crane, N/mm2	410
C_{df}	Duty factor for the load lifting motion in the vertical direction.	0.85
C_{bf}	Basic stress factor for the given loading condition	2.5
C_{sf}	safety factor for using the material FE410.	1.12

$$\sigma_a = \sigma_{ut}/(C_{df} \times C_{bf} \times C_{sf})$$
$$\sigma_a = 410/(0.85 \times 2.5 \times 1.12)$$
$$\sigma_a = 172.27\ N/mm^2 [17]$$
$$\tau = \text{Shear stress, } N/mm^2$$
$$\tau = 0.50 \times \sigma_t$$
$$= 0.50 \times 172.27$$
$$= 86.13\ N/mm^2$$

3.1.3 Design calculation of the Beam

Based on the structure and application, the load diagram for the crane is shown as under.

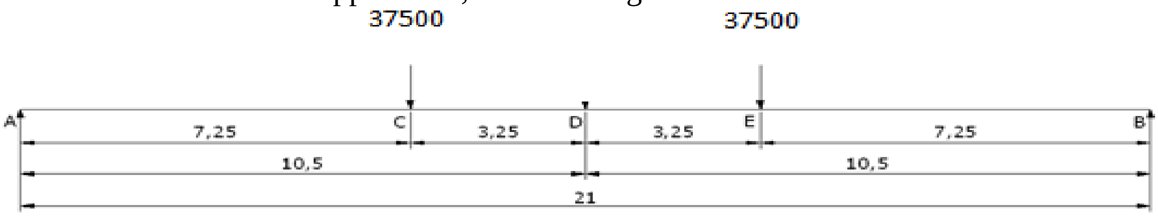

Fig3.2

Position Of Maximum Bending Moment

M1=bending moment generated on the beam based on the Live load.

$$M_1 = \Psi \times ((W_d + W_t) \times \left(S - \left(\frac{T_c}{2}\right)\right) \times 2)/8 \times S$$

$$M_1 = 1.32 \times ((123750 + 18750) \times \left(21000 - \left(\frac{6500}{2}\right)\right) \times 2)/8 \times 21000$$
$$= 3.53 \times 10^8 Kg \cdot mm$$

$M_2 = Bending\ Moment\ due\ to\ Impact\ Load$

$M_2 = 0.25 \times M_1$

$= 0.25 \times 3.53 \times 10^8$

$$= 8.83 \times 10^6\ Kg \cdot mm$$

$M_3 = Bending\ Moment\ due\ to\ self\ mass\ of\ girder$

$M_3 = 1.1 \times (W_g \times S)/8$

$$= 1.1 \times (37875 \times 21000)/8$$
$$= 1.1 \times 9.942 \times 10^7\ Kg \cdot mm$$
$$= 1.09 \times 10^8\ Kg \cdot mm$$

Maximum bending moment generated in the crane are as under:

$M_{max} = M_1 + M_2 + M_3$

$= 3.53 \times 10^8 + 8.83 \times 10^6 + 1.09 \times 10^8$

$= 4.71 \times 10^8 Kg \cdot mm$

$= 4.71 \times 10^8 N \cdot mm$P[1]

$$M_{max} = 4.71 \times 10^8 N \cdot mm$$

"I" Section used in the Crane Assembly is as under.

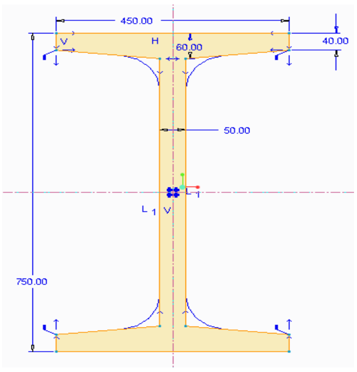

Fig 3.3 Dimension of the "I" section

Now to find out the stress generated in the I Section by the following equation.

Stress and Deformation for "I" Section 750×450

$M_{max} = 4.71 \times 10^8 \; N \cdot mm$

$\sigma_b = \left(\frac{M_{max}}{I}\right) \times Radius\ of\ Gyration$

$\sigma_b = \left(\frac{4.71 \times 10^8}{1.41 \times 10^{10}}\right) \times 3651.18$

$$= 121.96 \; N/mm^2$$

$\delta_l = P \times b \times (3l^2 - 4b^2)/48 \times E \times I \times 2$

$\delta_l = 37500 \times 9.81 \times 7000(3 \times 21000^2 - 4 \times 7000^2)/48 \times 2 \times 10^5 \times 1.41 \times 10^{10} \times 2$

$$= 10.72 \; mm$$

Now the stress generated in the beam is 121.96 N/mm^2 and the allowable stress is 172.27 N/mm^2. Which is higher than the generated bending stress. So the selected section is safe.

IV. ANALYSIS OF EXISTING SECTION OFTHE GIRDER

4.1.1 Analysis of the Crane

There are following main steps to analysis the crane model in the ANSYS.

Step 1: Apply the material property to the imported CAD neutral model. The material property for the Crane material are mentioned as under:

Material used for the Crane column and Crane Beam is FE410. The property of the FE410 are mentioned as under:

Table No. 4.1 Material Properties

σ_{ut}	410 N/mm^2
σ_y	250 N/mm^2
E	2 x 10^5 N/mm^2

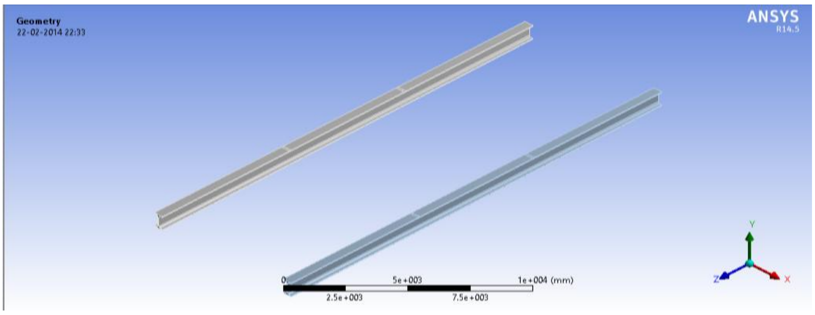

Fig 4.2: Crane model imported in the ANSYS environment.

Step 2: Import the Crane model in to ANSYS environment with *.iges model.
Step 3: Apply the material to imported CAD model.
Step 4: Apply the mesh to the Crane to divide the crane model in to no. of small part.

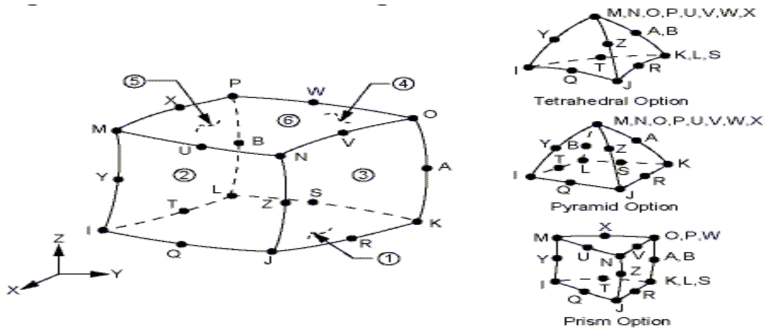

Fig 4.3 : Solid186 Element

SOLID186 is a higher order 3-D 20-node solid element that exhibits quadratic displacement behaviour. The element is defined by 20 nodes having three degrees of freedom per node: translations in the nodal x, y, and z directions. The element supports plasticity, hyper elasticity, creep, stress stiffening, large deflection, and large strain capabilities. It also has mixed formulation capability for simulating deformations of nearly incompressible elastoplastic materials, and fully incompressible hyper elastic materials.

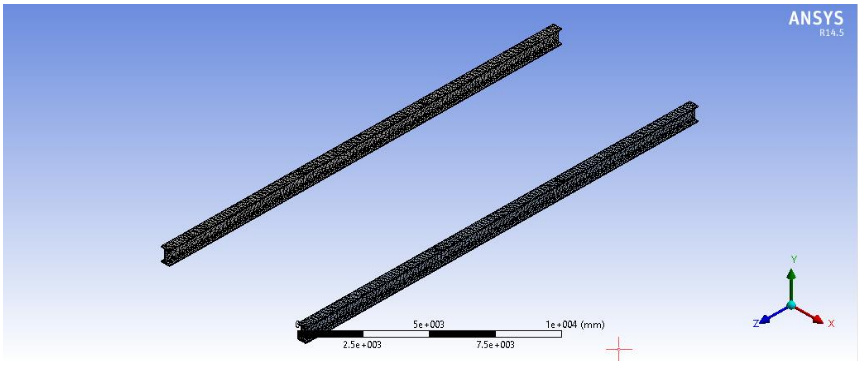

Fig 4.4: Mesh Crane model.

Step 5: Apply the boundary condition to CAD model.

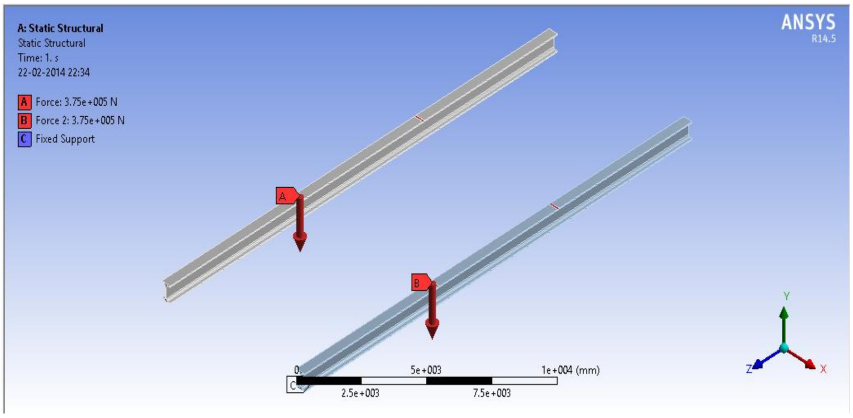

Fig 4.5: Load and Support applied on the Crane

After applying the mesh to the crane model, the boundary condition has been applied to the model as Load acting on the crane, support area to the crane which is require to generate the stresses and deformation on the crane model.

Step 6: Run the Analysis.

Step 7: Results of the Crane**.**

After applying all the information to the crane model, following results are observed for the crane model.

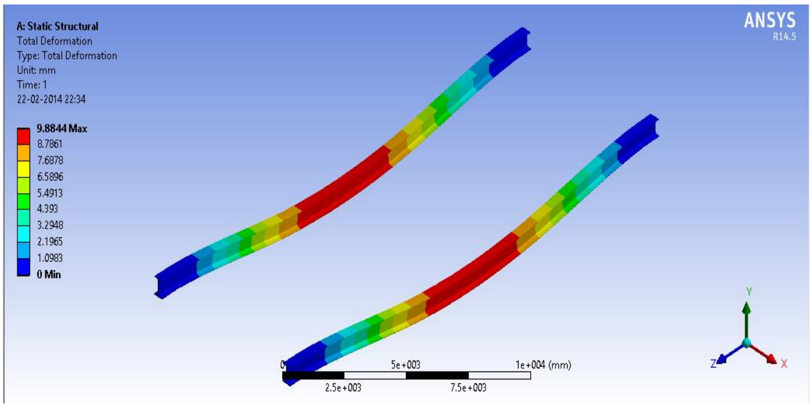

Fig 4.6 Total Deformation on 750 x 450 "I" Section

Fig shows the total deformation generated on the crane. Total maximum deformation generated on the crane is 9.8844mm and the minimum total deformation generated on the crane is 0mm.

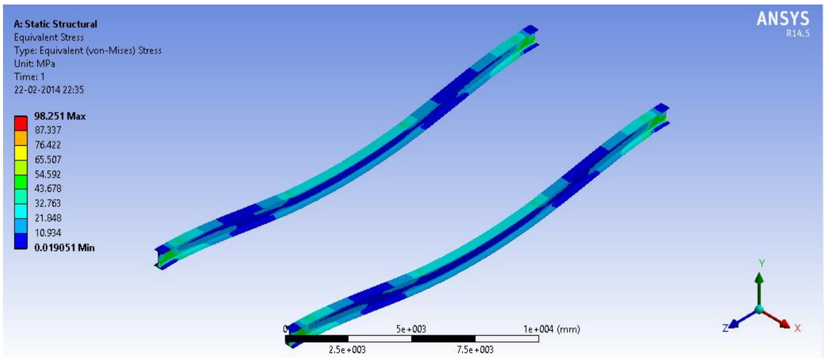

Fig 4.7 Equivalent Von-misses stress on 750 x 450 "I" Section

Fig shows the von-misses stress generated on the crane. The maximum von-misses stress generated on the crane is 98.251 N/mm2 and the minimum von-misses stress generated on the crane is 0.019051 N/mm2.

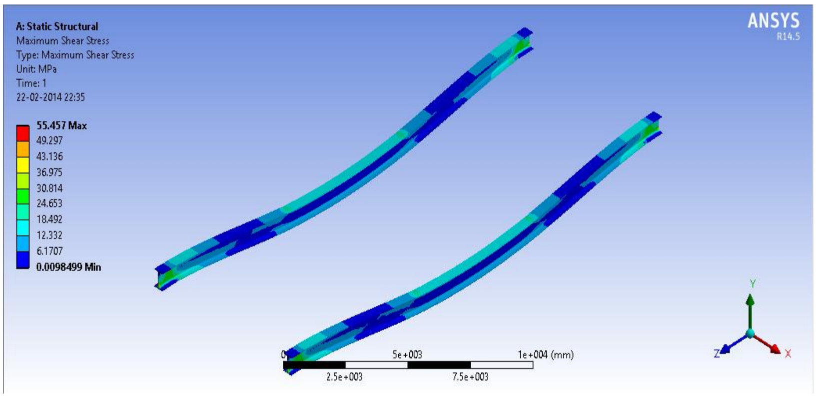

Fig 4.8 Maximum Shear Stress on 750 x 450 "I" Section

Fig shows the shear stress generated on the crane. The maximum shear stress generated on the crane is 55.457 N/mm2 and the minimum von-misses stress generated on the crane is 0.0098499 N/mm2.

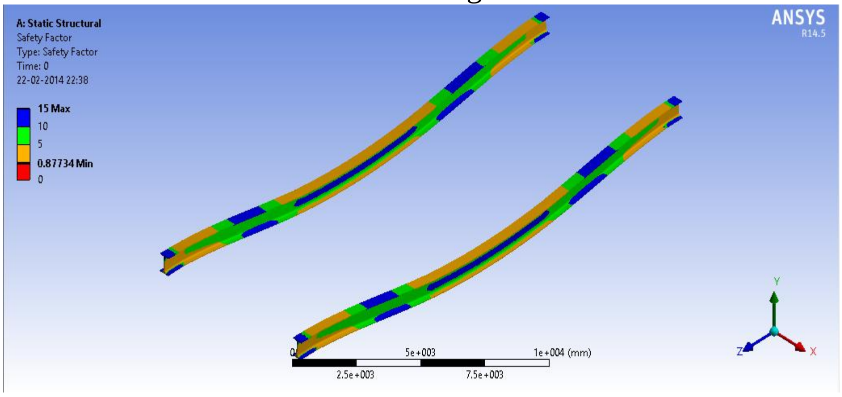

Fig 4.9 Safety Factor on 750 x 450 "I" Section

Fig shows the safety factor achieved with the allowable limit of the material. The maximum safety factor generated on the crane is 15 and the minimum safety factor generated on the crane is 0.87734.

V. OPTIMIZATION OFTHE CRANE

To reduce the weight of the "I" section used in the application, further analysis is performed by reducing the width and height of the "I" section by 10mm till the stress and deformation are achieved same as the stresses and deformation are generated till the allowable limit of the material.
After completion of all analysis of the different size "I" section used in the Crane assembly, the following summary has been derived

Table - 5.1 Summary of different size "I" section

Sr. No	Parameter	Allowable	"I" Section Size						% Change
			750 x 450	740 x 440	730 x 430	720 x 420	710 x 410	700 x 400	
1	Von-Misses Stress (N/mm2)	172.27	98.251	102.24	106.06	108.63	113.39	117.61	
2	Shear Stress (N/mm2)	86.13	55.457	57.737	59.876	61.25	63.939	66.279	
3	Deformation (mm)	16	9.8844	10.286	10.725	11.201	11.686	12.228	
4	Safety Factor		0.87734	0.84309	0.81273	0.79349	0.76018	0.7329	
5	Weight (Kg)		26976	26472	25969	25465	24960	24455	- 9.35

VI. CONCLUSION

After completion all the work as calculation, modelling and analysis work for the crane following summary has been derived.

Table - 6.1 Summary of different size "I" section

Sr.No.	Parameter	Section Size					
		750 x 450	**740 x 440**	**730 x 430**	**720 x 420**	**710 x 410**	**700 x 400**
1	Deformation (mm)	9.88	10.29	10.73	11.20	11.69	12.23
2	Analytical Deformation (mm)	10.72	11.27	11.85	12.48	13.15	13.88
3	Von-Misses Stress (N/mm^2)	98.251	102.24	106.06	108.63	113.39	117.61
4	Analytical Stress (N/mm^2)	121.96	128.18	134.84	141.97	149.63	157.86
5	Weight (Kg)	26976	26472	25969	25465	24960	24455

In thiswork, Existing section of girder is 750 x 450 I section. in the existing model the von-misses stress generated is 98.251 N/mm^2 against the allowable von-misses stress is 172.27 $N/mm^2$and same way the shear stress is 55.457 N/mm^2 against the allowable shear stress is 86.13 N/mm2 with this result the weight of the "I" section is 26976 Kg. Weight of this section is much more higher. So we are optimized the weight of section by using different size I section.

After analysis of the summary data, the von-misses stress and the deformation is matched with the data generated through the ANSYS data.

After completion of the analysis and optimization work, It is found out that Optimum Section is "700 x 400" and following conclusion has been derived.

1. Weight of the "I" section has been reduced from 26976 Kg. to 24455 Kg whereas the stress is increased from 98.251 N/mm2 to the 117.61 N/mm2 but it is near about the allowable stress limit of the "I" section material. Total weight saving is 9.35% and stress increases by 16.46%.
2. The shear stress is increased from 55.457 N/mm2 to 66.279 N/mm2 but within the allowable limit range. it is increased by 16.32%.
3. The deformation is increased from 9.8844mm to the 12.228mm. Increased by 19.16%.
4. The safety factor is reduced from 0.87734 to the 0.7329. it is reduced by 16.50%.
5. Finally the existing design has a possibility to reduce the weight by 9.35% by increase the performance parameter by 16.50%. But all the performance parameter are is within its allowable limit.

REFERENCES

[1], R. C. Patel, A. D. Pandya, “Machine Design” , Edition 4th Publisher Acharya, 1961 PP-190

[2]AbhinaySuratkar, Vishal Shukla, Dr. K. S. Zakiuddin, "Design Optimization of Overhead EOT Crane Box Girder Using Finite Element Analysis", International journal of engineering research and technology (IJERT), ISSN: 2278-0181.

[3]AbhinaySuratkar, Vishal Shukla,"3D Modelling and Finite Element Analysis of EOT Crane, International journal of Mechanical and Production engineering", ISSN :2320-2092.

[4] CameliaBretoteanPinca, GeluOvidiuTirian, Ana Josan, "Application of Finite Element Method toan Overhead Crane Bridge", Issue 2, Volume 4, 2009

[5] Ismail Gerdemeli, Serpil Kurt, Metinyıldırım M.Sc., "Calculations, Modeling And Analysis With Finite Element Method Of Rubber Tyred Container Stacking Crane", 14th International Research/Expert Conference, Trends In The Development Of Machinery And Associated Technology, 2010

[6] M. Euler, U. Kuhlmann, “Crane runways - Fatigue evaluation of crane rail welds using local concepts", International Journal of Fatigue, 2011