LATTICE (CAGE) BEAM DESIGN AND FINITE ELEMENT ANALYSIS OF GANTRY CRANE

İ. Gerdemeli [1], S. Kurt [2] and B. Bige[3]

[1, 2, 3] ITU Faculty of Mech. Eng., Mech. Eng. Department, İstanbul, Turkey.

Abstract: *Gantry cranes are particularly suited to lifting very heavy objects, especially in shipbuilding industry are used massive objects like ships' engines to be lifted and moved over the ship. Capacity of the overhead gantry cranes increased proportionally through the needs. In this study dual-trolley heavy duty overhead gantry crane that can carry loads up to 800 tons was designed and analyzed. Stress and displacement values of main beam were calculated by both Finite Element Method and using mechanical formulas. As a result, it has been seen that, F.E.M. is a quite practical method to handle stress analysis problem of gantry cranes.*

Keywords: GANTRY CRANE, CAGE STRUCTURE MAIN BEAM, DESIGN, ANALYSIS, FINITE ELEMENT METHOD

1. Introduction

Because of the fact that overhead gantry cranes are the structures, particularly designed according to the needs of the users, the parameters that are needed to be determined before the design. In this study a dual-trolley (2x400 tons) heavy duty overhead gantry crane that has 102 meters long cage structure main beam, 800 tons lifting capacity and 62 meters effective lifting height was designed and analyzed.

Classifications were made to determine coefficients to be used in analytical calculations by using FEM and DIN norms and the basic structural elements like ropes and pulleys were selected. All loads affecting the crane were determined by taking into consideration the chosen structural elements.

Occurring result of these affecting loads, maximum stresses were calculated analytically by using mechanical formulas. Calculated stresses were checked whether they are under allowable limits or not. For the case calculated stresses were over allowable limits the design was renovated and provided to remain under allowable limits.

All 3-D models were created with the help of the draft drawings which were formed by mechanical calculations and the selection of the structural elements. The gantry crane which was designed during this study is shown in the figure below (Fig. 1).

According to the various loading conditions the generated solid model was analyzed by the finite element method with the help of ANSYS Workbench 14.5.

Finally, the stress and displacement results obtained from analytical calculations were compared with the results obtained from the finite element program and percentages of errors were calculated.

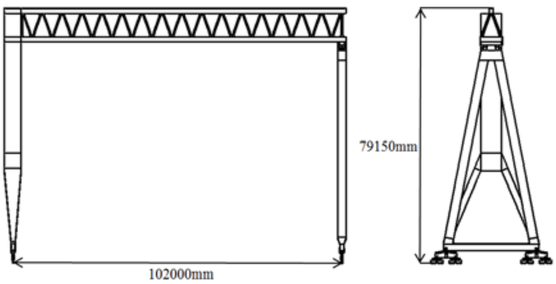

Fig. 1 *Designed gantry crane.*

2. Analytical Analysis of Gantry Crane

The free body diagram of the main beam was created as shown in the figure below (Fig. 2). Trolley position creating maximum moment was used in our calculations to see the stresses in the worst case. The loads were increased by the proper coefficients to add dynamic effects. The total load affecting from trolley's wheels (P=5824882N) and the distributed load coming from weight (w=117384N/m) of the main beam were calculated. The maximum moment occurred at 46m far from rigid leg and the moments are calculated as shown in equations below.

Equation 1: *Moment from the distributed load.*

$$M_{dis_{(46)}} = \frac{w.l.c}{2} - \frac{w.c^2}{2} = 151190592\,Nm$$

l =102m, w =117384N/m, c =46m

Equation 2: *Moment from the singular load.*

$$M_{snglr_{(46)}} = 2.P.\frac{c^2}{l} = 241675496.3\,Nm$$

l =102m, P =5824882N/m, c =46m

Also moments coming from rope effect and trolley's braking force were calculated and all moments were summed. Finally, maximum stress values occurring on the main beam by result of total moment was calculated by using mechanical formulas. For the case calculated stresses were over allowable limits the design was renovated.

According to the maximum shear force occurring on the main beam, compressive and tensile forces affecting cross bars that are forming cage structure were calculated individually and then stresses on bars were calculated.

Consequently, displacement occurring on main beam by weight and lifting capacity were calculated. The displacement was checked that it is under allowable limits. Reverse displacement value that will be used at production was calculated.

3. Analysis of the Gantry Crane with Finite Element Method

The next stage of the study includes the analysis of the gantry crane with finite element method which is one of the most common numerical methods that can solve many engineering problems. To analyze the gantry crane by this method, 3D solid model of gantry crane must be generated.

3.1 Modelling of Gantry Crane

All parts of the gantry crane were 3-D modeled by using the SolidWorks 2012 drawing program with the help of the draft drawings which had been formed by mechanical calculations and the selection of the structural elements. Then all modeled parts were assembled.

3D modeled gantry crane was shown in the figure below.

Fig. 2 *3D modeled gantry crane.*

3.2 Analysis of the Gantry Crane by Using ANSYS FEA Software

According to the various loading conditions the generated solid model were analyzed by the finite element method with the help of ANSYS Workbench 14.5 analyzing program. Loading conditions were created by taking maximum effects into consideration. For that reason for 4 different trolley position (left side, right side, center and maximum moment position) 8 different loading conditions were created and also for without trolley situation one more loading condition was created.

Static structural mode which allows static analysis with materials' linear behavior was used to analyze in ANSYS Workbench 14.5. In static analysis which is commonly used in engineering applications, it is thought that forces and supports are applied momentary and calculations are made for this moment. Dynamic coefficients to increase static forces and acceleration/braking forces were used to dynamic analyze in static structural mode.

To prepare finite element model, contacts were determined and *no separation* contact type for flexible joints and *bounded* contact type for fixed joints were assigned. Then the modeled body was meshed by using proper *sizing* as a result 836048 elements and 2492913 nodes were created. Meshed body was shown in the figure 3.

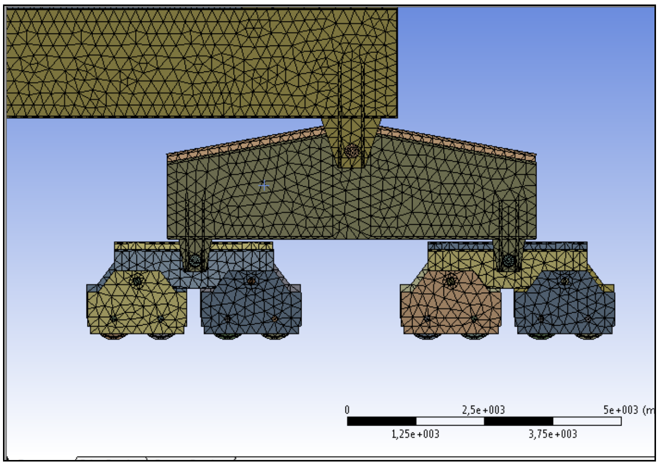

Fig. 3 Meshed model.

After meshed body was created, all of the loading conditions were applied. Earth gravity and acceleration coming from crane's move which equals to 1/30 of the crane's weight were applied to whole body. Working load (y direction) and the forces (x and z direction) coming from trolley's move which equals to 1/30 of the working load were applied to the touching points of the trolleys' wheels to the main beam. And also in the calculations the crane's wheels were considered fixed to the ground. Created finite element model was shown in the figure below (Fig 5).

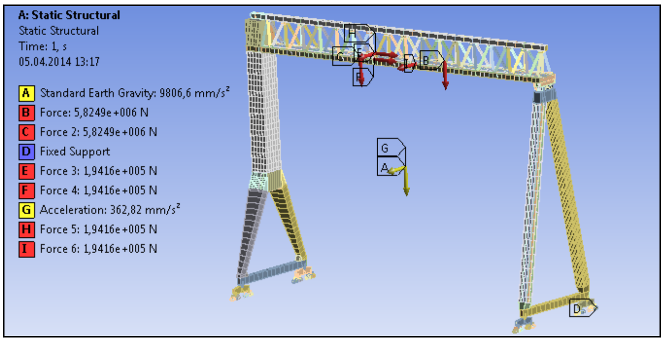

Fig. 4 *Finite element model.*

4. Results

From these static analysis, stress and displacement values on the crane body were obtained for all loading conditions. To obtain the exact deformation value of main beam, arithmetic mean value of legs' deformation was subtracted and the obtained value was checked that it is under allowable limits. The deformation value occurring as a result of the working load was calculated as 80.68mm. Exact value of deformation of the main beam was calculated as shown in the figure below by using formulas below.

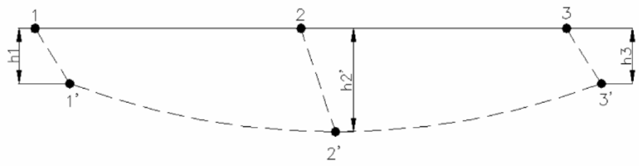

Fig. 5 *Deformation calculation.*

Equation 2: *Exact deformation of the main beam.*

$$h_2 = h_2' - \left(\frac{h_1 + h_3}{2}\right)$$

When evaluating the results of the finite element method, point defects occurring on generally corners were neglected. So, it's easily seen that maximum stress (172.67 MPa) occurred at upper beam.

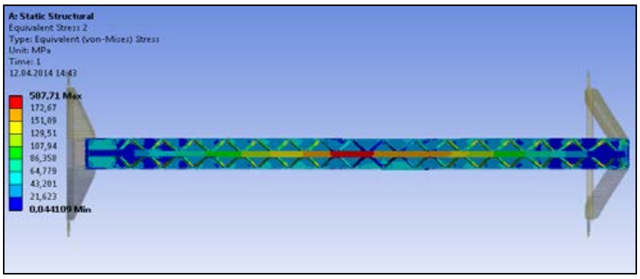

Fig. 5 *Stress distribution of main beam.*

When the result of stress distribution was examined in detailed manner, it was seen that high stresses occurred on some local zones at the balancing beams of carrying groups.

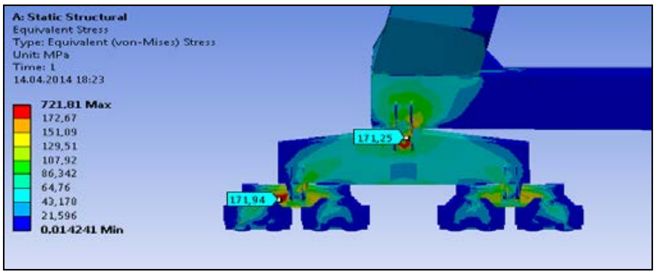

Fig. 5 *Stress distribution of carrying group.*

Finally, the stress values on main beam obtained from analytical calculations were compared with the results obtained from the finite element program and percentages of the error were calculated.

Table 1: *Comparison between analytical approach and finite element method.*

Loading Combination	Analytical Approach	Finite Element Method	
	σ_{max} [MPa]	σ_{max} [MPa]	Difference (%)
110	56.43	53.08	6
121	134.34	142.07	6
122	90.43	94.78	5
123	134.75	145.7	8
124	86.86	89.56	3
131	169.56	171.91	1
132	116.22	113.35	2
133	170.66	172.67	1
134	107.6	104.32	3

The basic reasons of the difference occurring between mechanical calculations and finite element method are the acceptances in creating mechanical model and the mathematical approaches in finite element method.

5. Conclusion

It was concluded that St 52-3 which has 210 MPa tensile strength is the proper material for the main beam. Determining nodal stresses provided us that dimensions and thicknesses at excessive safe areas to be simplified in order to reduce the manufacturing cost and at unsafe areas to be strengthened. As a conclusion, finite element method is a very effective and useful way to see applicability of design of gantry crane.

6. References

[1] Bayvas, M.Ş., (1987). General Mechanic, Turkish National Education Association, Istanbul.
[2] Kocaturk, T., (2008). Static Notes. YTU, Istanbul.
[3] Gerdemeli, İ., İmrak, C.E., (2008). Transport Technique Symposium Notes, ITU.
[4] Gerdemeli, İ., (2008). Advance Transport Technique Notes, ITU.
[5] F.E.M., (1987). 1001 3. Association 1987.10.01
[6] Oztepe, H., (1999). Transport Technique, Istanbul.
[7] Curgul, İ., (2010), Transport Technique, Birsen Association, Istanbul.
[8] Suner, F., (1981), Lifting and Handling Equipment, Istanbul.
[9] Kurt, S., Kutay, G.M., Aslan, R., (2008). Steel Construction At Cranes, Vol. II. Chamber of Mechanical Engineers Association, Ankara.
[10] Topcu, M., Tasgetiren, S., (1998), Finite Element Method For Engineers, Faculty of Engineering Association, Denizli.