10th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2006, Barcelona-Lloret de Mar, Spain, 11-15 September, 2006

LINEAR AND NONLINEAR FINITE ELEMENT MODELLING AND STRESS ANALYSIS OF PORTAL CRANES

C.Erdem IMRAK
ITU, Faculty of Mech. Eng.
Mech.Eng. Department
Gumussuyu, 34437
Istanbul, Turkey

Ismail GERDEMELİ
ITU, Faculty of Mech. Eng.
Mech.Eng. Department
Gumussuyu, 34437
Istanbul, Turkey

Atıl ERDIK
Otokar Otobus Karoseri
Sanayi A.Ş., Ataturk Cad.
No:9 Arifiye 54580
Sakarya, Turkey

ABSTRACT
Portal crane which is affected by a number of load combinations can be used in seaports, shipyards and warehouses with a wide range of usage areas during the operation. Design of portal crane with box girder has been investigated and a case study on a crane with 2x200 ton capacity and 100 m span length has been considered. In this study finite element model of the portal crane box girder is generated with full details. The linear static and nonlinear static stress analysis of the finite element model are computed under certain load combinations and boundary conditions. In order to compare the stress results from linear static and nonlinear static stress analysis, an illustrative example of portal crane box girder is given.
Keywords: portal crane, finite element method, stress analysis

1. INTRODUCTION
Cranes are the best way of providing a heavy lifting facility covering virtually the whole area of the building. Portal crane which is affected by a number of load combinations can be used in seaports, shipyards and warehouses with a wide range of usage areas during the operation [1]. Their design features vary widely according to their major operational specifications such as: the type of motion of crane structure, weight and type of the load, location of the crane, geometric features and environmental conditions. Since the crane design procedure is highly standardized with these components, main effort and time spent mostly for interpretation and implementation of available design standards. Considerable research studies have been carried out about structural and component stresses, safety under static loading and dynamic behavior of cranes [2 - 8]. In this study, the system behavior is considered to be linear elastic and nonlinear elastic separately and one of a finite element modeling and solver software Msc.Patran/Nastran was utilized for the linear static and nonlinear static stress analyses.

2. MODELING PORTAL CRANE BOX GIRDER
The finite element method is a numerical procedure that can be applied to obtain solutions to a variety of problems in engineering. Steady, transient, linear or nonlinear problems in stress analysis, heat transfer, fluid flow and electro mechanism problems may be analyzed with finite element methods. Basic steps in the finite element method are defined as follows: preprocessing phase, solution phase, and post-processing phase. Among the numerical techniques, the finite element method is widely used due to the availability of much user-friendly commercial software. The finite element method can analyze any geometry and solves both stresses and displacements [11-12]. The finite element method approximates the solution of the entire domain under study as an assemblage of discrete finite elements interconnected at nodal points on the element boundaries. The approximate solution is formulated over each element matrix and thereafter assembled to obtain the stiffness matrices, and displacement and force vectors of the entire domain.

To achieve more accurate results, the structural components of the crane have been modeled using a combination of shell and beam elements. This model can be used for the static, dynamic, natural frequency analysis. The geometric characteristics of the crane and its various elements are modeled and determined on the basis of the planned geometric layout. The support ends of the crane are assumed to be simple supported. They are generally located as contact face of wheels. The finite element discrimination includes 26126 nodes and 27219 elements [11]. Real crane data gathered from a Turkish company TRANSPORT TESISLERI Inc. where mass production of portal cranes is made have been used. First box girder is modeled as a surface. Because its geometry is suitable for this, and also long and thin parts should be modeled as a surface. Later, mesh is created. In this study quadratic element type is used as illustrated in Figure 1.

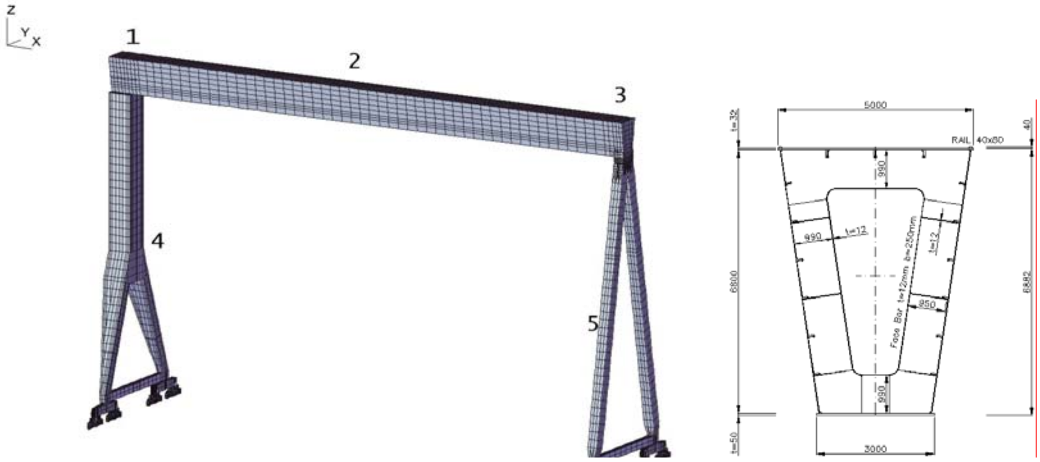


Figure 1. Isoparametric view of the model.

To model the portal crane box girder with four-node tetrahedral element, Msc.Patran/Nastran software was used to finite element analysis using the girder solid model generated by means of Msc.Patran/Nastran. Young’s Modulus (E) is $2.1x10^5$ N/mm^2 and Poisson Ratio (ν_{St}) is 0.3 for finite element analysis.

The boundary conditions are selected according to the working conditions. All wheel touch faces are regarded as simple supported. When the crane is loaded 2 cases can be considered: weather the crane is moving or not or braked. In the first case all touching faces can be considered as simple support and no roller support is presented. In the latter case the crane in the global Y direction is moving which means no support should be presented in the same direction. However no support condition causes singularity and no results are obtained. But the fact is that if the wheels in one bogie are fixed the other bogie can be considered as roller support. By reasoning the support condition can be defined.

3. LOADING CONDITIONS

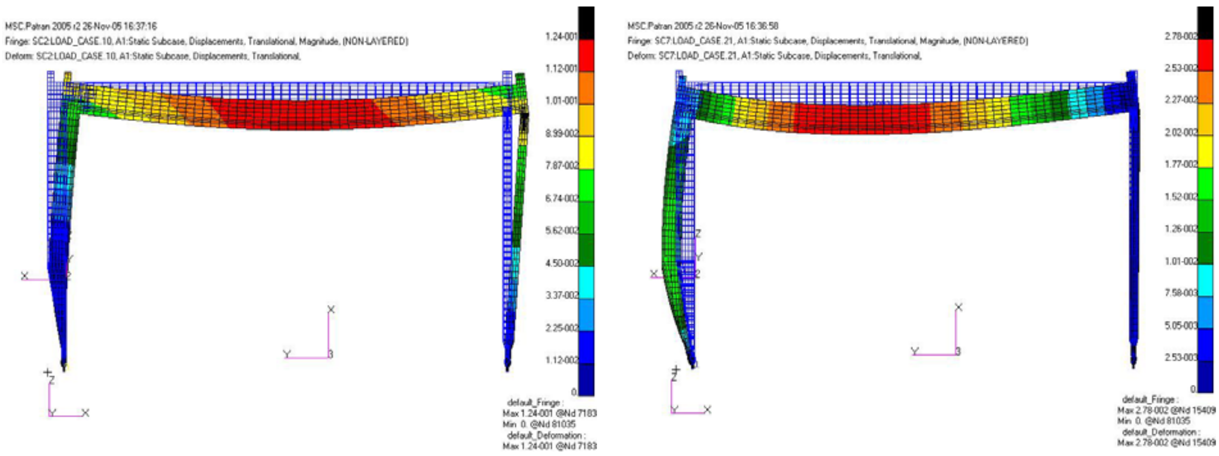
The gantry crane system is exposed to different type of applied external and internal loads. The loads acting on the gantry crane in its operating and non-operating conditions include the loads due to the dead weight, the wind load, and the dynamic loads caused by changes in the steady motion they act on every crane parts possessing mass. In the operating conditions the weight of the load being handled in addition to the forces induced by the resistance to motion of the crane are considered in the analysis [1,11]. The dead loads include the self weight of the crane, trolley and hook self weights. The last two loads move across the main girder beam. The two traveling trolleys are capable of carrying 200 tons of load each. This load is considered as a live load and moves across the main girder beam. Acceleration and breaking mechanisms of the crane introduce dynamic loads. The value of dynamic loads is given as 1/30th of self weight of crane. The wind load is computed by assuming it to act horizontally in all directions with the same force and the same probability. When the crane is moving with 400 tons there may be a relative deformation of the wheels to each other due to different velocity of rolling. According to the standards this can be account as %8 of vertical load act horizontally.

4. NUMERICAL EXAMPLE

A 400-ton-capacity portal crane of overall length 100 m was selected as a study object. The configuration of the portal crane is shown in Figure 1. The portal crane consists of one box girder, two saddles to connect them, a trolley moving in the longitudinal direction of the portal crane and wheels. The portal crane is supported by two rails and the runway girders installed on the shipyard site. The design values used in the crane analysis from F.E.M and DIN standards. There were used two types of element in the model including 32 one-dimensional beam element and 27187 two-dimensional shell (tria and quad) elements which are already found in Msc.Patran element library. Total degree of freedom of the model as shown in Figure 1 is 156653 and total number of nodes is 26126 for this finite element model of portal crane.

4.1. Linear Static Stress Analysis of Girder

The general deformations due to different load cases for three dimensional frame models are shown in Figure 2. The trolley loads moves on the girder beams and the deformations of the crane for trolley on different places on the girder beam is illustrated in Figure 2.



(a) self weight of the crane only (b) working loads of trolley at the left of crane

Figure 2. Deformations of portal crane.

4.2. Nonlinear Static Stress Analysis of Girder

The nonlinear static procedure is intended to provide a simplified approach for directly determining the nonlinear response behavior of a structure at different levels of lateral displacements, ranging from initial elastic response through development of a failure mechanism and initiation of collapse. Response behavior is gauged by measurement of the strength of the structure, at various increments of lateral displacement [11]. There only need to perform a nonlinear static analysis in Msc.Patran is stress – strain curve of related material. Deformation result of the nonlinear static analysis on the main girder beam of the crane for the some important load cases is given as comparative with deformation result of linear analysis in Table 2.

Table 2. Deformation results of linear and nonlinear static analyses

Displacement on main girder beam [m]							
Load Case	Analysis Type	Maximum Displacement			Minimum Displacement		
		DX	DY	DZ	DX	DY	DZ
A	Linear	0.160543	0.000925	-0.012316	0.140617	0.000514	-0.106640
	Nonlinear	0.556853	0.002671	-0.040215	0.497112	0.001455	-0.328780
B	Linear	0.000561	0.020710	0.000908	-0.000382	0.001137	-0.001529
	Nonlinear	0.001690	0.060887	0.002644	-0.001097	0.003420	-0.004561

All displacement results of nonlinear static analysis as seen above is approximately 3 times greater than linear static analysis displacement results. Large displacement rule is satisfied here and nonlinear static definition in this study is true as well.
The value of maximum Von Mises stress of the main girder is read 76.29 N/mm^2 to two decimal places for linear static analysis and the value of maximum Von Mises stress of the main girder is read 69.33 N/mm^2 to two decimal places for nonlinear static analysis. Nonlinear static analysis stress results decrease 9% with respect to linear static analysis stress results. It is obvious that nonlinear static analysis gives more accurate results than linear static analysis if boundary conditions and material properties are correctly applied. Maximum Von Mises stress results of nonlinear and linear static analyses for several load combinations are computed.

5. CONCLUSION

The design of a portal crane with a box girder has been investigated and a case study of a crane with 400 ton capacity and 100 m span length has been conducted. Nonlinear static analysis stress results decrease 9% with respect to linear static analysis stress results in all load combinations. Finite element analysis, not only help modifying the areas which are critical in terms of the durability of the structure, but also help the analyzer optimizing the over designed sections and thus simplifying the design. This optimization can be done by considering reliability, weight and material expenditures of the structure and by modifying the structure geometry or narrowing cross-sectional areas.

6. ACKNOWLEDGEMENT

This research is a part of work for the project funded by ITU Research Fund: Grand No: 00_05_47 “Modeling of Gantry Crane Frame with Finite Elements and Static Stress Analysis”. It is pleasure to acknowledge the support of TRANSPORT TESISLERI Inc. which provided the design data.

7. REFERENCES

[1] Alkin, C., Imrak, C.E, Kocabas, H.: Solid Modeling and Finite Element Analysis of an Overhead Crane Bridge, Acta Polytechnica, **45** (3), 61-67, 2005.
[2] Erden, A.: Computer Automated Access to the “F.E.M. rules” for Crane Design, Anadolu University Journal of Science and Technology, **3** (1), 115-130, 2002.
[3] Baker, J.: Cranes in Need of Change, Engineering, **211 (**3), 298, 1971.
[4] Demokritov V.N.: Selection of Optimal System Criteria for Crane Girders, Russian Engineering Journal, **54** (4), 7, 1974.
[5] Lemeur, M., Ritcher, C., Hesser, L.: Newest Methods Applied to Crane Wheel Calculations in Europe, Iron and Steel Engineer, **51** (9), 66, 1977.
[6] McCaffery, F.P.: Designing Overhead Cranes for Non-flat Runways, Iron and Steel Engineer, **62** (12), 32, 1985.
[7] Reemsyder, H.S., Demo, D.A.: Fatigue Cracking in Welded Crane Runway Girders, Causes and Repair Procedures, Iron and Steel Engineer, **55** (4), 52, 1978.
[8] Moustafa, K.A., Abou-El-yazid, T.G.: Load Sway Control of Overhead Cranes with Load Hoisting via Stability Analysis, JSME Int. Journal, Series C, **39** (1), 34 – 40, 1996.
[9] Oguamanam, D.C.D., Hansen, J.S., Heppler, G.R.: Dynamic of a Three-dimensional Overhead Crane System, Journal of Sound and Vibration, **242** (3), 411 – 426, 2001.
[10] Demirsoy, M.: Examination of the Motion Resistance of Bridge Cranes, PhD. Thesis, Dokuz Eylul University, Izmir, Turkey, 1994.
[11] Erdil, A.: Finite element modeling and static analysis of a portal crane, Ms.C. Thesis, Istanbul Technical University, Istanbul, Turkey, 2006.
[12] Moaveni, S.: Finite Element Analysis: Theory and Application with ANSYS, Prentice-Hall, New Jersey, 1999.