14th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2010, Mediterranean Cruise, 11-18 September 2010

FINITE ELEMENT ANALYSIS OF THE TOWER CRANE

Ismail Gerdemeli	Serpil Kurt	Okan Deliktaş
ITU Faculty of Mech. Eng.,	ITU Faculty of Mech. Eng.,	ITU Faculty of Mech. Eng.,
Mech.Eng.Department	Mech.Eng.Department	Mech.Eng.Department
Gümüşsuyu, 34437	Gümüşsuyu, 34437	Gümüşsuyu, 34437
Istanbul,	Istanbul,	Istanbul,
Turkey,	Turkey,	Turkey,

ABSTRACT
In heavy industries, tower cranes are used to lift and carry heavy materials. This study focuses on the prevention of crane damage which occurs due to heavy loads. In this study, the strength of the tower crane parts has been calculated according to FEM and DIN standards .In order to accomplish finite element analysis method, the tower crane parts are modelled one by one. Afterwards, the stress analysis of the crane parts are accomplished with ANSYS software considering the crane's self weight, payload, hook weight, trolley weight and the dynamic loads. As a conclusion, the results obtained from finite element method and analytical calculation are compared.
Keywords: Tower Crane, Strength Equations, Stress Analysis

1. INTRODUCTION

Cranes are widely used to transport heavy loads and hazardous materials in shipyards, factories, nuclear installations, and high-building construction and play an important role in production process and serve to transfer loads from one place to another. Cranes are the best way of providing a heavy lifting facility covering virtually the whole area of the industry. Their design features vary widely according to their major operational specifications such as the type of motion, dead weights and type of the load, location of the crane, geometric features and environmental conditions.[1] Since the crane design procedure is highly standardized with critical components, main effort and time spent mostly for interpretation and implementation of available design standards. A tower crane is a type of crane with a
hoist in a trolley which runs horizontally along gantry rails, usually fitted underneath a beam spanning between uprights which themselves have wheels so that the whole crane can move at right angles to the direction of the gantry rails.

In this study, a tower crane is modelled in 3D using Solidworks computer software. Then, the generated components are meshed in ANSYS Software. The meshed components are mounted to each other and the meshed model of the tower crane is obtained. Finite element analysis are accomplished considering the load combinations in FEM norms. The obtained data is compared with the analytical calculations.

2. MAIN BODY

The specifications of the tower crane are; the max. radius as 24.000 mm, the hoisting capacity as 1500 kg and the hoisting distance as 65.000 mm. The tower crane components are made of St-37.

The strength calculations of the crane parts have been accomplished due to FEM norms. The tower crane is exposed to different types of external and internal loads. The loads acting on the crane in its operating and non-operating conditions include the loads due to the dead weight, the wind load and the dynamic load. In the operating conditions, the load weight being handled and the friction forces are considered in the analysis. The crane is subjected to vertical and horizontal loads by the weight of the crane, the working (hook) load and the dynamic loads.

The dimensions of the crane and load are illustrated in Figure 1.

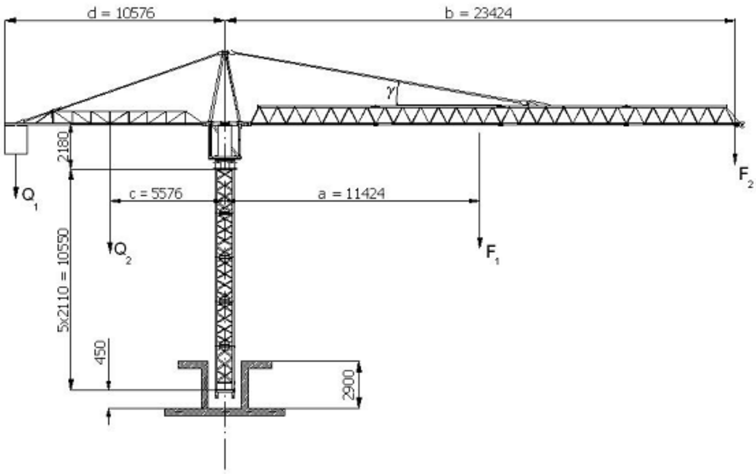

Figure 1. The dimensions of the crane and the loading case

The load values shown in Figure 1 are calculated as below.

$$F_1 = 1200 \cdot 1{,}1 = 1320 \text{ kg} \quad (1)$$

$$F_2 = 1500 \cdot 1{,}18 = 1770 \text{ kg} \quad (2)$$

$$Q_1 = 3000 \text{ kg}$$

$$Q_2 = 100 \text{ kg}$$

Total weight of the tower crane: $S_{crane} = 10500$ kg

The total moment of the tower crane is calculated as below;

$$\sum M = a \cdot F_1 + b \cdot F_2 - c \cdot Q_2 - d \cdot Q_1 \text{ tonm} \quad (3)$$

$$\sum M = 11{,}424m \cdot 1320kg + 23{,}424m \cdot 1770kg - 5{,}576m \cdot 100 - 10{,}576 \cdot 3000kg = 24{,}255 \text{ tonm} \quad (4)$$

The stress on the tower square profile resulting from the bending moment:

$$\sigma_{bending} = \frac{\sum M}{W_\zeta} \text{ [kg/cm}^2\text{]} \quad (5)$$

$$\sigma_{bending} = \frac{2435500 kgcm}{2567 cm^3} = 948{,}7 \text{ kg/cm}^2 \quad (6)$$

$$\sigma_{weight} = \frac{S_{crane}}{F_{section}} \text{ kg/cm}^2 \quad (7)$$

$$\sigma_{weight} = \frac{10500 kg}{24{,}6 cm^2} = 426{,}8 \text{ kg/cm}^2 \quad (8)$$

he total stress on the tower square profile;

$$\sum \sigma = \sigma_{bending} + \sigma_{weight} \ \text{kg/cm}^2 \quad (9)$$

$$\sum \sigma = 948,7 kg/cm^2 + 426,8 kg/cm^2 = 1375,5 \ \text{kg/cm}^2 \quad (10)$$

The components of the tower crane is modelled one by one using Solidworks and Autocad computer Software. During modelling, some geometrical simplifications have been done. The solid model of the gantry crane main beam is generated using Solidworks Computer Software Program as seen in Figure 2.

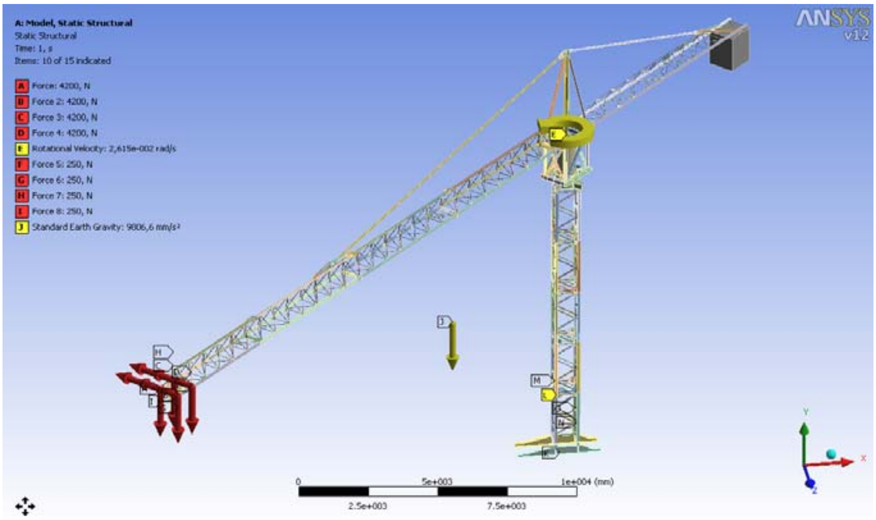

Figure 2. The 3D model of the tower crane

3. RESULTS

Static analysis is the most common analysis method which is used in engineering. As the loads are assumed to be applied instantly, the effects due to the time variation are neglected.

In the cage system, as the bars are placed in an order, the force acting on the main beam is balanced by a force flow from each bar. Thus, when a bar is under effect of compression, the other is under tension. The load combinations that include horizontal loads resulting from vertical and inertia loads applied on the main beam causes stress and sag.

In this study, the finite element analysis is accomplished due to these load combinations and then the maximum stress values are compared with the permissible stress values. The stress distribution which occurs on the tower crane is illustrated as in Figure 3. The red regions show the max. stress value as 175 MPa.

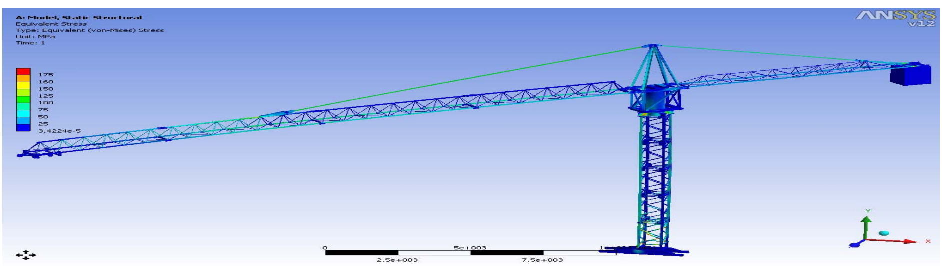

Figure 3. Stress distribution on the tower crane.

Figure 4 illustrates the deformation on the tower crane. The red region shows the max. stress values.

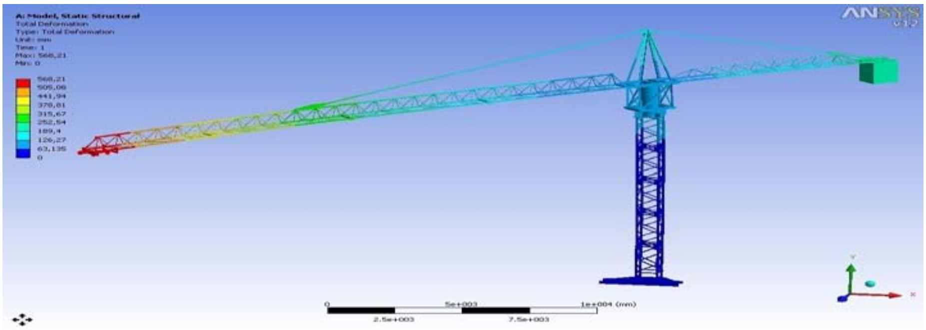

Figure 4. Deformation on the tower crane

4. CONCLUSION

In this study, the stress values on the components of tower crane are computed both by analytical and FEM method. As seen in table 1, the maximum deviation between these methods are not more than 5% except the tower connection part. This deviation occurs because of the assumptions made in analytical calculations and the numerical approach used in the finite element method. Since it is not easy to calculate the weight carried by the bolts and by the square plates positioned on the square profiles, the deviation on the tower connection part is unacceptable.

Table 1. The comparison of the analytical and FEM method results

Part Name	Analitical Cal. Compound Stress (Mpa)	FEM Analysis V.Misses (Mpa)	Deviation(%)
Boom upper tube	108,8	110	1,102941
Tower Square Profile (Compression)	134,9	128	5,1149
Tower Square Profile (Tension)	93	95	2,15
Tower Connection Part	216,3	100	53,76791

As the computed stress values are smaller than the allowable stress of the material of the crane components (175 MPa), it is observed that the gantry crane is safe according to DIN and FEM norms. As a result of the FEM analysis, the punctual stress occurs especially on the corners of the crane components. This proves that FEM analysis method is insufficent in some areas. The punctual loads are neglected in this study. The results obtained from FEM analysis shows that the element type and the boundary conditions have been selected in correctly. Considering the data obtained from the analyses, the material waste can be prevented in crane design. The construction is now more reliable, light and durable. This is crucially important in means of low cost production and low design duration.

5. REFERENCES

[1] Fetvacı, M. C.: Sonlu Elemanlar Metodu ile Modellemede Temel Prensipler, Mühendis ve Makina, Sayı: 470, Mart 1999