

COMPARISON OF ANALYTICAL AND FINITE ELEMENT ANALYSIS OF A TOWER CRANE BOOM

Umut Şahin

**Istanbul Technical University, Mechanical Engineering Faculty
ITU Makina Fakültesi Inonu Cd. 34437 Beyoglu, Istanbul
Turkey**

Adem Candaş

**Istanbul Technical University, Mechanical Engineering Faculty
ITU Makina Fakültesi Inonu Cd Beyoglu, Istanbul
Turkey**

Muharrem E. Boğoçlu

**Yıldız Technical University, Mechanical Engineering Faculty
Makina Fakültesi, Barbaros Blv. 34349 Yıldız, Istanbul
Turkey**

ABSTRACT

High-performance tower cranes have been required with the development of construction technology. Usage of cranes has been mandatory in an effort to build quicker these buildings. In order to prevent damage of the cranes during lifting heavy loads, the strength of the cranes is investigated by various methods. In this respect, comparing an analytical calculation with a computer aided analysis has been very important. In this study, the strength of the Liebherr 154 EC-H 6 tower crane boom was examined according to FEM standards. Firstly, the analytical calculations were performed to calculate the critical sections of the boom part. 3D modelling of each part of the tower crane parts was created. ANSYS software was used for finite element analysis of the boom 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 calculations are compared. The maximum stress difference between two methods is found 4.2% on the boom. The computed stress values are smaller than the allowable stress of the material of the crane components, the tower crane boom is safe according to FEM norms.

Keywords: tower crane, boom, finite element analysis

1. INTRODUCTION

With accelerating urbanization and growing construction projects, in particular with the scale enlargement of high-rise buildings, tower crane's roles in transport process are increasing. As a heavy lifting facility in a various area of the industry, their designs are changing depending on different requirements, for example, hoist boom length and height are increasing with the high-rise buildings. A tower crane is defined as a crane with a hoist in a trolley runs horizontally along gantry rails [1]. It 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 [1].

The crane booms are subjected to buckling failure quickly on the working condition of complex combination, such as gravity, wind load and inertia force. Their design features vary widely according to their primary operational specifications such as the type of motion, dead weights and type of load, location of the crane, geometric features and environmental conditions [2]. In this study, ANSYS

software was used to analyze a crane boom strength and stability and check analytical calculations. After the calculation results are obtained, the research conclusion can provide a particular reference value for other crane's design and optimization.

2. MODELING OF THE CRANE

Designing the crane complies with Liebherr 154 EC-H construction tower crane. The boom was formed by bringing together boom parts of about 10 m lengths with pin connections. The boom section consists of three main carrier profiles: two at the bottom and one at the top. These parts carry moment and force. These profiles are joined by hollow profiles with a circular cross-section with a diameter of 60 mm from the lateral surfaces. It has been noted that the combinations of these lateral profiles are in one spot on the same profile.

3. ANALYTICAL ANALYSIS OF THE CRANE

Tower crane had been analyzed analytically before it was analyzed with finite element method. After basic crane elements and their dimensions had been defined, the static balance of the crane was checked. The strength of the tower crane was calculated analytically according to FEM standards. Stress and deformation of the boom and stress of the crane mainframe were calculated while taking into account acceleration and breaking loads. According to these results, crane model was revised to achieve better strength values.

3.1. Tower Crane Specifications

The boom weight of the crane is 12.2 tons; total height is 45 m; construction material is ST37-2; maximum capacity at the end of the jib 1.92 tons; maximum radius is 61.49 m; counter jib weight is 19.1 tons. The boom is exposed to both static and dynamic loads under operating conditions. According to the FEM Norms, static loads are converted to dynamic loads by multiplying the predetermined magnification factor. Static and dynamic state working loads are 1,920 kg and 2,695 kg respectively. Points B and C are determined as critical control points on boom part.

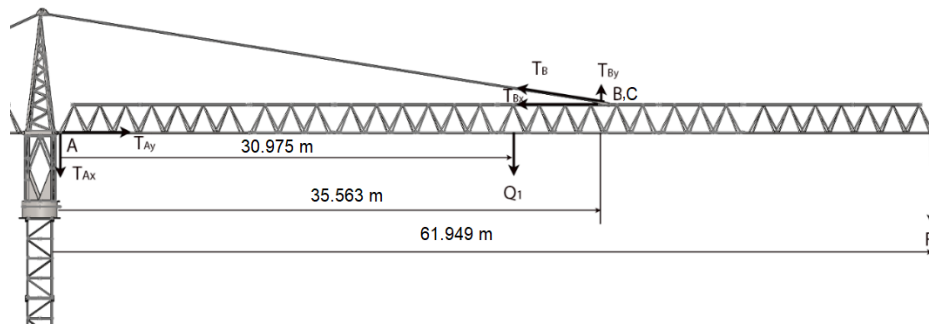


Figure 1. Boom of Tower Crane and Critical Points B and C.

The acting load on the boom's end is $F_1 = -2,695$ kg and there are A and B reaction forces from simple static equations. Since point A is connected to the pin, this point will not carry M_A moment (Eq. 1-10).

$$Q_1 \times 30,975 + F_1 \times 61,949 = T_{By} \times 35,563; \quad T_{By} = 15,324 \text{ kg} \quad (1)$$

$$\sum F_y = 0; \quad T_{Ay} + Q_1 + F_1 = T_{By}; \quad T_{Ay} = 425 \text{ kg} \quad (2)$$

$$\sum F_x = 0; \quad T_B = \frac{T_{By}}{\sin(14,4^\circ)} = \frac{15,324}{\sin(14,4^\circ)} = 61,618 \text{ kg} \quad (3)$$

$$T_{Bx} = T_B \times \cos(14,4^\circ) = 61,618 \times \cos(14,4^\circ) = 59,638 \text{ kg} \quad (4)$$

$$F_{k1} = -T_{Ay} - qz_1 = -425 - 0,197 \times z_1 = -7,430 \text{ kg} \quad (5)$$

$$M_{\epsilon 1} = -T_{Ay} \times z_1 - \frac{qz_1^2}{2} = -425 \times 35,56 - \frac{0,197 \times 35,56^2}{2} = -139,689 \text{ kg.m} \quad (6)$$

$$F_{k2} = -T_{Ay} - q(z_1 + z_2) + T_{By} = -425 - 0,197 \times 61,94 + 15,324 = 2,695 \text{ kg} \quad (7)$$

$$M_{e1} = -T_{Ay} \cdot (z_1 + z_2) - \frac{q(z_1 + z_2)^2}{2} + T_{By} \cdot z_2 \quad (8)$$

$$M_{e1} = -425 \times 61.49 - \frac{0.197 \times 61,94^2}{2} + 15,324 \times 26,36 = -0.603 \text{ kg.m} \quad (9)$$

Top and bottom profile cross-sectional area are $A_1=63 \text{ cm}^2$ and $B_1=51 \text{ cm}^2$ and $I_y = 1,250,299 \text{ cm}^4$.

$$x_g = 75.4 \text{ cm}; y_g = \frac{(12 \times 12 - 9 \times 9) \times 179}{(12 \times 12 - 9 \times 9) + 2 \times (10 \times 10 - 7 \times 7)} = 68.3 \text{ cm} \quad (10)$$

Maximum stresses in profiles at point B loaded is

$$M_{e1} = -13,968.9 \text{ kg.m} \quad (11)$$

$$M_{e2} = 59,638 \times 110.7 = 6,601.927 \text{ kg.m} \quad (12)$$

$$\sum M = -13,968,900 + 6,601,927 = -7,366.974 \text{ kg.m} \quad (13)$$

For A, B, and, C, maximum critical stress values are

$$\sigma_A = \frac{\sum M}{I_y} \times (-a) = \sigma_A = \frac{-7,366,974}{1,250,299} \times (-110.7) = 63.98 \text{ MPa} \quad (14)$$

$$\sigma_{B,C} = \frac{\sum M}{I_y} \times e = \frac{-7,366,974}{1,250,299} \times (+68.3) = 39.43 \text{ MPa} \quad (15)$$

4. FINITE ELEMENT ANALYSIS OF THE TOWER CRANE

The model which was designed with SolidWorks CAD software was transferred to ANSYS and prepared for the analysis. For defining the problem, firstly all parts meshed and it consisted of 1,027,273 elements and 3,054,210 nodes. After the setting fixings and loads in the Workbench page, the analysis was carried out and stresses were obtained from the point which was the most hazardous points in the analytic calculation.

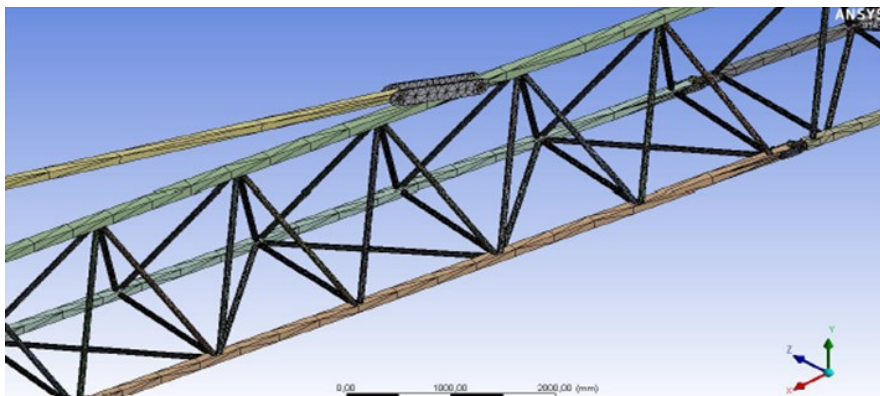


Figure 2. Mesh view of the crane's boom.

The boom consists of three profiles, one at the top and two at the bottom. These three profiles carry forces and moments. These cylindrical profiles which referred to as bars and connect the profiles don't carry moment because they join at the same point. It should be well known that the pins carry the rotational moment. During the modelling, the mast sections were connected with the bolts. The loads on the boom are the weight of the boom itself, the weight of the car, the braking force, the counterweight. The dynamic effects are multiplied by the dynamic coefficient as in the FEM standards included in ANSYS. The force created by the counterweight is also added to the counterpart.

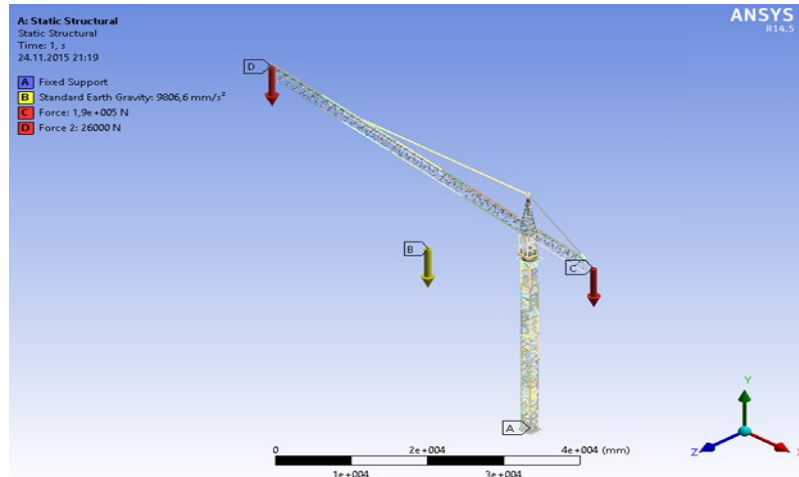


Figure 3. A loading case of the tower crane.

As shown in Figure 4, the top profile measures 61.2 MPa while the bottom profiles measure 38.2 MPa. These values both do not exceed the yield value of 175 MPa for the St37-2 and ensure that the safety coefficient is high, indicating that the system is safe.

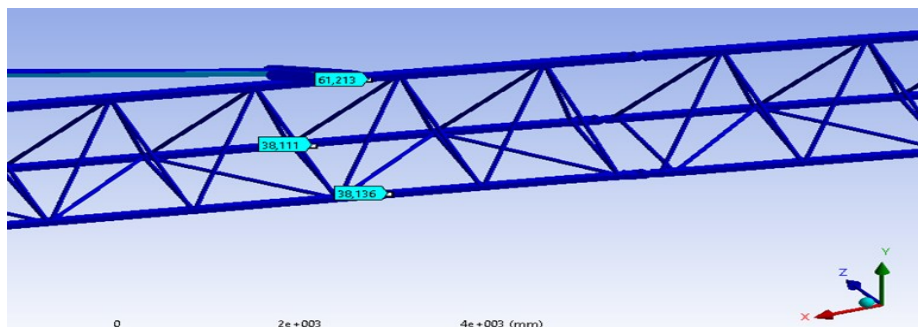


Figure 4. Stress values on critical points.

5. CONCLUSION

The stress values on the components of tower crane boom are computed both by analytical and FEM method. As seen in Table 1, the maximum difference between these methods is not more than 5% except the tower connection part.

Table 1. Comparing the analytical and Finite Element Critical Stresses.

Section / Loading Type	Analytical Calculation [MPa]	Finite Element Method [MPa]	Difference (%)
Boom Top	63.98	61.29	4.2
Boom Bottom	39.43	38.11	3.3

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 tower crane is safe according to FEM norms. In future studies, dynamic loading conditions can be examined.

6. REFERENCES

- [1] Gerdemeli, İ., Kurt, S. and, Deliktaş, O.: Finite Element Analysis of the Tower Crane, 14th International Research/Expert Conference "Trends in the Development of Machinery and Associated Technology" TMT 2010, Mediterranean Cruise, 11-18 September 2010.,
- [2] Li, B., Xu, S. S., Zhang, H., Wang, F. M. and, Zhao, X.: Research on the Virtual Test Technique of Tower Crane Jib. Advances in Manufacturing Technology, Vols. 220-223, pp. 521-525, 2012.