Stress Analysis for the Critical Metal Structure of Bridge Crane

To cite this article: Zhangwei Ling et al 2018 IOP Conf. Ser.: Earth Environ. Sci. 108 022056

View the article online for updates and enhancements.
Stress Analysis for the Critical Metal Structure of Bridge Crane

Zhangwei Ling, Min Wang, Junfang Xia, Songhua Wang and Xiaolian Guo
Zhejiang Provincial Special Equipment Inspection and Research Institute, Hangzhou, 310020, China

*Corresponding author e-mail: wangmin@zjtj.org

Abstract. Based on the type of connection between the main girder and end beam of electrical single beam crane, the finite element analysis model of a full portal crane was established. The stress distribution of the critical structure under different loading conditions was analyzed. The results showed that the maximum Mises stress and deflection of the main girder were within the allowable range. And the connecting location between end beam web and main girder had higher stress than other region, especially at the lower edge and upper edge of the end beam web and the area near the bolt hole of upper wing panel. Therefore it is important to inspect the connection status, the stress condition and the crack situation nearing connection location during the regular inspection process to ensure the safety of the connection between the main girder and end beam.

1. Introduction
The electrical single beam crane consists of main girder, end beam, electric hoist, electrical device and traveling mechanism critical piece [1]. For convenient transportation, the main girder and end beam are respectively independent components, and then combined to a full portal crane by high-strength bolt when arrived at destination. Since the crane structure affected by lifting capacity, fatigue load, material aging and overload using during service, the critical metal structure of bridge crane, such as mid-span of main girder, end beam web, bolt connection and welding heat affected zone [2,3], will produce inevitable damage accumulation. Therefore, it is very important to study the stress condition of the precarious position of crane metal structure. Mao Cong [4] studied the three-dimensional finite element analysis for the end carriage of the overhead traveling crane, calculated the deformation, maximum stress and displacement value of the end carriage. And the structure of the end beam was improved appropriately. Chen Feiyu [5] studied the mechanical performance of single girder crane’s overall structure, calculated the strength of the bridge structure and analyzed the deformation and stress distribution of the typical working conditions. And the result was shown that the dangerous section of bridge is not in the cross areas, but the girders connection between girders parts of the stress condition which is more complex. Tang Jianfu [6] analyzed the cause of one crane accident. The conclusion was shown that the crane had fatigue crack in the connection position of the web. And the continuous expansion of the crack leaded to the failure of the web to bear the load. Hou Xianqi [7] analyzed the mechanics of electric single-beam bridge
crane’s structure, and the stress concentration phenomenon in the beam and connecting plate was found.

Therefore, the paper established the finite element analysis model of a full portal crane based on the electrical single beam crane in mill building, and analyzed the stress distribution of main girder, end beam and connection position of electrical single beam crane under different loading conditions.

2. The finite element analysis model of crane

The electrical single beam crane in mill building was shown as Fig.1, with rated load lifting capacity 10t, span 6800mm, lifting altitude 6000mm, thickness of main girder web, upper wing panel and bottom wing panel is 5.5mm, 10mm and 20mm. The end beam has 3000mm length, 300mm high and web thickness 5.5mm.

![Figure 1. The electrical single beam crane.](image1)

Based on the configuration size and installation way, the finite element analysis model of a full portal crane was established as showed in Fig.2. Since the thickness ratio of the main girder, the main girder was modeled as the shell structure and the end beam was modeled as entities. The end beam was connected with main girder through the connecting plate and the high strength bolt M20. The bolt was considered as a simplified model. The contact between bolt and end beam, bolt and main girder and bolt and the connecting plate were defined by setting the contact property. And the bolt pretightening force was set to simulate bolt connection.

![Figure 2. The finite element analysis model.](image2)

The analytical model of the crane was considered as a simple beam model when simulating the boundary conditions. The lifting load was applied to the bottom wing panel of main girder by coupling loading in 4-wheel positions. And the gravity and load factor were considered during calculation. The material parameters of the crane were shown in Table 1.
Table 1. The material parameters of crane.

| Material | E/GPa | $\sigma_y$/MPa | $\sigma_b$/MPa | $\nu$ |
|----------|-------|----------------|----------------|------|
| Q235A    | 206   | 235            | 375            | 0.3  |

3. The theory analysis model for main girder

The load carried on the main girder and consisted of lifting weight and electric hoist could be applied as concentrated load when calculating the strength of main girder, since the track gauge of electric hoist was small relative to length of main girder. The theory analysis model for main girder could be simplified as showed in Fig.3. $P$ is the concentrated load, consisting of the weight of electric hoist $PQ$ and lifting weight $PH$. $q$ is uniform weight load of main girder along the length $L$.

![Figure 3. The theory analysis model of main girder.](image)

According to the Material Mechanics, the deflection of mid bottom wing panel is the algebraic sum of deflection under uniform load and concentrated load, ie, the deflection of mid bottom wing panel is,

$$f = \frac{5qL^4}{384EI} + \frac{P \times \phi}{48EI}$$  \hspace{1cm} (1)

Where, $L$- span of main girder. $I$- moment of inertia of mid span section. $\phi$ -the lifting load coefficient, taken as 1.17.

Based on calculation, the deflection of mid bottom wing panel is 6.04mm, with the 10t lifting weight applied at mid span.

4. The numerical analysis for main girder

The stress distribution of main girder with 10t lifting load applied at mid span, was presented in Fig.4(left figure). Maximum Mises stress was 76.74MPa, less than the allowable stress. The deflection of the mid bottom wing panel was 6.6mm, with difference of 8.45% compared with the theoretical calculation results, which show the accuracy of the crane analysis model established in this paper. The stress distribution of main girder with 10t lifting load applied at end span, was shown in Fig.5(right figure). Maximum Mises stress was 117.1MPa, less than the allowable stress too.
5. The stress analysis for connection part
The stress distribution of half end beam model was shown as Fig.5, with 10t lifting load applied at mid span. As shown in the figure, the stress of connecting local part between end beam and main girder was higher than that of other parts and the bolt hole had obvious stress concentration.

Figure 4. Stress distribution of main girder under 10t load applied at mid and end span.

Figure 5. Stress distribution of end beam.

Figure 6. Analysis path.

For analyzing the stress state of typical connecting local, the analysis path was chosen to study the stress distribution of end beam, as showed in Fig.6. The stress state along the path was shown in Fig.7 and Fig.8, under different lifting weight of 0t, 2t, 5t and 10t. As shown in figure, the stress on the analysis path was increased with the increase of weight. And in the end beam web region, showed in Fig.7, the stress of lower edge was greater than that of the upper edge and the maximum stress was 140.2MPa. This is mainly because of the large bending tensile stress of lower edge under the weight loading. There were two stresses peak during 100mm to 250mm along the path of end beam web, since the peak area was close to the bolt hole, in where had a large stress concentration. As showed in Fig.8, the stress along the path of upper wing panel also appeared two stress peaks, higher than that of end beam web.
Figure 7. The stress distribution along path in end beam web.

Figure 8. The stress distribution along path in upper wing panel.

From the results of Fig8(b) and (c), the critical structure of the lower edge and upper edge of end beam web and the area near the bolt hole of upper wing panel had relatively high stress value. If there’s fatigue crack in those critical structure, the crack may be propagated under the stress condition less than the strength stress, which will cause the end beam web to fracture, resulting in a crane accident. Therefore, it is important to inspect the connection position of end beam in the periodic inspection process, especially the crack condition near the bolt hole.

6. Conclusion
Through the numerical analysis and theoretical analysis for electrical single beam crane, the stress and deformation of main girder were all within the allowable range with the rated lifting weight. And the lower edge and upper edge of end beam web and the area near the bolt hole of upper wing panel, had relatively high stress value, especially the connecting bolt hole had obvious stress concentration. Therefore, it is important to inspect those critical structures in the periodic inspection process, especially the crack condition near the bolt hole.

Acknowledgments
This work was financially supported by the science and technology project of Zhejiang Provincial Administration of Quality and Technology Supervision (NO.20160228).

References
[1] Zhang Zhijian. Analysis and Research on Quality and Safety of Electric Single Girder Crane [D]. Zhejiang University of Technology, 2014.
[2] Tao Ye. Parameterization Design and Finite Element Analysis for Electric Hoist Single-Beam Bridge Crane [D]. Taiyuan University of Science and Technology.2014.
[3] Ren Yuan. Structural Analysis and Optimization Design of Single-beam Bridge Crane [D]. Taiyuan University of Technology, 2014.
[4] Mao Cong, Guo Kexi, Li Heqing. Three-Dimensional Finite Element Analysis for the end Carriage of the Overhead Traveling Crane [J]. Journal of Changsha University of Science and Technology (Natural Science), 2009, 6 (3):59-62.
[5] Chen Feiyu, Wei Guoqian, Xia Xuhui. Mechanical Performance Study of Single Girder Crane’s Overall Structure [J]. Machinery Design & Manufacture, 2012 (9):126-128.
[6] Tang Jianfu. Analysis of the cause of crane accident [J]. China Equipment, 2014 (S2).
[7] Hou Xianqi, Ren Yuan. Mechanical Analysis of Electric Single-beam Bridge Crane’s Structure[J]. Mechanical Engineering & Automation, 2016 (1): 60-61.