Sensors & Transducers, Vol. 21, Special Issue, May 2013, pp. 89-94

Sensors & Transducers

Finite Element Analysis on Crane Girder with Variable Cross Sections Based on ANSYS

Dan Meng, Gongfa Li, Dawei Tan, Shulin Yang
College of Machinery and Automation,
Wuhan University of Science and Technology, Wuhan 430081, Hubei, China
Tel.: 15671676471
E-mail: mengdan1023@163.com

Received: 20 March 2013 /Accepted: 14 May 2013 /Published: 30 May 2013

Abstract: To the 36 meters crane girder with trapezoidal variable cross-section in a steel plant, finite element model is established reasonably according its actual size. The calculation of crane girder structure was introduced by using the finite element analysis software ANSYS. The local stress state of the cross-section is investigated and the stress concentration position of variable cross-section is found, which can provide a theoretical basis for the establishment of monitoring system. At the same time, the former 6 natural frequencies and the corresponding vibration patterns of the crane beam are extracted in modal analysis of crane girder, which is to provide a basis and reference for the structure design of crane girder. *Copyright © 2013 IFSA.*

Keywords: Crane girder, Finite element analysis, Trapezoidal cross-section, ANSYS.

1. Introduction

Steel crane girder is a member which supports various types of cranes in factory buildings. Due to the complexity of crane girder stress, the metal structure of crane is easy to discover crack, propagation, and sudden rupture under cycles of alternating loads. Especially crane girder with trapezoidal variable cross-section, due to the poor fatigue resistance of the mutation of cross-section, fatigue fracture appears frequently in the practical engineering which results in reduced reliability and security of the crane beam as well as the increase of service life.

In recent years, the situation of fracture of crane beam caused by fatigue often appears in a steel mill. Such as dozens root crane beam cracking in the steel billet library of original rough rolling mill in 2003, crane girder at the top of the platform in iron making plant cracking because of fatigue in 2007, main span 22 # crane beam in iron making plant cracking suddenly in 2011.All of these are directly or indirectly affect the safety in production caused by fatigue crack or damage. Therefore, the stress test and modal analysis of the crane beam which has long service cycles and use of high frequency has important practical significance.

If the crane girder with trapezoidal variable cross section is calculated by using the traditional mechanical calculation method, the relevant parameters should be estimated and simplified, and then inaccurate calculation result is reflected. To this end, the static characteristics and the dynamic characteristics of the crane girder are simulated by using the finite element analysis software of ANSYS [1]. Through analytical data, the state of deformation and stress distribution of crane girder is researched, the stress concentration position of variable cross-section is found, the distribution of bending rigidity and torsional rigidity of crane girder is determined. The comparative analysis of analytical data and field results is performed, which can provide a theoretical basis for the establishment of monitoring system and maintenance and reinforcement of the crane girder. Finally, the rate of accidents can be reduced and the safe production can be promoted. 36 meters span crane girder structure with trapezoidal variable cross-section is shown as Fig. 1.

Fig. 1. Crane girder structure with trapezoidal variable cross-section.

2. Static Analysis of the Crane Girder

2.1. Establish the Model of Crane Girder

The reasonableness of finite element model of the crane girder structure directly affects the accuracy of the finite element results. Therefore, in this paper, the finite element model of the crane girder is drawn by the three-dimensional mapping software of PROE, then change into IGES format and import into ANSYS. The selected units of PROE is mm/N/s, coordinate origin is fixed at the center of the bottom flange. The material of crane girder is Q235, isotropic material, the elastic modulus E = 2.1E5 MPa, Poisson ratio μ = 0.3, density ρ = 7.8E-9 t/mm^3. The entire structure meshed by solid element (Solid 45). The schematic plan and the three-dimensional model of crane girder in PROE are shown as Fig. 2 and Fig. 3. The meshed map of crane girder is shown as Fig. 4.

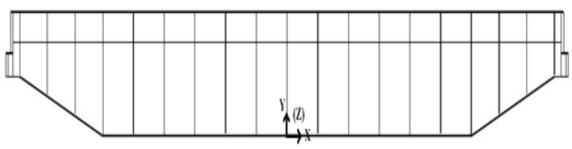


Fig. 2. The schematic plan of crane girder.

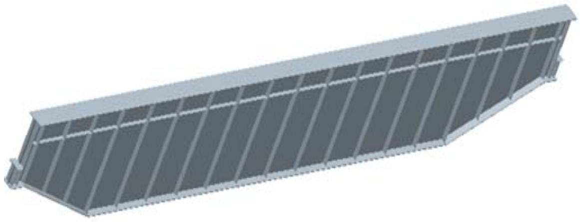


Fig. 3. The three-dimensional model of crane girder in PROE.

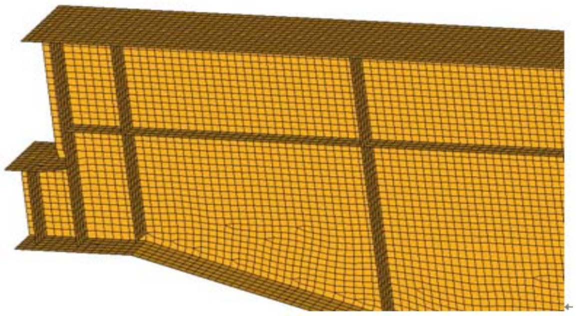


Fig. 4. The meshed map of crane girder.

2.2. Load Case of the Crane Girder

Because the size of crane girder is large, the structure is complex, so it is simplified to simply supported beams. One side is fixed hinged bearing and the other is horizontal movable hinged bearing [2]. In static analysis, the weight of crane girder is not considered, only hanging wheel pressure is considered. So the calculated stress is the stress amplitude. The working-level of crane is A7, there are four small wheels on each side, whose maximum wheel pressure is 272KN. The wheel pressure distribution is shown as Fig. 5.

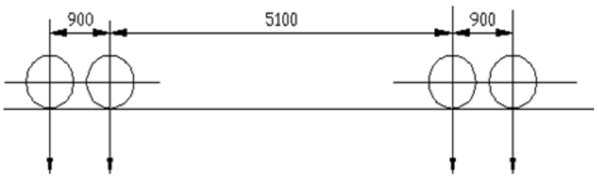


Fig. 5. The wheel pressure distribution.

Because the wheel pressure which acts on the beam is dynamic loads, the most unfavorable load position of the crane girder should be pointed, which produce maximum bending moment. According to the crane running and Steel Design Manual [3]: when the bending moment is maximum, the crane is not located in the center of the crane beam, but located in the position shown in Fig. 6.

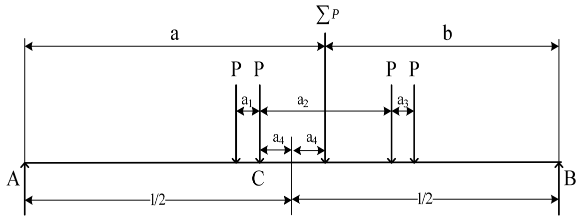


Fig. 6. The location of crane when the bending moment is the maximum.

Steel Design Manual [4] pointed out the force of beam when four wheels act on the beam.

1) The location of the maximum bending moment point (C):

$$a_4 = \frac{2a_2 + a_3 - a_1}{8}, \tag{1}$$

2) The maximum bending moment:

$$M^c_{\max} = \frac{\sum p\left(\frac{l}{2} - a_4\right)^2}{l} - Pa_1, \tag{2}$$

3) Radiation formula:

$$V^c = \frac{\sum p\left(\frac{l}{2} - a_4\right)}{l} - P, \tag{3}$$

where l is the span of crane girder, P is the wheel pressure which was distributed in the form of concentrated force, a is the distance from the application point of join forces to the A-side.

When $a_3 = a_1$, $a_4 = \frac{a_2}{4}$.

The maximum bending moment $M^c_{\max}$ and its corresponding shear V^c are all the same as the formula (2) and formula (3), while a_4 in the formula should be replaced by $\frac{a_2}{4}$.

The mechanical model should be loaded as the case shown in Fig. 6.

The length of the entire crane girder is 36 meters. When the bending moment is the maximum, the distance from application point of join forces to one end of crane girder is: $a = 19.275m, b = 16.725m$.

2.3. Calculation Results and Analysis of the Static Characteristics

Deformation of crane girder belongs to elastic deformation, using finite element method can calculate the stress and strain after loading. The finite element analysis software of ANSYS has powerful pretreatment, solving and post-processing functions. It has reliable calculation, high efficiency, and is a powerful tool in structural analysis [5]. The deformation of the vertical direction of the crane beam is shown in Fig. 7, Von Mises stress equivalent diagram of variable cross-section is shown in Fig. 8, and Von Mises stress contours of variable cross-section is shown in Fig. 9.

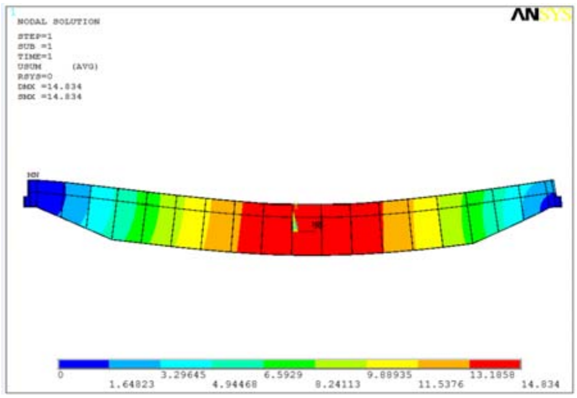


Fig. 7. The deformation of the vertical direction.

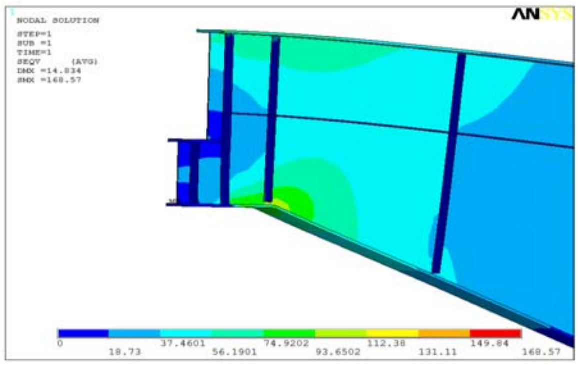


Fig. 8. Von Mises stress equivalent diagram of variable cross-section.

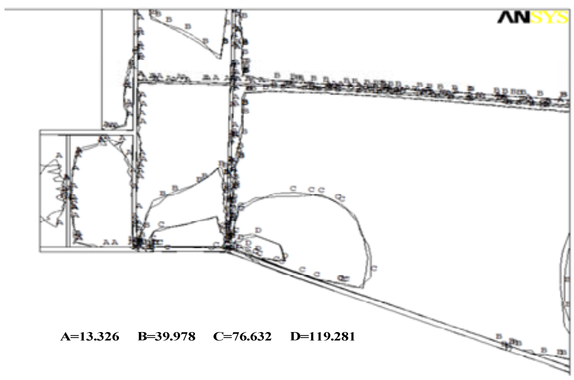


Fig. 9. Von Mises stress contours of variable cross-section.

When the crane is located in the position shown in Fig. 6, the maximum vertical deflection of the crane beam is 14.834 mm (L/2427), while allowable value is 30 mm (L/1200); and the maximum Von Mises stress of variable cross-section is 119 MPa. The stress concentration phenomenon appears on variable cross-section of crane girder, so crane girder easily fracture due to fatigue of variable cross-section. Under the effect of dynamic loads of the crane whose working-level is A7, crack of trapezoidal variable cross-section is generated. The crack is the same as the judgment of calculation result. The actual crack of trapezoidal variable cross-section crane girder is shown in Fig. 10.

Fig. 10. The actual cracks of trapezoidal variable cross-section crane girder.

3. Modal Analysis of the Crane Girder

3.1. The Theory of Modal Analysis

Finite element modal analysis is the progress to establish the modal model and conduct numerical analysis [7]. For the general structure system with multi degree of freedom, any movement can be synthesized by its free vibration modal. For one linear system with multi degree of freedom, the vibration equation can be represented as:

$$M\ddot{u}(t)+C\dot{u}(t)+Ku(t)=P(t)\,, \qquad (4)$$

where M is the mass matrix; C is the damping matrix; K is the stiffness matrix; they are all n order square. $\ddot{u}(t)$ is the acceleration vector; $\dot{u}(t)$ is the velocity vector; $u(t)$ is the displacement vector; $P(t)$ is the dynamic load vector.

The essence of the modal analysis is to solve the modal vector of motion equation with a finite number of degrees of freedom without external load. Structural damping can be ignored, the effect of it on its modal frequencies and mode shapes is small [8]. In the process of modal analysis, undamped free vibration equation is:

$$M\ddot{u}(t)+Ku(t)=0\,, \qquad (5)$$

ANSYS offers six modal extraction methods [9], the Block Lanczos method is selected to conduct modal analysis [9]. Block Lanczos method is the acquiescent solution method of ANSYS, it uses the Lanczos algorithm, which realizes recursive algorithm by a set of vectors. Its characteristic is that using sparse matrix equation solver which turns n*n order matrix by similar transformation into three diagonal matrix to get characteristic value. It is also with less input parameters, faster convergence speed and higher accuracy of characteristic value and characteristic vector solution. It is suitable for solving the problem of large symmetric matrix [10].

3.2. Calculation Results of Modal Analysis

In order to understand the dynamic characteristics of crane girder, natural frequencies and mode shapes, modal analysis of the crane girder is carried out. In modal analysis, unit types and constraints loaded remains the same as parameters of static stress analysis. As crane beam is low-frequency vibration structure, engineering value in use is several previous order nature frequencies. The 1st to 6th vibration modes are extracted in modal analysis of crane girder, and the results are shown as Table 1.

Table 1. The modal results of the top 6 steps.

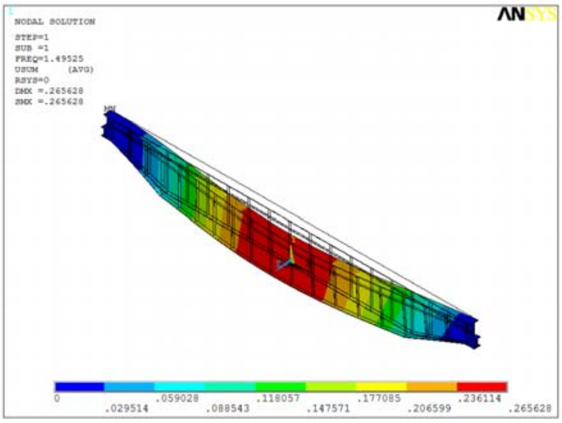
SET	1	2	3	4	5	6
FREQ(HZ)	1.50	3.28	4.29	7.35	8.43	10.51

The vibration diagrams of 1st to 6th are shown as Fig. 11.

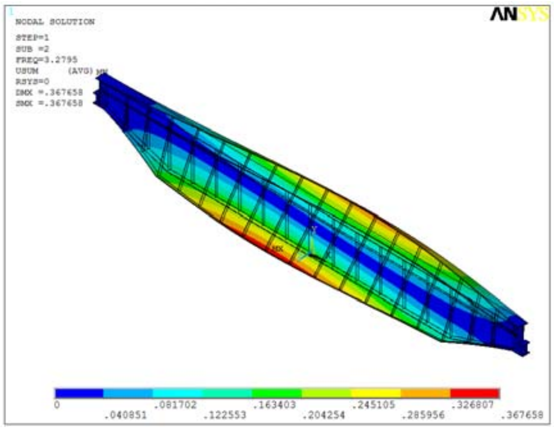
Known by the calculation results, because of the uneven distribution of crane beam stiffness, crane beam on the bottom flange is relatively weak. When the excitation frequency reaches its natural frequency, it produces resonance and cause jitter. However, within the scope of our analysis, the mode vibration deformation amount of crane beam is very small, and general resonance frequency is relatively high, so resonance is not produced, it is always consistent with requirements.

4. Conclusions

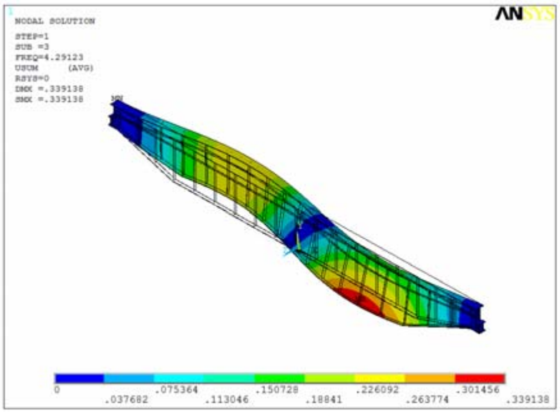
The results of the static analysis of crane beam show that stress concentration phenomenon appears on ladder variable cross-section. Under repeated loads, fatigue crack appears easily and the calculation result is consistent with the actual working condition. So the crane beam should be checked regularly in use process, which can contribute to find obvious deformation and steel plate cracking of the crane beam. These problems should be timely reported, and then take the necessary measures to avoid certain losses and damage.



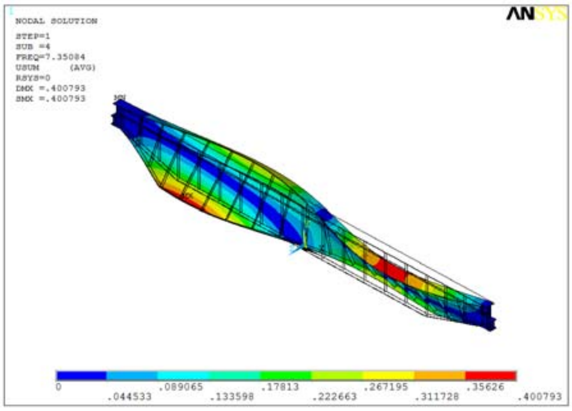
(a) The 1st vibration mode.



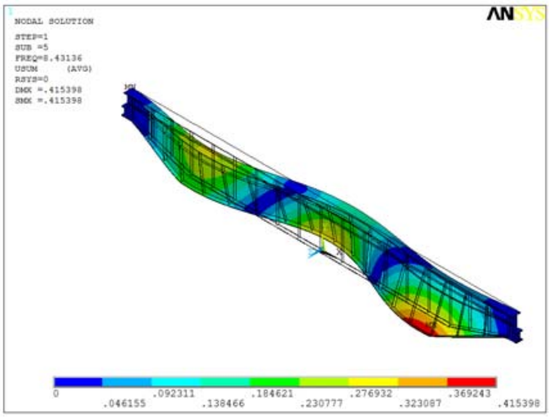
(b) The 2nd vibration mode.



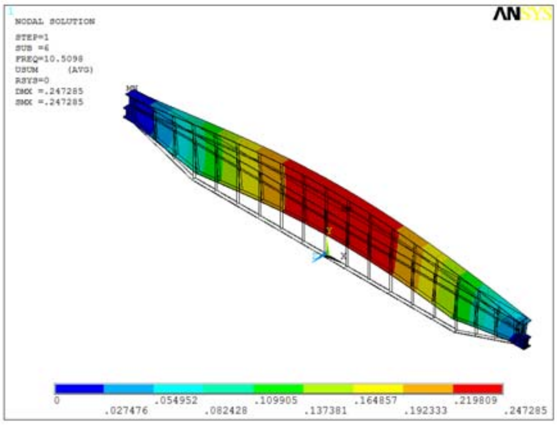
(c) The 3rd vibration mode.



(d) The 4th vibration mode.



(e) The 5th vibration mode.



(f) The 6th vibration mode.

Fig. 11. The vibration cloud modes of 1st to 6th.

Through modal analysis of the crane beam, the distribution of bending rigidity and torsional rigidity of crane beam can be determined. In order to ensure the safe use of trapezoidal cross-section of crane beam, the flange and web connection strength should be strengthened in the process of design and manufacturing under, especially the welding quality of these parts. At the same time, special attention should be paid to the working conditions of these areas in the daily maintenance of crane beam, especially to the detection of weld, which can improve the fatigue life of crane beam, reduce the accident rate and promote safety production.

References

[1]. H. S. Xu, ANSYS Application in Construction Projects, *Machinery Industry Press*, Beijing, 2005.

[2]. J. Chen, Z. H. Lu, F. Xu, Q. X. He., Performance Analysis of the Arc and Right-Angle Type Mutation

Bearing of Cross-Section Crane Beam, *Structural Engineer,* Vol. 26, Issue 1, 2010, pp. 22-26.
[3]. Y. J. Wang, Steel Structure Design Manual (Third Edition), *China Building Industry Press*, Beijing, 2004.
[4]. B. F . Luo, M. Z. Wei, Steel Structure Design Manual Beijing, *China Building Industry Press*, 1989.
[5]. H. M. Wang, Z. S. Liu, Dynamic Characteristics Analysis of the Four Hot Mill Rack, *Journal of Hefei University of Technology,* Vol. 32, Issue 3, 2009, pp. 40-45.
[6]. Y. F. Ding, ANSYS12.0 Finite Element Analyses, *Electronic Industry Press*, Beijing, 2011.
[7]. Y. Y. Cao, D. F. Zhao, Analysis Theory and Its Application of the Finite Element Modal, *Mechanical Engineering & Automation,* Vol. 1, Issue 1, 2007, pp. 352-354.
[8]. A. F. Yuan, J. Chen, The Application of Modal Analysis in ANSYS, *Manufacturing Technology & Machine Tool,* Vol. 1, Issue 8, 2007, pp. 102-105.
[9]. K. Chen, Modal Analysis of PCR-80 Cold Rolling Machine and Multiple Objective Optimization of Total Machine Rigidity, *Journal of Hefei University of Technology,* Vol. 1, Issue 1, 2005, pp. 26-27.
[10]. H. L. Jia, P. F. Du, X. P Ren, Modal Analysis of Crane Beam Structure Based on the ANSYS, *Industrial Buildings,* Vol. 40, 2010, pp. 406-408.