Sensors & Transducers, Vol. 21, Special Issue, May 2013, pp. 73-77

www.sensorsportal.com

Sensors & Transducers

http://www.sensorsportal.com

Anti-fatigue Performance Analysis on Steel Crane Beam

Yuanmin Xie
College of Machinery and Automation,
Wuhan University of Science and Technology, Wuhan 430081, Hubei, China
Tel.: +86-027-68862283
E-mail: wustxie@126.com

Received: 31 March 2013 /Accepted: 14 May 2013 /Published: 30 May 2013

Abstract: Loads and working conditions of the trapezoidal cross-section of 36 m span crane beam can be analyzed by finite element method. Firstly, analysis of the stress and deformation is completed under the dangerous condition of crane beam based on the ANSYS software; secondly, find out the stress concentration and fatigue crack sensitive area of the bearing; finally, capture the position of fatigue crack. All these methods can provide strong basis for implementing effective monitoring to the state of the crane beam stress. Thus, ensure safe operation of equipment and improve equipment utilization rate in work. *Copyright © 2013 IFSA.*

Keywords: Fatigue cracks, Crane beam, Stress concentration, Finite element analysis.

1. Introduction

In recent years, the steel structure is widely used in industry and civil construction. Steel structure of crane beam is one of them which are used in industrial workshop electric hoist of wall or bridge supporting beam, wall cranes and other type of cranes. In industrial workshop, cross section of the crane beam is needed to be changed at joints as well as the production technology and economy requirements. So, designers can use a variety of ways that can be summarized up as: cross-section of the end trapezoidal, cross-section of the end Angle variable and cross-section of the end arc variable. Trapezoid transition of variable cross-section bearing is widely applied in engineering practice because of its relatively high anti-fatigue performance.

The stress of steel crane beam structure is complex; the repeated actions of vertical wheel crane pressure often does not meet the design stress, so fatigue crack happened at the parts. China has begun to pay attention to study the cause of the cracks, the prevention and control method since the 1990 s. But until now, someone haven't seen a effective measures to stop this kind of crack, the current specifications for design of steel structure in our country has not explicitly put forward to prevent the cracks of concrete calculation method and construction measures [1].

Now, more and more crane beam fatigue cracks of heavy duty welded steel have been discovered, especially more serious on large tonnage of heavy duty steel crane beam. Because the early cracks that is tiny and more hidden, often ignored by people. Dozens of crane beams cracking in the original rough rolling mill of a steel plant in 2003, feeding oblique beam fatigue cracking in iron in 2007. All of these caused by fatigue crack or damage, are directly or indirectly affect the safety in production. So the unit should census earnestly to deal with this kind of heavy duty welded steel crane beams, who find the problem and deal with as soon as possible to ensure the safe use of this important component. So in this paper, the variable cross section of the crane beam should be simulated to find out the static characteristics and fatigue performance by application of finite element analysis software ANSYS, which can determine the trapezoidal variable cross-section stress concentration, produce the detection of cracks and establish an effective monitoring system. It can avoid unnecessary loss. The cracks often appear in the section of trapezoidal variable crane beam in actual engineering such as Fig. 1.

Fig. 1. The actual cracks of crane girder.

2. Overall Layout of Crane Beam

Crane beam is mainly composed of main girder and end beam, main beam and side beam is rigid connection, on both ends of the beam is equipped with wheel, in order to support bridge running on crane beam. Crane beam is generally designed as simply supported beam, whose characteristic is force clear, simple structure and convenient construction. The different column spacing in the workshop leads to the different heights of the adjacent crane beams. And to make the same column rail surface elevation at the same height, the crane beam is trapezoidal mutation type bearing, the beam end using the trapezoidal cross section.

Steel structure design specification [2, 3] is in accordance with the crane use status and level of work, divided the work level as 4 working system such as light, medium, weight and heavy duty. Crane design specification [4] and the crane load code divided crane work level into grade A1~A8. The crane beam span of this study is 36 meters, work level for the A7, namely the structure of heavy duty.

3. Finite Element Analysis

Finite element method [5] is a method of structural analysis; its basic idea is to disperse solving continuous area to unit assembly which is composed of a finite number of units connected together in a certain way to analyze. The current finite element software analysis function covers almost all engineering field; the use of the program is also very convenient. Currently, finite element analysis software which is widely used and very popular in Chinese engineering field contain: ANSYS, Abaqus, Marc and Algor etc.

Finite element equation principle: by the known element stiffness matrix and equivalent node load array assembled into the whole stiffness matrix of the whole structure and load array, assisted by a total stiffness matrix [K], total load vector {F} and integral nodal displacement vector $\{\delta\}$ showed balance equations.

$$[K]\{\delta\}=\{F\}, \tag{1}$$

The overall nodes displacement vector is obtained after introducing displacement boundary conditions. The finite element discrete equation is an algebraic system of equations, the symmetric stiffness matrix after introducing boundary conditions is a definite sparse equation, algebraic equations of such a can be solved by using a variety of methods.

ANSYS is one kind of engineering analysis software [6], mainly in the mechanical structure system by external load of reaction, such as displacement, temperature stress. The state of mechanical structure system can be got according to the reaction by external loads. General geometry of mechanical structure system is very complex; the load is considerable, so theory analysis often can not be conducted. In order to answer the analysis, the structure should first simplified by using the numerical simulation analysis method. Finite element software ANSYS has strong treatment before solving and post-processing function [7]. It has reliable calculation and high efficiency, and it also is a powerful tool in structural analysis. It is widely used in engineering, which can reduce the design cost and shorten the design time.

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.

3.1. Load Analysis

The calculation load of the crane beam is shown in Fig. 2, including: vertical loads generated by the crane, transverse horizontal loads and longitudinal horizontal loads. The longitudinal horizontal load is along the direction of crane rail, which is supported by column crane beam. So it can be ignored when calculating. Transverse horizontal load is mainly composed of crane beam on the flange of the directly to the brake structure, relative to the vertical load, its effect on crane beam is smaller, for which the influence of the flange and its nearby area is much smaller than top flange and bearing variable cross-section is generally close to the flange. Therefore, in order to simplify the calculation, only the vertical load on the web need considering.

Considering a variety of adverse conditions, the most dangerous situation of crane beam is when the bending moment is maximum. Then the crane is not located in the center of the crane beam, but located in the position shown in Fig. 3.

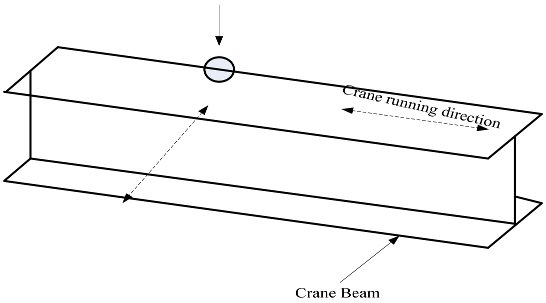

Fig. 2. Calculation load of the crane beam

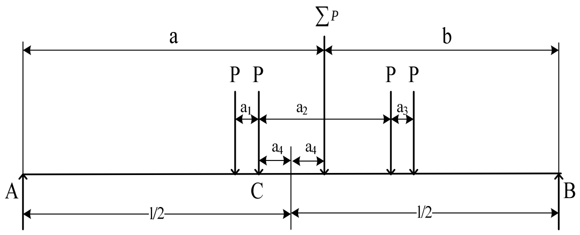

Fig. 3. The most dangerous situation.

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

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

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

(2)The maximum bending moment:

$$M^c_{\max} = \frac{\sum p\left(\frac{l}{2} - a_4\right)^2}{l} - Pa_1, \tag{3}$$

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 $M^c_{\max}$ is the same as the 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. 4.

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: $a = 19.275m, b = 16.725m$.

3.2. Material Setting and Modeling

The material setting of the crane beam is shown as Table 1.

Table 1. The material setting.

Material	Elastic Modulus (MPa)	Poisson Ratio	Density (t/mm^3)
Q235	2.1×10^5	0.3	7.85×10^{-9}

Steel crane beam is composed of web plate, flange plate, stiffener and many other small plates. It is very complicated if all parts are considering, and parts or less important small ribs can make the ANSYS calculation inaccurate even not able to be calculated. So it is necessary to make model simplified, 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.

3.3. Load Case of the Crane Girder

(1) Meshing. The entire structure meshed by solid element (Solid 45). The meshed map of crane girder is shown as Fig. 4.

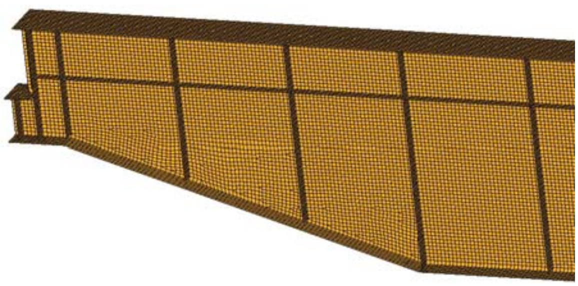

Fig. 4. The meshed map of crane girder.

(2) Constraint. 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 [8].

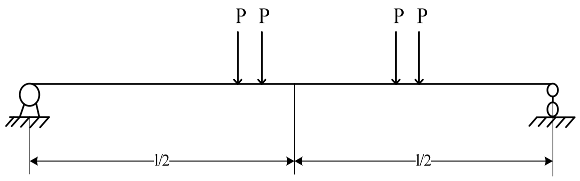

Fig. 5. Simplified as simply-supported beam.

(3) Loading. 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. There are four small wheels on each side, whose maximum wheel pressure is 272 KN. The wheel pressure distribution is shown as Fig. 6. The concentrated load of each wheel is evenly distributed to the corresponding nodes nearby, direction is vertically downward.

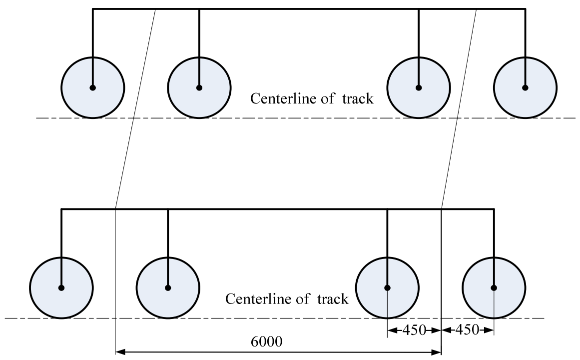

Fig. 6. The wheel pressure distribution

4. Calculation Results

Deformation of crane girder belongs to elastic deformation, using finite element method can calculate the stress and strain after loading. In order to reflect the characteristics of the crane beam more intuitively, the deformation of the vertical direction of the crane beam and Von Mises stress equivalent diagram are drawn out. Von Mises stress equivalent diagram of crane girder is shown in Fig. 7.

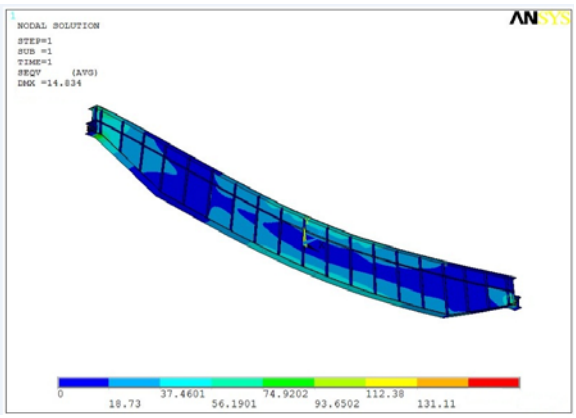

Fig. 7. Von Mises stress of crane girder.

Through finite element static analysis of 36 meters span crane girder structure with trapezoidal variable cross section, the following conclusions can be drawn:

When the crane beam is in the most dangerous conditions, the maximum vertical deflection of the crane beam is 14.834 mm (L/2427), while allowable value is 30 mm (L/1200); crane beam on the bottom flange is relatively weak because of the uneven distribution of crane beam stiffness, and the Von Mises stress is 55.163 MPa; the maximum Von Mises stress of variable cross-section is 119 MPa. The stress concentration phenomenon appears on variable cross-section of crane girder, because of residual stress due to the shrinkage of the weld, so crane girder easily fracture due to fatigue of variable cross-section.

The physical quantities of crane beam generated in the course of work are important test parameters of the monitoring system, such as stress, strain and others, which also can directly reflect working conditions. Determined by static analysis, the monitoring locations are the trapezoidal variable cross section and the middle of the bottom flange. The monitoring locations are shown in Fig. 8.

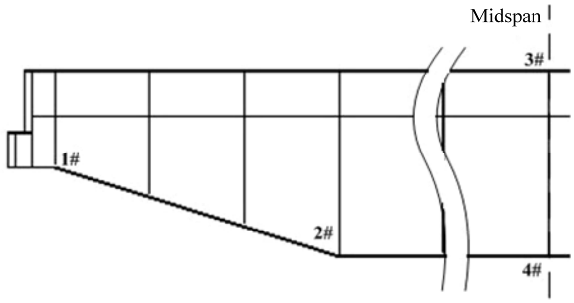

Fig. 8. The monitoring locations.

5. Conclusions

By static analysis of crane beam with trapezoidal variable cross section, the stress levels and performance indicators of the dangerous parts of crane beam are obtained. The welded crane girder destruction is mainly fatigue failure, and stress concentration is the main reason of fatigue failure [10]. The results show that stress concentration phenomenon appears on ladder variable cross section. Under repeated loads, fatigue crack appears easily. So the real-time monitoring of trapezoidal variable cross section is conducted to prevent fatigue failure. At the same time, 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.

In order to ensure the safe use of the crane girder with trapezoidal variable cross section, the real-time monitoring system is established by using advanced sensor technology, which can monitor the crane beam timely and effectively. Especially the trapezoidal variable cross section and the middle of the bottom flange are focal monitoring areas. Therefore, the flange and web connection strength should be strengthened in the process of design and manufacturing under, especially the welding quality of these parts, which can improve the fatigue life of crane beam, reduce the accident rate and promote safety production.

References

[1]. Y. Huang. Analysis and Treatment of Cracks of Steel Crane Beam, *Nanjing Science and Management,* Vol. 1, Issue 3, 2007, pp. 37-39.

[2]. B. F. Luo, M. Z. Wei, etc., Steel Structure Design Manual Beijing, *China Building Industry Press*, 1989, pp. 379-446.

[3]. Y. J. Wang etc., Steel Structure Design Manual (Third Edition), *China Building Industry Press*, Beijing, 2004, pp. 318-393.

[4]. Z. W. Zhang, H. Q. Yu, etc., the Manual Design of Crane, *China Railway Publishing House*, 1998.

[5]. Y. F. Hong, The Basic Principle and Application of Finite Element Method, *Higher Education Press,* 2010.

[6]. X. J. Gao, H. H Zhao, Brief Introduction to the Finite Element Analysis Software ANSYS, *Journal of Liaoning Shihua University*, Vol. 24, Issue 3, 2004, pp. 94-97.

[7]. Y. F. Ding, ANSYS12.0 Finite Element Analyses, Beijing, *Electronic Industry Press*, 2011.

[8]. 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.

[9]. J. Dai, Three Kinds of Variable Cross-Section Crane Beam Analysis by Finite Element, *The Structure of the Steel*, Vol. 26, Issue 11, 2011, pp. 44-46.

[10]. G. J. Xia, The Reason for Web Cracking of Welded Crane Girder and Prevention Measures, *The Structure of the Steel*, Vol. 25, Issue 4, 2010, pp. 65-68.

.