Journal of Physics: Conference Series

PAPER • OPEN ACCESS

Finite element analysis of crane truck main beam based on abacus

Finite element analysis of crane truck main beam based on abacus

Liuyuan[1,2*] Feiyunfeng[2] Hezhenxing[1] Tuqunzhang[3] Hechunping[1]

[1]Xi'an Research Institute of High Technology, Xi'an 710025, Shaanxi Province, China

[2]Beijing Special Engineering Design Institute, Beijing 100011, China

[3]The Army Engineering University of PLA,Nanjing,210000, China

*liuyuanbixue@163.com

Abstract. The load size of a crane's main girder during different working conditions is analysed and calculated. By using Solidworks three-dimensional software to establish the related model, and then the model is introduced into the abacus for finite element analysis. The stress and strain of the main girder of the crane in different conditions are obtained. According to the results of the finite element analysis, the risk position of the main girder in the working process can be found. The maximum stress 266.8MP of the most dangerous working condition of the crane is less than the allowable stress value of 282.8MP, which satisfies the strength requirement. The maximum deflection is 4.422mm, less than the limit of deflection 9.5mm, which satisfies the stiffness requirement. The analysis methods and results of this paper provide some basis and reference for the improvement and optimization of the main girder structure of the truck.

1. Introduction

With the vigorous development of the logistics industry, port wharf and steel plant, the demand for all kinds of cranes has increased rapidly. The traditional design of the traditional crane based on static design has been unable to meet the demand for the rapid update of the product [1]. In recent years, the finite element analysis method is introduced into the design calculation and analysis of the crane. It can not only check the strength and stiffness of the structure by the finite element analysis, but also realize the rapid design of the product [2-5].

In this paper, in view of the main girder of a large vehicle running mechanism of a type bridge crane, the relevant model is established by Solidworks three-dimensional software. The finite element analysis is carried out on the model under the abacus environment, and the stress and strain of the main girder model and the dangerous parts in the working process are obtained.

2. bridge crane structure

The bridge crane mainly consists of the crane cart traveling mechanism, crane trolley traveling mechanism, the lifting mechanism, the end beam of the crane cart, the running track of the trolley, the assembly of the bridge, the electrical equipment, and the railings and so on. The structure of the whole machine is shown in figure. 1.

IOP Conf. Series: Journal of Physics: Conf. Series **1087** (2018) 052032 doi:10.1088/1742-6596/1087/5/052032

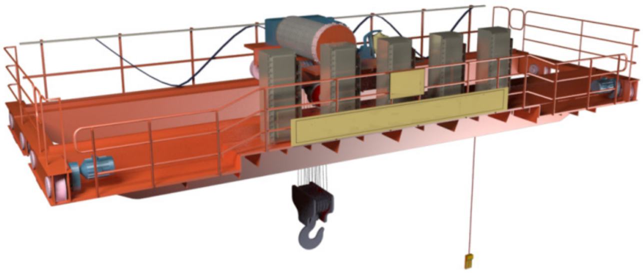

Figure. 1 bridge crane structure diagram

3. Stress analysis of the main girder

According to the actual analysis, it can be seen that when the crane is in the working state of hoisting, the crane trolley wheel is rolling along the track of the crane cart. The force of the main girder is different due to the position of the crane trolley. In this paper, two special working conditions of the crane trolley wheel in the middle and the right limit position were chosen.

The main beam of the crane cart is mainly subjected to three kinds of load: 1) the crane trolley wheel pressure to the guide rail of the main beam, 2) the shear force of the crane cart to the main girder side, 3) the gravity of the whole of main girder.

Lifting capacity of 1.25p (p=25t), and the effect after loading is shown in figure.2. Crane trolley wheel pressure load on the upper rails of the main beam is shown in Table 1. The weight of crane trolley is 1000kg. This paper decomposes the shear force on the side of the cart to the 28 sides of the side of the cart. So the loading area is 0.4032m^2, and the pressure load is 0.0243MPa. The gravity acceleration of 9.8m/s^2 is applied to the main beam of the crane cart.

Table 1 wheel pressure

Loading	wheel pressure /t	
	R_B	R_C
1.25p	21.06	0.97

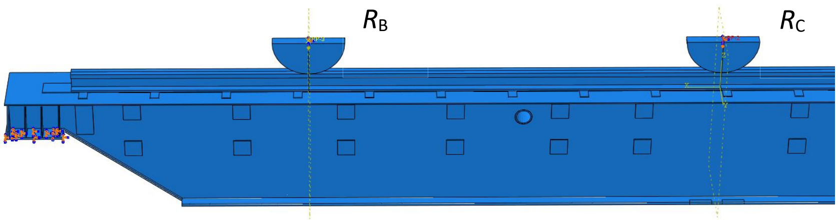

Figure. 2 the effect after loading

4. Finite element analysis of the main beam of the crane cart

4.1. The establishment of finite element simulation model

Based on the working condition analysis of the crane, the finite element simulation model of the small wheel in the middle of the main girder and the limit of the main beam on the right are set up. In order to accurately simulate the pressure of the guide rail of the wheel, the contact between the small wheel and the guide rail is defined, as shown in figure.3. The boundary conditions for strengthening the supporting surface of two main girders are discussed. The definition of boundary conditions under two working conditions is shown in figure.4. In order to facilitate the modelling, the structure of the main girder is simplified before modelling. The simplification follows the following principles:1) simplification has little effect on the calculation accuracy and analysis results of the model, 2) the simplified model is consistent with the size of the object.

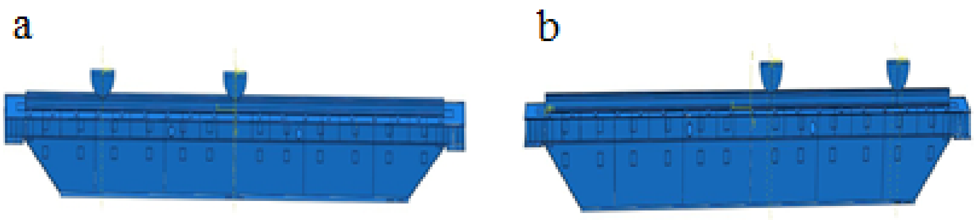

Figure. 3 the finite element simulation model of the crane trolley wheel in the middle position (a) and the right (b) position of main beam.

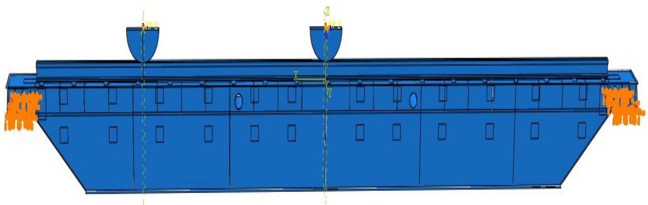

Figure. 4 The definition of boundary condition

4.2. Material definition and network partition

Material characteristics: modulus of elasticity is 210000MPa, poisson's ratio is 0.3,density is7.85×10^{-9}t/mm^3. The model is divided into two parts: the first part is the guide rail and the small wheel part; the second part is the lower part of the main girder (thin wall part). The first part is divided by three dimensional entity unit C3D8R, and the second part is divided by the shell unit S4R, the unit size is 5-30mm, the final total number of units is 170344, the total number of nodes is 193085. For thin-walled structures, shell element can reduce the computational cost of the model. The simulation model after dividing the grid is shown in figure.5.

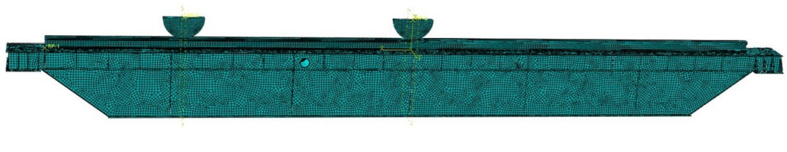

Figure. 5 Simulation model of partition grid

4.3. Finite element static analysis of the main beam

The permissible stress of elastoplastic materials is$[\sigma]=\sigma_s/n_s$, σs is the yield limit of the material. n_s is a safety factor[5]. The main beam material of the crane is Q345 steel. Under the working condition of a Class A, n_s takes 1.22, the allowable stress is 233.1 MPa; in the C working condition, n_s takes 1.22, then the allowable stress is 282.8MPa. For the bridge crane, the requirement of the stiffness of the main girder is to limit the deflection of the main beam, and the static deflection is f≤（1/700- 1/1000）L, and L is the span of the crane. It has been known that the crane L=9500m has a static deflection f≤1/1000*L, that is f≤9.5mm.

4.3.1. Finite element analysis of the main beam when the crane trolley wheel at the middle position

When the 1.25p load is hoisted, the stress distribution map and displacement distribution map of the main girder of the truck are shown in figure 6, 7, and 8 respectively. According to figure.6 and figure.7, under this condition, the maximum stress of the main girder of a big car appears to be 210MPa in the middle partition board, less than 282.8MPa, so the main beam of the big car meets the

IOP Conf. Series: Journal of Physics: Conf. Series **1087** (2018) 052032 doi:10.1088/1742-6596/1087/5/052032

strength design requirements under this condition. According to figure.8, under the working condition, the maximum deformation of the main girder of the truck appears in the middle position of the beam, which is 4.422mm, which can meet the requirements of stiffness design.

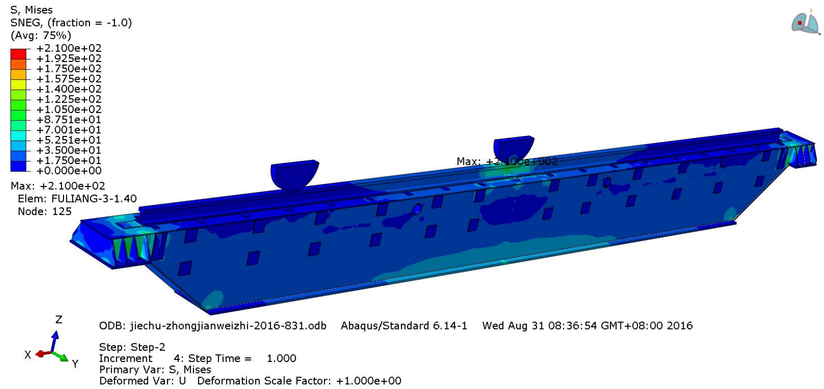

Figure. 6 the overall stress distribution of the main beam.

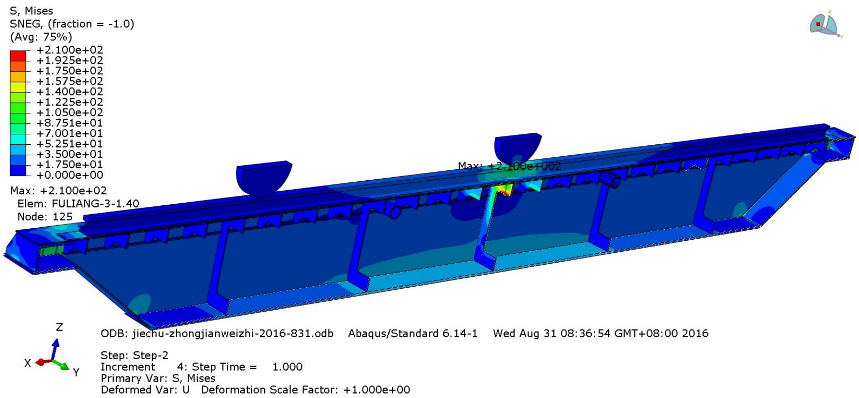

Figure. 7 the position of the maximum stress of the main beam.

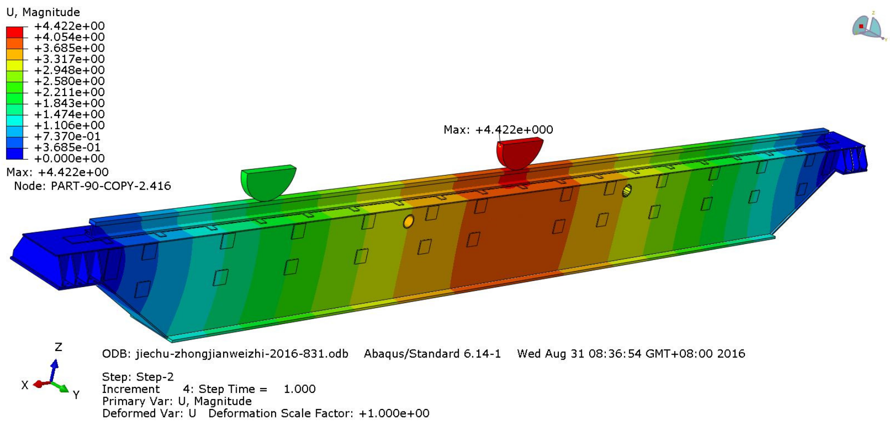

Figure. 8 Displacement cloud chart of main deam

4.3.2. Finite element analysis of the main beam when the crane trolley wheel at the extreme position on the right

When the 1.25p load is hoisted, the stress distribution map and displacement distribution map of the main girder of the truck are shown in figure.9, 10, and 11 respectively. From figure.9 and figure.10, it is known that the maximum stress of the main beam of a big car appears at the right small baffle at this condition, so the main beam of the big car can meet the requirements of the strength design under the condition of the working condition. According to figure.11, under the working condition, the maximum deformation of the main girder of the truck appears in the middle position of the beam, which is 2.070mm, which can meet the requirements of stiffness design.

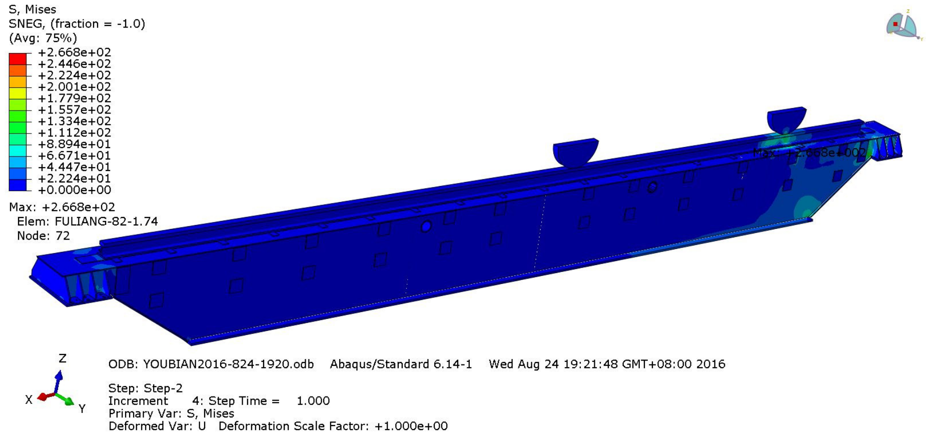

Figure. 9 the overall stress distribution of the main beam

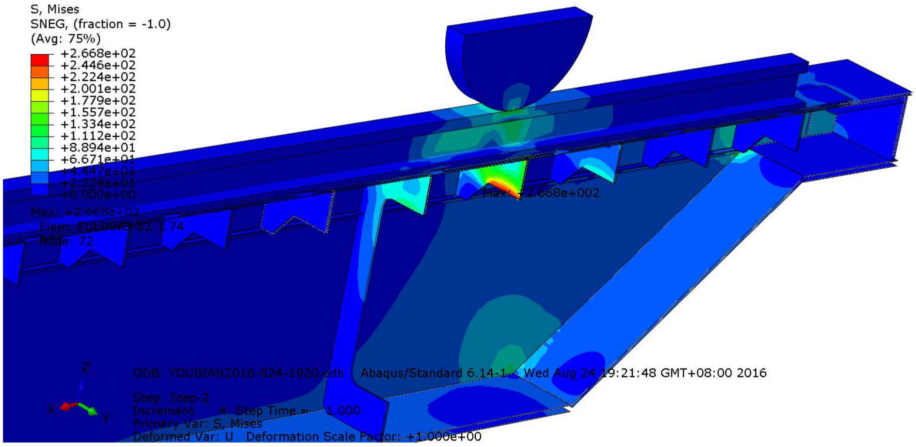

Figure. 10 the position of the maximum stress of the main beam.

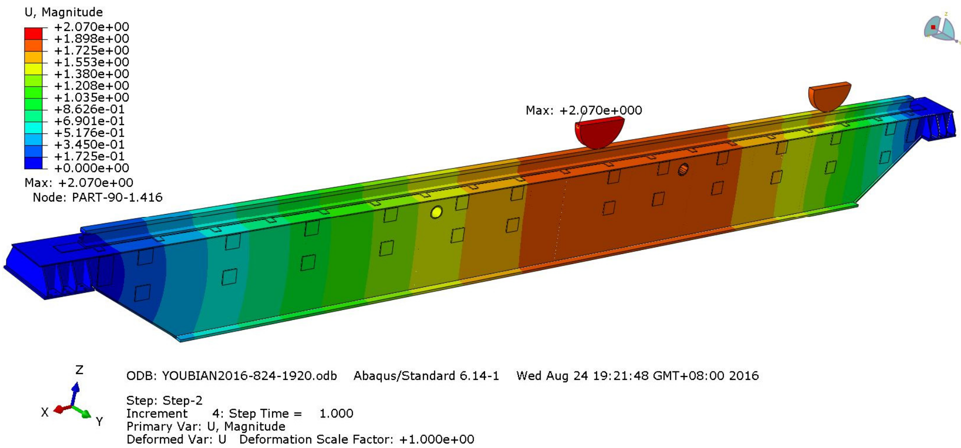

Figure. 11 Displacement cloud chart of main deam

5. Conclusion

The maximum stress 266.8MP of the most dangerous working condition of the crane is less than the allowable stress value of 282.8MP, which satisfies the strength requirement; the maximum deflection is 4.422mm, less than the limit of deflection 9.5mm, which satisfies the stiffness requirement. Through the static analysis of the main girder structure of the crane truck introduced in this paper, the Solidworks three-dimensional software was used to establish the related model and introduce the model into the abacus for the finite element analysis. It had important meaning to shorten the development cycle of the product, improve the design efficiency and optimize the structure of the product, and provide the design of the crane bridge structure.

Reference

[1] Ling Z, Wang M and Xia J 2018 *IOP Conference Series: Earth and Environmental Science* vol 3 p 406
[2] Cheng X F, Wu F Q and Tang G 2017 *Journal of Physics Conference Series* vol 1 p 205
[3] Li Y K, Wang L and Song Y 2017 *International Conference on Mechanical and Intelligent Manufacturing Technologies*. IEEE vol 1 p 69
[4] Liu J Y and Liu J 2012 *Mechanical Engineer* p 69
[5] Jun A N and Hongjie Y U 2012 *Modern Manufacturing Technology & Equipment* p106
[6] He Y B, Zhang Y and Yang B K 2013 *Applied Mechanics & Materials* p 70