Study on bridge structure of bridge crane

Li Hai-ying[1, a *], Wu Hua-liang[2, b]

[1] School of Applied Technology, University of Science and Technology Liaoning, Anshan, 114051, China

[2]Cold Rolling Plant, Angang Steel Co., Ltd, Anshan, 114021，China

[a]49911944@qq.com, [b]kexiao1221@sina.com

Keywords: bridge crane; bridge structure; finite element analysis

Abstract. In this paper, the large finite element analysis software - ANSYS is used to create the model of bridge-crane frame structure. Using ANSYS, we established bridge-crane frame structure of finite element model. Creates the stress and stiffness analysis under different conditions using the ANSYS models created before, obtains the maximum deformation and stress of main beam, and provides the theoretical basis for the design of bridge structure of the bridge crane.

Introduction

The bridge-crane can complete hoisting and lifting of the work in the condition of processing and manufacturing. Due to the special working environment: aerial work, wide operating range, the bridge-crane is an important tool in the production of enterprises, so the bridge-crane is facing a great development space. What the most important is that the bridge-crane is used in 90% of the enterprise, so the research and improvement of bridge-crane can promote its continuous optimization, and have a huge economic impetus to the enterprise [1].

The large-scale finite element analysis software – ANSYS is used to build a three-dimensional model of the structure of a bridge-crane. The static analysis model is simulated according to a specified type of bridge-crane frame structure, so the magnitude and distribution parameters can be obtained.

Structure analysis and finite element model of the main girder

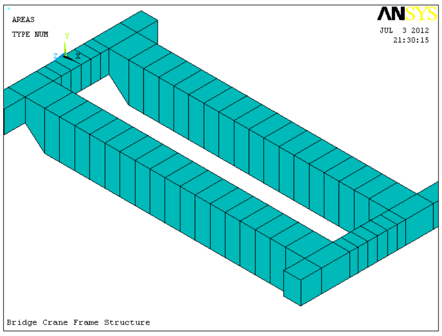
Unit selection of box girder structure. The box girder is welded using four plates which includes upper and lower coverred plates and the left and right side plates. The upper coverred plate of the box girder obeys the regulations, so it is appropriate to divide the finite element mesh with the shell element. And the lower coverred plates and the left and right sides of the beam are more complex. The shell element ANSYS of Shell63 has the following properties: Shell63 has the ability to bend and film. Both the loads and the normal loads are permitted.

The element has six degrees of freedom at each node: the freedom of movement along the direction of the three axes, the freedom of rotation around the three axes. In addition, the element has the ability to deal with the singular stiffness and large deformation [2,3].

The finite element model of the main girder.Using the bottom-up models established methods to build the model from the bottom up, first create the key point, and then create the relevant line, surface, and body and other elements.

This paper adopts GUI method to operate. Firstly, we define material properties, set the real constant and define analysis type, set the modulus of elasticity and Poisson constant, using preprocessor command, depending on the size of the drawings, in the active coordinate system (active CS) generated key point, in the active coordinate system has been set up as the world coordinate system (World CS).

Then, the key points are used to product the line and the flat. Because this model is welded with thin-walled material, it is enough to replace the entity with the surface. When the meshes are being built, we can use the setting dialog (user interface) to set the thickness,finally the generated plane are used to combinat the bridge crane girder model and mapped into double beam bridge [4].



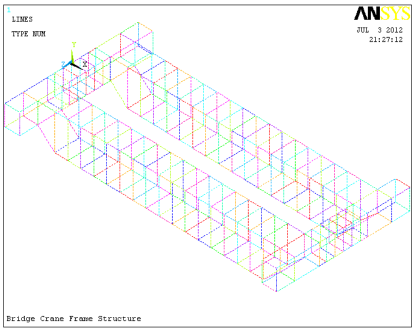


Figure 1 Double beam of bridge model Figure 2 the wireframe of bridge model

Through the steps above, a dual beam bridge model and tripod wireframe, respectively, as shown in Figure 1 and figure 2.

Division grid

Set analysis module. After setting the selected unit type, we use Shell63. The typical constants contain the thickness of the shell element, the cross-sectional area of the beam element, and the form of the real number of the market array. At present, the real constant of shell element (Shell63) is the thickness of the upper and lower cover, web and diaphragm.

Select the real constant to the shell element (Shell63), because the model has different thickness of the web and the plate,it is required to set the thickness of the 3 types: 0.08m, 0.01M, 0.012m.

Define the material properties. Material properties are the essential properties of the geometric model, such as young's modulus, density, etc. This paper only relates to a material Q235 steel, whose material properties are respectively: Young's modulus: $2\times10^{11}Pa$, density: $7800kg/m^3$, Poisson's ratio: 0.3, shear modulus: $7.6923\times10^{10}Pa$.

Mesh of geometric model. Mesh size control: because of the large size of the model, we need to separate respectively. But because most of the construction planes share the same property, so we use the size to control the geometric model of the middle section of the line of the grid division. Unit type: Shell63, unit size is 200mm, as shown in figure 3:

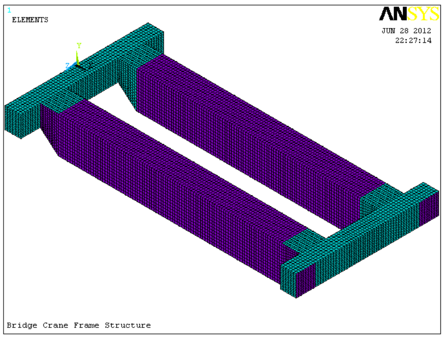


Figure 3 mesh bridge pattern

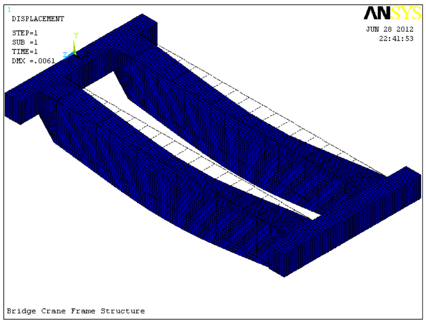
The built model has 21984 units; the number of nodes is 20844.

Static analysis of main girder of bridge crane

calculation and results analysis of the result of case 1. The working condition is: 35t rated load, wheel pressure is 441KN, the center of the main girder is cross section, and the main beam body weight.

(1)Loading.In the created model, select the cross section of the main girder, its distance is 2.27meters, and the two key points along the track line are applied to load -441KN in the FY direction.

(2)the static deformation of the main girder.In the graphics display dialog, select edge Def+nudef, which represents the boundary of the main girder and the deformation front, as shown in Figure 4.



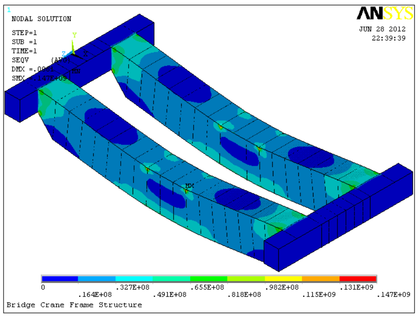


Figure 4 bridge deformation of case1 Figure 5 equivalent stress of case 1

As shown in the figure, the maximum distance is in the position, which across the interface, is 0.0061 meter, and the bridge displacement values is 0.022 meter, which within the allowable range.

The stress in the cross section of the main girder is shown in Figure 5.

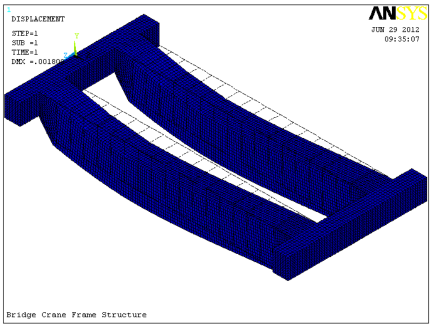
As shown in figure, the main girder is located near the upper cover plate, the maximum equivalent stress is 147MPa,which multiplied by the safety factor is 1.4, the stress value is less than the material allowable tensile stress $\left[\sigma\right]=400MPa$, it is also less than the material allowable stress $\left[\sigma\right]=240MPa$ [5].

According to the analysis above, the result shows that the bridge not only under the rated load stiffness and strength meet the requirements, and there is a sufficient reserve of strength.

calculation and results analysis of case 2. The working condition is: the 7T gravity load, the wheel pressure is 179KN, the center of the four wheels is in the end of the main girder and the main girder is the body weight.

(1) Loading.In the above model, the key points are selected in the position 4 meters far from the end of the main beam, another key points are selected from 2.27meters far from the prior points, and the -179KN is applied to the solution in the FY direction.

(2)Static stiffness of the main beam.In the graphics dialog, select Def+nudef edge, which represents the main beam after deformation and the boundary before deformation, as shown in Figure 6.



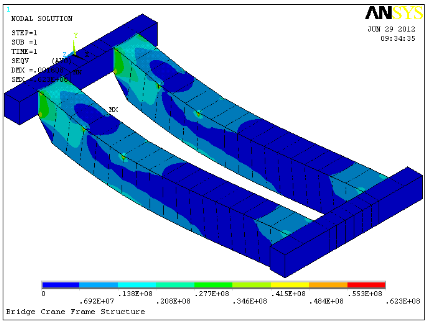


Figure 6 bridge deformation of case 2 Figure 7 equivalent stress of case 2

As shown in the figure, the maximum displacement, which at the middle of the crossed interface, is 0.0018meter, the calculation results and the test results are matched, and it satisfied the require of actual product deformation.

(3)The dangerous section of the main girder is shown in figure 7.

As shown in the figure, the maximum equivalent stress is 62.3MPa, which multiplied by the safety factor, and the result is less than the allowable stress of materials.

Summary

This paper based on the analysis of bridge-crane frame structure and specifications mechanical characteristics, the static mechanical characteristics of main beam is obtained fast and accurate by using finite element analysis software ANSYS. According to analysis and calculation, the maximum deformation and maximum stress value of the main girder are obtained, which provides the basis for the structural optimization of the late stage.

References

[1]. Xu Lei.The characteristics and development trend of modern cranes, lifting transportation machinery, 1997,10:3-7

[2] Liu Yunfeng. Research on the new structure of box girder of bridge crane [D], Kunming: Kunming University of Science and Technology, 2001

[3] Ling Guilong, Shen Zaiyang.ANSYS structure and material application manual [M]. (Tsinghua University press, 2013.04

[4] Gong Shuguang.ANSYS application examples [M], Beijing: Machinery Industry Press, 2003,1 ~ 183

[5] "crane design manual" compilation group. [M], Beijing: Machinery Industry Press, 15, 48 ~ 183, 204 ~ 494, 1980 ~ 510