ISSN: 2414-1895 DOI: 10.6919/ICJE.201909_5(10).0003

Finite Element Analysis of Gantry Crane under Moving Load Based on Ansys

Yifan Gao [a], Fangfang Lai

Logistics Engineering College, Shanghai Maritime University, Shanghai, 201306, China

[a]gaoyifan522@163.com

Abstract

The trolley travels on the gantry crane beam, and the beam is subjected to the moving load, which will cause the vibration of the front girders, which has a great influence on the strength and fatigue of the girders, and also affects the stability of the movement of the trolley on the girders. The static analysis of the girders can not be obtained by the commonly used static analysis. The Ansys finite element software transient analysis is used to simulate the influence of the trolley on the girders on the girders, which will provide reference for future research.

Keywords

Gantry crane, girders, Ansys, simulation.

1. Introduction

Intelligent manufacturing is the direction of modern manufacturing development. In order to improve efficiency and reduce production costs, people pay more and more attention to the intelligent development of manufacturing equipment. Bridge cranes play a vital role in the intelligent development of the manufacturing industry. They are large-scale equipment for material handling. The objects to be transported are of high quality and have high handling frequency. Practice has shown that with the rail-type container gantry crane, the same investment can increase at least one-third of the storage area, and the capacity can be increased by more than half. The main beam is the main component and load-bearing component of the bridge crane. The design of the main beam directly affects the manufacturing cost, working performance and service life of the whole machine.

The main beam of the bridge crane is the main working area for the dragging of the trolley and the movement of the container. It is generally believed that the forced vibration of the structure is the response under the action of the excitation force of a fixed period, but the structure will also vibrate under the action of the moving load. In actual work, when the car moves on the main beam, it will be found that the main beam does have vertical vibration. This vibration did not attract the attention of engineers, nor did the specification mention the calculation and prevention standards for such vibrations. Therefore, studying the dynamic response of the main beam under the action of moving trolleys and mastering the dynamic characteristics of the main beam are of great significance for further vibration control.

The traditional design method of the main beam is that the model is simplified and manually calculated, which has a certain deviation from the actual working conditions and is complicated to calculate. In the mechanical industry analysis and design, CAD/CAE technology is increasingly used for auxiliary design and auxiliary analysis. Ansys software effectively combines finite element numerical analysis technology with CAD/CAE to enable designers to visually analyze problems in structural design.

ISSN: 2414-1895 DOI: 10.6919/ICJE.201909_5(10).0003

In this paper, the Ansys software is used to analyze the transient response of moving loads, and the dynamic response of the main beam is studied to provide reference for further research.

2. Kinetic equation modeling

In the analysis of gantry crane, the gantry crane is usually simplified as a simply supported beam model. According to Bernoulli - Euler beam theory and relevant mechanical knowledge, the motion equation of the simply supported beam under general load $f(x,t)$ can be obtained:

$$EI\frac{\partial^4 w(x,t)}{\partial x^4}+\rho\frac{\partial^2 w(x,t)}{\partial t^2}=f(x,t) \tag{1}$$

Where, E is the elastic modulus, I is the moment of inertia of the section, ρ is the unit length mass of the beam element, w is the deflection of the beam, and x is the distance from the micro section to the left support of the simply supported beam, t is the time.

Considering the inertial acceleration of the car and the impact of the coupling vibration between the car and the beam, the analysis of the car and the suspension bell as the moving mass is as follows:

$$f(x,t)=M(g-\frac{\partial^2 w(x,t)}{\partial t^2})\delta(x-s) \tag{2}$$

Where, $\delta(x-s)$ is the Dirac delta function.

According to the modal superposition method, the modal equation applied to the j mode of simply supported beam under the action of moving mass can be simplified to:

$$\begin{aligned} M[g-\frac{\partial^2 w(x,t)}{\partial t^2}]\Phi_j(s)=\sum_{j=1}^{N}[\Psi_{ij}\frac{\partial^2 q_j(t)}{\partial t^2}] \\ +\sum_{j=1}^{N}[\frac{(j\pi)^4}{L^4}\frac{EI}{\rho}\Psi_{ij}q_j(t)] \end{aligned} \tag{3}$$

Where, $\Phi_j(s)$ is the j-order mode shape of simply supported beam, N is the highest order of mode shape; L is the beam length; $q_j(t)$ simply supported beam first modal coordinates.

Under the action of the moving load, the influence of the difference between the weight and its inertia force on the vibration of the main beam is considered in addition to the gravity of the weight itself.

3. Ansys modeling and simulation

3.1 Solid modeling

First of all, according to the design requirements of the crane and reasonable parameters in Ansys to build a three-dimensional solid model of the crane.Table 1 is the parameters of 3d solid model of crane and track.

Table 1 Geometric parameters of gantry cranes

Geometric parameters	Parameter meaning	Geometric parameters	Parameter meaning
B=40	gantry crane frame width	L_H=40	beam width
H=15	gantry crane frame height	B_H=2	beam width
L=120	gantry crane rigid frame span	H_H=2	beam height

ISSN: 2414-1895 DOI: 10.6919/ICJE.201909_5(10).0003

L_S=20	arm length	L_KZ=80	Mid span length
L_S_1=10	arm variable section length	H_KZ=4	Mid span height
H_S_1=4	arm variable section height	H_KZ=4	Mid span height
B_S_1=2	variable section width of extender	L_ZC=15	support length
HS_ZC=4	support height	B_ZC=2	support width
L_KJ=40	frame leg length	H_KJ=4	frame leg height
B_KJ=2	frame foot width		
Material	Parameter meaning	Material	Parameter meaning
E_L=2.1e10	elastic modulus	U_L=0.3	Poisson's ratio
DENS_L=7800	density (KG/mm)	T	shell element thickness

In solid modeling, the bottom-up modeling method is used to establish the key points first, then the points are generated, and finally the symmetry and mirroring functions are used to build the whole model. Since the double-foot structure is analyzed, the boundary conditions are imposed on the model. And the grid is divided, as shown in Figure 1.

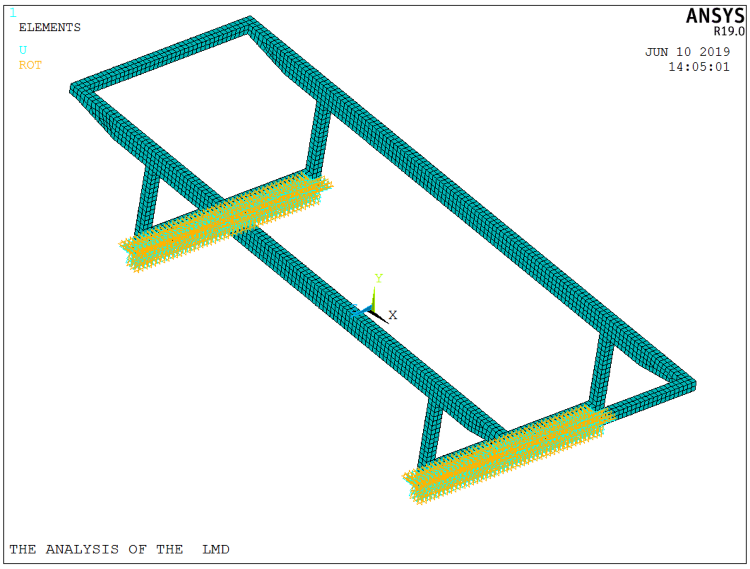

Fig. 1 Model meshing

3.2 Loading solution analysis

The load and application instructions of each analysis are summarized as shown in table 2.

Table 2 Load type

Analysis of the type	Load instructions	The main purpose
Static analysis	Gravitational stress field	Simulate the deformation and stress under static load

ISSN: 2414-1895 DOI: 10.6919/ICJE.201909_5(10).0003

Transient analysis	Both dead weight and moving load are considered	Determine the deformation and internal forces under moving loads
The modal analysis	The free modal	Determine the vibration information of the structure

3.2.1Static analysis

After the gravity acceleration is applied to the model, the deformation cloud diagram and stress cloud diagram obtained are shown in Figure 2 and Figure 3.

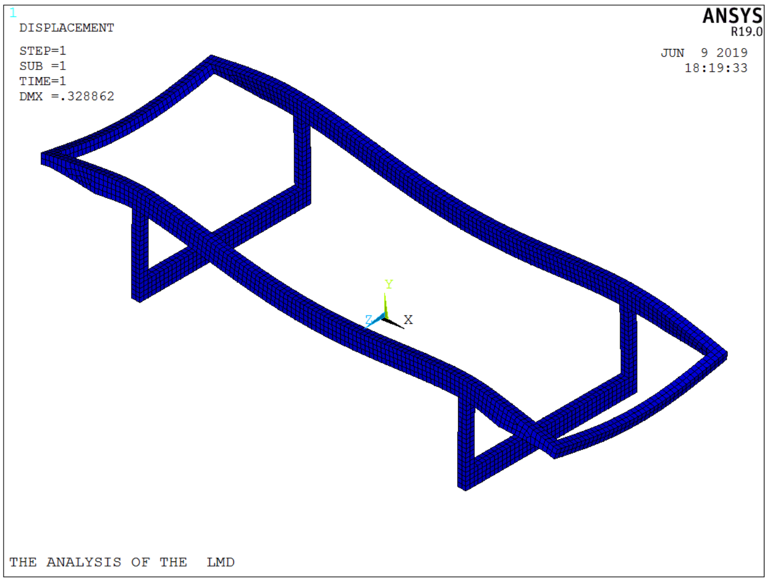

Fig. 2 Deformation nephogram

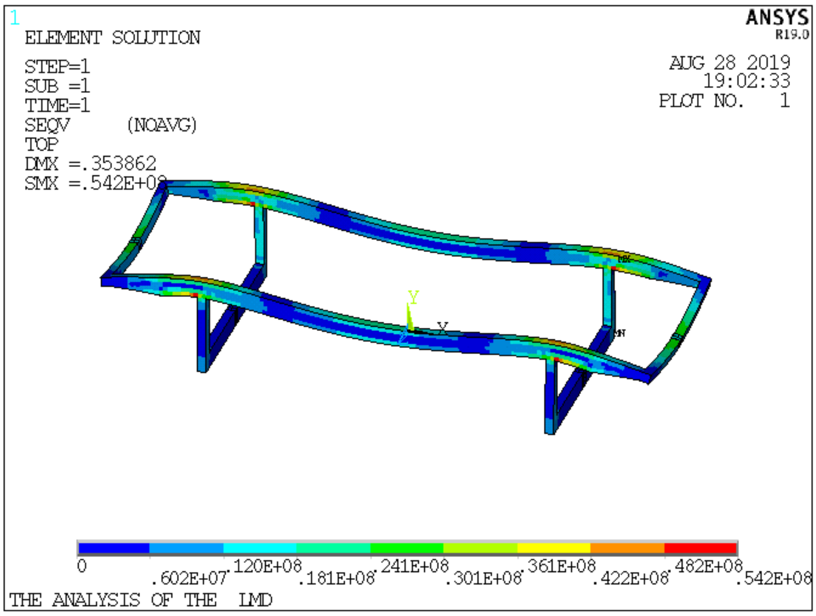

Fig. 3 Equivalent stress nephogram

The displacement cloud map can be used to observe the position where deformation can easily occur under the action of gravity and the displacement changes in all directions.

International Core Journal of Engineering Volume 5 Issue 10, 2019
ISSN: 2414-1895 DOI: 10.6919/ICJE.201909_5(10).0003

3.2.2 The modal analysis

Modal analysis can not only provide the vibration information of the structure (such as the natural frequency and various formations), but also serve as the starting point for dynamic analysis.Ansys simulation modal analysis is shown from figure 4 and figure 5.

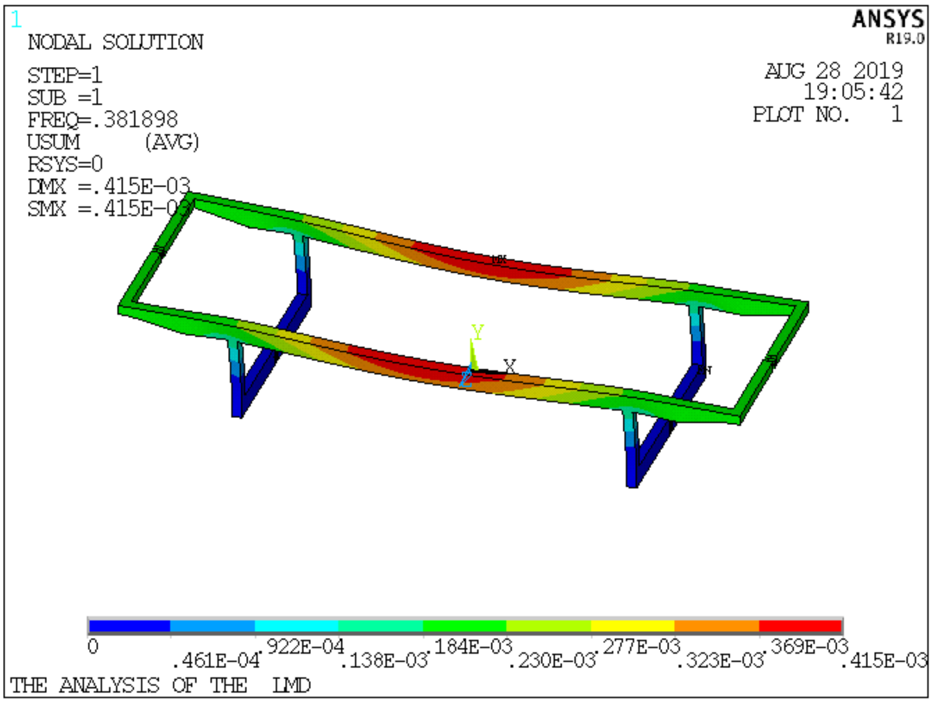

Fig. 4 First mode

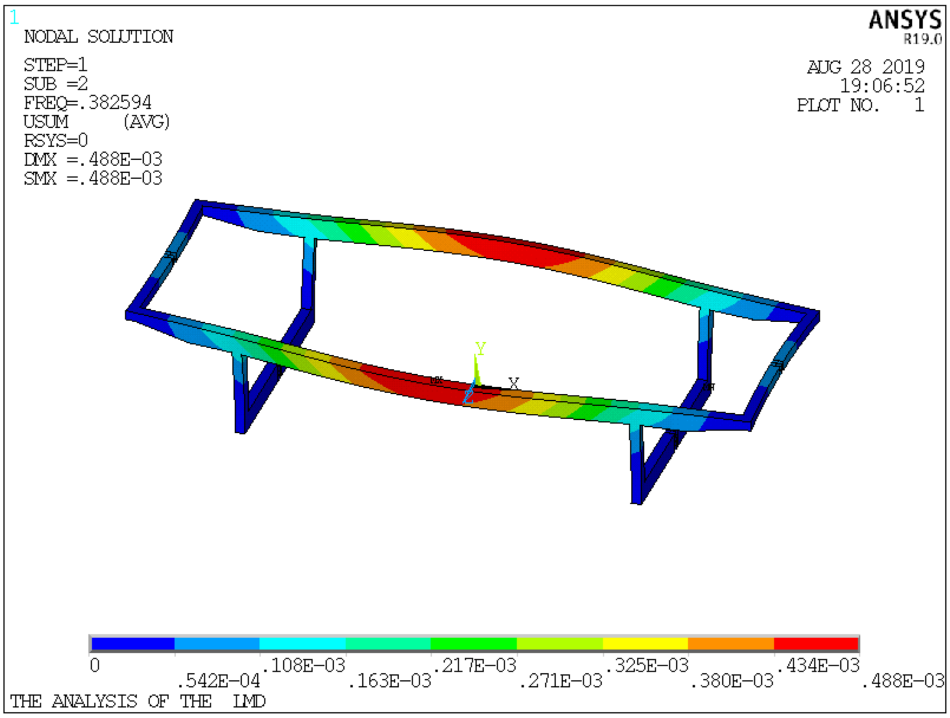

Fig. 5 Second mode

Meanwhile, the first 20 natural frequencies can be obtained as shown in table 3.

Table 3 The first twenty natural frequencies

Frequency orde	Frequency value	Frequency orde	Frequency value
1	0.3819	11	1.0888

ISSN: 2414-1895 DOI: 10.6919/ICJE.201909_5(10).0003

2	0.38259	12	1.1071
3	0.45188	13	1.3613
4	0.45381	14	1.4156
5	0.5821	15	1.5893
6	0.59817	16	1.8116
7	0.70756	17	1.934
8	0.7624	18	2.0575
9	0.84717	19	2.2656
10	0.95367	20	2.5078

3.2.3 Both dead weight and moving load are considered.

At this time, the influence of dead weight and moving load is taken into consideration, and the dynamic analysis is carried out on the load caused by adding the moving trolley at the same time by taking static solution as the initial condition (taking static balance as the reference).Figure 8 to figure 9 show the displacement history diagram of the trolley at the mid-span node and the cantilever endpoint.

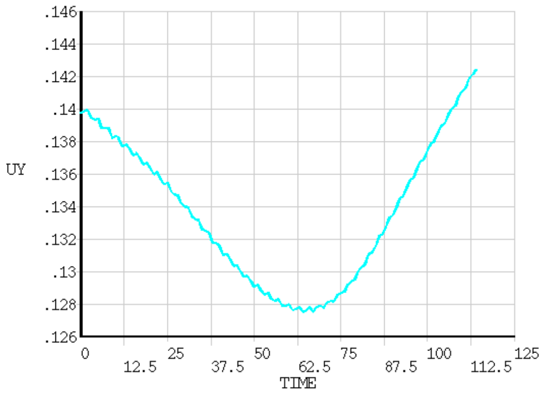

Fig. 6 The y-direction displacement history of the midspan node

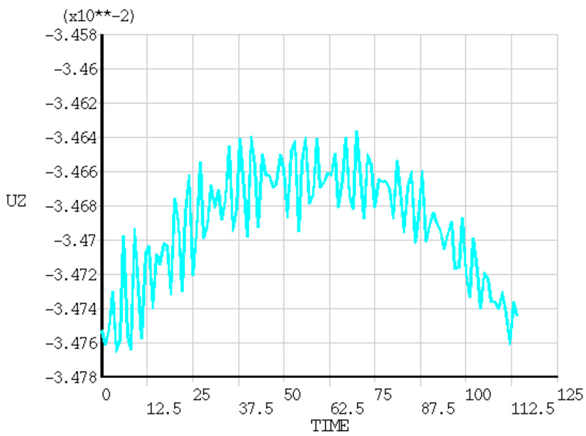

Fig. 7 The displacement process of Z direction of midspan node

It can be seen from the calculation that the deflection value (i.e., y-direction displacement) under the action of moving load varies with the position of moving load. It can be seen from the time-history curve with deflection that the deflection is the largest when the load is at the mid-span position. Meanwhile, although the x-direction and z-direction displacement of courseware is relatively small,

ISSN: 2414-1895 DOI: 10.6919/ICJE.201909_5(10).0003

it also varies with different load positions. Z-direction displacement is also known as lateral deformation, which needs to be strictly controlled in practice.

4. Conclusion

Through the analysis of the simulation study can be seen: gantry crane when subjected to dead weight of each displacement direction changes, provide a reference for crane, and being dead weight and operation in the process of moving load, maximum deformation position for the location in the cross, and the courseware although the X to close the Z displacement is small, but also with the different load location, Z displacement, that is, the lateral deformation, in practice, also must be strictly controlled.

Ansys software has a powerful simulation calculation function, and modeling ability, more easily used in daily design analysis, it can not only draw the force of the change diagram, but also can calculate the specific value. Ansys is based on the finite element method, can calculate static, dynamic, thermal analysis, fluid analysis, magnetic field and a series of problems, the object is divided into a number of grid cells for the corresponding analysis, each individual vibration and deformation (structural mechanics).In addition, while Ansys has its own modeling ability, it can also combine UG,ADAMS,MATLAB and other software. Using these software together will greatly enhance its function.

References

[1] Wang H.J, Chen H.J. Advanced examples of Ansys engineering analysis. China Water Conservancy and Hydropower Publishing, Beijing,2006.

[2] Meng D.L, Gong Fa, Da Wei,Yang S.L, Finite Element Analysis on Crane Girder with Variable Cross Sections Based on Ansys. Sensors & Transducers,2013,21(05).

[3] Kyuba H, Dong P. Equilibrium-equivalent structural stress approach to fatigue analysis of a rectangular hollow section joint. International Journal of Fatigue，2005, 27 (1) : 85 － 94.

[4] Guo B. Z, Wang J .M，YANG K Y. Dynamic Stabilization of an Euler -Bernoulli Beam under Boundary Control and Non-collocated Observation . Systems & Control Letters, 2008, 57(9)：740-749.

[5] BIERER T, BODE C. A Semi -analytical Model in Time Domain for Moving Loads. Soil Dynamics and Earthquake Engineering, 2007, 27(12):1073-1081.