Sensors & Transducers, Vol. 25, Special Issue, December 2013, pp. 23-29

www.sensorsportal.com

Sensors & Transducers

http://www.sensorsportal.com

Study on Structural Optimization Design for Band Sawing Machine

Miaofen Zhu, * Guojin Chen, Youping Gong, Tingting Liu, Huipeng Chen, Jing Ni
Hangzhou Dianzi University, Hangzhou, 310018, China
* Tel.: 008613805786793
E-mail: chenguojin@163.com

Received: 16 September 2013 /Accepted: 15 October 2013 /Published: 23 December 2013

Abstract: This paper takes the large band sawing machine GB42500H as the research object, and establishes the dynamics models of its various parts. Respectively, the relative research is done from the theoretical modal analysis and the dynamic performance testing. According to the analysis results, the structure or geometry improvement scheme of the parts is proposed. The paper also studies preliminarily the dynamic characteristics of the large band sawing machine GB42500H. During the analysis, the models of the parts and the whole are established using the finite element method to optimize the structures of the parts and the whole. That has undoubtedly a very important significance for improving the dynamic performance for the new machine tool's structure, increasing the processing accuracy, and shortening the development cycle and costs.

Keywords: Structural optimization design, Band sawing machine, Dynamics model, Finite element method, Static and dynamic simulation.

1. Introduction

That the dynamic characteristics of the band sawing machine are good or bad will directly affect the machining performance. The entire band sawing structure is a very complex structured system. The dynamic characteristics of the whole machine are the integrated results for the dynamic characteristics of the various key components and the dynamic parameters of the bonding surfaces between the various components.

This paper takes the large band sawing machine GB42500H as the research object, and establishes the dynamics models of its various parts. Respectively, the relative research is done from the theoretical modal analysis and the dynamic performance testing. According to the analysis results, the structure or geometry improvement scheme of the parts is proposed. The paper also studies preliminarily the dynamic characteristics of the large band sawing machine GB42500H. During the analysis, the models of the parts and the whole are established using the finite element method to optimize the structures of the parts and the whole. That has undoubtedly a very important significance for improving the dynamic performance for the new machine tool's structure, increasing the processing accuracy, and shortening the development cycle and costs.

2. Structure and Composition of Band Sawing Machine

The large and efficient band sawing machine GB42500H integrates the mechanical, hydraulic and electrical to implement the integrated control through the much structural innovation on the basis of the ordinary band sawing machine [1-2]. The unique workpiece's rotation sawing has high efficiency and many specifications (cutting the pipe diameter 5000mm). The machined parts have the high size, position, shape accuracy and the low surface roughness. The machine can greatly improve the processing accuracy and efficiency, and its energy-saving effect is significant. So it can be widely used in military, aerospace, wind power generation, machinery, shipbuilding and other industries, and is particularly suitable for sawing a variety of large flanges and oil pipelines [3]. Its structure is shown in Fig. 1.

3. Dynamic Structure Design Method and Processes

With the internal structure and the external shape of the large machine's parts as a starting point, the finite element method is used to study the relationship between their geometry and the dynamic characteristics. We should quantify the factors that affect the dynamic characteristics, and heighten the dynamic characteristics of the factors which are composed of the internal structure and shape of the large machine's parts, in order to improve the dynamic characteristics of the large machine's parts. It can ensure from the micro and macro that the large parts of the designed machine have the better dynamic characteristics. This approach is targeted to avoid the topology optimization for the complex structures, and has strong operability [4-5]. The research method proposed in the paper on the dynamic characteristics of the large band sawing GB42500H based on the finite element is shown in Fig. 2.

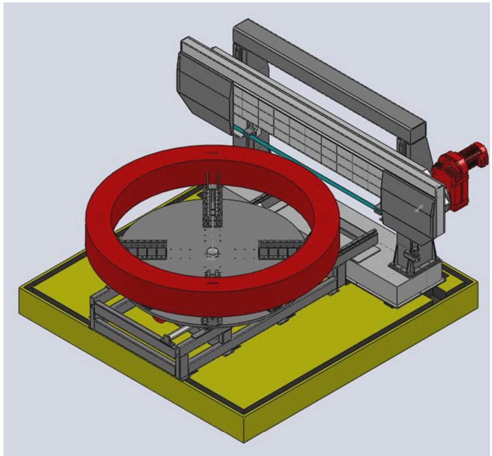

Fig. 1. The large and efficient band sawing machine GB42500H.

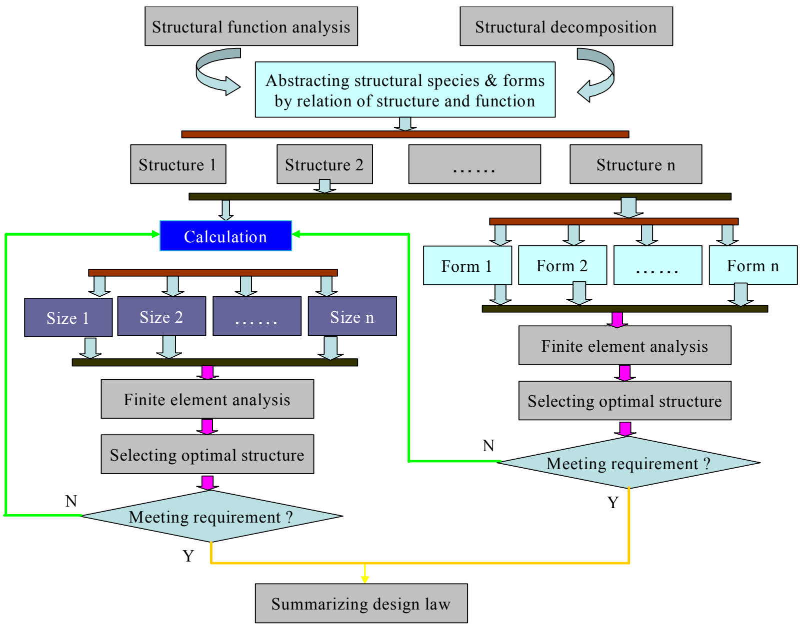

Fig. 2. The research method on the dynamic characteristics of the large band sawing machine.

First the big parts of the machine are decomposed according to their form and structure, and ultimately the basic cell structures which are little in form change and relatively independent in internal structure are gotten. For example, the pentahedron, hexahedron and column leg rib plates of the machine body. Optimizing these cell structures from the dynamic design angle can obtain the optimization parameters of the variable sizes. The module part of the machine tool has a basic configuration, such as the length, width and height of the body. The length and width are decided by the processing range, the height can be obtained by optimizing the natural frequency. The angle of inclination for constant strength column, the width and thickness of the column outrigger can obtained according to the similar law. The quality and natural frequency of the body can be determined by studying the quality and natural frequency distribution pattern of the machine tool. That is easy to control the structural parameters of the machine body. Then the structural dimensions of the various parts are calculated and the forms are classified. First the preliminary size is calculated to obtain the preliminary parameters. And these structural parameters are analyzed using the finite element to obtain the optimum size in a certain range. The forms are classified to get a variety of machine layouts. And the static stiffness and dynamic stiffness are structurally analyzed using the finite element for each layout. Finally the optimal layout is selected. If you modify the layout, the scantlings are re-calculated. And the component design is verified to ultimately get the design criteria of the machine.

4. Structural Design Calculation of Band Sawing Machine

In this paper, the analysis of the actual band sawing movement is achieved by combining the mechanical design, the dynamic analysis, the electro-hydraulic servo control and the multi-domain hybrid simulation. Its specific implementation steps are as follows.

1) Analyzing the actual band sawing process studies the affection of the cutting speed, the tension pressure, and the feed speed to the performances of the band saw blades and the saw disks.

2) Combining the relevant machinery design, the dynamic analysis, and the electro-hydraulic servo control and the multi-domain hybrid simulation studies the impact factors to the cutting speed and the saw blade's performance, namely establishes the actual sawing process model.

3) According to the design parameters, the sawing digital design model is established. Using the model can optimize the follow-up static stiffness, dynamic stiffness and the overall structure design.

5. Optimization Design based on Dynamic and Static Stiffness

The static rigidity of the sawing machine is mainly determined on the materials, shapes, sizes, and arrangements of ribs for the components and parts. It is an important indicator measuring the characteristics of the sawing machine. Increasing the sawing static stiffness value is very effective for improving the processing efficiency, machining accuracy and surface quality. Drawing the static stiffness value of the whole machine and its distribution by the finite element analysis is very useful and time-saving.

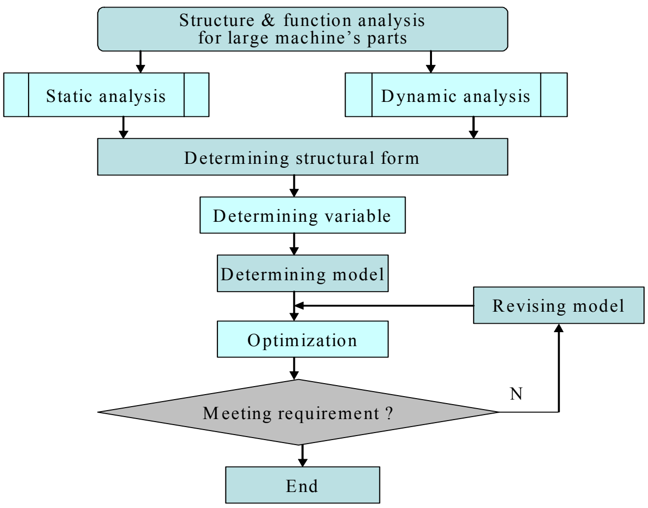

Fig. 3. Optimization design on dynamic and static stiffness.

The static characteristics of the structure can be obtained through the structural static analysis. But for the sawing machine, usually during the actual use, the cutting force and the other excitation force are acted on the structure in the form of the dynamic load. In order to ensure the processing accuracy and efficiency, the structural dynamic characteristics should be considered for the sawing machine on the basis of the certain static stiffness requirements, as shown in Fig. 3. That is, the whole machine has the high dynamic stiffness (in the resonance state, the ratio of the excitation force and the vibration amplitude) and the larger damping. That can make the forced vibration amplitude of the structure to be small under the certain amplitude of the periodic excitation force, and the natural frequency of the machine to not coincide with the excitation frequency.

6. CAD Models and Grid Models for Key Support Components

Taking into account the complexity of these models, the powerful CAD software of Solidworks is used. On the premise of reflecting the true mechanical properties for the main column structure of machine, the individual local areas of the column structure are simplified to facilitate the modeling. The top and bottom ends of the base have many holes of the small diameter. There are several bolt holes at the bottom of the base, the cooling plate and the end surface of the bed platform, etc. These holes have little effect on the stiffness of the column, but will affect the meshing. So they can be negligible in the modeling. The overall effect of the three-dimensional model for the machine is shown in Fig. 4. The machine's column supporting framework plays a main role for supporting the tool and machining forces. The motors saw boxes and other parts are placed on it. The overall dimensions are 2385 mm, 1020 mm and 720 mm. The material is HT200. The elastic modulus is E=6.61781e10, and the Poisson's ratio is u=0.27. The table support frame is connected with the table into a whole by the taper pins and the screws. The overall dimensions are 4870 mm, 1020 mm and 880 mm. The material is HT200. The elastic modulus is E=6.61781e10, and the Poisson's ratio is u=0.27. The assembly model for the machine's column supporting framework is shown in Fig. 5. The assembly model for the table support frame is shown in Fig. 6.

The base, the table support frame and the machine's column supporting frame are welded by a number of ribs. Some ribs are very thin and complex in the structure. So the tetrahedron with intermediate nodes and 10 nodes of Solid187 are used when meshing.

When meshing the model freely, the refinement level is 3. Note the number of nodes and units to increase the computing speed. Also check the finite element model to prevent coincident nodes, cracks and distortion units, etc. When the finite element mesh is completed, the overall grid model must also be checked to ensure the authenticity of the results. First, check the free unit edge. When one side of the unit is not in the other units, it is known as free unit edge. In the process of establishing the complex model, the various components are produced by the stretching and rotating operations. They sometimes are not connected together. That would lead to cracking the finite element model, and affecting the results. The serious condition will make calculation void. Second, check the repeating units and nodes. The element model for the machine's column supporting framework is shown in Fig. 7. The grid model for the table support frame is shown in Fig. 8. By calculation, the strain for the table support frame is shown in Fig. 9, and the equivalent stress for the machine's column supporting framework is shown in Fig. 10.

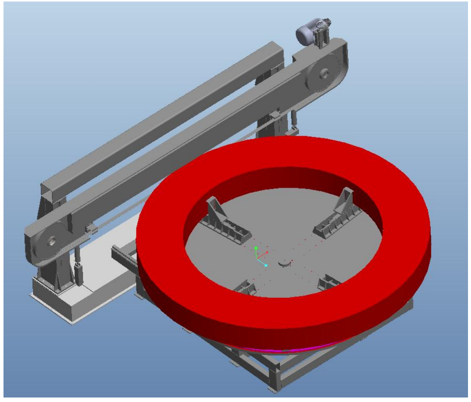

Fig. 4. The three-dimensional model for the machine.

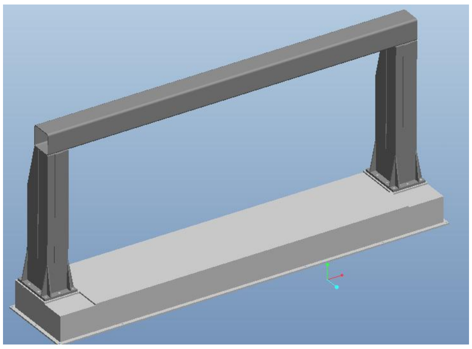

Fig. 5. The assembly model for the machine's column supporting framework.

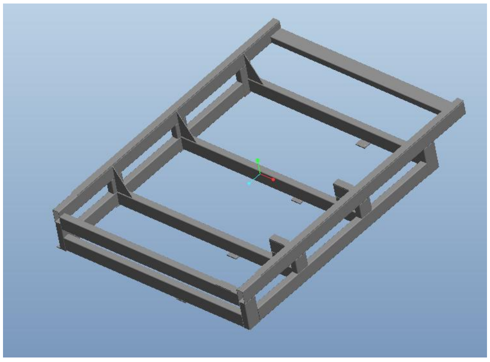

Fig. 6. The assembly model for the table support frame.

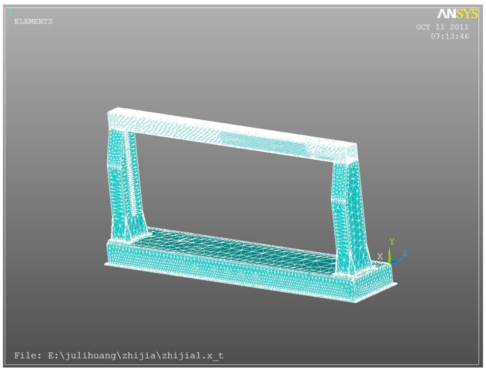

Fig. 7. The element model for the machine's column supporting framework.

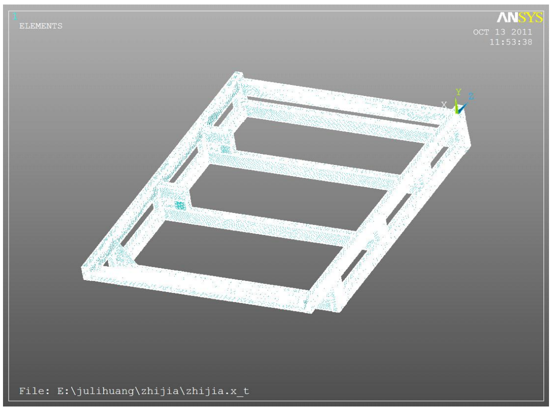

Fig. 8. The grid model for the table support frame.

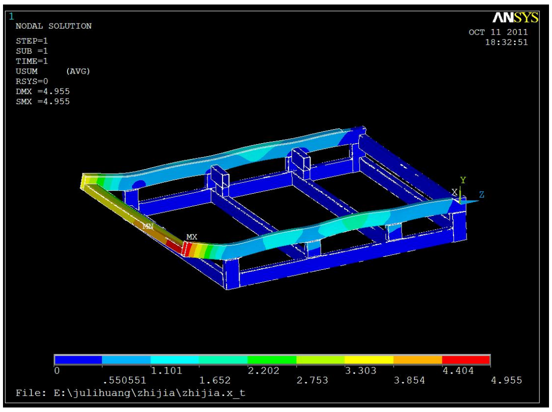

Fig. 9. The strain for the table support frame.

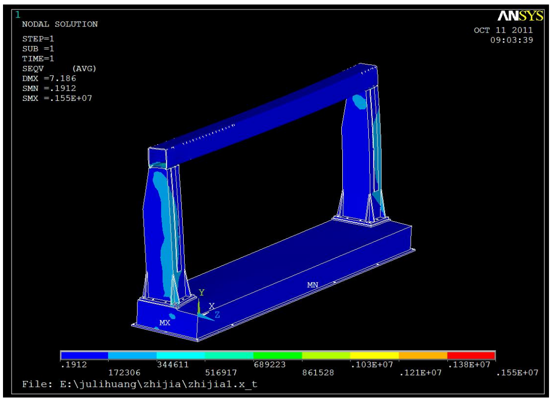

Fig. 10. The Vos Mises equivalent stress.

7. Static and Dynamic Simulation

The ultimate goal of the structural dynamic optimization for the sawing machine is to design the machine of the excellent dynamic characteristics. The dynamic characteristics of the machine are determined by the various modules' characteristics and the interface characteristics between the modules. Using the dynamic optimization method of the element structure and frame size, you can design the sawing module of the more excellent dynamic characteristics. The preliminary optimized modules and the corresponding interfaces are joined together to form the whole which has the better form of the structure. How to make the whole machine in the form of the optimal dynamic characteristics needs to be further optimized.

The finite element model of the sawing machine contains a lot of data. The dynamic optimization spends a lot of resources and time. So the computation amount must be reduced. Decreasing the number of the design variables and remaining the structure of the whole machine in the optimization process is an effective measure to reduce the calculation amount. When optimizing the whole sawing machine, first the internal structure and the whole layout of the machine modules must be determined, and then the some structural geometry and the dynamic parameters on the combination surfaces are taken as the design variables. After the ANSYS optimization calculations, the optimal solution can be obtained to meet the specific constraints (for example, the low-level natural frequency of the whole machine shall not be less than a certain value, etc.). According to the optimized dynamic parameters of the joint surfaces, the existing experimental values are combined to determine the technical requirements of the interface surface accuracy. The dynamic optimization process of the whole machine in the form of a fixed structure is shown in Fig. 11.

8. Modal Analysis of Sawing Machine

The modal analysis can be used to determine the natural frequencies and the corresponding modes of the structure, to provide the basis for the structural design, to avoid the resonance phenomenon under the dynamic loads, and to improve the vibration performance of the structural system. Therefore, the modal analysis is an important element of the structural dynamic analysis. The dynamic performance analysis of the system is primarily the modal analysis of the structure, including the natural frequencies and the corresponding modes of the structure. In the engineering practice, we often only find out the first few orders of smaller natural frequencies and mode shapes, because generally only the first few orders of the natural frequencies may cause resonance and danger. The established model is calculated using the ANSYS modal analysis module.

Modal analysis process includes:

1) Modeling.

Only linear behavior of modal analysis is valid. A non-linear element will be taken to be linear. Young's modulus EX must be specified (or some form of rigidity), and density DENS must be specified (or some form of quality). Material properties may be linear or nonlinear, isotropic or orthotropic, constant or temperature-dependent. The non-linear will be ignored.

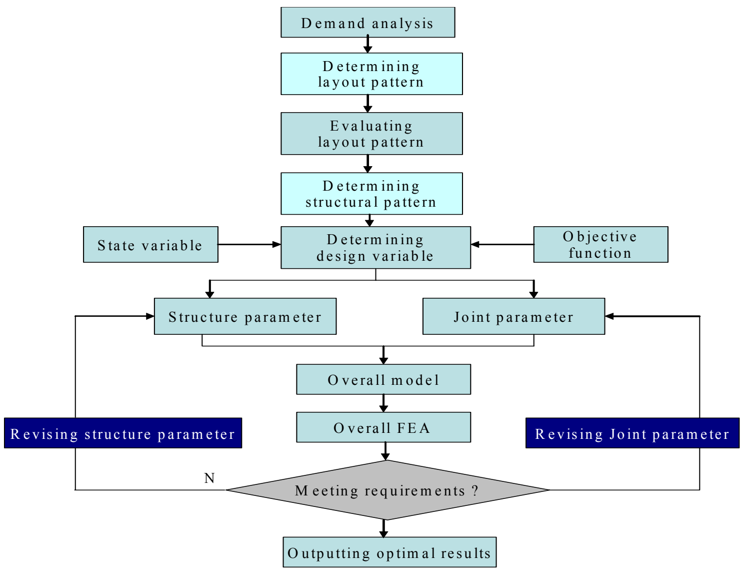

Fig. 11. The dynamic optimization process of the whole machine.

2) Loading and Solution.

In the typical modal analysis, the only effective load is the zero displacement constraint (if some DOF has a nonzero displacement constraint, the program will replace the DOF position with a zero displacement constraint). The other loads may be applied in addition to displacement constraints. But they will be ignored. On the direction of non constraints, the program will solve the rigid motion mode (zero frequency) and the free body mode of high-frequency (nonzero frequency).

3) Expanding the modes.

“Expanding” means that the reduction solution will be expanded to a complete set of DOF. “Reduction solution” is commonly expressed by the master DOF. In modal analysis, the “expanding” refers to write the modes into the resulting file. That is, the “expanded mode” is not only suitable for the reduced mode shape extracted by the reduction mode, but also applicable to the complete mode shape extracted by the other mode. Therefore, if you want to view the mode shapes in the post-processor, you must first expand the modes (that is, to write the mode shapes into the result file).

4) Observing the results.

The modal analysis results (namely, the treatment results of the expanded modes) are written into the result file for the structural analysis Jobname. RST. The analysis includes: the natural frequencies, the expanded mode shapes, the relative stress and the force distribution. You can observe the modal analysis results in POST1 (namely, the normal post-treatment).

Using the ANSYS modal analysis module, you can calculate the established model. The modal analysis for the key support structure of the sawing machine – the pillar framework can be done by the block lanczos method. The first 3 and second 2 order modal’s vibration modes are respectively shown in Fig. 12 and Fig. 13.

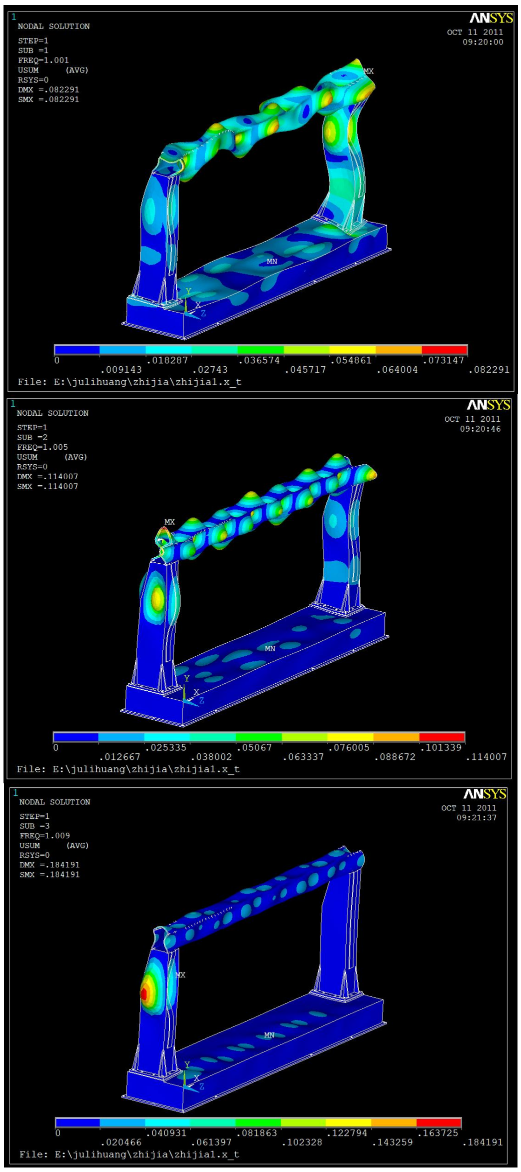

Fig. 12. The first 3 order modal’s vibration modes.

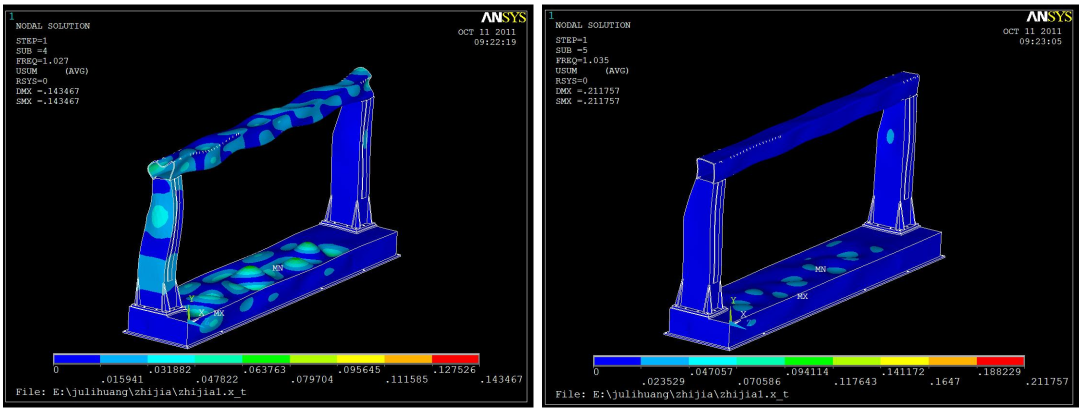

Fig. 13. The second 2 order modal's vibration modes.

References

[1]. Zhu Jie, Chen Guo-Jin, Wang Wan-Qiang, Applying FPGA on Frequency Convert System of SVPWM, *Mechanical & Electrical Engineering Magazine*, Vol. 22, No. 4, 2005, pp. 22-25.

[2]. Chen Guo-Jin, Yao Zhai-Rong, Sun Hong, Realization of AC Motor Vector Control Based on DSP, *Mechanical & Electrical Engineering Magazine,* Vol. 21, No. 6, 2004, pp. 27–30.

[3]. Wang Tao, Chen Guo-Jin, Ni Jing, Design and Application of Electric Control System on Pattern Drawing Machine, *Journal of Mechanical & Electrical Engineering,* Vol. 29, No. 4, 2012, pp. 413–416.

[4]. Chen Guo-Jin, Chen Hui-Peng, Chen Li-Ping, Research on Multidisciplinary Modeling and Robust Control for Electric Power Steering System, *Advanced Materials Research,* Vol. 179-180, 2011, pp. 179-185.

[5]. Chen Guo-Jin, Deng Kai-Ping, Gong You-Ping. The Design and Simulation of Walking Hydraulic Transmission System of Loader HT25J, *Mechanical Engineer,* No. 6, 2012, pp. 46–49.
