Sensors & Transducers, Vol. 159, Issue 11, November 2013, pp. 92-99

www.sensorsportal.com

Sensors & Transducers

http://www.sensorsportal.com

Research and Key Bearing Part Simulation of Finite Element Analysis Platform of Gantry Crane Based on ANSYS

He Bin-Hui

College of Mechanical & Electronic, Nanjing Forestry University, Nanjing 210037, China

Received: 9 September 2013 /Accepted: 25 October 2013 /Published: 30 November 2013

Abstract: In this paper, we firstly elaborate the Gantry Crane is a kind of door type frame and can move along the rail of the gate hoist and the finite element method. And then analysis overall planning of finite element analysis platform of gantry crane: structure and design characteristics and finite element analysis platform overall process planning. We also research the research of key technologies: parametric design technology, modular technology, the secondary development technology based on APDL. Through simulation analysis and calculate of key bearing part, we get the conclusion is the door frame stability coefficient is 9, the coefficient far outweigh the 1 that door frame under the above after loading charge in a stable state. *Copyright © 2013 IFSA.*

Keywords: Gantry Crane, Finite element analysis platform, ANSYS simulation.

1. Introduction

Gantry Crane is a kind of door type frame and can move along the rail of the gate hoist, also call door type hoist [1]. Metal structure is important constituent of door type hoist, some of the hoist in the weight of the metal structure of the overall weight is 40 % - 70 %, and some other giant hoist is above 90 % [2-5]. Door type hoist is a typical non-standard metal structure, different hydropower station are not the same on the performance requirements. Therefore, every time a new hydropower station need portal type hoist to redesign, from the drum and the car frame to door frame design, the entire process need to work again [6]. Secondly, as the weight and lifting height increase gradually, the original classic design method has been more and more not applicable, and the calculation is also more and more big, the cost of artificial are more and more.

The finite element method is the analysis method of the structure of a numerical calculation method, is the application and development of matrix method in structural mechanics and elastic mechanics fields [8]. The finite element method based on principle as the foundation of the development of a high efficient, commonly used numerical calculation method, through development of sixty years, is widely used in various engineering fields [7]. Using finite element analysis door type hoist metal structure stiffness and strength, can very good simulation door type hoist geometric model, working load and boundary conditions, the accuracy of calculation results will be greatly improved, so that the door type hoist design more optimization. The finite element method in mechanics and mathematics and other fields is the basis of the theory [10], units classification principle, the shape function selection and coordination; the finite element method involves the various numerical calculation method and error, convergence and stability. The finite element method developed early

general finite element program, suitable for lifting the transport machinery structure mechanics analysis [9].

Based on the portal type hoist structure and load form the basis of the study, using APDL parametric design, combined with the idea of modular design and the software of ANSYS secondary development, construct the door type hoist finite element analysis platform.

This paper is organized as follows. According to the door type hoist design calculation characteristics, puts forward the door type hoist includes reel group, car frame and door frame in three module design and analysis of finite element analysis process and overall planning in Section 2. In Section 3, constructing door type hoist finite element analysis platform key technology: parametric and modular based on APDL and the secondary development of technology. Combined with drum, car frame modeling process discussed the parametric and modular thought in door type hoist finite element of the application of the platform, and points out that the finite element analysis platform secondary development advantages. In Section 4, we will use finite element analysis platform portal type hoist key bearing part for stability analysis. Studied the traditional method to calculate the steel structural stability and finite element method of calculate the structural stability. The conclusions are given in Section 5.

2. Overall Planning of Finite Element Analysis Platform of Gantry Crane

2.1. The Basic Steps and Application of Finite Element Method

Using finite element method to solve the problems has many steps, including basic steps are:

1) Continuum discretization and unit type selection. First of all, we should according to the shape of the continuum to choose the appropriate unit. The second is unit division. The continuum cut into many units, the size of the unit, the continuum is divided into finite element composition by finite element model for cutting into node and units.

2) Choose displacement function. It must be able to reflect the unit of the rigid body displacement, reflect unit constant change and the continuity of displacement [11].

3) Define strain displacement and the relation between stress and strain. For the finite element equation is derived, need to build strain displacement relationship, such as in one-dimensional deformation and small strain of strain and displacement in the *x* direction relationship is as follows:

$$\varepsilon_x = \frac{du}{dx} \tag{1}$$

where ε_x is the stress of X direction, u is displacement.

The stress strain relationship between constitutive stress and strain will be connected Hooke's law is the most simple stress strain law:

$$\sigma_x = E\varepsilon_x \tag{2}$$

where ε_x is the strain of X direction, E is the Elastic modulus.

4) List the element stiffness matrices and equation.

5) List the overall equation and boundary conditions are introduced. Using the direct stiffness method is superposition method, the equation of the unit together assembly into general equation, the matrix form is:

$$\{F\} = [K]\{d\} \tag{3}$$

where $\{F\}$ is the whole node force matrix. $[K]$ is the overall stiffness matrix. $\{d\}$ is the node degree of freedom or generalized displacement

6) To solve the unknown degree of freedom.

7) Solving unit strain and stress.

8) View the results of the analysis. In the design process, usually it's important to determine the structure of displacement and stress maximum position, usually finite element program in the post-processing module to graphics mode display.

The finite element method is a numerical calculation method in engineering application. Mainly inspects the structure strength, stiffness of performance. ANSYS software can be seven kinds of structure analysis, including static analysis, structure analysis and transient dynamics analysis, buckling analysis, and so on.

2.2. Door Type Hoist Structure and Design Characteristics

2.2.1. Door Type Hoist Structure Characteristics

Door type hoist main components are: Drum, car frame, door frame, the driver room, rail ladder and reducer. The main bearing part is car frame and door frame. On the door there is a lifting trolley, lifting trolley and door frame in mutual vertical track were moving, so constituted two-way door type hoist; door type hoist topology structure shows in Fig. 1.

This paper studies the car frame beam structure consists of two root girders, and three root and girder vertical beam as well as several beams with vertical girder parallel small girder composition. Structure shows in Fig. 2.

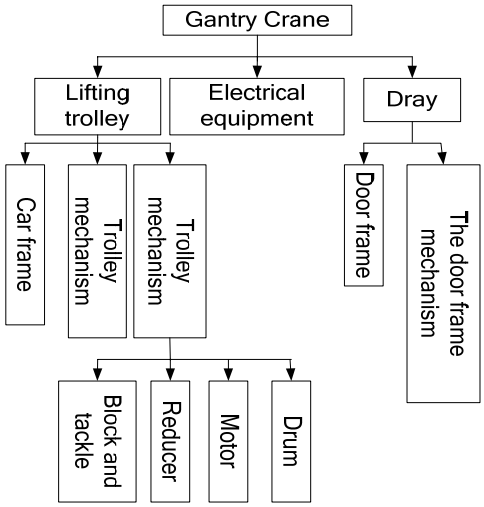


Fig. 1. Door type hoist topology structure.

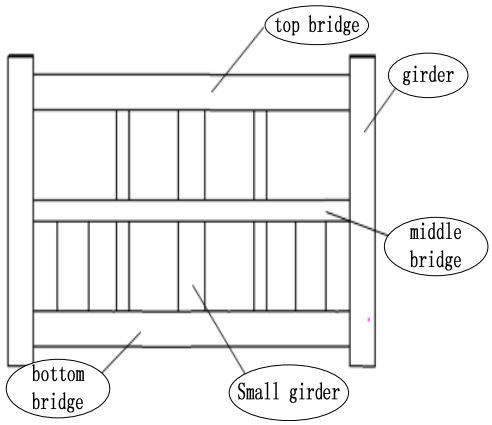


Fig. 2. The car frame metal structure diagram.

2.2.2. Door Type Hoist Design Process Characteristics

Door type hoist is a large metal structure, its design process and the traditional mechanical design process, compared with its own features:

1) Door type hoist design are the basis of the original hoist, according to the customer for the new door type hoist performance different requirements, the corresponding structure form or shape size must be modified, then check calculation strength, stiffness, until the generation meet customer demand design results.

2) The design process refer to the corresponding design manual, more depend on designers design experience.

3) Door type hoist is a large metal structure, the traditional design method is very conservative, already can't satisfy the design requirements of the design unit in the product design stage design cycle period, high quality.

4) Door type hoist structure form is agile changeable.

2.2.3. Finite Element Analysis Platform Overall Process Planning

The finite element calculation method results the calculation results more accurate, and to the secondary development of the finite element software, research and development of a finite element analysis platform. Door type hoist in the gate and the weight of the lifting process, the weight transfer power to roll, roll through the placement in the car frame roll bearing a transfer of load transfer to the car frame, lifting trolley through the door frame orbit car driving mechanism will be small wheel pressure transfer to the door frame. According to above analysis, drum, car frame and door frame finite element analysis need to analyses one by one, after the completion of a reaction is applied to extract a parts then on to the next part of the finite element analysis. Door type hoist overall analysis process is as follows Fig. 3 shows.

According to the analysis of the structure of platform is different, need to prepare a lot of mutual nested toolbar menus, such not only can expand the toolbar to the number of the abbreviation, it could still realize menu level, make the analysis process orderly. The toolbar button and macro command is one-to-one, the user through the operation custom toolbar button realize door type hoist the whole process of the finite element analysis. Door type hoist finite element analysis platform of the overall operation process shows in Fig. 4.

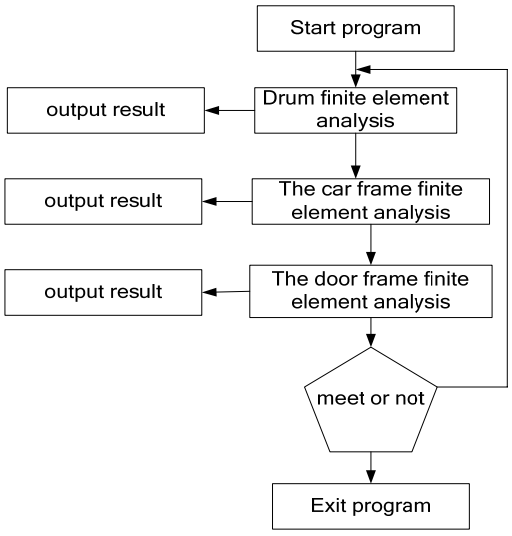


Fig. 3. Finite element analysis process.

3. The Research of Key Technologies

3.1. Parametric Design Technology

Parametric design technology refers to the shape that is standardized structure, with a set of parameters to restraint the geometry of a group of structure size sequence [12]. The advantages of parametric design are fast and accurate, particularly

in the application of some structure and connection fixed structure design. The same structure form just need relative size changes, can change the size of the modified by parametric, parametric sequence fast drive structure geometry change.

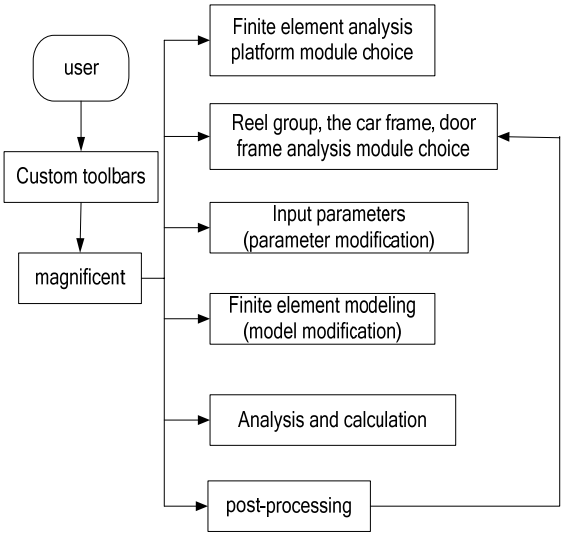
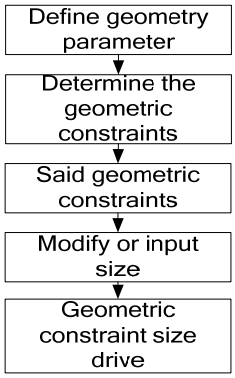


Fig. 4. Analysis platform overall operation process.

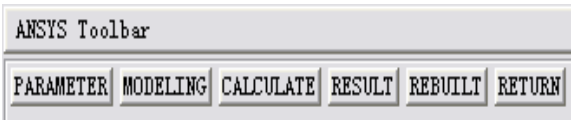
The methods of parametric design main are application and structure change less but geometry size need to change in product design. Common parametric design methods are: algebraic method, artificial intelligence method, direct manipulation and language description method. The method of parametric design from CAD in actual use process-develop, produced a kind of modern design method. The main goal of parametric design is to design a series of products of the requirements, in the design of a series of products. The advantages of parametric design embodied in the structure design of the variability, template repeatability and assembly convenience. It is precisely because of these advantages make parametric design method and is more and more widely used. Parametric design principle shows in Fig. 5 (a). Door type hoist due to its structure and design characteristics of process, in the design of parametric technology, based on web form die as the example, this paper introduces parametric technology in the application of finite element analysis platform. Reel group of finite element analysis process including roll group input parameters, parametric modeling and mesh division, the finite element calculation, the results of check steps, such as in the Toolbar (ANSYS Toolbar) as shown below Fig. 5 (b):

3.2. Modular Technology

Mechanical product Modular Design (Modular Design) is to point to the demand of the market research, product on the basis of analyzing the classification and design a series to meet the requirements of function module, and then according to demand choose corresponding module combination, get different function or structure of different products. The main way of modular design has four kinds: horizontal series modular design, longitudinal series modular design, span series modular design, the whole series modular design.



(a) Parametric design structure diagram.



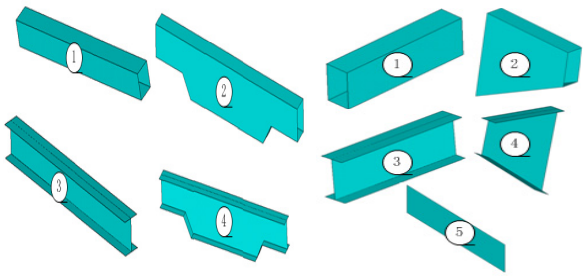
(b) Finite element analysis process.

Fig. 5. Parametric design principle.

Modular design is key module division and the interface of module between technologies. In division module is needed the system function and carefully structure analysis, module has direct influence on the stand or fall series design module. Mechanical products are made from components and parts by parts. According to the functional independence principle will product segmentation, get level subcomponent, then according to the function division, each separate subcomponent independent get secondary subpart until be divided into components.

Module partition has four principles: sketch positioning principle, independence principle, the principle of the stability and compatibility principle. Modular technology products of the finite element model has geometry sub-module and load sub-module, through the parameter modification, module interchange and realize the finite element model of geometric module and the change of the load module, and establishes the finite element model of the product.

The car frame structure and load complexity, decided to the diversity of the finite element model of the module. Small girder has Fig. 6 shows the structure.



(a) (b)

Fig. 6. Small girder structure.

3.3. The Secondary Development Technology Based on APDL

ANSYS software scalability, its own provides several secondary development tools: APDL parametric design language, UIDL user interface design and UPFs user programmable characteristic. APDL not only provides a general programming language function, such as parameter, macro, scalar, vector and matrix operation, branching, and circulation, repetition and ANSYS finite element database access, also provides a simple interface customization, realize the interaction parameters input, message mechanism, interface driver and run applications, etc. It can use the edit ASCII file software editing and generation, APDL command flow file from ANSYS software version or the operating system platform restrictions or influence.

ANSYS software allows users according to their own need to customize special graphics interface, the ANSYS software is in graphic interface aspects of the important advantages. ANSYS predefined default Toolbar, there are four buttons: SAVE_DB, RESUME_DB, QUIT, POWERGRAPH, through the modified ANSYS boot file start100.ans can let ANSYS loading user defined start Toolbar, and not the system default Toolbar.

1) Article tools of loading.

Start100.ans document control predefined toolbar commands are as follows:

There are five predefined ANSYS abbreviations in the Toolbar. The following commands can be used to remove them (by removing the exclamation point).

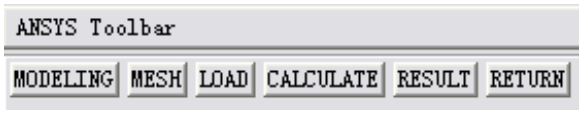
!*ABBR, SAVE_DB
!*ABBR, RESUM_DB
!*ABBR, QUIT
!*ABBR, POWRGRPH
!*ABBR, E-CAE
…

This paper studies is outlets hoist finite element analysis process, so the four macro file name are JUANTONG, CHEJIA, MENJIA and RETURN. ANSYS software commands are not case-sensitive, but in the user interface displayed on the toolbar names are capitalized. The revised command flow is as follows:

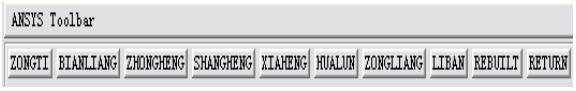
*abbr, juantong, juantong.mac
*abbr, chejia, xiaochejia.mac
*abbr, menjia, menjia.mac
*abbr, return, return.mac

ANSYS after rebooting, toolbar into custom good new toolbar, as below Fig. 7.

(a)



(b)



(c)

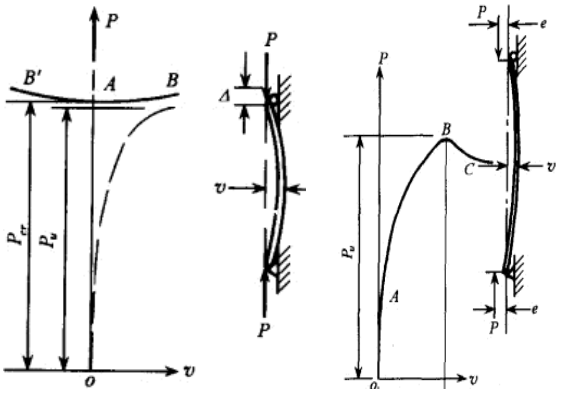
Fig. 7. The car frame modeling tool bar.

With the car frame, for example, in order to make the analysis process more regulations, the level of the toolbar button: only six MODELING, MESH, LOAD, CALCULATE, RESULT, RETURN, and MODELING button nested next layer of the toolbar, click on the toolbar in Fig. 7 (c).

4. The Simulation Analysis of Key Bearing Part

4.1. The Traditional Method to Calculate the Stability of the Door Frame

Steel structure, due to the advantages of high strength and light structure in engineering field widely be used. On the other hand, the stability problem is the most prominent problem in steel structure [13]. The stability of the structure is deformation problem. The main is to find the external load and internal resistance between the stable equilibrium states. Prevent structure appear instability and not make their actual stress below a certain value, but to prevent structure into the unstable equilibrium state. The stability of the steel structure can be divided into three categories: Equilibrium bifurcation buckling, the load-deflection curve is shows in Fig. 8 (a). Extreme value point instability, the load-deflection curve is shows in Fig. 8 (b). Steel structure stability analysis has the following characteristics: diversity, integrity and correlation.



(a) Equilibrium bifurcation buckling curve.

(b) Extreme value point instability load-deflection curve.

Fig. 8. Load-deflection curve.

4.2. The Simulation Analysis of Key Bearing Part

4.2.1. The Finite Element Method to Calculate the Structural Stability

ANSYS software provides two kinds of method to calculate the stability of the structure, eigenvalue buckling analysis and nonlinear buckling analysis. Eigenvalue buckling analysis is mainly used in the theory of structure buckling analysis, calculate the structure of theory buckling strength. Eigenvalue buckling analysis main steps are as follows:

1) Pretreatment stage. The process of the finite element analysis process of the pretreatment stage is the same. Chosen the right unit (door frame structure using shell63 unit with surface modeling), defined the good material properties (such as young's modulus, Poisson's ratio), as below Fig. 9.

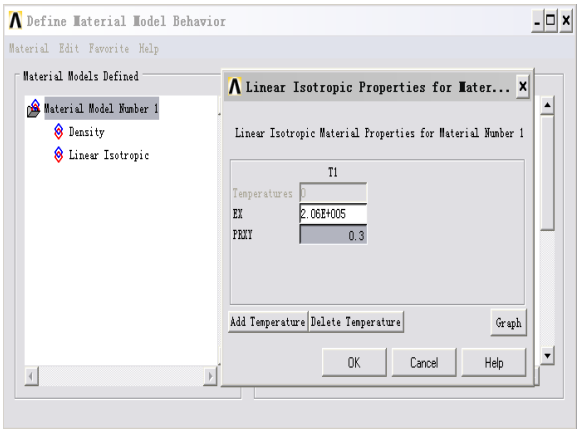


Fig. 9. Define material properties.

2) Static analysis. The eigenvalue buckling analysis is needed before the static analysis of the structure. The static analysis process need to consider the influence of stress, so need to open the stressed switch, other process and general static analysis process is the same: to model applied load and the degree of freedom constraint and solving. Open the stressed switch mode is as follows: Solution>Analysis Type>Sol'n Controls>Basic. In the Basis options checked Calculate stress effects.

3) Eigenvalue buckling analysis. Once again get into the solver, choose solution type into buckling analysis. We can see in Fig. 10.

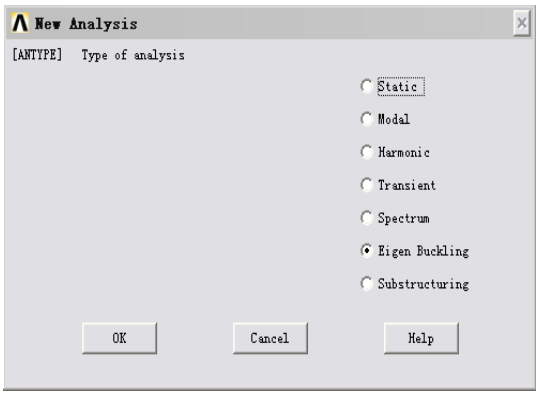


Fig. 10. Choose analysis type.

4.2.2. The Simulation Analysis of Key Bearing Part

Start door type hoist finite element analysis platform, in turn sequence reel group and car frame finite element static analysis, and then will return to the preservation, finite element analysis of the platform of the main menu. Click on the toolbar button door frame, enter key bearing part, is door frame finite element analysis, and then menu bar shows the door frame finite element analysis menu. Click on the first MODELING command start MODELING. The platform will require user input modeling need structure, size and at the same time will open a pair of schematic diagram prompts the user for each input parameter in the structure of the size of the corresponding. We can get structure diagram as shown in Fig. 11. Input parameter is obtained parametric finite element model in Fig. 12.

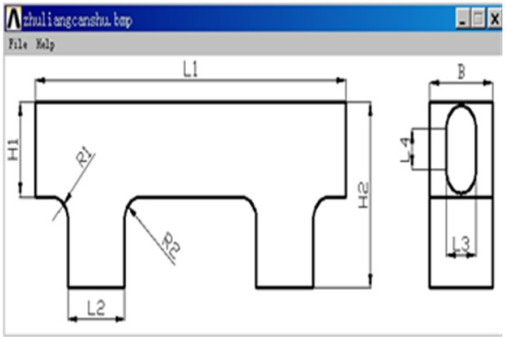


Fig. 11. Structure diagram.

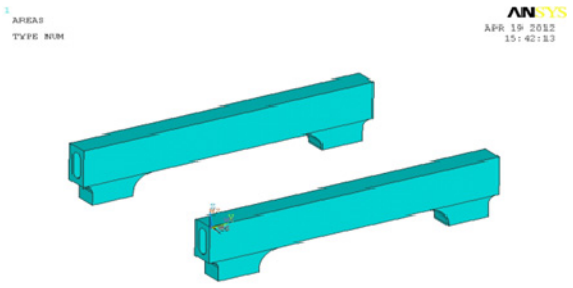


Fig. 12. Parametric finite element model.

Upon completion of the main girder, leg, beam, beam and floor beam modeling, door frame structure as below Fig. 13.

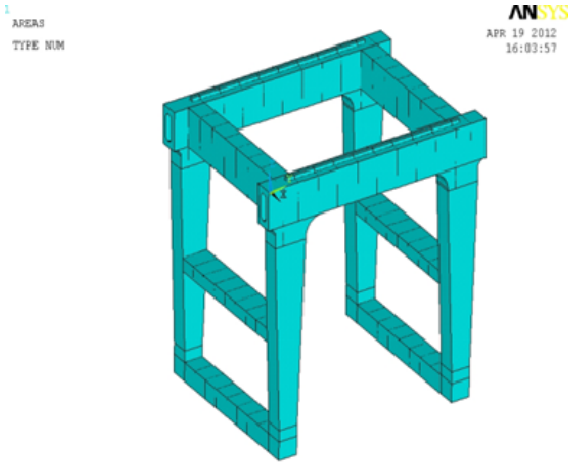


Fig. 13. The door frame structure.

The door frame structure modeling is completed, need to mesh model. Considering the large amount data of finite element calculation, using surface set up door frame model, so in the mesh need through the real constant assignment to the door frame the plate thickness of the assignment. After the mesh, the door frame finite element model as shown in Fig. 14.

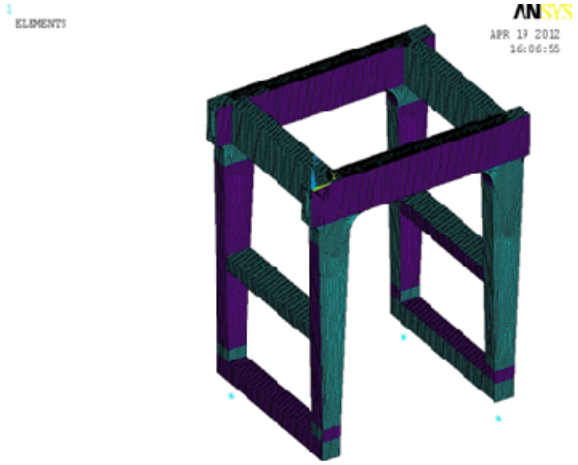


Fig. 14. Grid partition map.

Click on the button LOAD applied load, after completion of loading as Fig. 15. Diagram leg walk mechanism on the degree of freedom constraints, two roots on the girder put small wheel pressure, windward side put wind load. Before exit processor, into the static analysis, then need to pay attention to consider the influence of stress, open stressed switch.

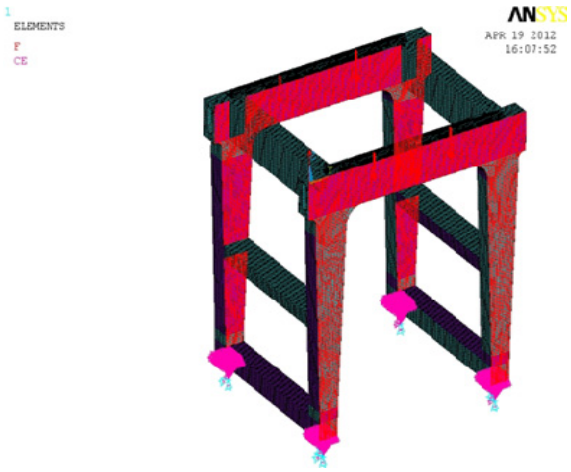


Fig. 15. The door frame after loading.

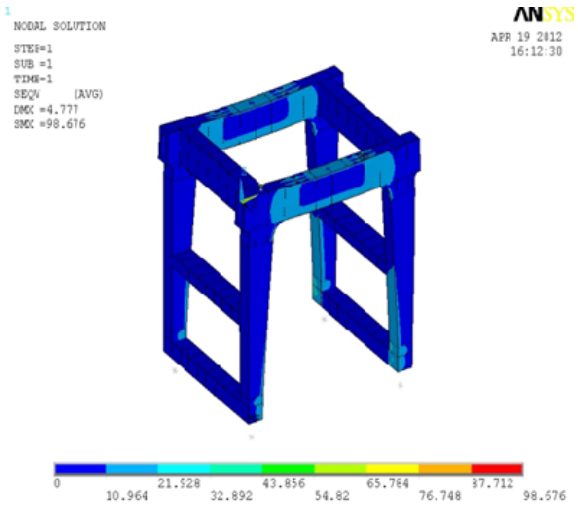


Fig. 16. The door frame VonMises stress.

Calculation after the completion of the user clicks RESULT can view static analysis results, as Fig. 16, the comprehensive strain in Fig. 17.

The simulation analysis part program of key bearing part is as follows:

```
/solu
antype, 1
bucopt, lanb, 8, 1, 0
mxpand, 8, 0, 1000, 1, 0.001
solve
save, menjiajieguo_wd,db,'d:\qbj\menshiqibiji\menjiaAPDL\menjia_save'
*get, buck_canshu, mode, 1, freq
finish
```

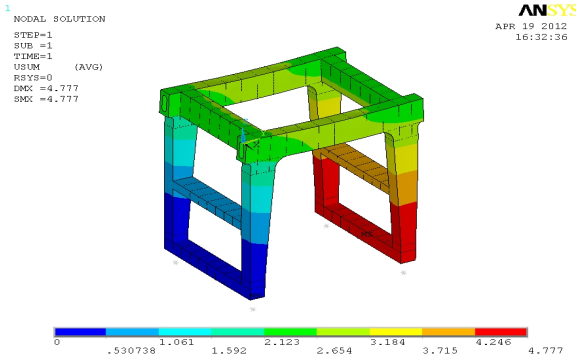


Fig. 17. The door frame integrated strain diagram.

5. Conclusions

The presented method is using the finite element method to improve the accuracy of the calculation, and enhance the reliability of the product; Considering the door type hoist structure type, size and condition changes, improve the generality of the software; Analysis process high automation, speed up the analysis speed, shorten the product development cycle.

Combined with drum, car frame modeling process discussed the parametric and modular thought in door type hoist finite element of the application of the platform, and points out that the finite element analysis platform secondary development advantages.

Through the calculation to get the door frame stability coefficient is 9, the coefficient far outweigh the 1 that door frame under the above after loading charge in a stable state.

References

[1]. Craig B. Chapman, Martyn Pinfold, The Application of a Knowledge Based Engineering, *Engineering Optimization*, 43, 10, 2011, pp. 1063-1077.

[2]. D. C. Uamanam, Shansen, Dynamics of a three-dimensional overhead crane system, *Journal of Sound and Vibration*, 242, 3, 2001, pp. 411-426.

[3]. Serrano D., Constraints in conceptual design, in *Proceedings of the IEEE PowerTech Conference*, Bologna, Italy, 2, 2003, pp. 678-683.

[4]. Thornton A. C, Constraint specification and satisfaction in embodiment design, PhD Thesis, *University of Cambridge*, 1993.

[5]. Kusiak A., Huang C. C., Design of Assembly Systems for Modular Products, *IEEE Transactions on Robotics and Automation,* 13, 5, 1997, pp. 646-656.

[6]. Erixon G., Modular function deployment a method for product modularization, *Royal in Statute of Technology*, Stockholm, 1998.

[7]. Pimmler T. U., Eppinger S. D., Integration Analysis of Product Decompositions, in *Proceedings of the ASME 6th International Conference on Design Theory and Methodology*, MN, Minneapolis, 1994.

[8]. Zeng Qing-Dun, Li Qin-E, Finite Element analysis of static characteristic for gantry crane, *Applied Mechanics and Materials*, 170-173, 2012, pp. 3077-3080.

[9]. Wu J. J., Whittaker A. R., Cartmell M. P., Dynamic responses of structures to moving bodies using combined finite element and analytical methods, *International Journal of Mechanical Sciences*, 43, 11, 2001, pp. 2555-2579.

[10]. Li Hua, Wang Chonghua, Rong Yimina, Service-oriented architecture based dockside container crane CAD/CAE integration system, in *Proceedings of the Design Engineering Technical Conferences*, 2, 2, 2009, pp. 549-557.

[11]. Wu Jia-Jang, Whittaker A. R., Cartmell M. P, Use of finite element techniques for calculating the dynamic response of structures to moving loads, *Computers and Structures*, 78, 6, 2000, pp. 789-799.

[12]. Lagaros Nikos D., Papadrakakis Manolis, Applied soft computing for optimum design of structures, *Structural and Multidisciplinary Optimization,* 45, 6, 2012, pp.789-799.

[13]. Wu Qingming, Zong Chi, Zhang Qiang, Yang Wei, Research on variant design and parameter transfer method for complicated products, *China Mechanical Engineering*, 19, 24, 2008, pp. 2955-2960.