Modal Analysis of Five-Layer Vertical Structure Platform

Beiyi Ma[1,a] , Hua Song[1,b] , Chengzhong Yu[2,c] , Qian Liu[1,a]

[1] School of Mechanical Engineering and Automation, University of Science and Technology Liaoning, Anshan, China;

[2]Anshan iron and steel group corporation, Anshan, Liaoning, China

[a]mbeiy@126.com, [b]songhua88@126.com, [c]tclan1987@163.com

Key words: structure platform, modal analysis, static analysis

Abstract. Using the function of static analysis and modal analysis of ANSYS software, the bearing capacity and vibration condition of high-rise buildings can be simulated and analyzed, and the safety for the design of high-rise steel structure can be ensured. This research is mainly aimed at modal analysis for the overloaded five-layer vertical structure platform, and proves the rationality of the structure design by analyzing the conclusion. The analysis content includes: the stress that structure platform is in the static load, the analysis results include analysis parameters such as stress, torque, strain; modal analysis of structural platform, and the analysis results are respectively outputed in forms and pictures. Through the analysis, it can be proved that the structure platform is designed reasonably, and can satisfy the requirement of actual production.

Introduction

At present, ANSYS software has become the mainstream of the CAE simulation analysis in civil construction industry, and has been widely used in the engineering of steel structure and reinforced concrete such as buildings, stadiums, bridges, dams, tunnels and underground buildings [1].

The modal analysis of ANSYS mainly is used to determine the design structure or the vibration characteristics of machine parts, namely, natural frequencies and vibration type of the structure, and gives the participation and the coefficient that they are not only the important parameters in structure design subjected to dynamic load, but also the starting point of other dynamics analysis problems [2]. The basic steps of building structure modal analysis that is the important method which ANSYS software is in building structural problem analysis include four steps: modeling-loading and solving-expanding modal- analysis results. It must note two problems in modal analysis: One is modal analysis belongs to linear analysis, and only linear behavior is effective in modal analysis. Any nonlinear problem will be ignored and processed in linear problem. The other one is material can be linear, isotropy or anisotropy, constant or relating with temperature in model analysis[1-3].

Static Analysis of the Structure Platform

Model Assumption. In this study the support platform include five layers. The supporting pillar is placed vertically and interactively with double steel I bean, and is chosen integrated structure with total height 15200mm. Each layer of beam includes main structure beam and strengthen beam. Among them, the main structure beam is constituted by double channel steel welded symmetrically, except each layer of platform composed the four edges supported frame, and some roots are placed parallelly between the inside of the platform frame and width direction constituting the main body structure of the supporting platform. Strengthen boom consititued by single channel steel is used to connect with the main structure, is parallel with the length direction of the platform, mainly plays the

role in reinforcing steel bar, and further strengthens the stability of the platform structure. In addition, in order to improve the stability of the support pillar, it will be added the channel steel placed vertically and interactively between each layer platform as pillar support.

In order to improve the accuracy of analysis, different unit types will be set up different structures of the support platform. The specific setup is: support pillar is as the structure undertaking the weight of platform and wind load, and two nodes linear beam element BEAM188 will be used to simulate in analysis; The main structure of the supporting platform and strengthen beam that is as fixed bracket of the supporting platform panel and coke powder tank will be simulated by BEAM44 used in the analysis; LINK10 will be used to simulate the supporting corner steel constituting a stable system with edge support column. Rigid connection method is adopted between the supporting pillar with the main structure beam and strengthen beam of the supporting platform, and hinge node connection is adopted between corner steel constituting a stable system with edge support column and supporting pillar; All steel structure materials adopted in simulation are assumed to rational elastic materials; All steel structure component materials are assumed to linear elastic materials, and meet Hooke's law, at the same time, in the simulation process Skin effect of steel plate is ignored.

Simulation Steps. Overall analysis consists of five steps, including: 1,Parameter Setting; 2, Formating model; 3, Static analysis; 4, Operating analysis calculation; 5, Analyzing the calculation results. Ultimately platform model of elements form is made, the rendering is shown in fig.1 Data statistics result is shown in table 1
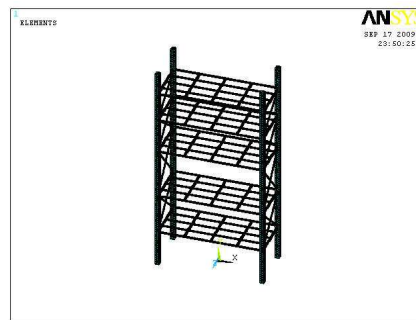

Fig. 1. Structure Platform Model

Table 1 Statistics of Static Analysis Result

Unit type		axial force	Element number	Axial strain	Element number	Beam bending moment	Element number
BEAM188	MIN	-0.699E8	210	-0.333E-3	213	-0.946E-5	179
	MAX	0.138E8	550	29990	550	29990	553
BEAM44	MIN	-0.106E9	4400	-0.505E-3	4400	-47748	865
	MAX	0.107E9	4819	0.508E-3	4819	47748	824

It can be seen obviously from the analysis result figures, whether the supporting for unit type BEAM188 or the main structure beam and the strengthen beam of unit type BEAM44, the larger stress appears on the bottom position, one-layer position and two-layer position under the role of the pressure. But as the building structure, the most structures of one-layer underpart contact directly with ground foundation that generates a big part of the force can be ensured that the ground base or embedded underground part undertake, so as to improve the safety of structures.

To sum up, it can be seen from the analysis result statistics, all the calculated data is in the safety range of materials allowed, namely, the integral design that the coke powder tank supports platform can satisfy the requirement of production.

Modal Analysis

The modal analysis to the model is block Lanczos method that ANSYS system defaults, and analysis result extractes the front four modes.

Loading and Solving of Modal Analysis. The difference with static analysis, in the modal analysis process four pillar-feet of the supporting pillar in the model are all fixed in the way of steel, namely: both mobiles in three directions of its x, y, z are restricted, and rotation around x axis, y axis and z axis are restricted. Adding the effect of this constraint is shown in figure 2.
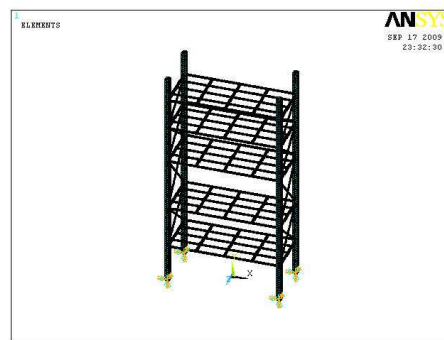

Fig. 2. Model Structure of Pillar-feet Steel

Modal Analysis Result. After modal analysis, using "SET, LIST" command can generate the the list of the calculation result, the result is shown in table 2. In the table 2, the frequencys of the structures are all lower, and far lower than the speed 61.29 r/min of the screw conveyor, therefore it won't happen the resonance phenomenon.

Table 2 The List of Ahead 20 Order Frequency Results

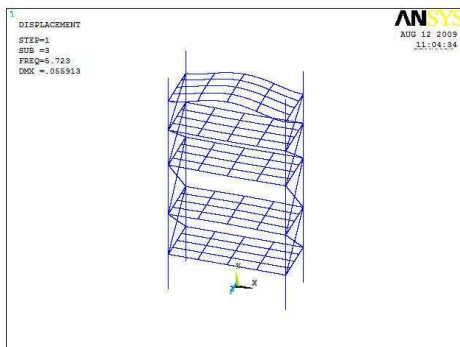| Frequency order times	Frequency value	Frequency order times	Frequency value
1	0.96925	11	8.6031
2	5.6463	12	8.6082
3	5.7233	13	8.6113
4	5.7316	14	8.6205
5	5.7361	15	8.9101
6	5.7384	16	9.6484
7	5.7404	17	11.475
8	5.7931	18	11.575
9	6.1118	19	11.608
10	8.5954	20	11.635

Block Lanczos method can extract the ahead four order vibration modes, the ahead four order vibration modes of modal analysis is shown in figure 3a, 3b, 3c, and 3d.
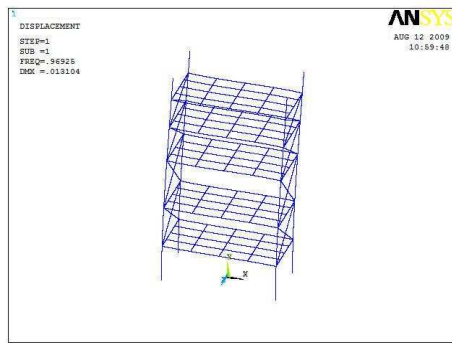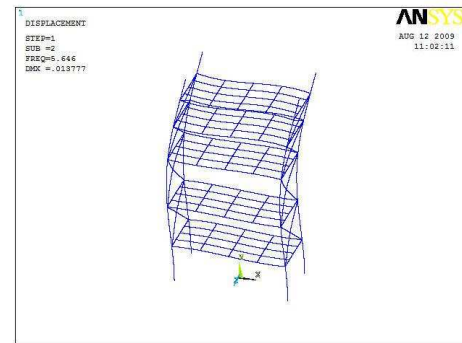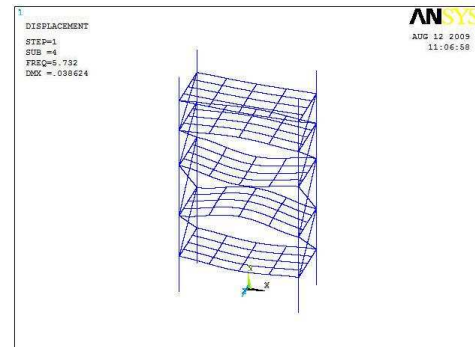
a First-order vibration mode	b Second-order vibration mode
c Third-order vibration mode	d Forth-order vibration mode

Fig. 3. Four Order Vibration Mode Shapes

From analyzing the vibration model figures, it can be seen: first-order vibration mode is mainly for the vibration of vertical direction, and mainly happens the top two layers; Second-order vibration mode is mainly for the vibration of horizontal direction, four supporting pillars appear vibration bending obviously; Third- order vibration mode and fourth-order vibration appear the vibration of the vertical direction again, and with the increase of frequency order times, vertical vibration gradually develops to the lower structure, forming overall vertical vibration. From the analysis figure, it can be seen, the vibration of the supporting platform with coke powder tank is mainly vibration in the vertical direction, vibration frequency is very small and far lower than the speed 61.29 r/min of the screw conveyor, and the whole structure will not appear resonance. This kind of mode meets the need of design. Four supporting pillars still guarantee the stability of the whole structure, and the stability of structure overall is good.

References

[1] Li Bingtao, Xie Xiaoli, and Zhang Ermao. ANSYS applicating in modal analysis of the giant steel frame structure. Shanxi Building. The 34th volume, the 8th period: 49,59,69, in March 2008

[2] Xiao Sa Workshop. The latest classic ANSYS and Workbench tutorial. Beijing: Electronics Industry press, 2004

[3] Zhang Kaiyin, Chen Jianhui and Liu Lijun. Based on the structure dynamic stress study of the modal analysis technology. Journal of Wuhan university of science and engineering (traffic science and engineering edition) the 31st volume, the 6th period:199,299,399,499, in December 2007