Available online www.jocpr.com

Journal of Chemical and Pharmaceutical Research, 2014, 6(3):849-852

Research Article

ISSN : 0975-7384
CODEN(USA) : JCPRC5

Random vibration analysis for the chassis frame of hydraulic truck based on ANSYS

Wang Hai-fei, Jia Kun-kun and Guo Zi-peng

School of Construction Machinery, Chang'an University, Xi'an, Shaanxi, China

---

ABSTRACT

Combined with the actual working environment, building chassis frame of the entity model by the Solid works software for FB45 hydraulic truck. The analysis software ANSYS is used to calculate the prestressed modal analysis and random vibration analysis(PSD), acquired the random vibration simulation results when the truck driving in dangerous conditions. These results could provide some useful suggestions for design and improvement for the truck'structure and to avoid resonance.

Key words: Hydraulic truck; Chassis frame; Finite element; Modal analysis; Random vibration analysis

---

INTRODUCTION

Hydraulic pallet truck is a kind of mechanical and electrical integration of liquid transportation vehicle, and is also an important part of tools of modernization. With the rapid development of economy in China, in military, bridges, ships, aircraft and petroleum chemical transportation, and other areas it has be widely used. FB45 hydraulic truck belongs to a kind of small and medium-sized hydraulic pallet transport vehicles, the truck frame includes chassis frame and subframe. The subframe is used to avoid goods produce concentration stress to bracket and suspension system, linking chassis frame, bracket and container load of parts. While, the chassis frame is the structure that cohesion and bearing bracket, suspension system, oil tank, engine, and the components of hydraulic electric system, the entity model is shown as figure 1.

Currently, there are exist these kinds of finite element analysis about hydraulic vehicle, such as Zhang Yinsheng of Central south university , CNHTC's group chengdu ace commercial vehicle co., LTD., Hu Qijian has done a finite element analysis of flat car. While, most of them are about statics or modal analysis of the subframe and suspension system structure.

Due to the FB45 hydraulic truck is usually used in bad working conditions, the external incentive is often complicated, considering the hydraulic truck may appear the middle wheel impending in the course of driving, therefore, the ability of bearing variable negative load is very important for the chassis frame[1-2]. This article based on the predecessors' theoretical analysis, using dynamic random vibration analysis for FB45 hydraulic truck when it is driving in macadam, thus we can get the result more reasonable and accurate.

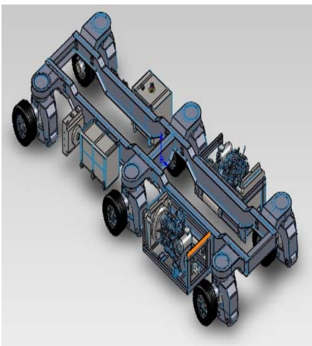

Fig. 1: FB45 hydraulic truck

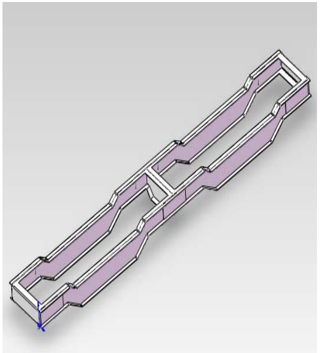

Fig. 2: The model of chassis frame

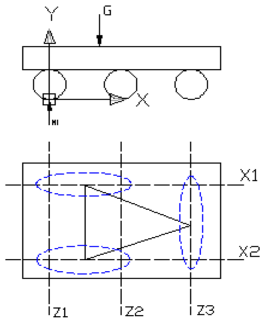

Fig. 3: The simple force diagram of vehicle level driving

Building the chassis frame model

According to the actual size of design, separating the chassis frame from the whole model and building it by Solidworks. The chassis frame length is 8500mm, the width is1200mm, weight is15t, kerb weight is 45t and maximum speed is 30km/h. Frame material select P150L steel, modulus of elasticity is 2.06e11Pa, poisson's ratio is 0.3, density is 7800kg/m^3, yield strength is 355MPa.

Chassis frame carries the engine, fuel tank, electrical equipment, etc. on two sides, but the weight of these structures are far less than the frame weight, so simplifying these, ignoring the bending moment function for the frame, regarding them as the vertical prestressed. And considering the vehicle in the transport process should ensure that certain safety coefficient, the sum of gravity by 4t uniform loading frame related on the surface. In the processing of modeling, we should simplify the factors according to the actual geometric structure, the entity, such as holes, chamfering etc. that not materially affect the calculation results as far as possible. The chassis frame model as shown in figure 2.

Hydraulic truck's simple force diagram as shown in figure 3 when it drives in level. When the vehicle driving, it mainly bear goods and their own self gravity, the ground supporting force Ni (i = 1 ~ 6), in the process of driving, the truck is supported by three point hydraulic cylinder (the dotted line in the figure shows), so the car hydraulic suspension system is divided into three groups, the hydraulic cylinder oil which in the same group and the pressure are the same, to make the same set of tyres under uniform load. We can obtain the wheel support according to the theoretical mechanics knowledge.

The establishment of finite element model

To establish a model in ANSYS, and do statics analysis first which makes preparations for prestressed modal analysis. The load is 4 t, equivalent to uniform force is 31566 Pa. A good finite element model is an important factor that effects the simulation results, so to acquire the relatively accurate results, we using entity unit (solid45) division for the grid division, choosing free division grid partition for division unit, divided into a total of about 25863 nodes, 75563 units, the finite element model as shown in figure 4. The chassis frame through the bracket structure and suspension system and tire connect, the connection way is riveting, so the treatment of boundary conditions is in the riveting area (simplified) set node constraints. From the calculation results we can get the maximum stress is 13.4 MPa and maximum deformation is 0.174 mm [3-5].

Fig.4: The finite element model mode of the chassis frame

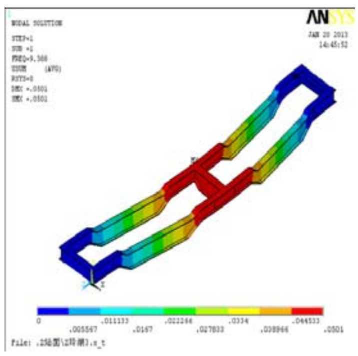

Fig.5: The first order natural vibration mode of the chassis frame

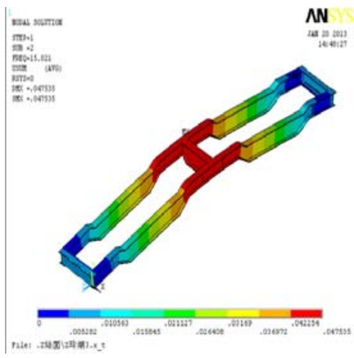

Fig.6: The second order natural vibration mode of the chassis frame

Modal analysis of the chassis frame

Modal analysis is a kind of linear analysis technology, used to determine the structure of the natural frequency and vibration mode. ANSYS provides seven modal extraction method, this paper uses Block Lanczos method to extract ten order modal of chassis frame that have prestressed firstly. In modal analysis, the only effective load is zero displacement constraints, if in a certain degree of freedom (DOF) specifies a non-zero displacement constraints, the program will replace the degree of freedom by zero displacement constraints [6].

Because the hydraulic truck is drived by three shafts, considering that in the course of driving, the middle wheels may appear suspended situation, therefore in the dynamic analysis, canceling their constraints to calculate the worst condition. The calculation results as is shown in table 1. From table 1 modal analysis results we can see that the frame of 1 ~ 10 orders modal natural frequencies are 0 ~ 110 Hz range. Images 5 and 6 are the first and the second order's natural frequency corresponding vibration modes.

Table 1. Each order's natural frequency and vibration mode characteristics of the chassis frame

numbers of	inherent frequencies	Characteristics of modal
1	9.3878	vertical downward bending along the Y axis
2	15.021	vertical upward bending along the Y axis
3	29.330	torsion in the plane
4	43.487	bending and torsion in all parts
5	47.951	torsion in middle and trail
6	59.444	bending and torsion in head and trail
7	70.965	torsion in middle and trail
8	72.476	bending and torsion in all parts
9	78.233	bending and torsion in all parts
10	113.54	bending and torsion in all parts

Random vibration analysis

Random vibration analysis is used to determine the structure response under random loading. ANSYS uses the power spectral density (PSD) spectrum as random vibration analysis of the load input. Power spectral density is a kind of probability statistics method, and is the root mean square value of random variables, including a measure of the random vibration energy and frequency information. Power spectrum that can be displacement, velocity, acceleration or force power spectral density and other forms.

The frame is a multiple degree of freedom and elastic vibration system, there are three factors mainly produce complex vibration of various turbulence for the frame: the first is random vibration caused by the uneven road, the second is the harmonic vibration when the engine working produced, the third is caused by the shaft. The hydraulic truck in the course of driving mainly bears uneven pavement that generates random vibration. Based on the former modal analysis, power spectral density uses displacement power spectral density, the spectrum value according to the national standard "Vehicle Vibration Input Pavement Level Representation Method" (GB7031-1986), shows in Table 2. Running speed is 30 km/h and driving road is macadam (D level).

Table 2. Spatial frequency power spectral density

Doubling frequency of center frequency n_c/m^{-1}	0.125	0.250	0.500	1.000	2.000	4.000
$G_d(n_c)/10^{-6}\ m^3$	655.4	163.8	40.96	10.24	2.56	0.64

Table 3. Time power spectral density

f/Hz	1.05	2.1	4.15	8.3	16.6	33.2
$G_d(f)/10^{-6}\ m^2/Hz$	78.96	19.74	4.93	1.23	0.31	0.08

Because the vehicle applications need to make the spatial frequency power spectral density into time power spectral density, and the transformation formula are:

$$f = v \times n_c \tag{1}$$

$$G_d(f) = \frac{G_d(n_c)}{v} \tag{2}$$

And the calculate results as table 3 shows [7-11]. Input above power spectral density values for the ANSYS random vibration analysis, incentive spectrum are applied on the frame and bracket institutions which connect related constraint nodes. The simulation results of load step 3 (1 σ)of the maximum displacement and stress (node 11938) diagram are as follows. Velocity, acceleration response spectrum analysis for the biggest displacement in the node, the results in turn as shown in figure 7 ~ 9 show.

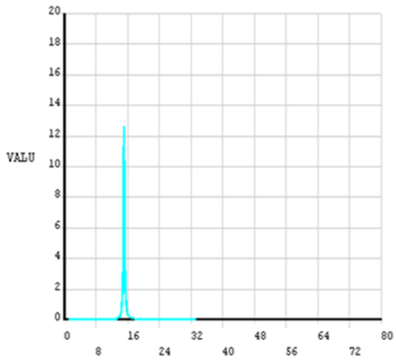

Fig.7:The results of displacement response spectrum analysis

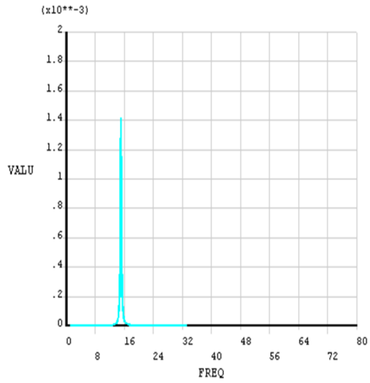

Fig.8:The results of velocity response spectrum analysis

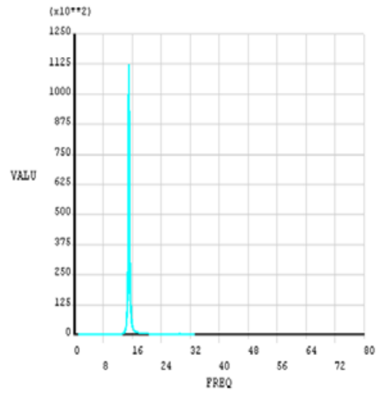

Fig.9:The results of acceleration response spectrum analysis

CONCLUSION

The inherent frequency value of the first order and second order frame are among sensitive frequency value ranges which we can get from modal analysis. For the vehicle driving in the maximum speed is 30 km/h, that close to the first order natural frequency when driving in macadam if regardless of the middle wheel constraints, thus we should pay attention to avoid resonance while driving. Large displacement and stress mainly concentrated in the central frame by random vibration stress and displacement nephogram. And the maximum is 0.02577 m, the biggest stress is 320 MPa, the biggest stress is less than the material yield strength. Figure 7~9 reflect the maximum displacement place node displacement, speed, acceleration response spectrum, by these diagrams can be seen among the 15 Hz the values are maximum, and time frequency is just the second order natural frequency of the chassis frame, so when design vehicle this should be the key consideration to avoid produce resonance. The simulation is the worst possible results when the middle of the wheel frame cancel their constraints, actually the frame in the middle part of the wheel have constraints. Considering this part of the support constraints, the first order natural frequency of the frame is about 50 Hz. Therefore, usually the truck can far avoid resonance arises.

Acknowledgments
The authors wish to thank the Key Laboratory of Road Construction Technology and Equipment of the Ministry of Education, Chang ' an university, under which the present work was possible.

REFERENCES

[1]Wang Jian,Hu Qijian.*2011 Western Automobile Industry·Academic BBS and The Tenth Car Academic Conference Proceedings of Sichuan Province Academic Essays,* pp.126-120, **2011**.
[2] Zhang Yinsheng, Wu Yunxin. *Modern Manufacturing Engineering,* no.1, pp.43- 46, **2009**.
[3]Si Jingping, Han Lu, Ren Qingshuang. *Journal of Inner Mongolia University of Technology,* vol.21, no.4, pp.328-333, **2011**.
[4] Tu Zhenfei. *ANSYS Finite Element Analysis Examples of Engineering Application Tutorial,***2010**.
[5] Wang Jinlong, Wang Qingming, Wang Weizhang. *ANSYS12.0 Finite Element and Example Analysis*,**2010**.
[6] Game Creation Studio. *ANSYS9.0 Foundation Course and Example Explanation*, **2006**.
[7] Sun Weiguang.S*cientific and Technological Information*, no.27,pp.465-467,**2008**.
[8] Cao Qunhao, *Master Degree Thesis of Shanghai Jiaotong University*, **2007**.
[9] MaRuixue, Wang xin, Zhang Kefeng, *Mechanical research and application,* no.1,pp.54-59,**2011**.
[10]Boudette, Neal E. *Industry Week*, v.238, n.7, p.70-73, Apr**1989**.
[11]Mukhopadhyay, M. *Journal of the Institution of Engineers (India), Part MR: Marine Engineering Division*, v. 67, p.8-13, Nov **1986.**