RESEARCH ARTICLE OPEN ACCESS

The Dynamic Response Analysis of Auto Body Sheets to Node Loads

Luo Yun*, Feng Guoying*, Du Yongzhao*, Zhou Shouhuan**
*(College of Electronic Information, Sichuan University, Chengdu 610064, China)
** North China Research Institute of ElectroOptics, Beijing 100015, China)

ABSTRACT
The 3D vehicle body model was built using UG NX6.0, then it was imported into the Workbench of Finite Elment Analyis Software ANSYS V12.1. In the Workbench, the modal analysis and harmonic response analysis of auto body sheets with 4 kinds of node load environment are implemented. Meanwhile the harmonic response analysis of engine's single sine vibration is obtained in stimulation processing based on the modal calculation. Then the rule of influence on the auto body sheets to node load environment was explored further. Node load environment increased the resonance amplitude of harmonic response analysis. What's more, the resonance amplitude increased as the increasing of node loads, which would increase the probability of vehicle structure failure.
Keywords-ANSYS-Workbench,vibration modal,dynamic response,harmonic response analysis,node load

1. INTRODUCTION

As computer hardware and software standards continue to improve, CAD / CAE / CAM has rapid development and wide application. Among them, the mechanical dynamic design analysis, mechanical finite element analysis and mechanical vibration analysis has a more broad application prospects[1]. Based on Computer-based finite element analysis software ANSYS, the dynamic analysis has been used in a variety of vibration analysis, structural design and other fields[2,3].With the birth of ANSYS-workbench module, the 3D model data interface extended. With the sharing data flow of 3D mechanical modeling software, and combined with Workbench's friendly windows operation interface, accurate analysis and subsequent processing can be performed with Finite Element Analysis [2, 4].

In the automotive dynamic design analysis field, a plenty of scholars used combination of inertial DOF matrix equation to solve dynamic stress analysis of vehicle [5]. However, in the process of solving freedom mathematical equations, as the high rate of simplified mathematical model and approximation, the large degree of freedom and inertia matrix, the amount of calculation is large and results are often rough. While conbining frequency analyzing method [6] with FEM thinking, Using ANSYS-Workbench software for dynamic design analysis, greatly reduced the amount of calculation and the calculation results are more beautiful. Currently, some scholars has made the relevant Finite Element Analysis of vehicle parts and body[7,8]. But the combination of precise 3D modeling and ANSYS-Workbench for overall body dynamics finite element analysis, has not been reported in the relevant literature [9].

In the use of vehicles , the engine cycle incentive and uneven road surface excitation will lead motor vehicles to the forced vibration [10]. If the excitation signal frequency and the resonance frequency of the the skin structure is closed, the mechanical structure will produce local resonance and deformation [11, 12]. Meanwhile, in the process of using and installing vehicle, bolts, screws and other small gripping fasteners will generating local node stress. Nodal stress may affect the generation of local vibration behavior, which have an impact on the vehicle body. Using Workbench's Harmonic Response module can well simulate the vehicle body forced vibration under the engine cycle driven incentives, and effectively grasp the forced oscillation amplitude distortion of the vehicle, to predict and prevent the deformation and damage [13,14].

In this paper, three-dimensional structure of the auto body sheets was established by using UG NX6.0, and the modal analysis was built with using ANSYS V12.1 Workbench. Lastly the forced vibration harmonic response analysis of the body sheets under the excitation of engine cycle is superimposed calculated. The auto body sheets deformation vibration amplitude-frequency curve of various nodes loads was obtained. The further influence of node load environment on the auto body sheets vibration characteristics was investigated.

2. THE ESTABLISHMENT OF FEM MODEL

Vehicle body is a complex assembly structure,and it cannot be directly modeled.Thus the model must been discreted and simplificated. Combined with the actual using conditions of vehicle structure, the modeling of auto body sheets has been simplified in this paper. In this paper, 3D modeling of the auto body sheets was built with Unigraphics NX6.0. Then it was imput ANSYS-Workbench with the common data interface IGES, and then the Modal and Harmonic Response calculation was made.

In the modeling, the auto body sheets was extracted. The door and windscreens were removed, ignoring the local structure which has no great influence on the performance of the whole automobile vibration. For the bottom of the vehicle frame, it was simplified based on the actual installation frame. The auto body, and bottom frame part is simplified as a whole, which produced a uniform structure.

In addition, combined with the actual situation, the bottom frame, door, car mounted beam edge stiffeners or reinforced beam on the body model were added, so that the vehicle dynamic analysis is more in line with the actual vibration effect situation.

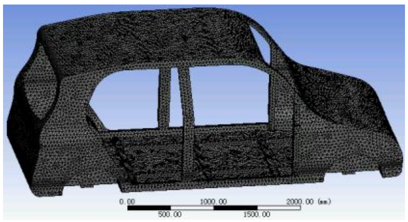

Fig.1 modal meshed by Workbench

Figure 1 is the model meshed by Workbench. Because the vehicle body is large, the model used the adaptive triangle mesh division, meshing size is 40mm, a total of 578 surface model, division consists of 222116 nodes and 114023 grid cells.

Material characteristic of experimental simulation vehicle is shown in Table 1.

Table.1 Material characteristic of experimental simulation vehicle

Main components	Elastic modulus (GPa)	Poisson's ratio	Density (kg/m^3)	Yield strength (MPa)
Structure steel	200	0.3	7850	245

3. MODAL RESULTS AND ANALYSIS

3.1 Results of the modal frequencies

In this paper, the auto body sheets model was calculated the first eight order natural modal frequency values. The calculated value of each order mode is specific shown in Table 2.

Table. 2 The first 8 natural frequency value of experimental simulation

Modal order	1	2	3	4	5	6	7	8
Frequency(Hz)	16.731	28.914	36.275	39.255	42.299	43.688	45.118	52.475

3.2 Modal shape figure analysis

Modal shape figure is the relative amplitude cloud of vibrational structure in the natural madal frequencies of Table 2, showing the characterization of the relative amplitude distribution of the structure under the natural vibration frequency. Borrowing model Animate can observed well the relatively weak area structures of the vehicle parts. By optimizing the material distribution, structure and shape of the target body can effectively avoid the resonance frequency region, thereby increasing the reliability and life of the structure [15].

This paper calculated the auto body sheets structure first 8 order mode figure, Where the typical first two vibration modes shown in Figure 2.

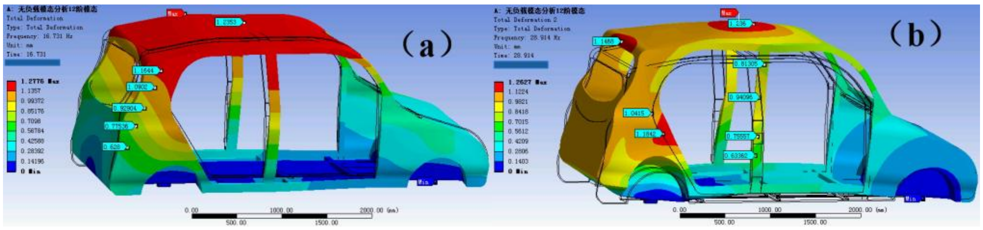

Fig.2 The modal shape of test vehicle listed (a)The first order;(b)The second order;

The black line in Figure 2 is the model contour before the deformation, and the color cloud represented each point relative amplitudes of vibration. Figure 2(a) is the first–order vibration modal figure. The vibration response of vehicle body is not obvious; maximum of lateral vibration amplitude is only 1.27760 mm; the vibration had no significant effect on the vehicle body. Thus, in actual use, the first order modal often is ignored artificially. Figure 2(b) is the second-order vibration modal figure, the body longitudinal vibration occurs; the maximum amplitude value of the roof local places is 1.26280 mm. In the condition of common idle (automobile common speed (850±50) r/min)) [16], the vehicle excitation frequency is f=(850±50)/60×2=28.33±1.67Hz. By the analysis of modal, it is close to the second order natural frequency of 28.914Hz, so the car in idling is prone to resonance, resulting to increase of amplitude, and affecting the driver and passenger comfort. Thus the two modes are most likely to be human perception.

Figure captions appear below the figure, are flush left, and are in lower case letters. When referring to a figure in the body of the text, the abbreviation "Fig." is used. Figures should be numbered in the order they appear in the text.

4. THE HARMONIC RESPONSE ANALYSIS

To further determine the structure vibration response of the engine incentive, and to study the forced vibration response law of the auto body sheets to node environment, In the basis of above modal calculation, this paper calculated the subsequent four kinds superimposed nodal loads environments harmonic response analysis. By analyzing the results, the response law of body forced vibration in node load condition is recived.

In the bottom of the vehicle front cover, engine parameters were added to motivate and simulate the forced vibration response of the vehicle body. Simulation engine chose Toyota 3VZ-FE. The maximum output torque is 195/4400Ft.Lbs/r.Min-1; maximum power is 185/5200Hp/r.Min-1. When harmonic response analysis superimposed calculated frequency is 1.5 times of the maximum of modal calculated frequency, it is possible to obtain accurate frequency response[10]. Thus this study calculated the frequency is (0~100Hz); the frequency increments is 1Hz; the number of sub-steps is 100. Damping coefficient is 0.0279; testing resistance is 260N.s / m [17].

Figure 4 is deformation vibration amplitude frequency curve of auto body sheets focus on various node load conditions. Selecting location were car roof, rear of the car, the car front windshield and the right door beams.

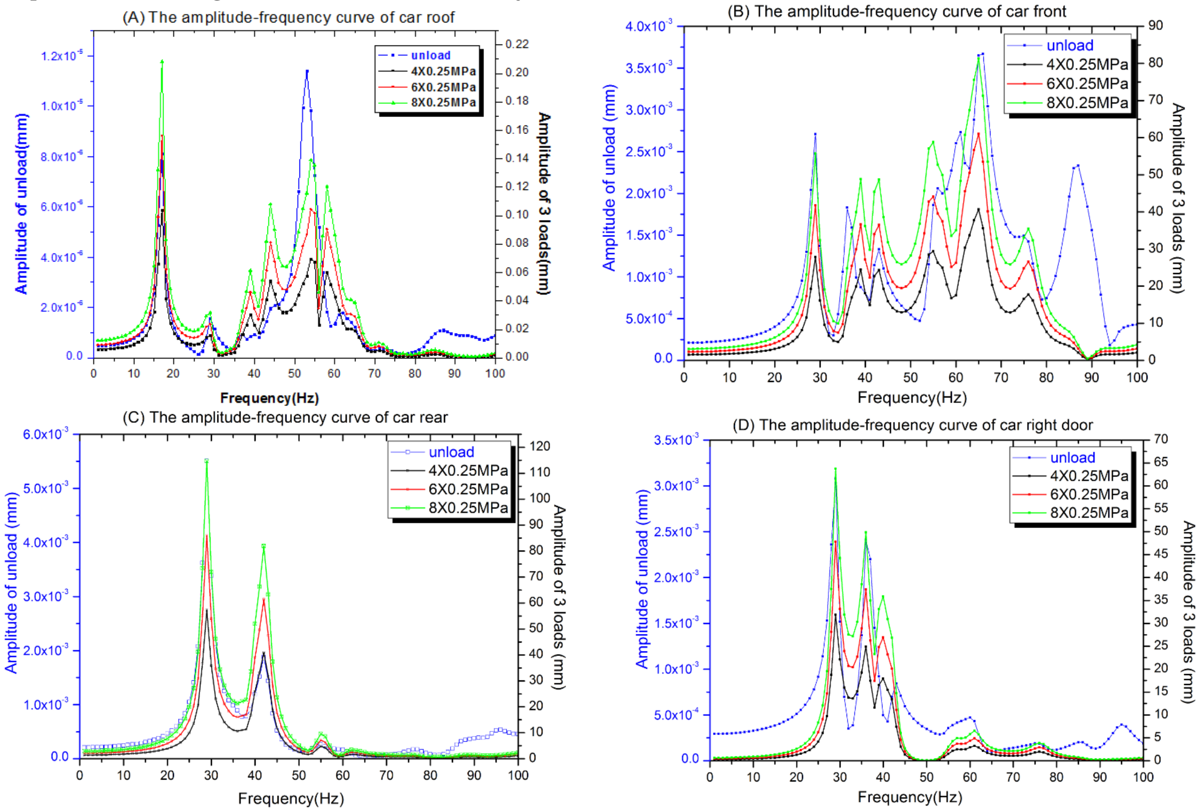

Fig. 4 Deformation vibration amplitude-frequency curve of auto body sheets focus on various node load

In Figure 4, left side blue ordinate corresponds to the various parts of the body amplitude-frequency response curve in natural conditions. The right black ordinate corresponds to the response curve of three node load environment. The abscissa is the engine converted excitation frequency. Curvature of the curve in Figure 4 shows that the beginning of curve corresponding to rotation of the engine at low speed, the vibration of the body is simple vibration forced by excitation. The forced vibration frequency is equal to the engine driving frequency, and the response amplitude of the body is small. As the engine speed increasing, the drive frequency gradually increased. The auto body sheets exhibited the multi-stage resonance, where the amplitude distortion increased sharply.

Contrasting the same part of the body in the node load and natural conditions, the response curve trend shows that the peak point of the curves corresponding to the same basic frequency, the amplitude of the response has been significantly enlarged. Figure 4 (a) shows that in natural condition, the first-order resonance amplitude of car roof is $8.14000\times e^{-6}$ mm. While in the node load environment, the response amplitude increases with the increasing of node load. When the node load is 2Mpa(8×0.25Mpa), it increased to $2.08410\times e^{-6}$ mm. Fig. 4 (c) shows that, at the rear of the vehicle, the maximum response amplitude corresponds to the second order modal, and the response frequency is 28.914Hz.The amplitude response of nodal load environment increased from $5.51000\times e^{-3}$ mm increased to $1.14650\times e^{2}$ mm, which occured an obvious resonance response. From Figure 4 (c) and Figure 4 (d), we can also reached the same conclusion that in node load condition, the auto body sheets of forced vibration compared to the natural conditions, the response amplitude increased in each modal, which the increasing is more obvious In each resonance frequency. Additionally, as the load node increased, the body edge forced vibration response amplitude increased.

Analysis shows that the node load environment will affect the auto body sheets edges reaction which are away from the node， and resulted the foced vibration increased compared to the node load points.

5. CONCLUSION

The body modal simulation was made using Ansys-Workbench,and then each order mode vibration of the auto body sheets structure is obtained. After the analysis we get that the relatively weak point of auto body sheets structure stiffness is mainly the installation beam door parts. In the design of the actual skin structure, it should be reinforced and optimize the door beam structure, to improve the structural stability of the parts. On the basis of modal analysis, superimposed various nodes load environments harmonic response analysis, the frequency response curve of auto body sheets forced vibration under driving torque of the engine is obtained, and it more accurately reflects the auto body sheets structure forced vibration response to the node load environment.

Through the comparison body dynamics response results of natural and node load environment, obtained the influence dynamics response rule of bottom node load environment on the whole auto body sheets structure. The forced vibration under the engine sinusoidal excitation, The node load will increase the edge body esonance response amplitude, thereby increasing the the damage or distortion probability of the parts. Thus in body design due to the effect of bolts, screws,etc, the result of node load environment on the auto body sheets structural vibration response need to be considered. Consequently, in the design, we should optimize and rational distribution of node load, and increase the vibration resistance of relatively weak position, to prevent excessive vibration and damage is indispensable.

Combined harmonic response analysis curve, and comparing the amplitude of the vibration of the same location under the same frequency, it can distinguish the vibration objectives from different structures and nodal load environment, and preliminary plays a reference role for automotive vibration information measurements and identification.

6. ACKNOWLEDGEMENTS

This research was financially Supported by the National Natural Science Foundation of China (60890203, 10676023),Fund project of National Key Laboratory of solid state laser,China.

REFERENCES

[1] Lin, N. I., Z. H. O. N. G. Hui, and D. U. A. N. Chao. "Research on RFID Application Mode in Automotive Manufacturing Production Line." *Computer Engineering* 4 (2012): 074.

[2] Wang, Gang, and Feng Sun. "FEA and optimize design of a new type structure of FOG." *Mechanical and Electronics Engineering* (ICMEE), 2010 2nd International Conference on. Vol. 1. IEEE, 2010.

[1] Kong, Xiang Wei, et al. "Finite Element Analysis of Sieve Box Structure in Fluidized Bed." *Advanced Materials Research* 118 (2010): 497-501.

[2] Gao, Dong Qiang, et al. "Research on the Application of Periodic Truss-Core Structures to Structural Design of Machine Tool." *Advanced Materials Research* 194 (2011): 1977-1981.

[3] [5] Wanqiong, Guan Fei Wang Lie Li, and Feng Benyi Li Baocheng. "A SIMPLIFIED MODEL OF A WHOLE VEHICLE AND DYNAMIC STRESS ANALYSIS OF A VEHICLE FRAME." *Chinese Journal of Applied Mechanics* 2 (1985): 006.(in Chinese)

[4] (Japan) Togawa Hayato,Yin Yin dragons. Vibration analysis of the finite element method [M]. *Seismological Press*, 1985:1-5.

[5] Shao, Kang Li, Feng Wang, and Yong Hai Wu. "Modal and Vibration Analysis of a Trator Frame Based on FEM." *Applied Mechanics and Materials* 373 (2013): 16-19.

[8] Senthil Kumar, M., et al. "Vibration Analysis and Improvement of a Vehicle Chassis Structure." *Applied Mechanics and Materials* 372 (2013): 528-532.

[9] ZHANG Yang, GUI Liang-jin, FAN Zi-jie. Dynamics Modeling and Random Response Analysis for Bus Body Structure. *Automotive Engineering*, 2008,(12):1091-1094.(in Chinese)

[10] Ferreira, P. A., and A. López-Pita. "Numerical Modeling of High-Speed Train/Track System to Assess Track Vibrations and Settlement Prediction." *Journal of Transportation Engineering* 139.3 (2012): 330-337.

[11] XIA Guo-lin, ZHANG Dai-sheng. Analysis of BIW and Its Shape Correlation Research [J].*Auto Mobile Science &Technology*, 2008(10):39-43.

[12] Yang, Yang, et al. "Shock wave impact simulation of a vehicle occupant using fluid/structure/dynamics interactions." International Journal of Impact Engineering (2012).

[13] Zhang, Xiaopeng, and Zhan Kang. "Topology optimization of damping layers for minimizing sound radiation of shell structures." *Journal of Sound and Vibration* (2013).

[14] Luo, Ji Man, Yang Jiang, and Zhi Hui Xing. "Modal Analysis of Mast Section of Hoist Based on the Vibration Theory." *Advanced Materials Research* 457 (2012): 150-155.

[15] ZHOU Chang-lu, FAN Zi-jie, CHEN Zhong-yun, et al. Modal Analysis for Body-in-White of a Mini Bus. *Automotive Engineering*, 2004(1):78-81. (in Chinese)

[16] ZHOU Ding-lu. Comparison of the FEM Modal Analysis and Test Modal Analysis for Body of Mini-Truck. *MSCsoftware Chinese user papers*,2002. (in Chinese)

[17] WEI Zi, CHeng Yao-dong. Analysis on effcets of engine to vhicle. *Journal of Vibration and Shock*,1997, (04):74-80. (in Chinese)