A Study of the Dynamic Characteristics of Transport Bus Frame using Finite Element Analysis

Shuli Sun*1, Jun Zhang1, Zhenhao Peng1

1 School of Engineering, Zhejiang University City College, Hangzhou, Zhejiang, 310015, P.R.China

Abstract — The topic carries on the analysis in order to get the static and dynamic characteristics of the bus frame, and do the lightweight research. In this paper, based on the finite element theory, the finite element model of the bus frame is established using the finite element software ANSYS14.5. Static analysis mainly analyzes the bus frame under full load and bending condition, so as to get the deformation of the frame, and the stress distribution. About modal analysis, the analyze analysis of the first ten order natural frequency and vibration mode about the frame to avoid the resonance of the chassis. About lightweight, mainly optimization is in beam thickness.

Keywords - Frame; Finite element; Static analysis; Modal analysis

□. INTRODUCTION

The frame is the Skeleton of the car. The static and dynamic performance of the frame is not only affected the occupant comfort, vehicle vibration characteristics, but also directly affect the safety of the car. In addition, the performance analysis of the frame is also provided a technical reference for the structural optimization design of the automobile frame [1].

The finite element analysis method is an effective and efficient tool in modern engineering design. The finite element analysis method is applied in the design of automobile structure. It can help to improve the static and dynamic characteristics of the vehicle, and can optimize the structure of the body and the frame, shorten the development cycle and save the cost[2].

□. FRAME FINITE ELEMENT MODEL

In this paper, the finite element model can choose beam element, plate element, shell element and solid element. For the beam element it can simplify the finite element model of the frame, the utility model has the advantages: model is simple, pre-processing work less, than the shell and solid elements are more efficient, and beam element is regular, constitute model unit number and the number of nodes are relatively less.

Therefore, the computational speed is fast, but it often can’t accurately reflect the section shape and geometry of frame beam, and will not be able to accurately calculate the stress concentration.

The defects of the plate element is the node with only three degrees of freedom, unable to accurately simulate the tensile, compression, bending, torsion, shear and so on of longeron and crossbeam. Therefore, the plate element cannot simulate various working conditions of the frame. Solid element can be used to solve the problem of beam element, and the stress concentration in the model can be refined to get a more accurate finite element model, but the size of the problem solving data is very large, it may be cause the analysis fail [3,4].

In view of the bus frame is consist of longitudinal beams and cross beams, they are all thin-walled structure, which can guarantee frame finite element model can be a very good approximation frame solid model and a reasonable amount of computation, in computing scale between computational accuracy and can be a very good balance. So this paper chooses the SHELL181 element to discrete frame structure provided by ANSYS 14.5.
SHELL181 is suitable for the analysis of thin to medium thickness of shell structure. It is the 4 node element with 6 degrees of freedom for each node. The 6 degree of freedom refers to the displacement of X, Z, Y axis and the rotation angle of the axes around X, Y, Z (if the selected film is the unit, the unit has only displacement degrees of freedom). Unit properties as shown in Figure 1.

The feasibility and reliability of the finite element analysis is not only related to the accuracy of the model, but also has a close relationship with the time and efficiency of calculation, and the time and efficiency of calculation are closely related to the size and the complexity of the model. Therefore, in the establishment of the finite element model, it is necessary to simplify and process the model according to the specific conditions of the task [3].

The frame is 12.6 meters long, 2.2 meters wide. Wheelbase is 5736mm. Quality is about 7236kg when full load on the beam. And constraint conditions of the frame: front suspension is 2512mm (distance to the front end of the frame is 2512mm) and front axle support (2 constraints); rear suspension is 4350mm (r distance to the back end of the frame is 4350mm) and steel plate spring support (4 constraints). Front and rear suspensions are no rotational restraint. The specific constraints are shown in Figure 2.

The bus engine mounted at rear, frames consist of two longitudinal beams, 9 crossbeams, 19 edge beams. Concrete structure as shown in Figure 2. Longitudinal beams adopt Z-shaped beam, and crossbeams and edge beams on the left side of the 4, 5, 8 and on the right side of the 4, 7 is Box girder, other beams were U-shaped beam, the frame adopts the material is 16Mn.

The load on the frame is shown in Table 1.

<table>
<thead>
<tr>
<th>Name</th>
<th>Quality /kg</th>
<th>Name</th>
<th>Quality /kg</th>
</tr>
</thead>
<tbody>
<tr>
<td>Body</td>
<td>1980</td>
<td>The passenger seat and full house</td>
<td>2442</td>
</tr>
<tr>
<td>Luggage box and luggage</td>
<td>1200</td>
<td>Engines and their accessories</td>
<td>547</td>
</tr>
<tr>
<td>Direction machine and its mechanism</td>
<td>128</td>
<td>Oil tank and oil</td>
<td>260</td>
</tr>
<tr>
<td>The spare tire and its bracket</td>
<td>78</td>
<td>Transmission and clutch</td>
<td>228</td>
</tr>
<tr>
<td>Reservoir pump and pipeline</td>
<td>168</td>
<td>Water tank and water</td>
<td>150</td>
</tr>
<tr>
<td>Battery</td>
<td>35</td>
<td>Air filter</td>
<td>20</td>
</tr>
</tbody>
</table>

### FRAME FINITE ELEMENT ANALYSIS

The load on the frame is shown in Table 1.

<table>
<thead>
<tr>
<th>Order number</th>
<th>Frequency /HZ</th>
<th>Order number</th>
<th>Frequency /HZ</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5.1589</td>
<td>6</td>
<td>19.782</td>
</tr>
<tr>
<td>2</td>
<td>8.5396</td>
<td>7</td>
<td>19.818</td>
</tr>
<tr>
<td>3</td>
<td>10.297</td>
<td>8</td>
<td>22.231</td>
</tr>
<tr>
<td>4</td>
<td>14.453</td>
<td>9</td>
<td>24.063</td>
</tr>
<tr>
<td>5</td>
<td>16.126</td>
<td>10</td>
<td>24.794</td>
</tr>
</tbody>
</table>
According to the technical parameters of the frame, and selecting the appropriate mesh size to establish the model. Depending on the location and load parameters in Table 2 and Figure 2. Then, get the model shown in figure 3.

The model consists of 46800 nodes and 45447 units after the completion of the pre-processing. Each unit is roughly 20mm*20mm. The results obtained are as follows:

Frame displacement and rotation are as follows:

The stress of the frame is as follows:
According to the strain diagram of the frame, the deformation of the front part of the frame is very small, and the maximum deformation occurs in the rear of the frame, $\delta_{\text{max}} = 1.4\text{mm}$. The deformation amount is in accordance with the frame.

From the displacement view of the frame, the whole model has little deformation. The maximum displacement of the whole frame is 30mm, which is in the position of the engine mounting at the end of the frame.

From the frame stress analysis results, full of bending and torsion condition of high stress level, the maximum equivalent stress occurs in the side of the plate spring support, it's in the installation position of the device, and the maximum stress is 226Mpa. Compared to the strength limit of the material can know that the strength of the frame meet the strength requirements.

### FRAME FINITE ELEMENT MODAL ANALYSIS

The modal analysis is basically consistent with the static analysis on the constraint and load. The difference is that the modal analysis must input the material density, and the static analysis is unnecessary to enter. Table 2 is the first 10 frequency of the bus frame extracted by the block Lanczos method of the modes analysis.

The results of modal analysis are as follows:
From the first to tenth order modal vibration shape diagram of the bus frame can be seen, tail beam and side beam in the XZ plane transverse swing is the main form of the first order vibration of frame, and it accompanied by around the X axis of the torsion; The main form of the second order vibration of frame is the longitudinal oscillation of the end of frame in the XY plane; the main form of the third order vibration is similar to the first order; The fourth order vibration is front part of the longitudinal beam and crossbeam in the XZ plane transverse swing mainly, but also with the XY plane displacement and torsion along the Z axis direction; The frequency of the fifth order vibration is 15.920Hz, the vibration form is the upper and lower of the front part of the frame in the XY plane, and it has the deformation in the middle part; Sixth order vibration frequency is 17.372Hz, the main form of vibration is to reverse the tail of the frame around the X axis direction of the longitudinal beam; The vibration form of the seventh order and eighth order is similar to that of the sixth order, and the seventh order and eighth order have the form that the middle of the frame are rotated around the X axis, and the torsion degree is more than sixth; The ninth order vibration shape is the front part of the frame and central around the X axis rotation; the tenth order vibration form is rear longitudinal beam in the XY plane transverse swing, the direction of two longitudinal beams’ swing is opposite.
V. CONCLUSIONS

According to the results of the modal analysis, the following schemes are put forward to improve the performance of the frame:

A. Increase the number of beam on the back part of the frame, in order to enhance the longitudinal beam’s torsional capacity;

B. Increasing the supporting area of the engine to reducing the stress concentration;

C. Increase the longitudinal section’s thickness; enhance the ability of its bending and torsion.

After changing longitudinal thickness, each order of natural frequency occurred some changes and after longitudinal beam thickness is increased, compared with the original, each order natural frequencies are on the rise. Before the sixth order, lifting of the scheme on the frame’s performance is significant, because of first order mainly as a rigid body deformation, so change the longitudinal thickness can enhance the stiffness. But this scheme has no significant effect on the performance of high order. The reason is that the deformation of the high order is not only a rigid body deformation, it cannot be improved. For higher order deformations it should be improved by other means.

REFERENCE


