DYNAMIC ANALYSIS OF AUTOMOTIVE CHASSIS USING FEA

Kallappa Khannukar¹, Vinayak Kallannavar², Dr. B. S. Manjunath³

¹ Assistant Professor, Mechanical Engg. Dept, MMEC Belgaum, Karnataka, India
² Trainee Engineer, Quest Global Engineering Pvt. Ltd, Karnataka, India
³ Professor, Mechanical Engg. Dept, KLE’s Dr. MSSCET Belgaum, Karnataka, India

Abstract – Chassis provide the structural integrity to automotive vehicles, where as it is also important to have the stiff and light weight structure. The present study lay the emphasis on dynamic analysis of chassis. The modal analysis is carried out using both analytical and FEA techniques, and the results were found to be in good agreement with each other. Also the chassis structure is optimized by varying the design parameters with the help of acceleration response of the system. It was observed that the acceleration levels were reduced to 20.24% by increasing the thickness of ribs and horizontal channel to twice of their initial dimensions.

Key Words: FEA, Hypermesh, Chassis, Stiffness.

1. Introduction

Chassis is the structural backbone of a passenger vehicle. When a vehicle travels, the chassis experience the dynamic forces due to the road roughness, vibrations of engine, transmission systems etc. Due to the dynamic excitations, chassis will tend to vibrate and can lead to ride discomfort, ride safety and stability problems. It is difficult to mathematically model and solve these mechanical problems by analytical techniques as they include complex nonlinear boundary conditions. This difficulty is overcome by seeking some form of numerical solutions. The finite element method (FEM) is the most appropriate and accurate numerical technique to solve the compound problems.

In past few decades many academicians, researchers have worked to analyze and understand the dynamic nature of the vehicle chassis using numerical, analytical and experimental methods under various loading conditions [1-4]. The sensitivity analysis is carried out by Hirak. P et al. [5] on the truck chassis to optimize the weight. They achieved 17% of weight reduction in the truck chassis by analyzing it at different sections. Dr. R. Rajappan et al. [6] simulated the dynamic characteristics of truck chassis using ANSYS FEA tool. The mode shapes and natural frequencies at mounting locations of the chassis are studied and the modified design is proposed. S. Ganesan et al. [7] carried out modal analysis of bus chassis using different shapes of reinforcement plates to increase its stiffness. The present work addresses the optimization of ladder type chassis under dynamic loading conditions.

2. Finite Element Analysis

The CAD model of the chassis is prepared in Catia V5 modeling environment as shown in Fig -1 and MSC Nastran simulation tool is used for the FEA analysis. For the ease of analysis the ladder type chassis is divided into subpart like side rails, C-channels, horizontal channels, cross members, and brackets as shown in Fig -2. Steel AISI 1015 is used as the chassis material and its material properties are listed Table 1.

![Fig -1: CAD model of the chassis](image)

2.1 Modal Analysis

The modal analysis is the field of measuring and analyzing the dynamic response of structure when excited by an input disturbance. Modal analysis of the system is closely associated with the natural frequency of the system. Stiffness (k) and mass (M) of the system are the deciding factors for the natural frequency and can be given by equation 1,

\[ \omega_n = \frac{1}{2\pi} \sqrt{\frac{k}{M}} \]

As the stiffness of system depends on the arrangement of the components (Series/Parallel), the stiffness of individual components are calculated and listed in Table 2.

**Table -2: Name of the Table**

<table>
<thead>
<tr>
<th>SL No</th>
<th>Natural frequency</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>5.173275x10^{-04}</td>
<td>Rigid body translation along x – axis</td>
</tr>
<tr>
<td>2</td>
<td>3.474783x10^{-04}</td>
<td>Rigid body translation along y – axis</td>
</tr>
<tr>
<td>3</td>
<td>2.004669x10^{-04}</td>
<td>Rigid body translation along z – axis</td>
</tr>
<tr>
<td>4</td>
<td>1.759311x10^{-04}</td>
<td>Rigid body rotation along x – axis</td>
</tr>
<tr>
<td>5</td>
<td>3.893044x10^{-04}</td>
<td>Rigid body rotation along y – axis</td>
</tr>
<tr>
<td>6</td>
<td>5.696454x10^{-04}</td>
<td>Rigid body rotation along z – axis</td>
</tr>
<tr>
<td>7</td>
<td>7.931197x10+00</td>
<td>Torsional mode along x-axis</td>
</tr>
<tr>
<td>8</td>
<td>1.599375x10+01</td>
<td>Torsional mode along y-axis</td>
</tr>
<tr>
<td>9</td>
<td>2.364815x10+01</td>
<td>Bending mode along x – axis</td>
</tr>
<tr>
<td>10</td>
<td>2.456802x10+01</td>
<td>Bending mode along y – axis</td>
</tr>
<tr>
<td>11</td>
<td>3.060098x10+01</td>
<td>Torsional mode along z-axis</td>
</tr>
<tr>
<td>12</td>
<td>4.08740x10+01</td>
<td>Torsion + Bending mode along z – axis</td>
</tr>
<tr>
<td>13</td>
<td>9.04825x10+01</td>
<td>Torsion + Bending mode along x – axis</td>
</tr>
</tbody>
</table>

From Table 2, Equivalent stiffness Keq = 136.98x10^6 N/m

Total mass of the chassis = 438 Kg
Substituting these values in equation 1,

\[ \omega_n = \frac{1}{2\pi} \sqrt{\frac{136.98 \times 10^6}{438}} = 89Hz \]

The finite element model was generated using HYPERMESH which includes ~88500 elements, and the free-free boundary condition was adopted in order to obtain the natural frequencies and mode shapes.

There were thirteen natural frequencies calculated for normal mode analysis and are tabulated in Table -2. 23rd natural frequency was near to the value calculated by analytical method. Normally, the operating frequency is always related to dynamic forces induced by road roughness, bumps, engine, transmission and many more. Each of these forces has its own excitation frequency. Fig - 3 shows the mode shapes obtained for some of the natural frequencies of the chassis.
Fig -3.6: Mode shape at 40.39 Hz

From the above results it is observed that natural frequency calculated by analytical method (89 Hz) is almost close to the 23rd natural frequency i.e., 90.48 Hz. Table 3 shows result comparison of analytical & FEA methods.

<table>
<thead>
<tr>
<th>Sl. No</th>
<th>Natural frequency (Hz)</th>
<th>Percentage difference</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Theoretical</td>
<td>FEA</td>
</tr>
<tr>
<td>1</td>
<td>89</td>
<td>90.48</td>
</tr>
</tbody>
</table>

### 2.2. Transfer Path Analysis

Transfer path analysis (TPA) is used to assess the structure borne energy paths between excitation source and receiver location. Transfer path analysis is a systematic method to understand the relation between multiple sources of noise and vibration and their effect on perceived user comfort and health.

#### 2.2.1 Acceleration Response of Chassis

Acceleration response is the acceleration levels of the chassis when excited by external force. From the results it was observed that maximum acceleration was at the horizontal channel. Fig -4 shows the fringe plot of the chassis when excited by unit acceleration and Fig -5 illustrates the acceleration response of the chassis at node 522127 where acceleration is maximum.
The acceleration Vs frequency plot recorded in Fig -5 shows the maximum acceleration of 27.48 mm/sec² at a frequency of 162 Hz. The effort has been made to enhance the attenuation characteristics of the chassis by making small changes in the design parameters.

**a. Increasing thickness of horizontal channel**

Thickness of the horizontal channel was 5 mm, to improve its vibratory characteristics its thickness was doubled as shown in Fig -6.1. From Fig -6.2 it is evident that the maximum acceleration value for the present load case is 21.22 mm/sec².

**b. Addition of ribs along long rail members and horizontal channel only**

The ribs of 5 mm thickness were added on the horizontal channel and long rail members (Fig -7.1). Reduction in acceleration level of 12.78% is observed (Fig -7.2).

**c. Addition of ribs along other C-channel sections**
Ribs were added along C-channels of the chassis as shown in Fig -8.1. Reduction in acceleration level of 13.14% is observed for the present loading condition (Fig -8.2).

**Fig -8.1:** Addition of ribs along other C-channel sections

**Fig -8.2:** Acceleration response comparison of modified & original model

**d. Increasing the number of ribs to 3 on horizontal channel only**

To reduce the acceleration levels three ribs of 5 mm thickness were added to the horizontal channel (Fig -9.1). 12.27% of reduction in acceleration level is evident from Fig -9.2.

**Fig -9.1:** Increasing the number of ribs to 3 on horizontal channel only

**Fig -9.2:** Acceleration response comparison of modified & original model

**e. Increasing thickness of ribs to twice of initial thickness**

To study the effect of thickness of ribs on horizontal channel thickness of the rib was doubled. Present load case was solved with increasing the thickness of the rib to twice that of initial thickness (Fig -10.1). Thickness maintained during this load case was 10 mm. Reduction in acceleration level of 7.03% is observed from Fig -10.2.

**Fig -10.1:** Increasing thickness of ribs to twice of initial thickness
f. Increasing thickness of ribs and horizontal channel to twice of that initial section

Present load case determines the combined effect of thickness of rib and horizontal channel on the acceleration levels of the chassis. In the present load acceleration levels of the chassis was determined by increasing the thickness of ribs and horizontal channel. From Fig -11.2 it is apparent that reduction in acceleration level of 20.24% is achieved.

3. CONCLUSIONS

In the present study dynamic characteristics of a chassis like natural frequency and mode shape are evaluated by finite element analysis. Values of natural frequency obtained by finite element method were in good agreement with analytical results. Acceleration responses of the chassis excited by road surface were analyzed. From the results it was observed that by increasing thickness of ribs and horizontal channel to twice of that initial section acceleration levels were reduced by 20.24%.

REFERENCES