Design and Finite Element Analysis of the Main Frame of Intelligent Fire Truck

Bai-lin FAN*, Xue-dao CHENG and Fan CHEN
School of Mechanical Engineering, University of Science and Technology Beijing,
30 Xueyuan Road, Haidian District, Beijing, China

*Corresponding author

Keywords: Intelligent fire truck, Main frame, Static analysis, Modal analysis.

Abstract. According to the parameters given, the main frame design of the intelligent fire truck is carried out, and the three-dimensional model is established. It is transformed into a general format by the established 3D model, and the finite element model is established. The static and mechanical analysis of the finite element model by ANSYS is performed to calculate whether the stress and deformation meet the strength and stiffness requirements; See if resonance occurs and what the mode shape is if resonance occurs.

Introduction

Fire is a disaster faced by human society[1]. Industrial fires (especially those that occur in petrochemicals) are characterized by sudden bursts of fire, large fires, large amounts of toxic gases in the combustion process, and difficulty in extinguishing[2, 3]. Through research and inspection of relevant data, it is found that all the fire trucks currently on the market are fired by connecting corresponding large fire trucks or water pipes, and there is no fire truck using a foam fire extinguisher for firefighting[4-7].

The independent driving of the intelligent motor of the hub motor has a great advantage in terms of energy or performance compared with the conventional automobile[8], but the structural size of the vehicle chassis changes greatly due to the structural influence of the hub motor. In addition, the use of the special smart car must have a large impact force when it starts to work. Therefore, it is required that the frame must have sufficient strength and rigidity to withstand the quality of each assembly of the vehicle and various loads generated during driving[9], and should have reasonable dynamic characteristics to control vibration[10].

In this paper, the static analysis and modal analysis of the frame under various working conditions are carried out by using the finite element ANSYS software. The corresponding stress and strain conditions and vibration conditions are obtained through these analyses, which provides a theoretical basis for the design and analysis of the special vehicle[11-14].

Finite Element Analysis of the Main Frame

Establishment of Finite Element Model of Main Frame

The three-dimensional model of the frame was established by SOLIDWORKS. On the basis of ensuring the main mechanical characteristics of the frame structure, the model was simplified in order to facilitate the next finite element analysis, that is, the process hole was omitted. The out and the chamfers are all replaced with right angles.

The three-dimensional finite element model of the main frame was established by Workbench in ANSYS software. The main frame adopts a tetrahedral mesh, the entire main frame model is divided into 1199716 nodes and 621331 units. The material of the main frame is Q235, and the specific material parameters are shown in Table 1.
Table 1. Q235 material parameters.

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Q235</td>
<td>200[GPa]</td>
<td>0.3</td>
<td>235[MPa]</td>
<td>7850[\text{Kg/m}^3]</td>
</tr>
</tbody>
</table>

The finite element model of the main frame is shown in Figure 1.

Figure 1. Finite element model of the main frame.

Static Analysis under Six Working Conditions

Bending Conditions

The analysis of the bending condition is the basic analysis condition of the finite element static analysis of the frame. It is mainly to simulate the force analysis of the vehicle during normal straight-line straight driving or when it is forbidden. The finite element model is calculated in ANSYS to obtain the corresponding stress and deformation.

Analysis of Torsion Conditions

The torsion condition analysis is also the basic analysis condition for the static analysis of the frame. It simulates the vehicle traveling on an uneven road. The finite element model is calculated in ANSYS, and the corresponding stress and deformation are obtained. The stress and deformation cloud diagram under this condition is shown in Figure 2 and Figure 3.

Analysis of Sharp Turn Conditions

The analysis of the sharp turn condition is mainly to simulate the situation that the vehicle needs an emergency turn when encountering an unexpected event during driving. The finite element model is calculated in ANSYS to obtain the corresponding stress and deformation.

Analysis of Emergency Braking Conditions

The analysis of the emergency braking condition is mainly to simulate the situation that the vehicle needs emergency stop when encountering special occasions during driving. The finite element model is calculated in ANSYS to obtain the corresponding stress and deformation.
Analysis of Accelerated Conditions

The acceleration condition analysis is mainly used to simulate the force and deformation of the vehicle in the case of sudden acceleration. The finite element model is calculated in ANSYS to obtain the corresponding stress and deformation.

Analysis of Launch Impact Conditions

The analysis of the launching impact condition is specifically for the analysis of the force and deformation of the frame during the firefighting of the special vehicle during the operation. The finite element model is calculated in ANSYS, and the corresponding stress and deformation are obtained. The stress and deformation cloud diagram under this condition is shown in Figure 3 and Figure 4.

![Figure 4. Stress cloud diagram of the launching impact condition.](image)

![Figure 5. Deformation cloud image of launching impact condition.](image)

The stress and strain data under these six working conditions are shown in Table 2.

Table 2. Stress and deformation of each working condition.

<table>
<thead>
<tr>
<th>Working condition</th>
<th>Maximum stress Location</th>
<th>Maximum stress value [MPa]</th>
<th>Maximum deformation Location</th>
<th>Maximum deformation Quantity [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bending condition</td>
<td>The junction of the middle side beam and the vertical beam supporting the suspension longitudinal beam</td>
<td>20.586</td>
<td>Middle of the main longitudinal beam and the two main beams</td>
<td>0.1154</td>
</tr>
<tr>
<td>Torsion condition</td>
<td>The junction of the side beam at the left rear wheel and the vertical beam supporting the suspension stringer</td>
<td>23.792</td>
<td>The side beam of the left front wheel and the vertical beam supporting the sub-frame on the left front wheel</td>
<td>0.1747</td>
</tr>
<tr>
<td>Sharp turn condition</td>
<td>The junction of the middle side beams</td>
<td>15.879</td>
<td>Middle of the main longitudinal beam and the two main beams</td>
<td>0.0986</td>
</tr>
<tr>
<td>Emergency brake condition</td>
<td>The junction of the middle side beams and the bracket of the rear wheel mounting suspension</td>
<td>18.012</td>
<td>Middle of the main longitudinal beam and the two main beams</td>
<td>0.1125</td>
</tr>
<tr>
<td>Accelerate</td>
<td>The junction of the middle side</td>
<td>17.987</td>
<td>Middle of the main</td>
<td>0.1124</td>
</tr>
</tbody>
</table>
It can be seen from the table that the maximum deformation under various working conditions occurs in other places under torsional conditions, and other working conditions all occur on the middle main beam, so the design of two main beams in the middle of the design to reduce the overall deformation is reasonable and necessary, and the maximum stress is basically occurred in the welding of the middle side beams. Here, the design is also the stress concentration. In summary, the overall design is reasonable.

**Main Frame Modal Analysis**

When the vehicle is driving on the road, the vehicle will vibrate due to the unevenness of the road surface. When the frequency is just the same as the natural frequency of the vehicle, resonance will occur. Because the resonance will cause great damage, it needs to pass. Modal analysis to obtain the natural frequency and vibration mode of the main frame, compare and analyze whether the frequency of each mode of the main frame is in the resonance zone. For example, in the resonance zone, it is necessary to optimize the structural design of the main frame to avoid it. Resonance occurs. The modal results of the analyzed main frame are shown in Table 3. The first-order mode is shown in Figure 6.

![Figure 6. First-order mode.](image)

<table>
<thead>
<tr>
<th>Order</th>
<th>Frequency [Hz]</th>
<th>Mode shape</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>162.46</td>
<td>Reverse</td>
</tr>
<tr>
<td>2</td>
<td>179.03</td>
<td>Bending and twisting combination</td>
</tr>
<tr>
<td>3</td>
<td>235.01</td>
<td>Reverse</td>
</tr>
<tr>
<td>4</td>
<td>239.16</td>
<td>Reverse</td>
</tr>
<tr>
<td>5</td>
<td>285.57</td>
<td>Bending</td>
</tr>
<tr>
<td>6</td>
<td>301.84</td>
<td>Reverse</td>
</tr>
<tr>
<td>7</td>
<td>320.19</td>
<td>Reverse</td>
</tr>
<tr>
<td>8</td>
<td>320.28</td>
<td>Bending and twisting combination</td>
</tr>
<tr>
<td>9</td>
<td>347.43</td>
<td>Bending and twisting combination</td>
</tr>
<tr>
<td>10</td>
<td>353.36</td>
<td>Bending and twisting combination</td>
</tr>
</tbody>
</table>

Since the excitation caused by the operation of the driving motor is relatively small, the resonance is generally caused by the external excitation caused by the unevenness of the road surface, and the excitation due to the road surface is generally between 1 Hz and 20 Hz, and the minimum frequency of the main frame. Both 162.46 Hz are much larger than the external excitation, so the main frame does not resonate.

According to the design scheme of the intelligent fire truck, other components of the intelligent fire truck were also trial-produced and assembled as shown in Figure 7. The results show that the frame of the smart fire truck prototype meets the predetermined strength and stiffness requirements, which proves the rationality of the structural design.
Conclusions

Through the design of the intelligent fire truck main frame structure and finite element analysis. Firstly, the stress and deformation of the main frame can be obtained by static analysis. From the analysis results, it can be known that the design of the main frame meets the requirements, and the strength and stiffness also meet the requirements. Then, the natural frequency and vibration mode of the main frame are obtained through modal analysis. It can be seen from analysis and comparison that the main frame does not resonate, so the overall design of the main frame is reasonable. Provide a theoretical basis for the next production.

References


