Finite Element Analysis of Automobile Steering Knuckle Based on ANSYS

Xuelong Yuan a, Minglong Li
College of Transportation, Shandong University of Science and Technology, Qingdao 266590, China.
a 2247693618@qq.com

Abstract: Automotive steering knuckle is one of the main parts of the bridge, its function is to bear the front load of the vehicle, to support and drive the front wheel to rotate around the main pin and make steering. Under the condition of vehicle driving, it bears a variety of impact loads, so it requires high strength requirements. In this paper, the steering knuckle is designed and analyzed, and the three-dimensional parts drawing is completed by CAD software. Then, the steering knuckle model is introduced into the ANSYS software. The strength and stress analysis of the steering knuckle under sideslip and emergency braking and two combinations of three conditions are carried out to get the finite element analysis results under each working condition, and the technical scheme for optimizing the steering knuckle is put forward.

Keywords: steering knuckle, finite element, ANSYS, optimization design.

1. Introduction
The finite element analysis is a common method to analyze the stress strength of the structure and is also widely used in the stress analysis of the automobile parts. Because this analysis method uses the convenient general program, and the calculation precision is high, the analysis results also become the basis of some product design and performance analysis, and the software is easy to operate and suitable for the analysis. Simulation analysis of parts.

2. Finite Element Analysis of Automobile Steering Knuckle
2.1 3D Solid Modeling of Steering Knuckle
The steering knuckle is one of the main parts of automobile, its main function is to load the front of the car, support and drive the front wheel steer around the drive, the car driving stability and sensitive transfer to the structure, as shown in Fig.1.
2.2 Finite Element Analysis

2.2.1 Calculation of Use Parameters

According to the specific parameters of the car, the parameters used in the calculation are shown in Table 1.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Sign</th>
<th>Numerically</th>
</tr>
</thead>
<tbody>
<tr>
<td>Front axle load</td>
<td>G</td>
<td>15371.202N</td>
</tr>
<tr>
<td>Wheelbase</td>
<td>L</td>
<td>2530mm</td>
</tr>
<tr>
<td>Centroid height</td>
<td>hg</td>
<td>90mm</td>
</tr>
<tr>
<td>Front wheel wheelbase</td>
<td>B</td>
<td>1480mm</td>
</tr>
<tr>
<td>Rim diameter</td>
<td>r</td>
<td>381mm</td>
</tr>
<tr>
<td>Dynamic load coefficient</td>
<td>kd</td>
<td>2.1</td>
</tr>
<tr>
<td>Centroid to wheelbase</td>
<td>b</td>
<td>1375mm</td>
</tr>
<tr>
<td>Lateral sliding coefficient</td>
<td>φ</td>
<td>1</td>
</tr>
</tbody>
</table>

2.2.2 Finite Element Analysis of the Single Working Condition of the Steering Knuckle

When the car in the driving state, it also bears the impact of changing loads, which requires the steering knuckle has high strength, so it has practical significance for finite element analysis of steering section.

The steering knuckle of the automobile requires sufficient toughness and strength to ensure its stability. The material is 42CrMo, the density is 7850kg/m3, and the yield strength is 930Mpa. The 3D model of the steering knuckle is inserted into the ANSYS software and divided into 26089 nodes and 13763 units by 0.5mm unit. As shown in Fig.2.
When the car is sideslip, take left turning and side slip as an example. The steering knuckle is subjected to vertical reaction and lateral reaction, and a torque is generated when the two forces are shifted. The upper end of the steering knuckle is connected with the shock absorber bolt hole and the lower end of the fixed platform is mainly subjected to the torque. The lateral force and vertical back force of the left front wheel are much larger than that of the right front wheel.

Vertical force:

\[ F_{vl} = \frac{G\left(\frac{2h_{sl}Q}{B} + 1\right)}{2} \]

Lateral force:

\[ F_{yl} = F_{v} \phi \]

Extra torque:

\[ M_{R} = rF_{yl} \]

Substituting numerical value \( F_{zl} = 8339.92 \text{N} \); \( F_{yl} = 8339.92 \); \( M_{R} = 3177509.52 \text{N} \cdot \text{mm} \).

In the workbench on the constraint and loading of the two force and torque as shown in Fig.3, analysis of the strain diagram under this condition and stress, as shown in Fig.4 and Fig.5.
In summary, the sideslip condition, the stress mainly concentrated in the turn around the bolt hole section and the upper part of the shock absorber connecting and steering fixed on the lower part of the section, the maximum stress 682.3MPa, meet the strength requirements.

When the automobile is in the emergency braking condition, the tire will be transferred to the steering joint by the vertical force and the vertical force, and the central hole of the steering joint will have vertical and longitudinal forces, and the two forces will produce torque after the translation, which is mainly borne by the steering knuckle lower lug. At this time the steering knuckle bearing capacity is as follows:

Vertical force:
\[ F_z = \frac{G \times (b + \varphi \times h_{z})}{L} \]

Lateral force:
\[ F_x = \frac{F_z}{2} \]

Extra torque:
\[ M_y = rF_x \]

Substituting numerical value
\[ F_z = 8281N; \quad F_x = 3312.4N; \quad M_y = 1262024.4N \cdot mm. \]

In the workbench on the steering knuckle constraints and loading of the two force and torque as shown in Fig.6, analysis of the strain diagram under this condition and stress, as shown in Fig.7 and Fig.8.
In emergency braking condition, the stress mainly concentrated in the turnaround under lug bolt and lug and the steering knuckle of the center hole under the contact surface, the maximum stress 623.68MPa, meet the strength requirements.

### 3. Conclusion

The results show that the braking sideslip steering knuckle of the high stress position in sideslip condition under high stress, by contrast to all conditions compared to cars in the steering braking sideslip condition for steering a greater impact.

According to the stress diagram of steering knuckle under different working conditions, the optimization of steering knuckle can be carried out from two aspects:

1. To reduce the weight of the steering knuckle, from the ANSYS analysis can be seen in many parts of the existing steering Festival material surplus by suitable design, further eliminates redundant material, saving production costs, further realize the lightweight of automobile.
(2) Reduce local stress and reduce stress concentration. The failure mode of steering knuckle is fatigue fracture. Stress analysis shows that local stress is relatively concentrated. Not at the expense of stress and reduce weight, because the local stress will increase the likelihood of fatigue damage increases, the optimization time as possible to relieve stress concentration.

References


