Performance Analysis and Lightweight Design of Steering Knuckle of Formula Student Racing

Zhangzhe Mao, Longtian Ma, Loufei Guo *, Jianming Wu
Wuhan University of Technology, Wuhan, China
* Corresponding Author Email: guoloufei@whut.edu.cn

Abstract. In this paper, the FSC Formula car ELSA of Wuhan University of Technology in 2020 is taken as the research object. On the premise of complying with the requirements of 2021 China Formula Student Car Competition Rules, CATIA modeling software is used to complete the model establishment and overall assembly of the steering knuckle. It is imported into the finite element analysis software Ansys for grid division and static analysis, analysis of the stress and strain of the steering knuckle, and based on this, the structure design of the steering knuckle. In order to meet the performance requirements of the competition, lightweight design should be carried out as far as possible.

Keywords: Formula student racing, automobile steering knuckle, lightweight design.

1. Introduction

Formula University of China (China FSC) is an automobile design and manufacturing competition organized by a team of university students majoring in automotive engineering or automotive related majors. It has been held for ten years. Every year, many universities independently design and manufacture a Formula car to participate in the race. The lightweight degree of the car can reflect the design level of the car, more related to the power and fuel economy of the car, directly affect the performance of the competition. The lightweight of racing car is a system engineering, in which the integration of parts is an effective method.

In FSC racing cars, steering is used to connect the hub to the suspension and is a key component affecting the car's stability, ride and maneuverability. Under different working conditions, the overall force condition of the car is different, and the influence on the steering knuckle is different. The design requirements of the knuckle should reduce its weight as far as possible under the premise of meeting the stiffness and strength requirements under different stress conditions. Low density aluminum alloy is generally used for steering knuckle materials. Based on the finite element method, this project analyzes the force and deformation characteristics of the aluminum alloy column under different racing conditions, and adopts the finite element optimization algorithm to optimize its topology structure and carry out the lightweight design of the column.

2. Design process

2.1. Overall process

In order to get a proper middle value in the design of the car in line with the actual demand and the ultimate lightweight, it is necessary to have a scientific and reliable design scheme, so as to ensure the scientific design. The following is the flow chart of the design of this project.
2.2. Part size limitation and installation

ADAMS software was used to optimize the simulation of the suspension steering system, so as to obtain the ideal internal and external hard points data. At the internal and external hard points, an arm was used to connect the column, and the upper and lower connection position coordinates of the column were determined. The size of brake disc is calculated according to the braking force matching, and the installation position of caliper is determined based on it.

Table 1. Parameters of outer hard points

<table>
<thead>
<tr>
<th></th>
<th>X</th>
<th>Y</th>
<th>Z</th>
</tr>
</thead>
<tbody>
<tr>
<td>hpl_lca_outer</td>
<td>-003.220</td>
<td>-538.700</td>
<td>85</td>
</tr>
<tr>
<td>hpl_uca_outer</td>
<td>006.218</td>
<td>-521.778</td>
<td>246</td>
</tr>
<tr>
<td>hpl_lca_outer</td>
<td>156012</td>
<td>-518.030</td>
<td>70.6</td>
</tr>
<tr>
<td>hpl_uca_outer</td>
<td>1600</td>
<td>-508.441</td>
<td>250.6</td>
</tr>
<tr>
<td>hpl_tierod_outer</td>
<td>-086.971</td>
<td>-576.088</td>
<td>126.544</td>
</tr>
</tbody>
</table>

2.3. Initial establishment of model

According to the existing size of hard points and the selected size of deep groove ball bearing 16012, the preliminary model of steering knuckle is established. Figure 1 is a preliminary model.
3. Working condition calculation

Before analyzing the model, the first thing is to determine the actual working conditions of the model, the force and load under the ultimate conditions. In this way, appropriate and light density materials can be selected according to the properties of materials after stress calculation.

Through the suspension motion analysis can be obtained under various conditions of stress, so as to calculate the stress of each point on the knuckle, in the preliminary design, the need to constantly analyze and calculate, fixed to select the relative worst limit conditions for preliminary check, so as to ensure the maximum safety. The ultimate condition was selected as the braking bending condition according to the acceleration sensor of the real car. At this time, the acceleration was divided into 1.5g braking deceleration and 1.2g lateral acceleration, which acted on the car at the same time. The calculated force data are as follows:

<table>
<thead>
<tr>
<th>Brake condition</th>
<th>force</th>
</tr>
</thead>
<tbody>
<tr>
<td>ΔF1</td>
<td>675,2476489</td>
</tr>
<tr>
<td>Frv2</td>
<td>336,1261755</td>
</tr>
<tr>
<td>Fbr</td>
<td>596,2075</td>
</tr>
<tr>
<td>Fbmr</td>
<td>136,2930345</td>
</tr>
<tr>
<td>Fbr</td>
<td>1397,877277</td>
</tr>
</tbody>
</table>

The load transferred from the rear axle to the front axle during braking is ΔF1, the vertical load exerted by a single rear wheel during braking is Frv2, the braking force exerted by the ground to a single rear wheel during braking is Fbr, the braking torque of a single rear wheel during braking is Fbmr, and the braking force generated by the rear caliper during braking is Fbr.

4. Model building and finite element analysis

After the text edit has been completed, the paper is ready for the template. Duplicate the template file by using the Save As command, and use the naming convention prescribed by your conference for the name of your paper. In this newly created file, highlight all of the contents and import your prepared text file. You are now ready to style your paper; use the scroll down window on the left of the MS Word Formatting toolbar.

4.1. Model building

According to the application requirements, CATIA was used to preliminarily optimize the structure, and the steering knuckle model meeting the actual use requirements was established.

![Figure 2. Structure optimization model](image)
4.2. Identify the Headings

According to the limit working condition of wheel edge, the structure is simulated and optimized. This analysis mainly refers to three factors: equivalent deformation, equivalent stress and safety factor. For the consideration of lightweight, the material of the column is 7075 aluminum alloy, and 1mm mesh is used according to the accuracy. 7075al has an extremely high compressive and tensile elastic modulus of 7.20E+10N/m^2 and yield strength of 5.05E+8N/m^2, while the density is only 2810kg/m^3.

Front column: Load is applied to the column model in ANSYS simulation software as follows:
The axial force of the hole on the front column toward the car is 299.2N.
The axial force of the lower hole of the front column is 1896.6N and the Z-axis is forward to 1230N.
Caliper lifting ear hole Z-axis forward 3850.9N.
Axial force of inner bearing hole 463.05N radial force 10990N.
Axial force of the outer bearing hole is 463.05N and radial force is 8437.6N.
So the constraints are applied as shown in the figure:

![Figure 3. Models that impose constraints](image)

Mesh module is used to complete mesh division of the model, and hexagonal mesh is used here. The mesh size is 2.5mm, and the mesh quality is not good at some key points, especially at the chamfering point when the steering knuckle structure is optimized. The finer mesh is divided separately, so as not to cause excessive stress concentration due to mesh deformation.

![Figure 4. A Grid model](image)
4.3. Simulation analysis and safety check

After the constraints are imposed and the grid is drawn, the simulation analysis and calculation of the whole model are carried out. In the calculation target, the equivalent deformation, equivalent stress and safety factor are set as the target, and the overall calculation and check are carried out.

The results of equivalent deformation, equivalent stress and safety factor obtained by simulation analysis are shown in the figure below:

Figure 5. Equivalent elastic deformation diagram

Figure 6. Equivalent stress deformation diagram

Figure 7. Safety factor analysis diagram
The maximum equivalent elastic deformation is 0.0020545mm. The maximum stress is 142.68MPa, less than the maximum allowable stress 505Pa; The minimum safety factor of 3.9951 is greater than the minimum safety factor of 1.8. The three parameters are in line with the requirements, so the column design scheme meets the requirements.

The safety check of the whole steering knuckle was carried out by finite element analysis with ANSYS software, which basically conforms to its important indexes. All the characteristics, such as maximum shape variable, equivalent deformation and equivalent stress indexes, are in good performance. After the preliminary completion of the model design, is the actual processing and assembly.

5. Actual machining and assembly

The steering knuckle is at the junction of suspension, steering hitch arm, wheel core and brake caliper. The steering knuckle is connected with the wheel core through bearings, and the brake caliper is connected at the lifting lug of the steering knuckle. The torque is larger here, and the fixing is completed by plugging to reduce the wear of the lifting ear hole caused by long-term use and frequent disassembly. The steering arm is connected by adding a lifting lug through two lower holes. In the upper and lower connection with the A arm using double bolt lifting lugs, sacrificing part of the stability, more adjustment space. By this method, the beam Angle size can be adjusted according to the dynamic condition of the car and the feedback of the driver at any time, so as to obtain a more matching chassis adjustment.

5.1. Physical processing

The drawings were drawn through cad and sent to the parts processor, then each part was mused out on the lathe and sent to the team for assembly. The wheel rim model was installed according to the spatial relationship, as shown in the figure.

![Figure 8. Wheel rim assembly material](image)

5.2. Physical processing

At present, the design of the column has been completed processing, and installed in the real car. The lightweight design of the column is to reduce the unsprung mass as much as possible under the premise of ensuring safety, so as to optimize the maneuverability and stability of the car. All the optimization must always be implemented to the operation of the real car, the real car running data will be further feedback to the calculated optimization.
At the end of the installation of each part of the vehicle, the running test was carried out, and the fitting and safety of each part were verified. After nearly half a year of running practice, there was no obvious damage to the whole wheel rim.

5.3. Optimization result

Using CATIA software, the workpiece weight was approximately 414g and the column weight was 14% lighter than last season. Each season through the vehicle mass design goal is more lightweight, so that in the design based on the lighter vehicle mass force calculation, parts safety check, in the design of parts, respectively in the premise of ensuring safety and function, to complete the lightweight goal, complete the closed-loop lightweight design goal.

Acknowledgments

This work was financially supported by National Innovation and Entrepreneurship Training Program for College Students S202110497062.
References

