Research on Lightweight Technology of new carbon fiber wheel hub structure

Dianming Chu 1, Wenjuan Bai 1, Yan He 1, *, Mingrui Ye 2, Junwei Li 1

1 Shandong Engineering Laboratory for Preparation and Application of High-performance Carbon-Materials, College of Electromechanical Engineering, Qingdao University of Science and Technology, Shandong Province 266061, China
2 Zhejiang Jizhi New Energy Automobile Technology Co., Ltd, Hangzhou, Zhejiang Province 31122, China

*Corresponding author e-mail: heyan@qust.edu.cn

Abstract. In this work, carbon fiber is used as rim and magnesium alloy as wheel hub for new design of wheel structure, making the material give full play to its properties. We use the technology of simulation analysis, dimension optimization, and topology optimization to analyse the performance of new structure. When the wheel hub meets the requirements of commercial vehicle wheel performance and test methods (GB/T5909-2009), the material weight loss of the product is 48% and the structure weight loss is 13.8%. The accuracy of the simulation and the superiority of the new technology are proved. These methods provide a new way of thinking and method for the study of lightweight technology in the future.

1. Introduction

Today, the automobile industry is developing rapidly, and the energy saving, emission reduction and safety problems brought by the automobile have gradually aroused the attention of the people of all countries. With the aggravation of the environment and energy problems, it is urgent to save energy and protect the environment. Lightweight is an inevitable choice [1]. Lightweight not only saves materials and energy, but also directly affects the power performance of cars [3]. According to authoritative statistics, every 10% reduction in vehicle quality can reduce vehicle emissions by 8%. As the most important and key safety component of the whole automobile, the performance and service life of the wheel hub determine the safety and reliability of the automobile to a certain extent [2]. Therefore, the lightweight of automobile wheel hub is of great significance.

Two main methods are used: one is to optimize the structure to reduce the mass of materials, the other is to change the materials and use lightweight engineering materials [4]. In the field of structural optimization, Miloslav, Riesner et al used shell element to analyze under bending fatigue condition [5]. Hamid and Behrooz fitted the curve on the basis of a large amount of data analysis, and established a model for the design variable and optimized it [6]. Yuqing Zhou took magnesium aluminum alloy as the research object. The thickness of spoke plate and rim was taken as design variable, and the yield strength was optimized as state variable [7]. In the field of material application, Guojun Liu took the composite materials as the research object and optimized the size of the composite materials, and achieved good results [8]. Carbon fiber (CF) hub is a new type of wheel hub, which is mainly used

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
in advanced models such as GT and koenigsegg. Because of the price, it is less used, but the material has outstanding advantages, once mass production will reduce the cost [9].

In the field of lightweight research, although many scholars have made a detailed analysis of some key points, few studies have been done on materials and structures at the same time. At the international level, there are few references in this field.

In this work, the method of material and structure optimization is used to analyze the hub systematically, as shown in Figure 1. Firstly, we carry on the skillful collocation of the material function; then, the maximum force position is optimized reasonably by using the dimension optimization technique. Finally, the topology optimization technique is adopted to further lighten the weight. The simulation and verification are carried out by processing the experimental samples.

![Figure 1. Hub lightweight process](image)

2. Simulation and Experiment

2.1. Material Optimization
This product adopts the combination of CF and Mg to design wheels hub, in which, CF materials are processed into wheel rim and Mg materials are processed wheel spoke. The performance of wheel rim is enhanced. The strength is greatly improved and the car body bearing capacity is greatly strengthened. The comprehensive properties of electric conduction, heat conduction and fatigue resistance are also taken into account. Using Mg materials, the spokes can absorb energy better, and play a buffer role, lighter quality and excellent performance. It is can give full play to the performance of the material by using two different materials to design wheel hub. At the same time, learn from each other and let lightweight play to the extreme.

In this study, the wheel hub structure is composed of CF composites and Mg materials, which not only realizes the maximum lightweight, but also ensures the performance of the hub structure. Compared with the steel structure and the Al structure, the weight reduction is 32.7% and 48%, respectively. Therefore, on the basis of ensuring the strength, the weight loss ratio of the replacement material is very obvious. Under the same structure, the difference of the utilization ratio of different materials is not obvious. According to the principle of segmentation, the mechanical structure can be designed with different materials to maximize the performance of the material.

2.2. Dimension Optimization
According to the standard commercial vehicle wheel performance requirements and test methods (GB/T5909-2009), we carry out the simulation analysis. Under the condition of experiment 1, the equivalent stress of spokes is the largest. Under the condition of experiment 2, the region of applied force and the zone of bolt connection reach the yield state first, so the emphasis is optimized to make the width of spoke and bolt joint reduce to close to the permissible utilization, ensuring these areas do not exceed permission. In order to better lightening other parts, the hub performance should be strictly checked according to international standards, so that it can meet the relevant requirements of the standard.
The width of the two regions, spokes and connectors are defined as T1 and T2 parameters, and the objective function is equivalent stress setting value. Under the condition of experiment, the object function of spoke is set to 140 MPa. according to the result, the software continues to be optimized until the strength condition is satisfied. Under the condition of experiment 2, the objective function is set to 120 MPa, and the life and death of each layer unit is defined.

According to the size optimization calculation, the optimization result diagram is shown in Figure 2. Under the stress conditions defined above, the final spoke width T1 is reduced from 24 mm to 15 mm, and the connection width T2 is increased from 14 mm to 19 mm. The equivalent stress reaches the target set value and meets the standard requirement. Finally, the structure optimization of the most dangerous parts is realized.

![Figure 2. Visual drawings before and after dimension optimization](image)

| Table 1. Comparison of performance data before and after dimension optimization |
|---------------------------------------------------------------|
| Equivalent stress | Before dimension optimization | After dimension optimization | Displacement | Before dimension optimization | After dimension optimization |
| Experimental 1 | 101MPa | 139 MPa | 0.497mm | 0.794 mm |
| Experimental 1 | 69 MPa | 120 MPa | 0.8 mm | 0.739 mm |

Under the condition that the performance is satisfied, the dangerous part can be improved and the weight can be reduced as much as possible by the dimension optimization. The width of the spoke part becomes smaller and the width of the rim connection part becomes larger.

2.3. Topology Optimization

The structure model after size optimization is optimized under two working conditions. The objective function is set as the mass, and the variable density method is used to optimize the topology. As shown in Figure 4, the material distribution cloud diagram under different topological density is obtained.

![Figure 3. original design](image)  ![Figure 4. Preserves 70% and 50% material optimization results](image)
According to the results, there were different degrees of material loss in the rim part, but the loss was irregular, and the material itself was in contact with the tire; thus, the retention was necessary on the spokes. The material is lost in different degree in the edge region, and the topology optimization results are further determined by comparing the data of different groups. After topology optimization, both displacement and equivalent stress meet the requirements, as shown in Table 2.

![Figure 5. Optimized results](image)

| Structure optimization | Topology optimization |
|------------------------|-----------------------|
| Equivalent stress     | 139 MPa               |
| Displacement           | 0.79 mm               |
|                        | 143 MPa               |
|                        | 0.88 mm               |

| Experimental 1 | | Experimental 1 |
|----------------|----------------|
| 120 MPa        | 0.74 mm        |
| 152 MPa        | 1.2 mm         |

**Table 2. Comparison of performance data before and after topology optimization**

**Table 3. Hub product lightweight data**

| Initial structure | Size optimization | Topology optimization |
|-------------------|-------------------|-----------------------|
|                   | Steel             | Al                    | CF and Mg | 1750g | 1750g |
| Rim mass          | 7178g             | 2489g                 | 1748g     | 1750g | 1750g |
| Spoke mass        | 6104g             | 2117g                 | 1361g     | 1052g | 930g  |
| Hub mass          | 13282g            | 4606g                 | 3109g     | 2802g | 2680g |
| Weight loss (%)   | 32.7%             | 48%                   | --        | 9.87% | 13.80% |
hammer impact test was added to verify the rationality of the sample structure. And compared with the simulation and verified, Figures. 6-8 is three kinds of experimental instruments.

The test rig has a rotating device for rotating the wheels with a constant bending moment. In this experiment, the wheel is stationary and subjected to a rotating bending moment. Similarly, the radial load Fr is determined according to the following Eq. (2), in which the test load and inflatable pressure should be determined according to the rating of the wheel.

\[ F_r = F_v K \]

\[ F_r = 2950N, \quad F_v = 1743N, \quad K = 1.69 \]

The heavy hammer and hub are 13 ° and close to the spoke according to the experimental requirements. The static stress and deformation of objects subjected to collision and extrusion are simulated.

**Figure 6. Bending fatigue experiment**

**Figure 7. Radial fatigue experiment**

**Figure 8. Impact test**

3. Comparative study between Simulation and experiment

3.1. Bending fatigue experimental calibration

Table 4 shows the calibration of the bending fatigue experiment. According to the numerical value of the test results, the allowable stress is not reached. The displacement increment of the installation surface is 0.08 mm, the finite element simulation result is 0.19 mm, and the fatigue value is not reached by the stress condition simulation and the test. The finite element results of displacement value are larger because of the simplification error in the simplification of the model, the error of the static substitution dynamic method, the error of the boundary condition program.

**Table 4. Bending fatigue experimental calibration**

| Sample number | test load | Required speed | Final deviation |
|---------------|-----------|----------------|----------------|
| 2017-115-01   | 484.2N.m  | 1.0×10⁵        | 14.31mm        |
| Initial migration | 14.23mm  |                |                |
| test result   | The wheel is free of cracks and the offset increment is not more than 10% of the initial offset after the completion of 1.0×10⁵ revolutions. |
| simulation result | The maximum equivalent stress of simulation is 143mpa, the displacement increment of installation surface is 0.19mm. |
According to the analysis results in Figure 9 and referring to the principle that V&V is relatively correct, it is assumed that the experimental results are correct. According to the analysis above mentioned, there are many kinds of error factors from modeling to the result of finite element analysis, and the whole value should be larger; thus, this calculation data is more reliable. According to the results, the maximum displacement is on the spoke, and the location of the maximum deformation is consistent with that of the simulation, which show that the simulation data has high reliability.

![Figure 9. Bending fatigue experiment and simulation results](image)

3.2. Radial fatigue experimental calibration

According to the experimental results in Table 5, the loading conditions are simulated in accordance with the experimental requirements. As shown in Figure 10, the maximum equivalent stress is 152 MPa. The maximum equivalent stress is within the allowable fatigue stress of the material, which is completely consistent with the experimental results of no cracks detected. The maximum displacement is 1.2 mm, which also occurs on the rim, which is consistent with the predicted position of the previous simulation. The displacement offset is not tested in this experiment, and the simulation results show that it meets the requirements of the experiment.

| Sample number | Specification / model | 2017-115-01 | 16*4T |
|----------------|-----------------------|-------------|-------|
| test load      | Required speed        | 2950N       | 5.0*105 |
| Use of tyres   | charged pressure      | T125/70R16  | 450KPa |
| Tyre final     |                       |             |       |
| pressure       |                       |             |       |
| test result    |                       |             |       |
| simulation     |                       |             |       |
| result         | Wheel free of cracks after completion of 5.0*10^5 revolutions | The maximum equivalent stress of simulation is 152 MPa, and the displacement increment of installation surface is 1.2mm. |

According to the simulation data and the experimental data, the crack is not detected in the experimental data, and the range is relatively wide. Since the maximum equivalent stress of the simulation data lies in the fatigue allowable stress, the simulation data is equally credible and verified by the experiment site. The maximum displacement exists on the rim, so the reliability of the data is high.
3.3. Impact test calibration

As shown in Table 6, the results of this simulation are only concerned with the structural performance of the spoke and hub, and the plastic wheel rim is not included in the simulation, and the stress displacement diagram is obtained by directly impacting the wheel spokes with the weight hammer. As shown in Figure 11, the stress of the spoke is 314 MPa under the action of the weight hammer impact, which does not reach the tensile strength limit (380 MPa). Therefore, the spoke will not break, which is consistent with the experiment.

Table 6. 13° impact test calibration

| Sample number | Specification / model | test load | Impact height | charged pressure |
|---------------|-----------------------|-----------|---------------|-----------------|
| 2017-115-01   | 16*4T                 | 300Kg     | 196 mm        | 200 KPa         |
| T125/70R16    |                       |           |               |                 |
| test result   | There are cracks in the impact position of the wheel rim, no crack in the spoke, no air leakage in the tire | | | |
| simulation    | The simulation results show that the spokes do not reach the allowable stress, and the displacement increment of the mounting surface is 0.19 mm. | | | |

The experimental simulation data are also highly consistent with the experimental data. In terms of setting the conditions, this simulation analysis is to directly reduce the weight hammer action and the spoke. However, in the experiment, the weight hammer directly acts on the wheel rim, and this calculation simplifies the wheel rim. So the simulation data cannot get the data of the wheel rim finding crack. But according to the simulation results, it can be judged that if the wheel rim is not installed in the experiment, and acts directly on the spoke, there will also be no crack because of this kind of working condition, which show that the tensile limit is not reached.

By comparing the experimental results with the simulation ones, it is proved that the simulation analysis has a high reliability.
4. Conclusion
In the present study, the material and technology of hub products are studied, which provides a new method for hub manufacturers to lighten weight. The optimization of structure, stress redistribution and quality improvement of current products can be realized by using this method. The reduction of cost and energy consumption have important reference significance.

Acknowledgements
This work is supported by the National Natural Science Foundation of China (No. 51676103) and Taishan Scholar Project of Shandong Province (No. ts20190937).

References
[1] Yong Zhang. Application of Automotive lightweight and Aluminum Alloy in Modern Automobile production [J]. Internal combustion engines and accessories, 2017(14):38-39.
[2] Xiaoyan Song. Finite element Analysis of Automobile Wheel Hub and its structural Optimization Design [J]. Value Engineering, 2018, 37 (16): 123-124.
[3] Wenlong Sun. Study on lightweight Technology of Light weight material used in Automobile Wheel Hub [D]. Beijing Institute of Technology, 2016.
[4] Liang Zhao. Structure Optimization Design of magnesium Alloy Wheel based on ansys [D]. Yan Shan University, 2013.
[5] Kang W.J, Kim A.K, Kim G.H. Fatigue failure prediction of press fitted parts subjected to a cyclic loading condition by finite element methods[J]. Fatigue & Fracture of Engineering Materials &Structures, 2007, 30(12):1194-1202.
[6] Hamid Jahed, Behrooz Farshi, Mohammad A. Eshraghi, AsgharNasr. A numerical optimization technique for design of wheel profiles[J]. Iran University of Science & Technology, 2008, (10):4-7.
[7] Yuqing Hou. Prediction and Optimization of fatigue Life of magnesium Alloy Wheel [D]. Master thesis, Chongqing University, 2008.
[8] Guojun Liu. Strength Analysis and Optimum Design of Composite Wheel [D]. Harbin University of Technology, 2006.
[9] Chao Sun. Brief Analysis of Aluminum Alloy Wheel Hub and New Carbon fiber Wheel Hub [J]. Shandong Industrial Technology, 2018 (14): 64.