Impact test simulation and structural optimization of aluminum alloy A356.2 wheel hub

liang Yang¹,², Haibo Yang¹, Hongjian Tan³, Peng Hu¹, Xuewen Cheng³ and Junlong Lu¹

¹ University of Science and Technology Beijing, 100083, China;
² Guangdong DCenti Auto-Parts Stock Limited Company, 529200, China;
³ Guangdong Engineering Polytechnic College 529200, China.

4 E-mail: yl08_08@163.com

Abstract. In order to improve the design efficiency of automobile hub, finite element simulation analysis is performed on the impact test of automobile hub. According to the national standard of impact test, Ansys Workbench software is used to load and constrain the explicit dynamics elastoplastic model. According to the results of impact strength analysis, the deficiency of the structure is studied, and put forward the corresponding structural optimization design. The actual test verifies the accuracy of the impact test simulation. Ansys software is used again to analyse the impact stress of the improved wheel hub to verify the rationality of the improved structure. The experimental results show that the method of dynamic analysis is practical and effective in calculating the stress and strain of automobile hub impact test. The research results provide theoretical basis and practical significance for the design and development of automotive aluminum alloy wheels.

1. Introduction
The wheel is an important part of the car that loads full weight of the car. In order to ensure the wheel performance is qualified, the wheel must pass the impact test, bending fatigue test and radial fatigue test. In the actual development and manufacturing process, we found that the main influencing factors of hub performance is its impact resistance. The main method of impact testing is the 13 ° impact test, which simulates the lateral impact of a wheel or other objects on the side of a road during collision [1]. In this paper, we use the ANSYS Workbench software to simulate the wheel impact test, and verify the analysis results by the actual experimental results. The weak part of the wheel hub that should be modified could be found out, which provides reliable design basis for wheel hub developers, thereby shortening the development cycle, reduce development costs and enhancing the competitiveness of enterprises.

2. Hub impact test model
2.1. wheel impact test device
This paper studies the 13 ° impact test of automobile aluminum alloy wheel hub, which is the main test method to test the impact resistance of aluminum alloy wheels. The impact test device is shown in figure 1. It consists of impact block release device, impact block, Hub and hub fixtures. The trial hub is a new finished wheel that has not been tested or used. The ambient temperature is 10-38 °C [1].
During the test, the axis of the hub and the impact block vertical drop direction are \((13 \pm 1) \degree\). According to the requirements of the industry standard, after the impact, the spoke section can’t be visually observed cracks, spokes can’t be separated from the rim [1].

![Figure 1. Impact test device diagram.](image)

2.2. Aluminum wheel material properties
By the company's testing machine, the material of the aluminum alloy wheel test chemical composition as follows table 1:

| Table 1. Aluminum alloy A356.2 wheel chemical composition. (wt.%) |
|-----------------|--------|--------|--------|--------|--------|--------|--------|--------|
| Element        | Si     | Mg     | Ti     | Mn     | Fe     | Cu     | Zn     | Al     |
| Standard ingredients | 6.5-7.5 | 0.30-0.45 | ≤0.20 | ≤0.05 | ≤0.12 | ≤0.10 | ≤0.05 | Remaining |
| The actual composition | 7.26  | 0.2938 | 0.1143 | 0.001 | 0.1148 | 0.001 | 0.024 | Remaining |

According to GB / T228.1-2010, the tensile testing machine is used to test the material of the hub material to obtain the test curve figure 2 a). It can be known that the strength of the material of spokes is weaker than that of the wheel rim. In order to ensure the accuracy of the impact model, the wheel's material properties are defined as spoke materials. The stress-strain curve of the spokes material obtained from the tensile test data is as follows table 2:

| Table 2. Aluminum alloy A356.2 wheel hub’s tensile test data. |
|-----------------|--------|--------|--------|--------|
| Specimen name   | Specimen area/mm² | The maximum strength value/kgf | Yield strength/MPa | Ultimate tensile strength72.4th/MPa |
| Rim             | 19.63  | 673.73 | 229.41 | 257.25 |
| Spoke           | 19.63  | 520.49 | 213.64 | 233.27 |
2.3. Finite element model
According to the requirements of the industry on the impact test, Shown in figure3, the wheel is to be bolted on the fixing device. Therefore, when analyzing, the five bolt holes of the hub are subjected to a fixed constraint and fixed to 6 degrees of freedom; the downward acceleration of gravity is set for the simulation environment; According to the actual situation of the experiment, the impact block is set to move vertically only. The finite element model is shown in figure 4. When the bottom surface of the impact block is in contact with the rim, the kinetic energy and potential energy of the impact block begin to be absorbed and consumed by the hub, and the velocity of the impact block at this time is \( v = \sqrt{2GH} \). G is acceleration of gravity, take 9.8m / s\(^2\); H is impact block height, take 0.23m.

3. Finite element simulation results and experimental verification

3.1. Finite element analysis of the results
After analysis and calculation, using the workbench post-processing module animation playback function can clearly see the whole process of stress and strain changes under the impact load. When the velocity of the impact block in the vertical direction is zero, the whole energy of the impact block is absorbed and consumed by the hub [3]. At this moment, the equivalent stress and strain of the hub are maximum. In the impact test model, the impact block is set as a rigid body, we can take the centroid of the impact block to understand the movement of the impact block. And look at the corresponding curve of the wheel hub maximum stress over time.
Shown in figure 5 and figure 6, when the impact block down to 7.5ms its speed is 0, and began to move in the opposite direction. This moment the impact load of the hub is maximum. Look over the wheel's equivalent stress cloud at this moment.
When the impact block comes into contact with the hub, the contact area between the impact block and the hub generates contact stress and stress concentration phenomenon. Since the volume of material in this section is very small, we think the stress concentration at this location is unrealistic and the hazardous area of wheel hub impact test will not be this place.

According to the analysis results, Shown in figure 7, it can be seen that In the process of impact load on the wheel hub, the equivalent stress of the wheel hub reaches the maximum at 7.5 ms, and the areas subject to the maximum stress are mainly located about 6 cm near the bolt hole at the junction of the spokes and the wheel disc. The maximum nonlinear effective stress reached 230.7 MPa, which is very close to the material’s tensile strength 233 MPa, Here is the risk of cracks. Due to various reasons, such as the shape and casting process, the wheel hub material distribution is not ideal and uniform, however, the finite element model is the ideal model. The equivalent stress of finite element analysis must ensure that there is a certain margin of safety stress. Therefore, we think this wheel hub’s structure design is unreasonable, can’t meet the qualified requirements of the impact test.

3.2. Test verification
From the figure 8, we can clearly see the distribution of the failure part on the wheel hub. After the wheel test, Cracks appear on the spokes where near the roulette. Finite element analysis results are in good agreement with the actual test, which can verify the rationality of the finite element analysis model of the wheel hub impact test established in this paper.

![Figure 8. Impact test results pictures.](image)

3.3. Wheel structure optimization
According to the engineering experience and the principle of optimal design, to reduce the stress of the weak parts of the spokes as the goal. Reduce the outer flange rim thickness, plastic deformation by increasing the rim to absorb more energy [4]. Reducing the diameter of the roulette so that lengthening the length of the spokes is also to increase the plastic deformation of the spokes to absorb more energy and disperse the localized stresses in the spokes. Increase the width of the spokes, Prevent the deformation of the material is too large lead to cracks, the stress distribution more well-proportioned [5]. The initial design and improvement of the wheel hub section line are shown in figure 9.
After improving the design of the wheel hub, perform finite element analysis again, the stress distribution shown in figure 10. The maximum stress at the spokes appeared on the front side and the maximum value was reduced to 180 MPa, which is lower than the tensile strength of the A356.2 material and has certain safety. The actual test also proves this point, the improved wheel hub through the 13° impact test. The accuracy of the finite element model of the impact test is verified. The structural strength of an 18 × 7.5J wheel has been successfully optimized and safely applied in practical engineering.

4. Conclusions
Based on the Ansys Workbench software, this paper simulates the wheel hub impact test by finite element method and adopts the explicit dynamics elasto-plastic mode. This method correctly predict the wheel hub structure’s weak position where at the junction of the spoke and the wheel disc. Improve the wheel structure, by increasing the plastic deformation to absorb more energy and make the stress distribution more evenly to improve the ability of the hub to resist fracture. The maximum equivalent stress of the impact process is obviously reduced, the overall structure of the hub tends to
be reasonable, and the maximum equivalent stress decreases from 230.7MPa to 180MPa. The improved hub successfully passed the impact test, further illustrating the effectiveness of the numerical simulation model. From the practical effect of the project, the impact test simulation and structural optimization of the aluminum alloy wheel hub can improve product design efficiency, reduce design errors, shorten the development cycle of new products and reduce costs, and have important practical application value.

Acknowledgments
I am deeply indebted to a number of people without whose encouragement and assistance this thesis would not have been completed. I am profoundly grateful to my supervisor, Yang haibo, whose illuminating instruction and expert advice have guided me through every step of my writing of this thesis. My great gratitude also goes to some of my friends and classmates who have selflessly and generously helped me with my thesis.

References
[1] Mengyan Z and Tao Q 2010 Simulation Analysis of Car A-alloy Wheel 13° Impact Test Chin.J Mech.Eng-En.46 83-7
[2] Hongxiang W, Xiaokang M and Yang B 2014 Simulation Analysis of Car Wheel 13° Impact Test Machinery Design & Manufacture.10 197
[3] Guozhi Z 2015 Study on Mechanical Model of Impact Test of Vehicle Wheels Vehicle & Power Technology 4 11-4
[4] Akbulut H 2003 On optimization of a car rim using finite element method Finite Elements in Analysis and Design 39 (5-6) 433-443
[5] Firat M Kocabicak U 2004 Analytical durability modeling and evaluation—complementary techniques for physical testing of automotive components Engineering Failure Analysis 11(4) 655-674