Impact Testing of Casting Aluminum Wheels

Fanxin Kong1,2*
1School of Materials Science and Engineering, Nanjing Institute of Technology, Nanjing 211167, Jiangsu, China
2Jiangsu Key Laboratory of Advanced Structural Materials and Application Technology, Nanjing, 211167, Jiangsu, China
*Corresponding author (E-mail: kfx@njit.edu.cn)

Jinhua Hu
School of Mechanical Engineering, Shanghai Dian Ji University, Shanghai, 201306 China

Zhenhong Li
School of Materials Science and Engineering, Nanjing Institute of Technology, Nanjing 211167, Jiangsu, China

Chen Wang
School of Materials Science and Engineering, Nanjing Institute of Technology, Nanjing 211167, Jiangsu, China

Abstract
The microstructure of the casting wheel is not uniform in different regions. Therefore, different fracture criterions should be taken for different regions of a same wheel. The finite element model was set up for the wheel’s impact testing. The results agree well with the experiments. Through the recent several years’ analyses and experimental verification, the wheel’s transient impact tracking data were accumulated according to the wheel’s structures. The criteria was determined from the tracking data. Based on this criteria whether the designed wheel is able to pass the transient impact test may be judged.

Key words: Impact, Failure, Finite Element, Simulation.

1. INTRODUCTION
The numerical simulations of wheel impact testing are essential in wheel development process, which can reduce design lead time, especially for prototype build or concept prove-out in the early stage of product development. When it is appropriately carried out, nonlinear finite element analysis can help to validate wheel design.

It has been over twenty years for the production of aluminum wheels in China. A whole set of evaluation methods have been formulated. For wheels, there are three types of testings, i.e. the impact testing, the bending fatigue testing and the radial fatigue testing. Aiming at the three types of testings, a large amount of engineering simulation research work have been carried out by the scholars throughout the world.

The implicit finite element method and the testing were applied respectively to carry out the research on the impact testing of wheels by Yan etc (Yan S.Z., 2007). They found that the cracking trend from FEA agreed very well with that from the testing.

Hongmei Sun from Yanshan University established the structural optimal design model of the vehicle wheel based on constrained variable metric optimization algorithm which comprehensively considered the rim stiffness, bending stress and the wheel vibration mode. the structural optimal design of the aluminum alloy wheel are studied with the thickness of rim and spokes as the design variables, the lightest of wheel quality as objective function.

Chang etc. set up the FEA model for the 13 degree impact testing of a wheel (Chang C.L., 2009). The total plasticity work was applied to predict the impact failure of a wheel. The influence of the microstructure and the defects in a casting wheel on the wheel’s impact character were researched by Merlin etc (Merlin M., 2009). Lu carried out comprehensive research on the 13 degree compact testing (Lu B., 2010). He concluded that the aging treatment had important influence on the compact property of the aluminum wheels. The crack plastic strain was taken as the failure criterion in his finite element model. The authors of this paper have carried out systematic numerical research on the wheel’s impact testings. The proposed finite element analysis procedure for the wheel’s impact testing has been successfully applied on the design of new wheels in Citic Dicastal Wheel Manufacturing Company.

2. THE MICROSTRUCTURE OF THE FRACTURE SURFACE

The crack region’s metallographs of a wheel spoke and a wheel hub were shown in Fig. 1 and Fig. 2
respectively. From Fig. 1a, the deficiency in this region is 0.34%. The maximum size of the shrinkage hole is 459.9μm. From Fig. 1b, the deficiency and segregation in this region are minor. From Fig. 2a, there exists a large amount of shrinkage cavities in the wheel’s hub region. The grain size in the wheel’s hub region is markedly larger than that in the wheel’s spoke region. The difference of the microstructure between the wheel’s spoke region and the wheel’s hub region results in the difference of the material properties between the two regions. Actually, the microstructure of the casting wheel is not uniform in different regions. Therefore, different fracture criterions should be taken for different regions of a same wheel.

![Figure 1. The original image](image)

3. VERIFICATION OF THE FINITE ELEMENT MODEL

3.1. Experimental Procedures

Wheel impact test is defined by the test procedure SAE J2530 for evaluating the impact performance of aluminium curb wheels. As shown in Figure 3, a wheel assembly with a tire is fixed on a stationary fixture, at an angle of 13 degrees to the horizontal. The flange is the first region which comes in contact with the striker upon impact. The striker falls freely from a 230mm height and has a 25mm overlap with the flange of the wheel and tire assembly. To evaluate the wheel performance in the impact test, two failure criteria are given in SAE: no fracture in centre member of the wheel and no air pressure loss in tire.

![Figure 3. Schematic of wheel impact test](image)

An aluminium wheel is consisted by 5 regions, which are the hub, spoke, rim, flange and windows (as shown in Figure 4).
3.2. The numerical model of the wheel’s impact testing

The 13 degree impact testing was selected as the research model. The practical impact testing device was simplified in order to save the computation time (Fig. 5). By simplifying in this way, though the computed value may not be the same as those from the testing, the problem’s nature was maintained. The uniform impact block was applied in the FE analysis. The practical weight of the impact block was taken in the computation. Thus the density of the block has to be modified factitiously. In this example, 10-node tetrahedron element was used to mesh the model (Fig. 6). In total, there are 156358 elements and 345886 nodes for this model.

For the wheel’s impact testing, the stresses due to the impact load will be larger than the material’s yield strength. The plasticity deformation will be produced. Therefore the elastic-plastic material property (Fig. 7), which was obtained from the material testing, was set up for the aluminum wheel. The wheel was casted using the low pressure casting technology. The material of the casting billet was A356, i.e. ZAISi7Mg. The elastic modulus was assumed to be 70 GPa. The Poisson’s ratio was taken as 0.33, while the density was taken as 2.67×10−6kg/mm3.

Based on the practical impact testing, the 5 bolt holes and the installation face of the wheel were constrained. The initial speed of the impact block was calculated from the impact height. The compact gravity was applied on the block. The contact pair was set up between the lower surface of the block and the upper surface of the wheel.
3.3. Analysis of the computing results

According to large amount analyses and comparison with the practical impact testing, the first principal strain was applied to judge the influence of the impact testing. For this wheel, the maximum first principal strain with the value of 9.2% located at the corner of the weight loss fossa (Fig. 8). Based on previous large amount of analyses for the impact testing, the crack may be produced at the red region shown in Fig. 8.

3.4. Experimental verification

The practical failure location as shown in Fig. 9 may be clearly observed after the impact testing. With the comparison between Fig. 8 and Fig. 9, it can be concluded that the numerical model can reflect the practical compact testing. Therefore, the numerical model can be applied in the practical design of a new wheel.
4. THE DATA TRACKING TABLE AND THE FAILURE CRITERION

Based on the research on the low pressure casting technology for the wheels, it can be concluded that material property of the casting wheel part cannot be uniform, i.e. different parts of the wheel have different material properties. Therefore it is not possible to apply a single criterion for the whole wheel.

![Figure 10. Classification of the wheels according to the spoke type:](image)
a) five-spoke wheel; b) six-spoke wheel; c) seven-spoke wheel; d) ten-spoke wheel e) five group spokes wheel

Through several years of analyses and verifications with the practical testings, the data tracking table can be obtained for the aluminum wheel’s impact testing according to the spoke type as shown in Fig. 10. A typical data tracking table was shown as Fig. 10 for the five-spoke wheels. The impact criterion table shown as Table I can be obtained according to Fig. 11.

![Table 1. The impact criteria table for the wheel’s impact testing](image)

<table>
<thead>
<tr>
<th>Type of the spoke</th>
<th>The 1st max. prin. strain %</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>back</td>
</tr>
<tr>
<td>5 spokes</td>
<td>7.7</td>
</tr>
<tr>
<td>6 spokes</td>
<td>7.5</td>
</tr>
<tr>
<td>7 spokes</td>
<td>7.3</td>
</tr>
<tr>
<td>10 spokes</td>
<td>7.4</td>
</tr>
<tr>
<td>5 group spokes</td>
<td>7.8</td>
</tr>
</tbody>
</table>

![Figure 11. The tracking table of the dynamic impact analysis data for the five-spoke wheels](image)

5. CONCLUSIONS

For the aluminum wheels, the failure ratio is relatively high during the impact testing. In order to quickly and accurately predict the impact testing, a simplified finite element model was proposed for the wheel’s impact testing. Through the comparison between the results from the finite element analysis and the practical testing, the finite element model proposed in this paper was demonstrated. Through large amount of
analyses and verifications with the testings, the transient impact data tracking table as well as the criterion table were obtained, which can be applied to guide the design of new types of aluminum wheels.

Acknowledgements

This work was supported by the Opening Project of Jiangsu Key Laboratory of Advanced Structural Materials and Application Technology (Project No. ASMA201507).
This work was supported by Students Innovation Project of Jiangsu Province (Project No. 201711276070).
This work was supported by Outstanding Scientific and Technological Innovation Team in Colleges and Universities of Jiangsu Province.

REFERENCES

Lu B. (2010) “The failure cases of the impact testing and the finite element analysis for the aluminum wheel”, Light vehicle technology, 245, pp.27~32