Damage model for the impact test of an automotive aluminum wheel

A Zapata², O A González-Estrada¹, and A Pertuz²
¹ Grupo de Investigación GIC, Universidad Industrial de Santander, Bucaramanga, Colombia
² Grupo de Investigación GIEMA, Universidad Industrial de Santander, Bucaramanga, Colombia

E-mail: agonzale@uis.edu.co

Abstract. The wheel is a critical component for car safety. To achieve maximum performance, product development includes a number of tests, such as rotating bending, radial fatigue, and impact. In general, the design process involving physical testing of prototypes following standards is time-consuming and cost-intensive. Nowadays, several industry sectors are including virtual testing as part of the certification process, e.g., the United States Food and Drug Administration now recognizes numerical simulations as a valuable tool for the design of implants. Numerical testing reduces time to market and improves cost control during the design stage. In this work, we investigate the use of finite element models to assess damage in an aluminum wheel, in agreement with the impact test according to the ICONTEC standard NTC 4177. We compare the results with the experimental values obtained for two different wheel models.

1. Introduction

The use of mechanical parts manufactured with aluminum alloy have increased in the automotive industry due to its high strength and weight ratio, low cost, and innovative designs. In recent years, the market for aluminum wheels has grown considerably for transport vehicles. In the design departments, the geometry of the wheel is constantly modified and must meet certain design criteria, related to its shape, weight and manufacture process in order to guarantee safety. It is necessary to consider that the wheels support static and dynamic loads present during the operation of the vehicle [1]. Thus, the design of the wheel is constrained by performance parameters related to its strength and functionality.

The automotive industry is in permanent search of ways to reduce manufacturing times and the number of tests for product deployment. The design process can be improved through the use of computer-assisted engineering [2]. In the literature, many papers investigate the use of numerical tools to validate the design standards, the design criteria or propose modifications to the standards [3-5]. Numerical models formulated using the finite element method (FEM) can be used to evaluate the performance of prototypes, such as wheels, under standard tests [1,6-8]. Using the results of FEM, we can see the effect of modifications of the design of the wheel on its performance [4]. Riesner et al. [9] proposed the application of FEM for aluminum wheel structures for the simulation of the impact test. The FEM and fracture mechanics were used to simulate the impact test, such that the performance of the wheel could be predicted by using a deformation energy density approach. The results indicated that the impact resistance of the wheel could be evaluated by using a ductile fracture criterion.
Chen [10] pointed out that by using FEM and a total plastic work approach, we can determine the impact resistance of an aluminum rim with an orientation between 13 and 30 degrees of load impact. It was found that the use of a static analysis with a load equivalent to the dynamic impact can be achieved by an energy conservation method. Shang et al. [11] also proposed a simplified approach to the impact tests of the wheel without the tire portion. In the simplified approach, a percentage of the kinetic energy of the striker was reduced to compensate for the absence of the tire.

Russo [12] used a nonlinear dynamic approach to simulate the impact behavior in the wheel. He proposed that the influence of the tire on the absorption of part of the impact energy is approximated by subtracting a fixed percentage of the impact energy induced by the striker. The impact behavior on the wheels was strongly dependent on the properties of the material and the geometry of the rim edge. To reduce plastic deformation induced by the impact on the arms and the central region, the thickness of the rim edge must be adjusted to promote greater plastic deformation in the rim edge areas.

In this work, we investigate the use of a numerical model to evaluate damage in wheels following the established by NTC 4177 [13] standard, analyzing the effect of the impact on the wheel assembly. We perform experimental impact tests and compare the results. The paper is organized as follows: in the next section, we show the methodology, which includes a description of the numerical method, the failure criteria, and the experimental setup. In section 3, we compare the results generated by the numerical and experimental tests. The conclusions are given in the final section.

2. Methods and materials

2.1. Finite element analysis and failure criteria
Modeling the mechanical response of the wheel assembly is a complex task as it involves non-linear analyses. We use an explicit finite element formulation to consider dynamic effects and contact.

There are two failure criteria established in the standard for the impact test of the wheel: (i) there are no visible fractures that penetrate the central component of the tire assembly. No separation can be generated between the central component of the support and the rim, (ii) it is required to maintain the pressure at a specified value during a lapse of time of one minute after performing the impact test. In addition, the damage or deformation in the area of the edge of the rim that is in direct contact with the face of the striker is not considered as a fracture. The wheels are visually inspected using penetrating liquids after the impact test [13].

2.2. Experimental setup
The tests follow the NTC 4177 standard [13]. For the tests, only new wheels representing the wheels intended for passenger vehicle applications are used. In order to evaluate the mechanical performance, two impact tests are carried out. The scheme of the experiment device for impact testing is illustrated in Figure 1. The whole of the wheel and the tire is mounted on a fixing structure at an angle of 13° to the horizontal, such that its highest point is opposite to the striker that is in the vertical. Then, the striker, with a considerable mass, is allowed to fall freely from a height of 230mm, coming into contact with the outside edge of the wheel.

All indications to properly perform the test are described in the NTC 4177 standard. However, the most relevant for this study are summarized here for completeness. For the tire, it is recommended to remain at a pressure specified by the manufacturer or inflated at 200KPa. The surface of the striker must be at least 125mm wide and 375mm long. The striker is located on the tire and rim assembly, at a horizontal distance of 25mm from the edge of the wheel as shown in Figure 1. The fixing bolts must be manually tightened to a value of 80N.m or by a method recommended by the vehicle manufacturer.

To evaluate the properties of the AlSi10 aluminum alloy of the wheels, a specimen was taken from the center of the wheels. The chemical composition after the lab tests is shown in Table 1.

Specimens were fabricated from the spokes of the finished aluminum alloy wheels for a tensile test. The elastic modulus of the material is 73GPa, which is close to that reported by aluminum alloys in
the literature, around 70GPa. The elastic modulus $E$, yield stress $S_y$ and ultimate stress $S_u$ are shown in Table 2.

![Figure 1. Experimental setup for impact testing [13], (a) front view, (b) lateral view.](image)

### Table 1. Composition of AlSi10 aluminum.

| Si   | Fe | Cu | Mn | Mg | Zn | Ni | Pb | Cr | Ti | Ca | Na | Sn |
|------|----|----|----|----|----|----|----|----|----|----|----|----|
| 10   | 0  | 0.5| 0  | 0.07| 0  | 0  | 0  | 0.05| 0  | 0.002| 0  |
| 10.75| 0.22| 0.6| 0.03| 0.09| 0.01| 0.014| 0.02| 0.01| 0.07| 0.002| 0.02| 0.02|

### Table 2. Test results for mechanical properties.

|          | $E$ (MPa) | $S_y$ (MPa) | $S_u$ (MPa) |
|----------|-----------|-------------|-------------|
| Test 1   | 76.875.00 | 107.25      | 161.41      |
| Test 2   | 74.485.00 | 106.32      | 209.35      |
| Test 3   | 67.435.00 | 107.25      | 209.79      |
| Average  | 72.931.67 | 106.94      | 193.52      |

### 3. Results

The impact tests were performed on each of the two wheels, with a height of the striker of 400mm. To force damage, in a second test we remove the rubbers from the assembly. In Figure 2 we can see the results of this impact. Notice the places where the 604 model has fractures in its three arms. For model 706 5H the fracture can be seen on one side of the rim.

To predict the impact response of the two wheels we use the finite element software Ansys v18.1. The model 706 5H has a diameter of 17in and the model 604 has 15in, the two models are manufactured in the AlSi10 aluminum alloy. Contact conditions were assigned to the bolts and the holes, concentrically locating the wheel and the support. In addition, the lower face of the rim and the upper part of the support were joined, the complete set is shown in Figure 3.

We used unstructured meshes of quadratic tetrahedral elements with patch independent conditions. Convergence of the mesh is evaluated in order to find the optimum element size for each model, minimizing the discretization error [14]. An element size of 5mm was used for the mesh in the rim to
optimize the computation times, resulting in a reduction of 3.5 times the time of the process compared with finer meshes. For model 706, the mesh converges for an element size of 5.5mm, and 5mm for model 604. These values correspond to the meshes where the variation of the strain energy is considered low [15]. We also controlled the skewness of the elements to be below the value of 0.95.

![Figure 2. Fractured wheels: models (a) 706 5H and (b) 604.](image)

![Figure 3. (a) Model 706 5H and (b) 604 complete set [8].](image)

According to the NTC 4177 standard, the initial parameters are found for each simulation, such as the velocity, height of the striker and the mass of the same, see Table 3. For the tire, a normal pressure to the surface with a value of 0.2MPa was used, which is recommended in the standard. To simulate the tightening that occurs between the lower part of the wheel and the support, it was necessary to apply a preload of 28.57KN.

| N       | Mass (Kg) | Height (m) | Velocity (ms⁻¹) |
|---------|-----------|------------|-----------------|
| Simulation 1 | 490       | 0.23       | 2.12            |
| Simulation 2 | 490       | 0.4        | 2.8             |

Using the values obtained in the tensile test we calculate the strain energy density for damage. The result is 10.5N.mm⁻², for a fracture stress $\sigma_f = 193.52$MPa, a fracture strain $\varepsilon_f = 0.0563$ m.m⁻¹ and a value of the coefficient of hardening by deformation $n = 0.037$.  

4
Because the geometries of the wheels are based on real models, when applying the impact test there are no fractures present. As shown in Figure 4, the maximum value of the strain energy density that was presented in the model was 9.07 N.mm$^{-2}$, which was less than the critical value of 10.5N.mm$^{-2}$.

For the second simulation, we enforce damage conditions with a speed of 2.8m$s^{-1}$. It can be seen that the maximum value of the strain energy density is 16.76N.mm$^{-2}$, which is greater than 10.5N.mm$^{-2}$, thus, the rim presents fracture. In Figure 5, the rim presents three fractures in the arms, similar to the experimental data visualized in Figure 2.

The model 706 5H presented a behavior similar to the previous model. For the first simulation, there were no fractures. The maximum value was 5.96N.mm$^{-2}$. This model is more rigid, and it has a
lower strain energy density than the 604 model. It shows a lower plasticity and the stresses are greater compared with the other model. For the second simulation, the maximum value was 15.5N.mm\(^2\) and occurs only in one of the arms of the rim. In the model, the fracture appears on one side of the rim, as can be seen in Figure 6, with a value of 10.8N.mm\(^2\), which is greater than the critical value. The fracture is shown in a similar place to that observed for the experimental test in Figure 2, showing good agreement between the numerical and the experimental results.

In order to analyze the incidence of tire pressure on the computational model, another model was made, and it was concluded that the deformation energy density decreases to 11.22N.mm\(^2\). This is equivalent to 33% less the density obtained in the previous model. A similar way of working the previous model would be to apply what was established by Shang [11], and reduce the kinetic energy of the striker to compensate for the absence of the tire. From simulation 2, model 604, we observe that the strain energy density was reduced by 30%, the processing time was 20.33h.

To verify that the results obtained are in accordance with the experimental test proposed by the standard, we made the static model that resembles the calibration process. We apply a load of 1000Kg in the upper part of the support and verify the vertical displacement. The total displacement has a value of 3.94mm, and this is lower than that established in the standard of 7.5mm.

![Figure 6. Simulation 2: Model 706 5H. Strain energy density.](image)

4. Conclusions
We defined a numerical model for the impact test of wheels, with a mass of 490Kg and a speed of 2.12ms\(^{-1}\), as indicated by NTC 4177. We used the concept of strain energy density for determining damage, the wheel did not present fractures as it did not exceed the critical value of 10.5N.mm\(^{-2}\). The numerical model of the wheel presented a strain energy density greater than 10.5N.mm\(^{-2}\) when a speed of 2.8ms\(^{-1}\) was applied, the same was done in the experimental test to enforce damage conditions, verifying good agreement between both tests. The design of the support used for the numerical model complies with the parameters established in the NTC 4177 standard, with a displacement value of 3.99mm, less than the maximum allowable value of 7.5mm, for the static test.

Based on the data obtained by the numerical model, we can predict if the wheel needs a different geometrical configuration to guarantee the stipulated in the NTC 4177. The simulation times for the computational model depend highly on the complexity of each geometry to be used. The numerical model allows predicting in which places of the rim the fractures are more likely to occur, but not in all the situations the failure occurs exactly where the maximum value of the deformation energy density is located, as this also depends on the homogeneous conditions of the material.

Acknowledgments
The authors acknowledge the support given by the project Capital Semilla 1742, VIE, Universidad Industrial de Santander.
References

[1] Cerit M 2010 Numerical simulation of dynamic side impact test for an aluminium alloy wheel Sci. Res. Essays 5 2694–2701

[2] Mohamad M L, Rahman M T A, Khan S F, Basha M H, Adom A H and Hashim M S M 2017 Design and static structural analysis of a race car chassis for Formula Society of Automotive Engineers (FSAE) event J. Phys. Conf. Ser. 908 12042

[3] Ayestarán A, Graciano C and González-Estrada O A 2017 Resistencia de vigas esbeltas de acero inoxidable bajo cargas concentradas mediante elementos finitos Rev. UIS Ing. 16 61–70

[4] Chang C L and Yang S H 2008 Finite element simulation of wheel impact test J. Achiev. Mater. Manuf. Eng. 28 167–170

[5] Martínez J, Casanova E, Graciano C and González-Estrada O A 2018 Sensitivity analysis of a member under compression via Monte Carlo method Rev. UIS Ing. 17 179–184

[6] Chang C-L and Yang S-H 2009 Simulation of wheel impact test using finite element method Eng. Fail. Anal. 16 1711–19

[7] Naeimi M, Li Z, Petrov R H, Sietsma J and Dollevoet R 2018 Development of a new downscale setup for wheel-rail contact experiments under impact loading conditions Exp. Tech. 42 1–17

[8] Gómez-Rodríguez G D 2015 Modelado computacional del ensayo de impacto de un rin automotriz por el método de los elementos finitos según la norma ICONTEC 4177 (Bucaramanga: Universidad Industrial de Santander)

[9] Riesner M, Zebrowski M P and Gavalier R J 1986 Computer simulation of wheel impact test SAE Trans. 95 351–357

[10] Chen W 1996 Computational analysis of impact testing on aluminum wheels Hua Kang J. Eng. 10 163–184

[11] Shang R, Altenhof W, Li N and Hu H 2005 Wheel impact performance with consideration of material inhomogeneity and a simplified approach for modeling Int. J. Crashworthiness 10 137–150

[12] Russo C J 2001 The design and processing of cast aluminum wheels for impact performance SAE Technical Paper 2001-01-0749 (Detroit: Society of Automotive Engineers) pp 1–9

[13] Instituto Colombiano de Normas Técnicas y Certificación (ICONTEC) 1997 Vehículos automotores. Ruedas. Procedimiento para el ensayo de impacto, Norma Técnica Colombiana NTC 4177 (Colombia: Instituto Colombiano de Normas Técnicas y Certificación)

[14] Jin Y, González-Estrada O A, Pierard O and Bordas S P A 2017 Error-controlled adaptive extended finite element method for 3D linear elastic crack propagation Comput. Methods Appl. Mech. Eng. 318 319–348

[15] Rodríguez-Calderón W, Rojas-Aguero R, Yépez-Aguirre J and Pallares-Muñoz M 2017 Análisis de sensibilidad y de estabilidad numérica en el cálculo de factores de intensidad de tensiones en un caso de mecánica de fractura Rev. UIS Ing. 16 151–160