Finite element simulation and experimental validation of the cracking phenomenon in aluminium alloy wheels during the impact test

S. Otarawanna, P. Uttamung, and A. Malatip

Citation: AIP Conference Proceedings **2030**, 020303 (2018); doi: 10.1063/1.5066944
View online: https://doi.org/10.1063/1.5066944
View Table of Contents: http://aip.scitation.org/toc/apc/2030/1
Published by the American Institute of Physics

Finite Element Simulation and Experimental Validation of the Cracking Phenomenon in Aluminium Alloy Wheels during the Impact Test

S Otarawanna[1,a)], P Uttamung[1,b)] and A Malatip[1,c)]

[1]*Computer–Aided Engineering Laboratory, Design and Engineering Research Unit,*
National Metal and Materials Technology Center (MTEC),
National Science and Technology Development Agency (NSTDA),
114 Thailand Science Park, Thanon Pahonyothin,
Tambon Khlong Nueng, Amphoe Khlong Luang, Pathum Thani 12120, Thailand

[a)]somboono@mtec.or.th
[b)]piyamabu@mtec.or.th
[c)]atipongm@mtec.or.th

Abstract. The impact test is a mandatory mechanical test used to ensure the structural integrity of alloy wheels produced. In the impact test, a specified mass is dropped from a specified height onto the wheel-tyre assembly. In the wheel manufacturing industry, finite element analysis (FEA) is often used to reduce time and cost associated with the wheel design process. This study aims to find suitable methods for modelling the wheel impact test by FEA. In this research, two modelling methods (with and without the tyre) were used to simulate the impact test of a passenger-car wheel model. The wheel impact test was performed in order to validate the real test results with the FEA results. The results in this work suggest that both modelling methods are capable of predicting the cracking phenomenon in the wheel.

INTRODUCTION

Alloy wheels are a load-bearing component of the automobile which is critical to driving safety. According to widely accepted standards, such as SAE [1], JWL [2] and ISO [3], the impact test is one of the mandatory mechanical tests that wheels need to pass. The impact test is intended to ensure the structural integrity of wheels during a typical lateral curb impact event. In the impact test, a mass is dropped from a specified height onto the wheel-tyre assembly (Fig. 1).

In order to design alloy wheels to pass the impact test, finite element analysis (FEA) is often used. The use of FEA can save cost and time related to the design and testing process of wheels. Moreover, the wheel properties, such as strength and weight, can be improved by the redesigning process via FEA.

Considering the FEA of the wheel impact test, there are two main methods generally used in the literature which are (1) simulation with the tyre [4-6] and (2) simulation without the tyre [6]. Apparently, the simulation with the tyre represents the real physics better than the one without the tyre. However, the simulation with the tyre needs considerable effort in creating the tyre model consisting of different layers of materials, such as rubber, steel and canvas. Furthermore, the simulation with the tyre uses more computational resource than the simulation without the tyre does. The question then arises whether the two simulation methods provide similar results. As far as the present authors are concerned, there has been no report in the open literature directly investigating this issue.

In this work, we studied a passenger-car wheel model in which cracks were observed during the impact test. The simulations with the tyre and without the tyre were performed on this wheel model. After comparing the FEA results with the experimental results, the effectiveness of each simulation method in predicting wheel cracking in the impact test was evaluated.

Green Design and Manufacture: Advanced and Emerging Applications
AIP Conf. Proc. 2030, 020303-1–020303-7; https://doi.org/10.1063/1.5066944
Published by AIP Publishing. 978-0-7354-1752-6/$30.00

MATERIALS AND METHODS

Fig. 2(a) shows the A356.2 aluminium alloy passenger-car wheel model used in this research. The wheel was cast by the low-pressure die casting process and subsequently heat treated to the T6 condition. In order to obtain the Young's modulus value and the stress-strain relationship in the plastic region for the FEA, specimens were machined from finished wheels for uniaxial tensile testing. The uniaxial tensile testing was performed in compliance with the ASTM B557 – 06 standard [7]. Table 1 and Table 2 show the Young's modulus value and the stress-strain relationship obtained from the testing, respectively. For other properties required for the FEA, the values of Poisson's ratio and density were taken from Ref. [8] (Table 1).

The wheel impact test was setup according to the SAE J175 – 1996 standard [9] as shown in Fig. 1. In the test, the wheel-tyre assembly with the tyre inflation pressure of 0.45 MPa was mounted at 13 ± 1° to the horizontal direction. The striker was freely dropped from a height of 230 ± 2 mm above the highest point of the wheel-tyre assembly. When the striker touched the tyre, the edge of the striker overlapped the rim flange by 25 ± 1 mm. In this research, a number of different striker weights were used in order to find the critical weight causing wheel cracking. It was found that the critical load for the wheel model in this study is somewhere between 690 kg and 850 kg (Table 3). For the wheel tested with the 850 kg striker mass, cracks were observed at the so-called rim-to-spoke junction (the region between the rim and spoke) (Fig. 2(b)).

Regarding the FEA of the wheel impact test, commercial finite element software Altair RADIOSS was employed to perform the transient dynamic analysis with the explicit solver. Due to the symmetry of geometry and boundary conditions, only one half of the wheel was modelled. Figs. 3(a)-(b) show the finite element models for the simulations with the tyre and without the tyre, respectively. In both modelling methods, the wheel model consists of 29,453 nodes and 97,965 elements. In the simulation with the tyre, the tyre model consists of 105,018 nodes and 63,810 elements. The wheel model was mainly meshed with hexahedral elements. Some portion of wheel was constructed by tetrahedral elements due to its relatively complex geometry. For the tyre model, it consists of rubber compounds, fabric and steel belts. The striker was modelled as a rigid body.

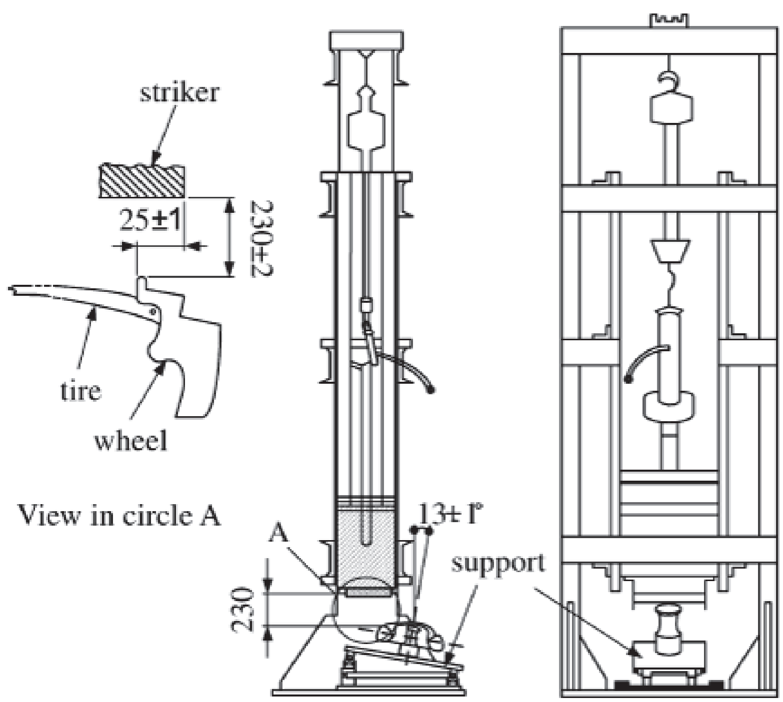

FIGURE 1. Setup of the wheel impact test (The figure is reproduced from Ref. [4].

FIGURE 2. (a) Wheel geometry and (b) crack locations. In (b), all arrows mark the crack locations whilst the white one also indicates the valve hole (v/h) location.

TABLE 1. Values of material properties of the wheel material used in this work

Material property	Value
Young's modulus	72 GPa
Yield strength	146 MPa
Ultimate tensile strength	230 MPa
Poisson's ratio	0.33
Density	2,670 kg/m^3

TABLE 2. Relationship between plastic strain and stress of the wheel material used in this work

Plastic strain (%)	Stress (MPa)
0	146
0.5	155
1	165
1.5	173
2	182
2.5	189
3	197
3.5	203

TABLE 2. Relationship between plastic strain and stress of the wheel material used in this work (continued)

Plastic strain (%)	Stress (MPa)
4	210
4.5	215
5	221
5.5	225
6	230

TABLE 3. Summary of the wheel impact test and FE results

Striker weight (kg)	Test result	Max stress from FEA simulation with tyre	Max stress from FEA simulation without tyre
690	Pass	221 MPa	226 MPa
850	Cracks observed at rim-to-spoke junction	231 MPa	236 MPa

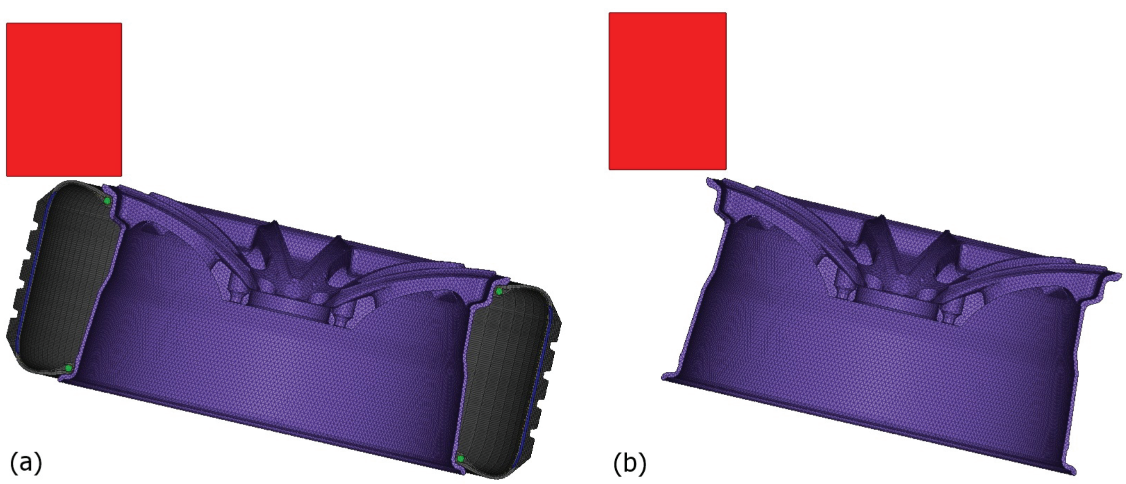

FIGURE 3. FEA modelling (a) with the tyre and (b) without the tyre

RESULTS AND DISCUSSION

The FEA simulations with the tyre and without the tyre were performed at the striker loads of 690 and 850 kg. In the simulation results in Figs. 4-7, it is noted that we should ignore the location where the striker directly contact the wheel when considering the location of the maximum von Mises stress. This is because the stress direction in the direct impact location is mainly compressive and therefore cracks are not likely to form in this location. Figs. 6-7 show that both simulation methods give the location of the maximum von Mises stress correspond with the crack location observed in the real test with the striker load of 850 kg (Fig. 2(b)). As summarised in Table 3, the simulation with the tyre gives a lower value of the maximum von Mises stress compared to the simulation without the tyre in both striker load cases (≈5 MPa less in both the 690 and 850 kg cases). This is related to the fact that the tyre absorbs the impact energy from the striker resulting in a less amount of the impact energy transferring to the wheel.

Considering the stress value causing cracking, both simulation methods give the maximum von Mises stress value of 231 and 236 MPa for the 850 kg case (Table 3). As these predicted stress values are higher than the ultimate tensile strength of the wheel in this study (230 MPa), the simulation results sound reasonable in predicting the crack phenomenon in this wheel. For the 690 kg case, both simulation methods give the maximum von Mises stress value of 221 and 226 MPa (Table 3) which are less than the ultimate tensile strength of the wheel. Therefore, both simulation methods are accurate in predicting no crack in the 690 kg case.

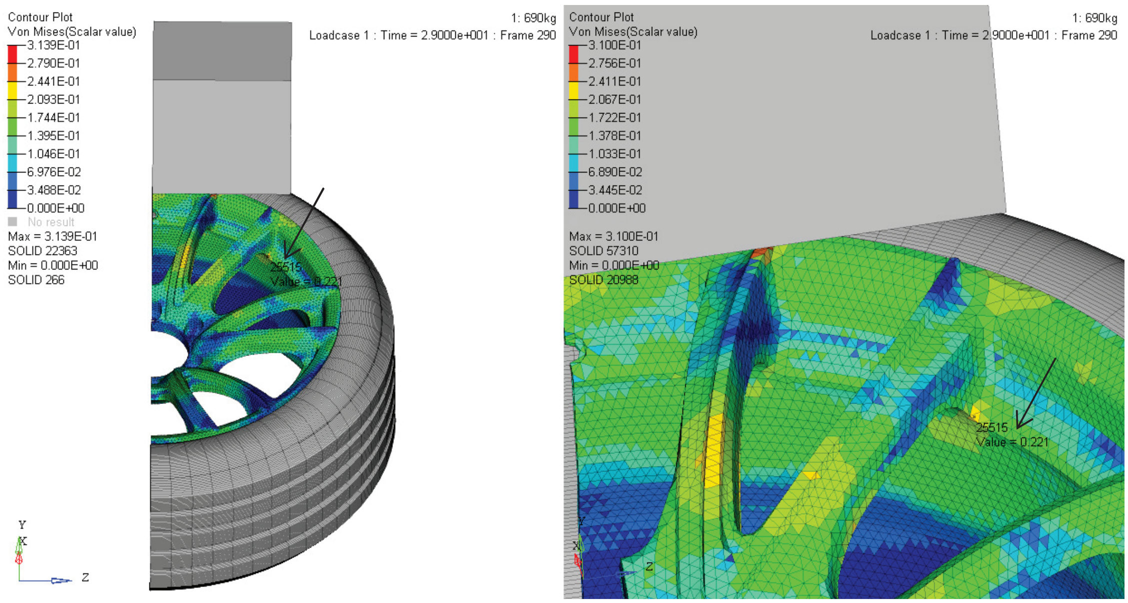

FIGURE 4. Contour of the von Mises stress obtained from the simulation with the tyre at 690 kg. Arrows mark the location where the von Mises stress is maximum (ignoring the direct impact region).

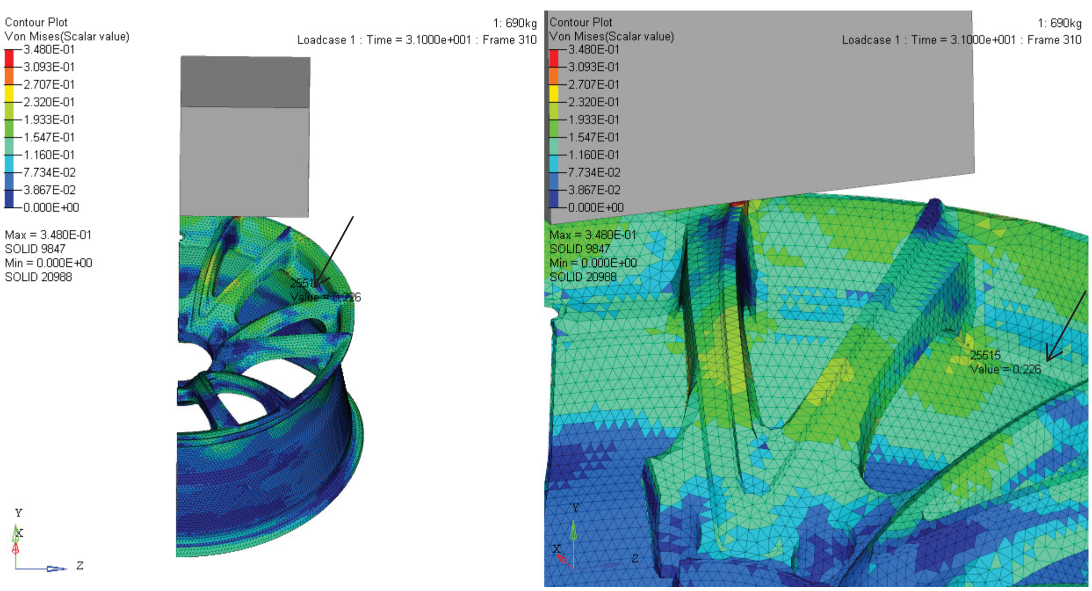

FIGURE 5. Contour of the von Mises stress obtained from the simulation without the tyre at 690 kg. Arrows mark the location where the von Mises stress is maximum (ignoring the direct impact region).

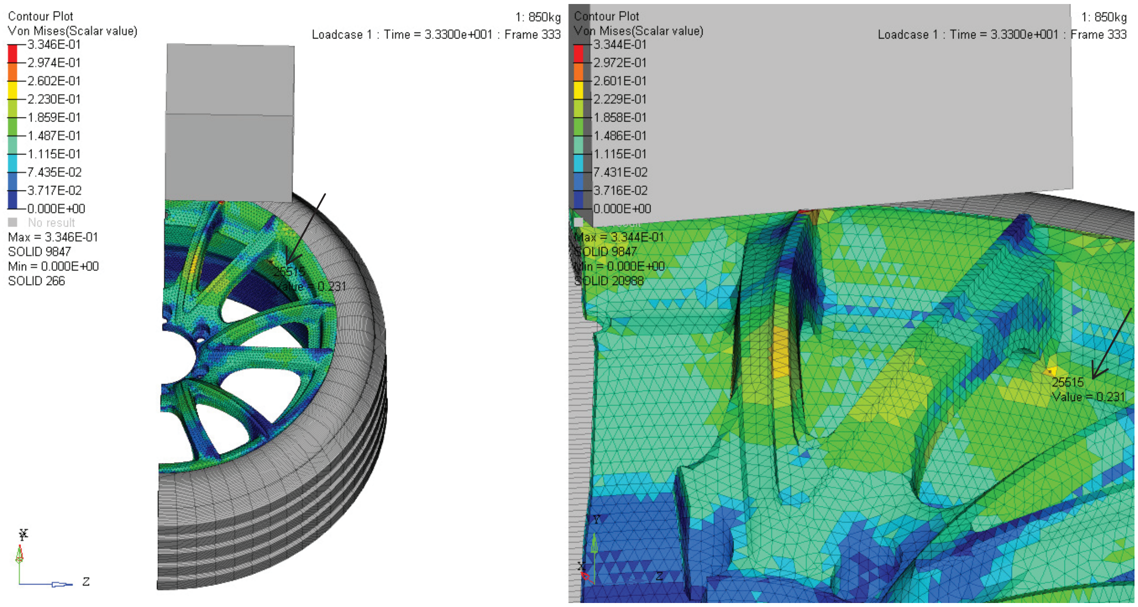

FIGURE 6. Contour of the von Mises stress obtained from the simulation with the tyre at 850 kg. Arrows mark the location where the von Mises stress is maximum (ignoring the direct impact region).

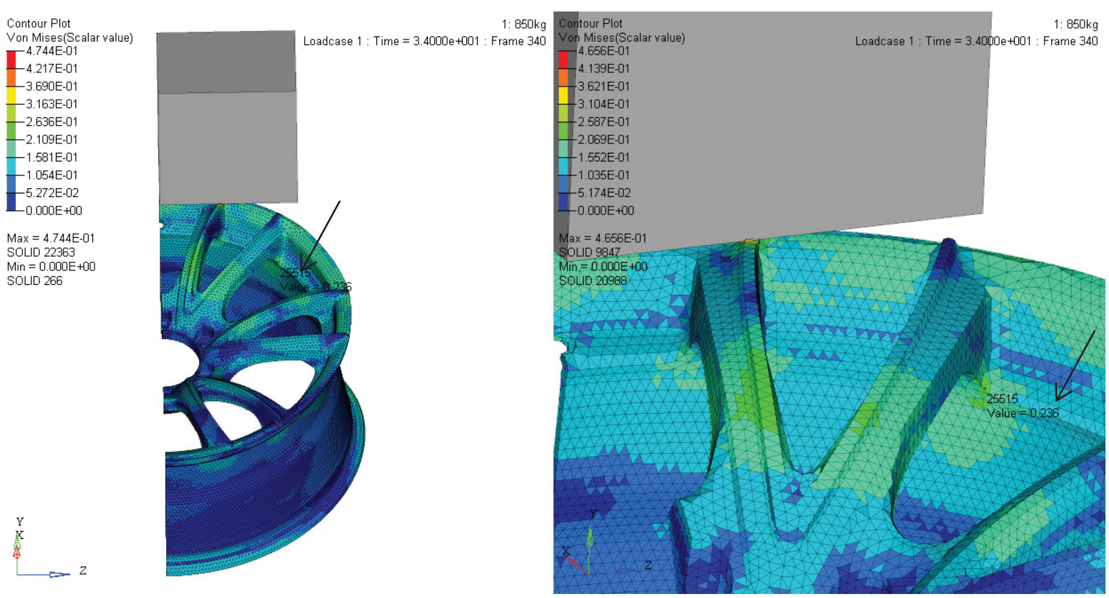

FIGURE 7. Contour of the von Mises stress obtained from the simulation without the tyre at 850 kg. Arrows mark the location where the von Mises stress is maximum (ignoring the direct impact region).

CONCLUSIONS

In this work, two FEA modelling methods (with and without the tyre) were used to simulate the impact test of a wheel model. From the simulation and real test results, it can be concluded that both methods are capable of predicting the cracking phenomenon in this wheel. In both the 690 and 850 kg striker load cases, the simulation with the tyre gives a lower value of the maximum von Mises stress (≈5 MPa less) compared to the simulation without the tyre. This is attributed to the fact that the tyre absorbs the impact energy from the striker resulting in a less amount of the impact energy transferring to the wheel.

ACKNOWLEDGEMENTS

This work was supported by Lenso Wheel Co., Ltd., Thailand.

REFERENCES

1. SAE J328_201603 Wheels – Passenger Car and Light Truck Performance Requirements and Test Procedures (Society of Automotive Engineers, Warrendale, PA, 2016).
2. Information on https://www.jwtc.jp/open/html/e/situmon1.html.
3. ISO 3006:2015 Road vehicles – Passenger car wheels for road use – Test methods (International Organization for Standardization, 2015).
4. C.-L. Chang and S.H. Yang: Eng. Fail. Anal., **16**, p. 1711 (2009)
5. M. Cerit: Scientific Research and Essays, **5**, p. 2694 (2010)
6. W.-C. Chang, "Modelling for Impact Test of Aluminum Wheels," Ph.D. Thesis, Department of Mechanical Engineering, National Central University, Taiwan, 2008.
7. ASTM B557 – 06 Standard Test Methods for Tension Testing Wrought and Cast Aluminum- and Magnesium-Alloy Products (ASTM International, West Conshohocken, PA, 2006).
8. J.G. Kaufman and E.L. Rooy, "Aluminum Alloy Castings: Properties, Processes and Applications (ASM International)," 2004, p. 76.
9. SAE J175 – 1996 Wheels – Impact Test Procedure – Road Vehicles (Society of Automotive Engineers, Warrendale, PA, 1996).