

Application of Impact Analysis for Aluminum Wheel with Inflated Tire

Satoshi ISHIKAWA¹, Yasuki SAKAI², and Norimasa NOSAKA²

[1]IDAJ Co., Ltd. and [2]BBS Japan Co., Ltd.

Abstract: Aluminum Wheel is one of the critical components in order to improve convenience and safety in an automobile vehicle. The impact test has been one of the most important subjects for the strength requirements of Aluminum wheel. The 13 degree lateral impact test was regulated by SAE and/or JWL. Furthermore, 90 degree vertical impact test was required by Automobile manufacturers in recent years. BBS Japan Co., Ltd has conducted not only 13 degree impact test but also 90 degree test. While these impact tests caused the expensive progress of work, the necessity of preliminary investigation was highly demanded on the design stage. Therefore, BBS Japan and IDAJ Co., Ltd have worked in close cooperation to analyze the wheel impact test by using CAE (Abaqus/Standard and Explicit), and improved the accuracy of analysis results by comparing with the experiments.

In this paper, we discussed the application of impact analysis of aluminum wheel. The analysis model consisted of aluminum wheel and tire structure. The tire structure model was naturally inflated with pressure by Abaqus/Standard, and the results of implicit analysis were imported to Abaqus/Explicit for the dynamical impact analysis afterward. Both results of 13 and 90 degree impact analyses were compared with the experimental data; the performance and accuracy were examined.

Keywords: Aluminum Wheel, Dynamics, Impact, and Tire.

1. Introduction

Aluminum Wheel is one of the critical components in order to improve convenience and safety of automobile vehicles, therefore, impact tests have been required for the guarantee of safety condition. SAE (SAE, 2003) regulates the 13 degree lateral impact test which is shown in Figure 1. In recent years, Automobile manufacturers require more arduous test which is 90 degree vertical impact test shown in Figure 2. By contrast, those impact experiments require high cost including human resources.

To achieve a higher level of design, the practical use of CAE system is significant. In this paper, we discussed the application of impact analysis of aluminum wheel. The analysis model was composed of aluminum wheel and tire structure. The tire structure model was naturally inflated with pressure by Abaqus/Standard, and the results of implicit analysis were imported to Abaqus/Explicit for the dynamical impact analysis afterward. Both results of 13 and 90 degree impact analyses were compared with the experimental data; the performance and accuracy were

examined. We emphasize that our proposal analysis methods can help us design more robust and sophisticated Aluminum Wheel.

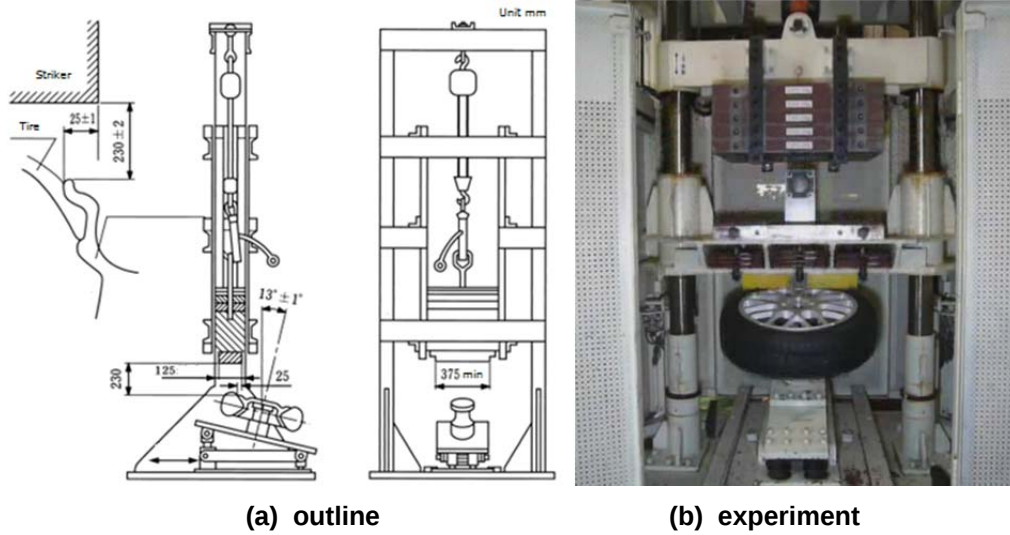


Figure 1. 13 degree test.

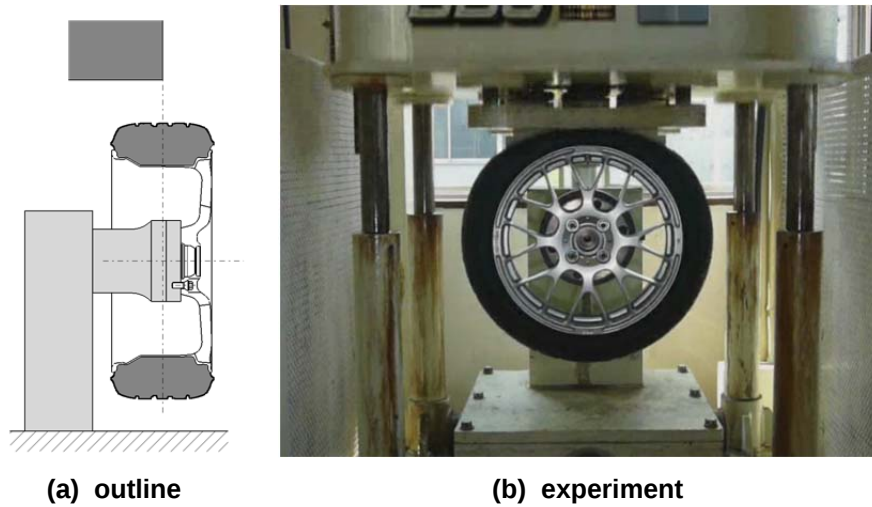


Figure 2. 90 degree test.

2. Model Description

2.1 13 degree model

To meet the SAE standard requirements shown in Figure 3, FEA model was assembled with four parts in our modeling study. Figure 4 shows the 13 degree impact test model. The model consisted of Aluminum wheel, test jig, rigid striker and tire.

This analysis model was formed with two steps. In the first step, the tire was statically inflated to regulated 200kPa pressure; this process was analyzed by Abaqus/Standard. In the second step, the impact dynamic analysis was executed by Abaqus/Explicit. Naturally, the first step static results were imported to the second dynamic analysis as an initial condition.

In the first simulation, the inflation step was performed on the half model of three dimensional tire. Figure 5 shows the detailed structure of tire. The main part of tire was modeled with three dimensional hexahedron solid elements which were applied to the simple Neo hookean hyper elastic material. The belts and carcass were modeled with rebar in surface elements embedded in continuum elements. Furthermore, the steel bead was modeled with three dimensional solid elements. The most important wheel model was generated as 10nodes modified tetrahedron solid element. The sample test piece for the stress-strain behavior was cut out from the formed wheel.

In order to set clear boundary and loading conditions in the dynamic procedure, the FE model involved test jig part as Figure 4. In addition, the test jig model was supported by blue dashed connector elements which figured the rubber inelastic behavior. The initial velocity which was assumed the free dropping above 230mm height was applied to the striker. This height is provided by SAE.

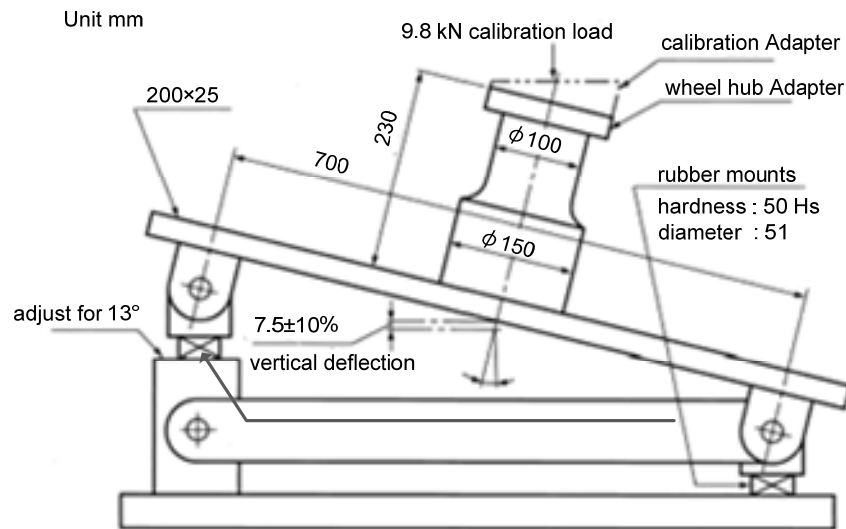


Figure 3. Specification of aluminum wheel 13 degree impact test.

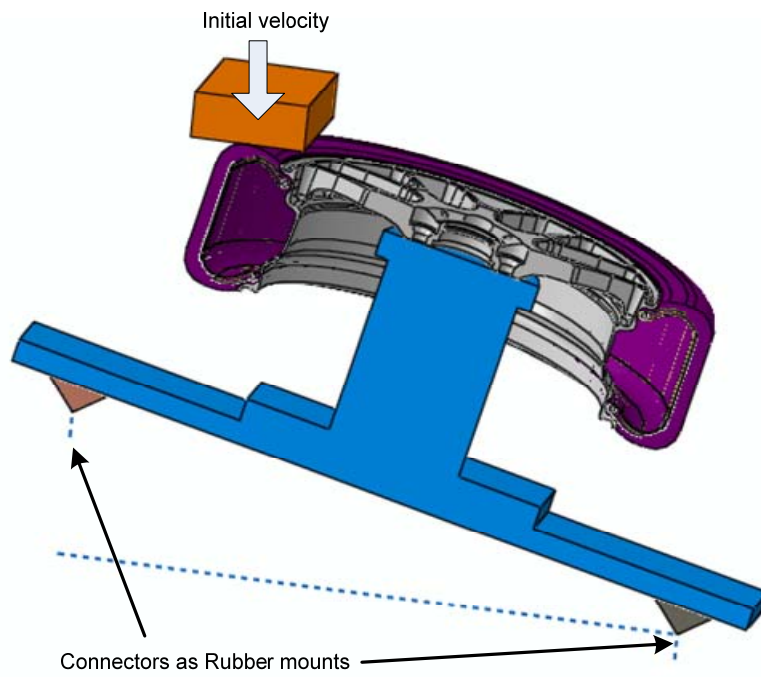


Figure 4. 13 degree analysis model.

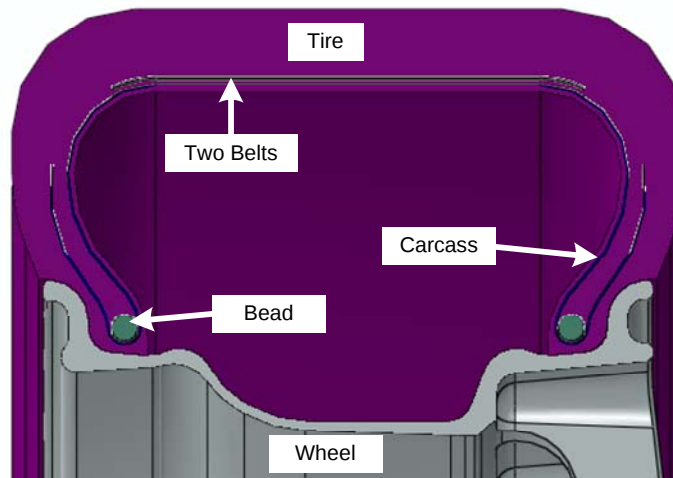


Figure 5. Structure of tire model.

2.2 90 degree model

Figure 6 shows the 90 degree analysis model. This model was formed with two steps as same as 13 degree analysis model. The first step was tire inflation analysis by Abaqus/Standard, following, the impact dynamic analysis was executed by Abaqus/Explicit at the second step. The tire pressure varied with the size of wheel in the actual test. In this analysis, the tire pressure was applied to 260kPa.

The tire structure and the material of wheel were almost same as 13 degree model. The entire fixed boundary condition was applied on the center face of the wheel. The initial velocity which was assumed the free dropping above 101.6mm height was applied to the striker under the explicit dynamic analysis.

The option of *CONTACT PAIR with friction conditions were invoked between tire and wheel, and also between striker and tire. While general contact condition was applied to whole area such as tire's self-contact condition in different coefficient of friction.

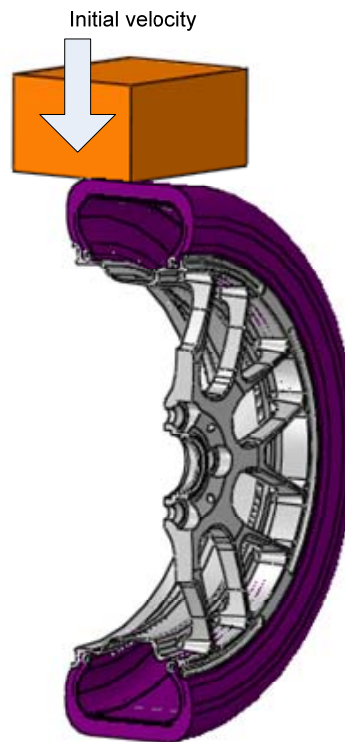


Figure 6. 90 degree analysis model.

2.3 Model size

Table 1 shows the typical model size and element types of each parts. The material of wheel was Aluminum alloy which was composed from Al-Mg-Si. The tire was modeled as a slightly compressible hyper elastic material, and the fiber reinforcement was modeled as a linear elastic material at the parts of belts and carcass. The bead and the base part which was used in only 13 degree analysis were modeled as ordinary steel.

Table 1. Model size.

Part	Element type	Number of nodes	Number of elements
Wheel	C3D10M	150249	86465
Tire	C3D8R	10080	6958
Bead	C3D8R	1188	520
Belt	SFM3D4R	2040	1848
Carcass	SFM3D4R	2880	2769
Base (13 degree model only)	C3D10M	59455	38968

3. Results

3.1 13 degree model

In order to examine our proposal model, two analyses were performed. One analysis was with tire model, and the other was without tire model. The section outlines of original shape and deformed configurations are shown in Figure 7. The black thin line shows the original shape, the black solid line shows the deformed shape which was real experimental result, the blue line indicates the result of FEA model with inflated tire, and the red line indicates the result without tire. The result of FEA model with inflated tire coincided with the experimental result. On the other hand, if the tire model was omitted, the permanent displacement result was calculated larger than it with tire model. These results insist that even in the 13 degree analysis case the tire model is essential.

Figure 8 shows the principal strain contour of dynamic implicit analysis, while Figure 9 shows the real test result which had the crack on the side of spoke. In this investigation, we adopted the trial wheel model for reducing weight. As Figure 8 describes, the analysis model was able to estimate the actual crack area.

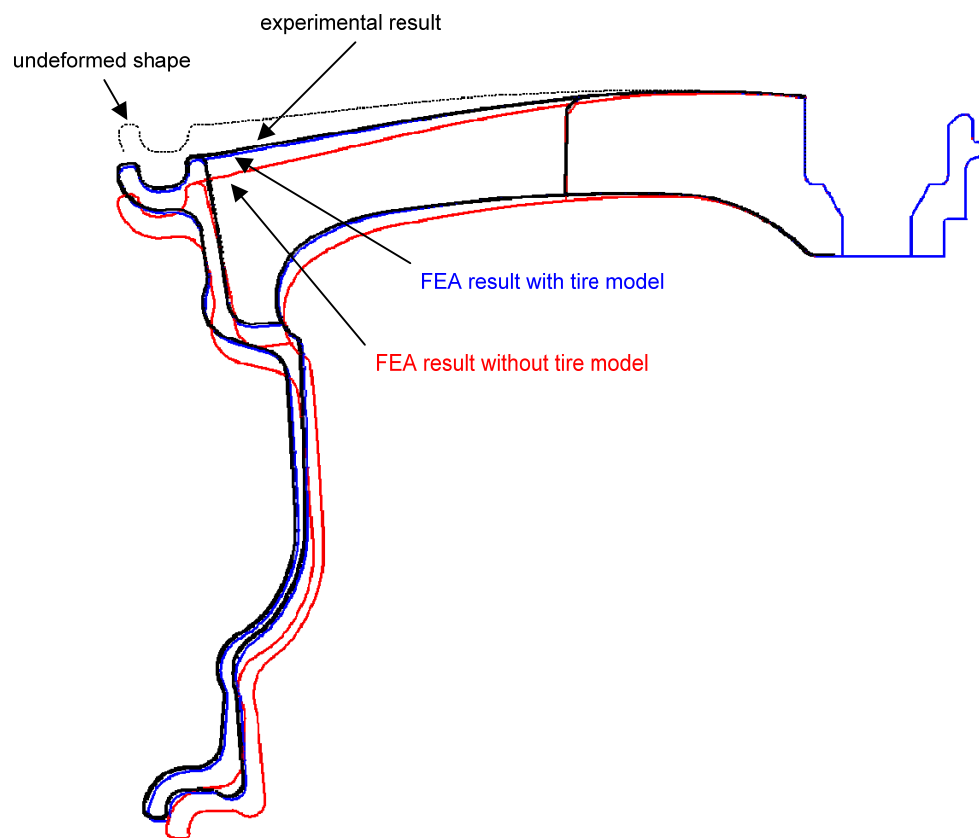


Figure 7. Comparison of deformed outlines (13 degree analysis).

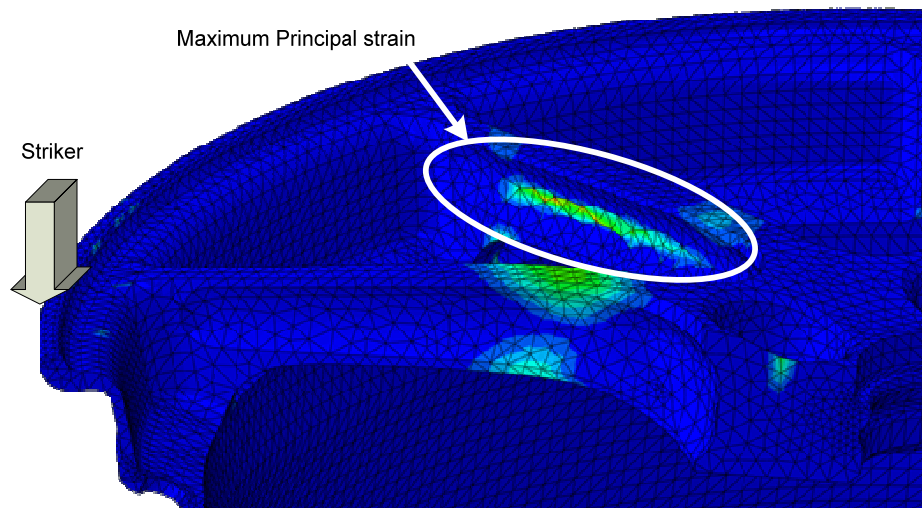


Figure 8. Thin shaped wheel model (Principal strain).

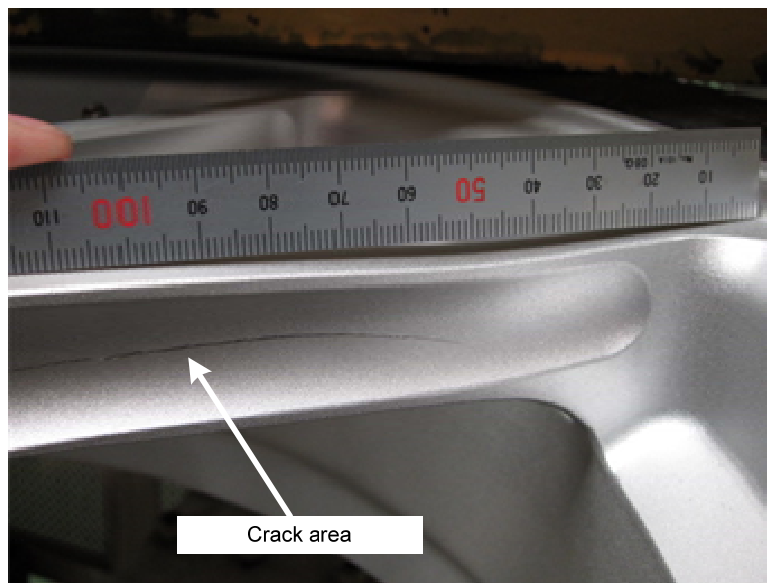


Figure 9. Thin shaped wheel (Real test).

3.2 90 degree model

The section outlines of original shape and deformed configurations are shown in Figure 10. The black line shows the original shape, the red line shows the deformed shape which was real experimental result, and the blue line indicates the result of FEA model with inflated tire. There was a slight difference between the FEA model and the experimental result. It was thought that the ratio of energy absorb of FEA model was larger than it of real phenomena.

Figure 11 and 12 show the deformation configuration as a whole and a part model respectively. Both Figures illustrate the maximum deformation instance. Figure 13 shows the mirror deformation configuration, while Figure 14 shows the deformation of real experimental test.

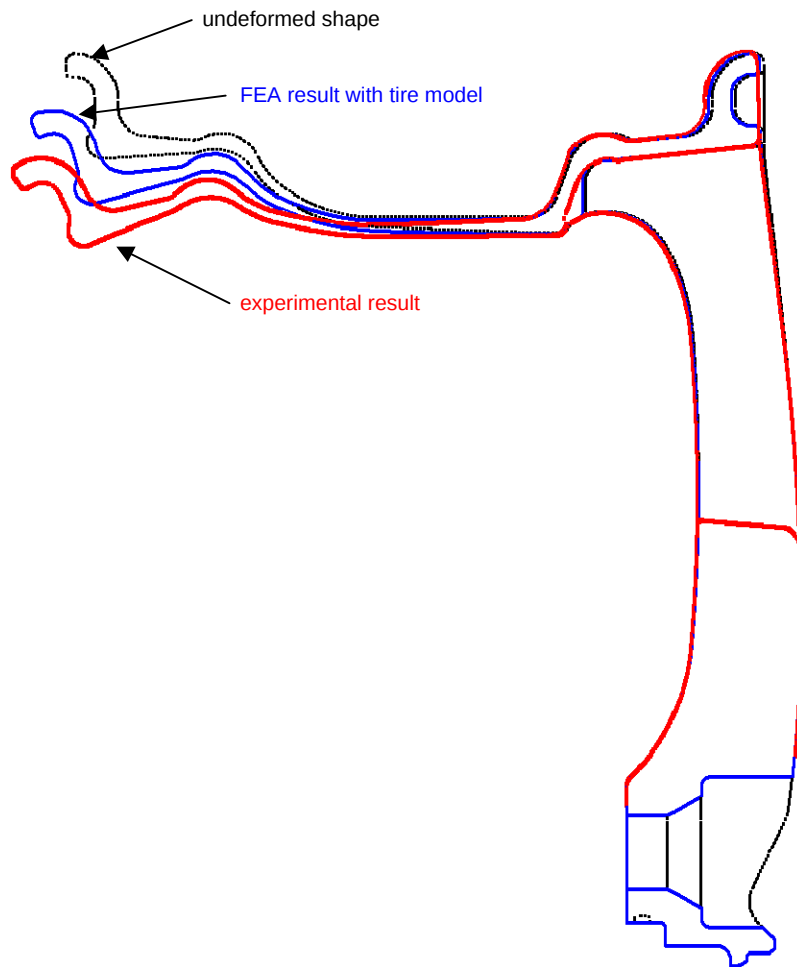


Figure 10. Comparison of deformed outlines (90 degree impact test).

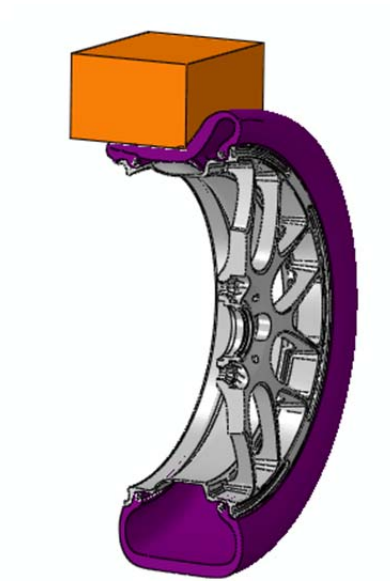


Figure 11. Whole deformation (90 degree model).

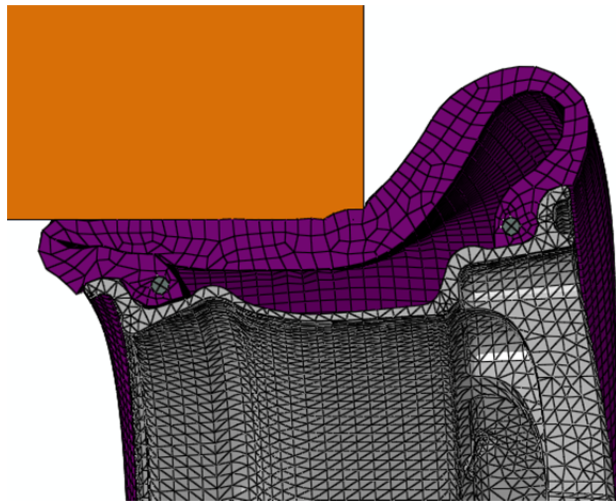


Figure 12. Part of deformation (90 degree model).

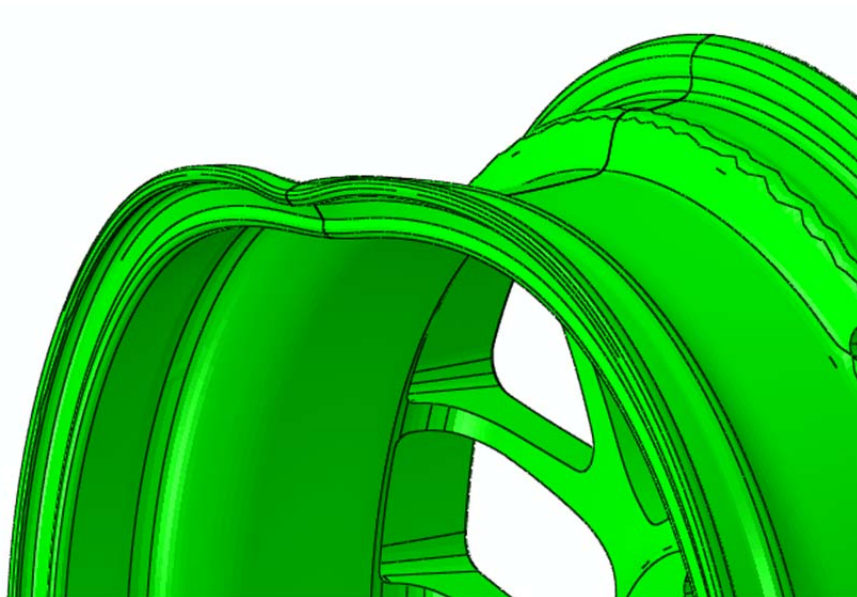


Figure 13. Mirror deformation configuration.



Figure 24. Result of real experimental test.

3.3 Analysis time

Table 2 shows the analysis time (wallclock time in the Abaqus output file) with parallel processing under Linux operating system and a computer having two Intel Xeon E5645s (2.40GHz processor six cores). The version of Abaqus was 6.12-3.

Table 2. Analysis time.

Analysis model	Analysis type	Phenomenon time [sec]	Cores	Wall clock time [hh:mm]
Tire inflation	Static	-----	4	00:45
			8	00:26
13 degree impact	Explicit	0.10	4	63:50
			8	49:55
90 degree impact	Explicit	0.16	4	130:34
			8	87:49

4. Conclusion

We established the FE analysis model for 13 and 90 degree Aluminum impact test with inflated tire. Abaqus has the seamless connection between implicit and explicit procedures. This function is substantial for integrating impact analysis with inflated tire. The study shows the following results:

- 13 degree test
 - o The results using inflated tire model simulate the real impact test (Comparing without tire model).
 - o Obtained results are in good agreement with observed crack zones.
- 90 degree test
 - o There was a subtle difference between the calculation and the experiment.
 - o As one reason, it was thought that there were some problems for the tire model construction.
 - o Further research on tire modeling would improve the difference.

5. References

1. SAE J 175, "Wheel – Impact Test Procedure – Road Vehicle", 2003.
2. Abaqus User's Manual, Version 6.12, Dassault Systèmes Simulia Corp., Providence, RI.