Application of CAE to Aluminum Wheel Impact Test Analysis Using Dynamic Implicit

Juntak Shin, Yeonsang Yoo, Sungjin Yoon

Hyundai Motor Company

Abstract: Aluminum Wheel Impact Test (13 Degree Lateral, SAE J157) is the way in order to guarantee the aluminum wheel strength. CAE Team in Hyundai Motor Company has been carrying out the wheel impact test analyses using CAE (Abaqus/Standard), and developing the accuracy of analysis results by comparing with the tests.

Before this study, we carried out the aluminum wheel impact test analysis with only wheel model (without test jig). And, we used static analysis. The impact load which used in static analysis was calculated by equation related to the impact theory. But, this analysis was not sufficient condition in order to simulate the real test. It must be considered the test jig stiffness. Sometimes, not considering test jig analysis result didn't show the highest stress area (especially, nearby bolt hole). Also, dynamic behavior in analysis is more effective if considering.

In this study, aluminum wheel impact test analysis will carry out with test jig model. And we'll use dynamic implicit analysis (Abaqus/Standard) in order to simulate the rear impact test.

Keywords: Aluminum Wheel, Aluminum Wheel Impact Test (13 Degree Lateral), Aluminum Wheel Impact Test Jig, Dynamic Implicit Analysis

1. Introduction

Aluminum Wheel Impact Test (13 Degree Lateral, SAE J157) is the way to prove its impact ability when the sudden crash (curb impact) of the wheel. This test establishes minimum performance requirements and related uniform laboratory test procedures for evaluating axial (lateral) curb impact collision properties of all wheels intended for use on passenger cars.

Figure 1. shows the Aluminum Wheel Impact Test (13 Degree Lateral, SAE J157) Machine.

The test machine shall be one in which an impact loading is applied to the rim flange of a wheel complete tire. The wheel shall be mounted with its axis at an angle of 13 degrees ± 1 degree to the vertical so that its highest point is presented to the vertically acting striker. The impacting face of the striker system shall be at least 125mm wide and at least 375mm long.

Before this study, we carried out the aluminum wheel impact test analysis with only wheel model (without test jig). Also, we used static analysis. The impact load which used in static analysis was calculated by equation related to the impact theory. But, this analysis didn't have good boundary & loading conditions for considering test jig stiffness and simulating the real test. Sometimes, not considering test jig analysis result didn't show the highest stress area (especially, nearby bolt hole). Also, dynamic behavior in analysis is more effective if considering.

In this study, aluminum wheel impact test analysis will carry out with test jig model. And we'll use dynamic implicit analysis (Abaqus/Standard) in order to simulate the rear impact test. And, we'll confirm the below.

1) The results using test jig model analysis show the real highest stress area (especially, nearby bolt hole).
2) The results using dynamic implicit analysis simulate the rear impact test (Comparing with test results).

Also, we assumed that there is no impact energy absorption of the tire. We couldn't consider the tire (EF) model in analysis. At this study time we didn't get sufficient tire (FE) models and material (rubber) information. Considering the tire model study is expected in the future.

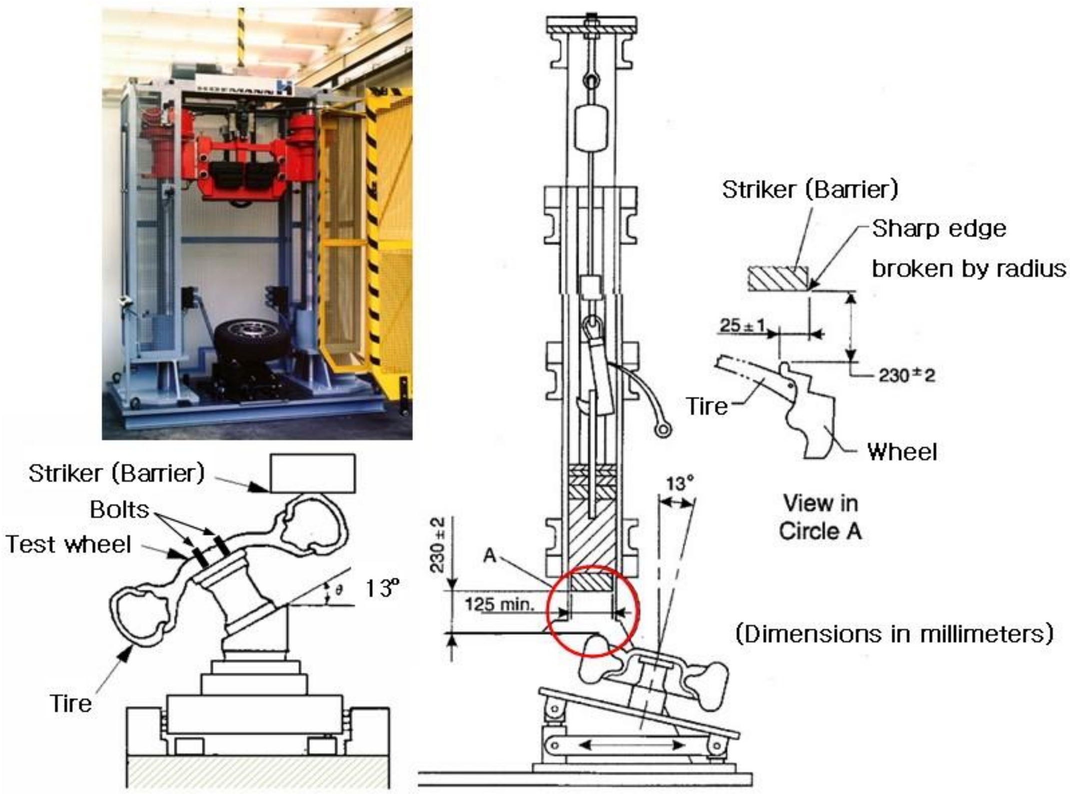


Figure 1. Aluminum Wheel Impact Test(13 Degree Lateral, SAE J157) Machine

2. Impact Analysis Using Static (Without Test Jig Model)

This analysis method is how to do aluminum wheel impact analysis in the past when we had no information of test jig and material.

From the impact theory, the impact load is a function of velocity and square root of the mass. In this test, the impact striker's height is always same (230mm). Therefore, the velocity of the moment when the striker contacts the aluminum wheel is constant. In conclusion, the impact load is a function of square root of the striker mass. The striker mass is determined by the weight of the vehicle.

$$F = \alpha\sqrt{M}$$

where, α is the correlation coefficient from the test results.

(This coefficient is confidential in our company)

Figure 2. shows the boundary and loading conditions of the aluminum wheel impact analysis using static. We used only aluminum wheel model (without test jig model). Also, there were no bolts in this analysis model. The impact load applied mentioned equation. Aluminum wheel hub side and bolt holes side were assumed displacement constraint. It was the boundary condition in this analysis method. Contact was defined between the aluminum wheel flange and striker.

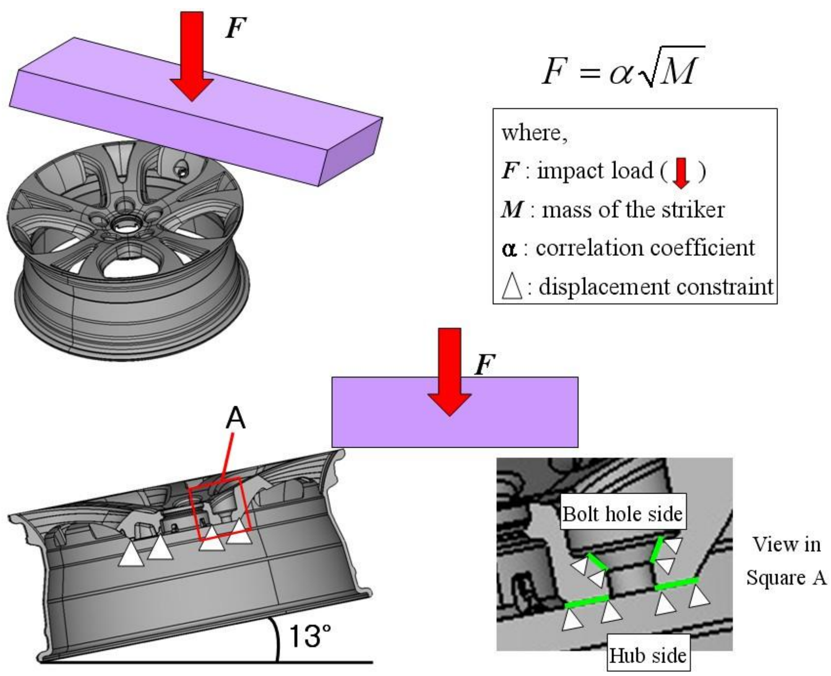


Figure 2. Impact Analysis Boundary & Loading Conditions Using Static (Without Test Jig Model)

3. Impact Analysis With Test Jig Model Using Dynamic Implicit

There were several assumptions in impact analysis using static. The impact load was from the mentioned equation. Also, there was no test jig model. Clear boundary and loading conditions were needed in order to ensure accurate analysis result.

In order to set clear boundary and loading conditions, FE test jig model was applied in this study. In addition, for more accurate impact simulation, the dynamic analysis was applied.

Figure 3. shows the specifications of the aluminum wheel impact test jig.

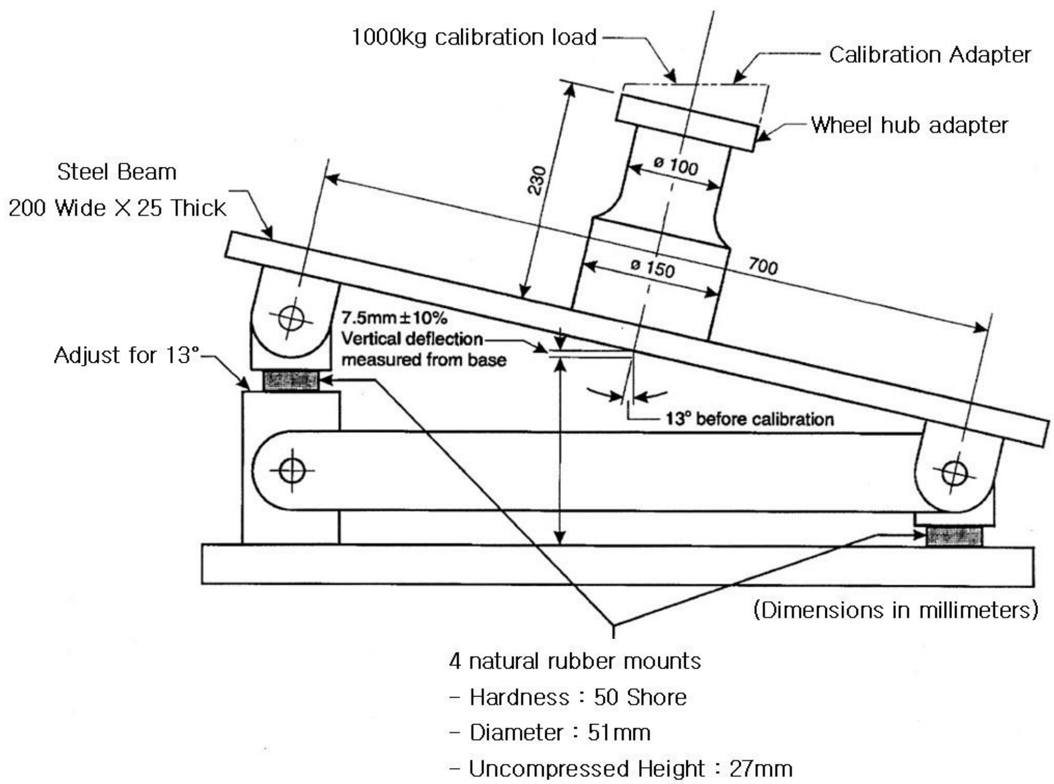


Figure 3. The Specifications of The Aluminum Wheel Impact Test Jig

The wheel mount fixture dimensions are shown in Figure 3. The width of the support beam is 200mm. Although the specified material for the beam is steel, high yield strength steel is recommended to prevent deformation. All pivot joint in the wheel mount fixture should be free to rotate. Shoulder bolts are recommended to prevent binding. With the test calibration adapter located at the mid-span of the beam, a vertical mass of 1000kg shall be applied to the center of the wheel mount as shown Figure 3. The vertical central deflection shall be 7.5mm±0.75mm when measured at the center of the beam.

Figure 4. shows the FE test jig model for aluminum wheel impact analysis using dynamic implicit. The material information of steel and rubber in test jig was provided from the test lab.

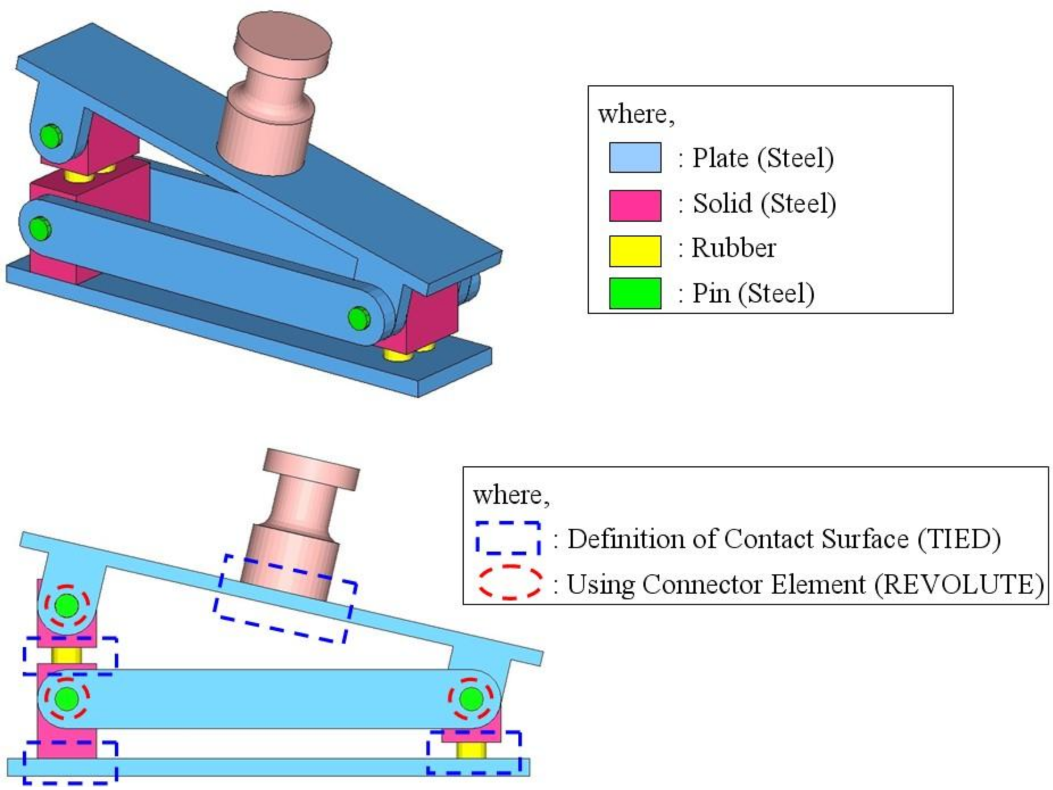


Figure 4. The FE Test Jig Model for Aluminum Wheel Impact Analysis

Figure 5. shows the step of the aluminum wheel impact analysis using dynamic implicit with FE test jig model. Impact analysis using test jig model proceeded to step 2.

In first step, aluminum wheel was bolted to the test jig using static analysis (bolt pretension). Contact was defined between the aluminum wheel hub side and the wheel hub adapter in test jig. In second step, the striker impacts to the aluminum wheel rim flange with initial specific velocity using dynamic implicit analysis.

We internally evaluate that dynamic implicit analysis is more reliable than dynamic explicit analysis in evaluating stress. This is the reason we used dynamic implicit analysis.

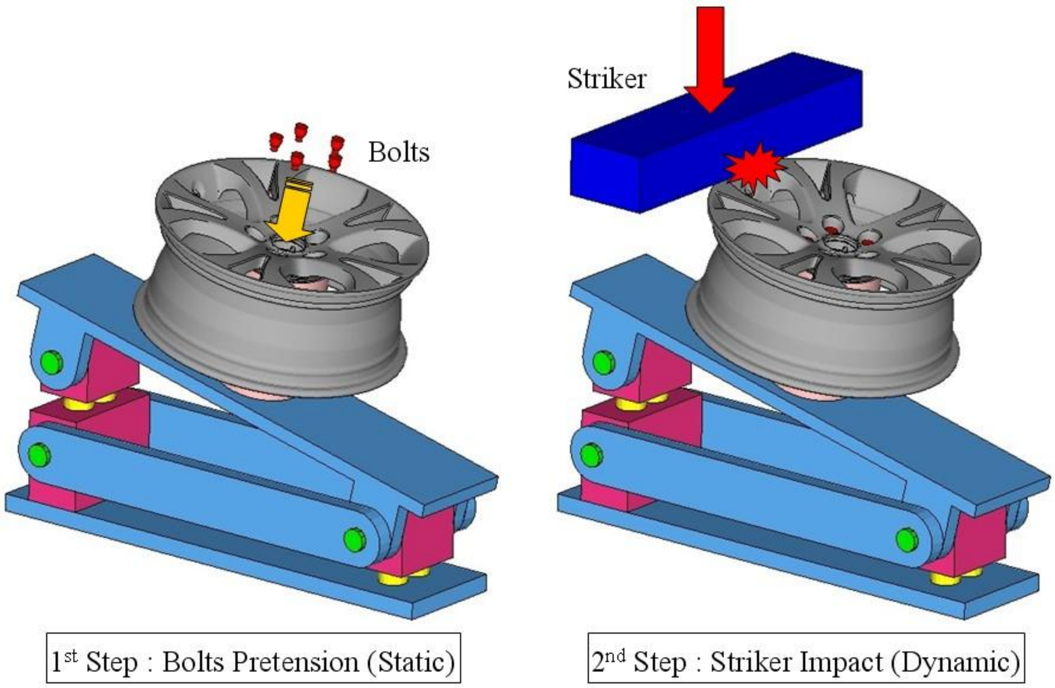


Figure 5. The Step of Aluminum Wheel Impact Analysis Using Dynamic Implicit

4. Analysis Results

CAE tool for this study was used Abaqus 6.10EF / Standard. Aluminum wheel FE model was composed of second order tetra element (C3D10M). The mesh size of aluminum wheel was 3mm~4mm (average).

4.1 Comparing The Analysis Results Using Test Jig Model (Dynamic Implicit) With Not Using Test Jig Model (Static)

Sometimes, the results of impact analysis without test jig model did not show the real weak area (the highest stress – the crack point). Since, this analysis could not consider the test jig stiffness. Especially, when crack point was located on nearby bolt hole.

However, the results using test jig model showed the real crack points. Figure 6. shows that the results using test jig model found the real crack points. (Comparing with not using test jig model) In this comparing, we did not consider the effect of dynamic and static.

(S/W: Abaqus 6.10EF, Element type: C3D10M, Element size: 3mm~4mm (average))

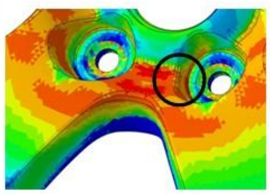
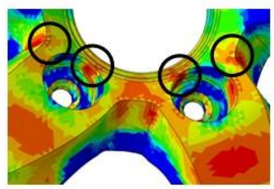
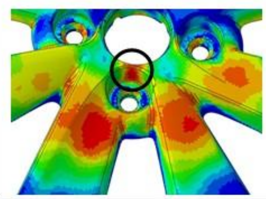
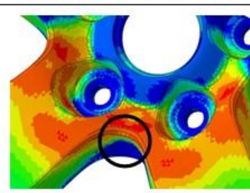
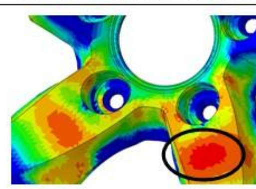
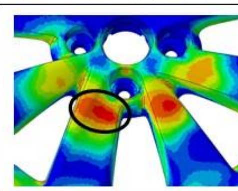
	Wheel of Vehicle "A" (15inch)	Wheel of Vehicle "B" (16inch)	Wheel of Vehicle "C" (16inch)
Test Result			
Using Test Jig Model (Dynamic Analysis)			
	No. of elements: 816,806 CPU Run Time (sec): 358,728	No. of elements: 377,193 CPU Run Time (sec): 86,692	No. of elements: 737,167 CPU Run Time (sec): 280,586
Not Using Test Jig Model (Static Analysis)			
	No. of elements: 816,806 CPU Run Time (sec): 257,621	No. of elements: 377,193 CPU Run Time (sec): 55,780	No. of elements: 737,167 CPU Run Time (sec): 78,241

Figure 6. The Analysis Results Using Test Jig Model and Not Using Test Jig Model

4.2 Comparing The Dynamic Implicit Analysis Results Using Test Jig Model With Real Test Results

In order to simulate the real impact test, we applied dynamic implicit analysis using test jig model.

Figure 7. shows the results of dynamic implicit analysis using test jig model (Comparing with real test results). Most analysis results show the real crack points.

(S/W: Abaqus 6.10EF, Element type: C3D10M, Element size: 3mm~4mm (average))

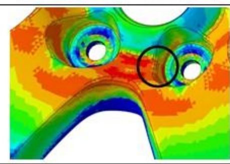
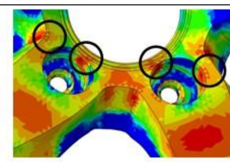
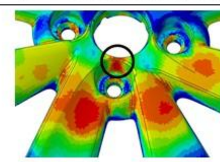
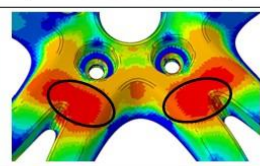
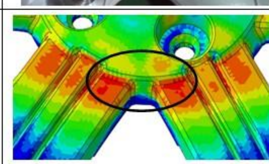
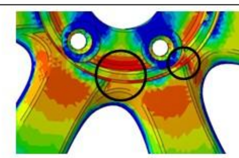
	Wheel of Vehicle "A" (15inch)	Wheel of Vehicle "B" (16inch)	Wheel of Vehicle "C" (16inch)
Test Result			
Dynamic Analysis (Using Test Jig Model)			
	No. of elements: 816,806 CPU Run Time (sec): 358,728	No. of elements: 377,193 CPU Run Time (sec): 86,692	No. of elements: 737,167 CPU Run Time (sec): 280,586
	Wheel of Vehicle "D" (17inch)	Wheel of Vehicle "E" (15inch)	Wheel of Vehicle "F" (17inch)
Test Result			
Dynamic Analysis (Using Test Jig Model)			
	No. of elements: 773,382 CPU Run Time (sec): 297,466	No. of elements: 581,002 CPU Run Time (sec): 213,728	No. of elements: 391,010 CPU Run Time (sec): 69,525

Figure 7. The Results of Dynamic Implicit Analysis Using Test Jig Model (Comparing with Real Test)

5. Conclusion

In Aluminum Wheel Impact Test(13 Degree Lateral, SAE J157), this paper shows the below.

1) The results using test jig model analysis show the real highest stress area (especially, nearby bolt hole).
2) The results using dynamic implicit analysis simulate the rear impact test (Comparing with test results).

6. References

1. SAE J 175, “Wheel – Impact Test Procedure – Road Vehicle”, 2003.
2. Chia-Lung Chang, “Simulation of wheel impact test using finite element method”, Engineering Failure Analysis 16 (2009) 1711-1719, Taiwan, 2009.
3. Kilju Ko, “A Study on the Stiffness of 13 degree-type Impact Tester for Aluminum Wheels”, KSAE, Vol. 14, No. 4 pp. 12-19 (2006), Korea, 2006.
4. Abaqus Users Manual, Version 6.10-1, Dassault Systémes Simulia Corp., Providence, RI.