

Topology Optimization of Aluminium Alloy Wheel

Ch. P. V. Ravi Kumar¹, Prof. R. Satya Meher

¹ M.Tech Student, QIS College of Engineering & Technology, Ongole – 523272, Andhra Pradesh

² Professors, Department Of Mechanical Engineering, QIS College of Engineering & Technology, Ongole – 523272, Andhra Pradesh

Abstract: A great number of wheel tests are required in designing and manufacturing of wheels to meet satisfactory requirements. The impact performance of a wheel is the major concern. Numerical implementation of impact test is essential to shorten the design time, improve the mechanical performance and lower development cost.

Project includes the “Topology Optimization of Cast Aluminium Alloy wheel” using impact analysis. Since the fail value of plastic strain for standard Cast Aluminium Alloy Wheel is 4.0%, cracks will appear if the Plastic Strain value is greater than 4%. This analysis will predict the plastic strains induced during impact testing. Topology Optimization is carried out by increasing the thickness of the rim until the plastic strain value is below 4%.

The main objective of the project is to generate a Finite element model (Hexa & Penta elements) using Hypermesh V10.0 with all the properties, materials, loads and Boundary conditions as specified by the client. Impact analysis is carried out using LS-Dyna software to predict the plastic strains during impact test.

Topology Optimization is carried out by changing the thickness of the rim of the Cast Aluminium Alloy Wheel until the value of the plastic strain is less than 4.0%.

Keywords: Topology Optimization, Impact test, Finite element analysis, Plastic Strain.

I. Introduction

Road wheel is an important structural member of the vehicular suspension system that supports the static and dynamic loads encountered during vehicle operation. Since the rims, on which cars move, are the most vital elements in a vehicle, they must be designed carefully. Safety and economy are particularly of major concerns when designing a mechanical structure so that the people could use them safely and economically. Style, weight, manufacturability and performance are the four major technical issues related to the design of a new wheel and/or its optimization. The wheels are made of either steel or cast/forged aluminium alloys. Aluminium is the metal with features of excellent lightness, corrosion resistance, etc. In particular, the rims, which are made of aluminium casting alloys, are more preferable because of their weight and cost.

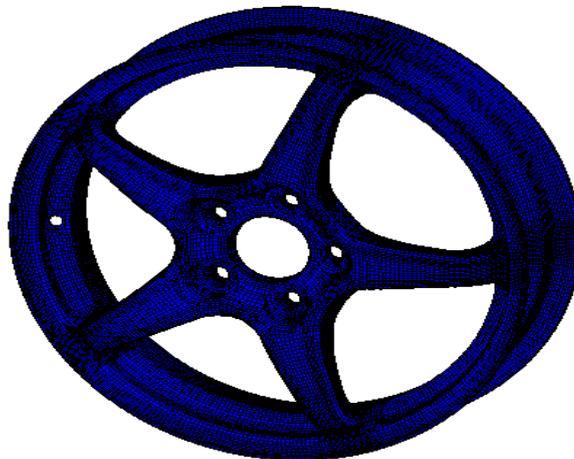


Figure 1. The Finite element model of Aluminium Alloy Wheel

Automotive manufacturers have been developing safe, fuel efficient and lightweight vehicular components to meet governmental regulations and industry standards. In the real service conditions, the determination of mechanical behaviour of the wheel is important, but the testing and inspection of the wheels during their development process is time consuming and costly. For economic reasons, it is important to reduce the time spent during the development and testing phase of a new wheel. A 3-D stress analysis of aluminium car road wheels involves complicated geometry. Therefore, it is difficult to estimate the stresses by using elementary mechanical approximations. For this purpose, Finite Element Analysis (FEA) is generally used in the design stage of product development to investigate the mechanical performance of prototype designs. FEA simulation of the wheel tests can significantly reduce the time and cost required to finalise the wheel design. Thus, the design modifications could be conducted on a component to examine how the change would influence its performance, without making costly alteration to tooling and equipment in real production. Therefore, in order to replace the physical test, the FEA simulation of the impact test should supply reliable results and sufficient information. In this regard, it is important to evaluate the effect of wheel impact performance during the impact test.

In this study, finite element analysis was conducted to simulate a cast aluminium wheel, shown in Figure 1 for the impact test according to the standard ISO 7141, using commercial LS-Dyna. The numerical model of an aluminium alloy wheel and striker were generated taking into account, the large deformable and highly non-linear material properties. In the case of strike that changes its magnitude and direction within a very short time, explicit coded software that considers dynamic forces as well as static forces is employed rather than implicit method used for static problems. The model includes elasto-plastic material for aluminium.

II. Impact Test Equipment And Procedure

Mechanical performance of road wheels under normal or severe driving conditions is evaluated by using three standard methods, such as impact, radial fatigue and rotary fatigue tests. The rotating bending test simulates cornering induced loads by applying a constant rotating bending moment to the wheel. In the radial fatigue test, the wheel and tire assembly are loaded radially against a constantly rotating drum to simulate the radial loading on the wheel. The wheel impact test is used to evaluate the impact performance, in which the striker is dropped from a specified height above the tire-wheel assembly. It is considered to be the case where the wheel collides with the curb of the road or a large obstacle. The test is designed to evaluate the frontal impact resistance of wheel and tire assemblies used in all cars and multi-purpose vehicles. The test is specifically related to vehicle pothole tests that are undertaken by most vehicle manufacturers. The scope has been expanded to allow the use of a striker that can be angled to preferentially impact the inboard and outboard wheel flange. Before the test, a wheel undergoes complete visual inspection to ensure that no cracks exist in the body. In order to pass the impact test, the wheel must meet the following minimum performance standards: There will no visible fracture of the central member of the wheel assembly, no separation of the central member from the rim, no sudden loss of tire air pressure and deformation of the wheel assembly, or fracture in the area of the rim section contracted by the faceplate weight system do not constitute a failure (International standard, 1995).

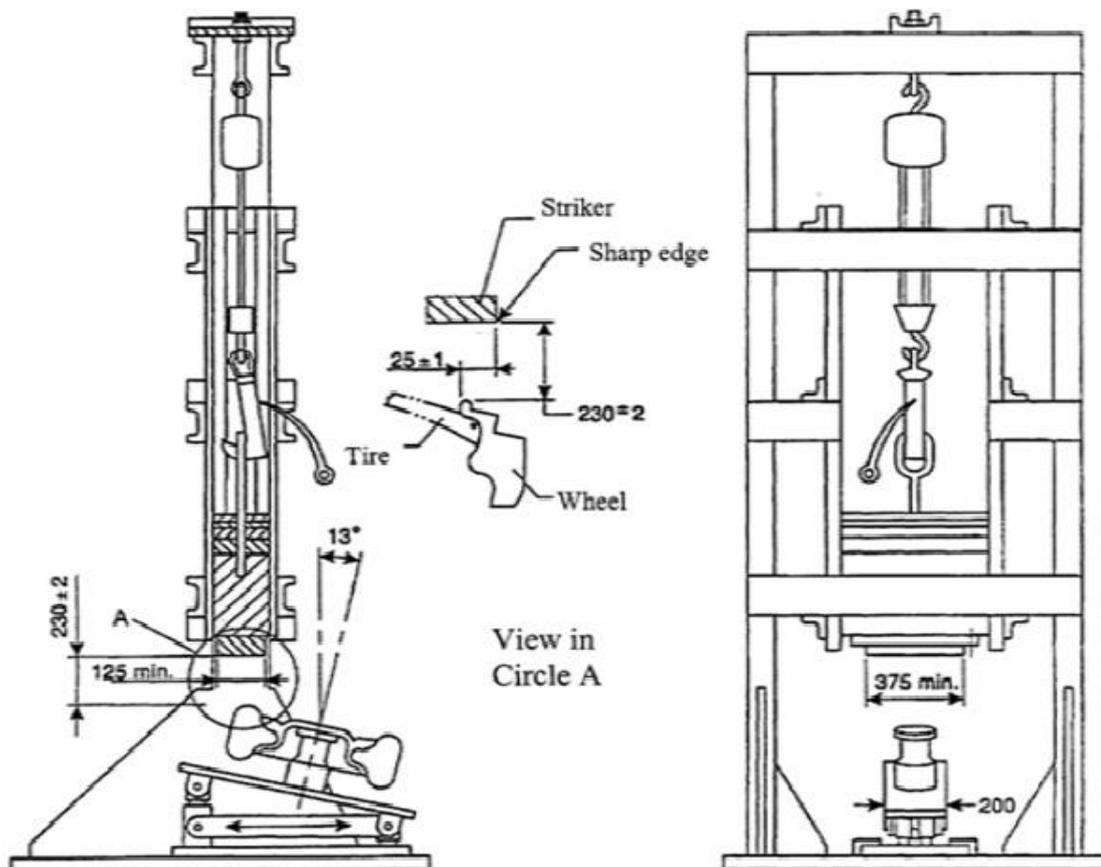


Figure 2. Schematic diagram of Wheel Impact test machine

The impact test standard provides detailed test procedures and equipment description for the impact test. A test machine is shown schematically in Figure 2. The test set up, in which a striker applies an impact to the rim flange of a wheel. The wheels are 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 is at least 125 mm wide and 375 mm long. The freely dropping height of the striker is 230 mm (± 2 mm) above the highest point of the rim flange. The striker is placed over the tire and its edge overlaps the rim flange by 25 mm. The inflation pressure of the tire can be specified by manufacturer taking into account, the serves conditions. An inflation pressure of 200 kPa, which in real service condition, was applied on the inner surface of tire and portion rim (International Standard, 1995).

III. Material Properties

The material properties of A356 Cast Aluminium Alloy that is widely used in automotive engineering industry was considered in the FE simulation. Mechanical properties of aluminium alloy are given in Table 1.

Table 1. Mechanical Properties of Aluminium Alloy

Elasticity Modulus	Poisson's ratio	Density	Yield stress	Ultimate stress
72.4 GPa	0.33	2.67E-09 tonne/mm ³	218 MPa	283 MPa

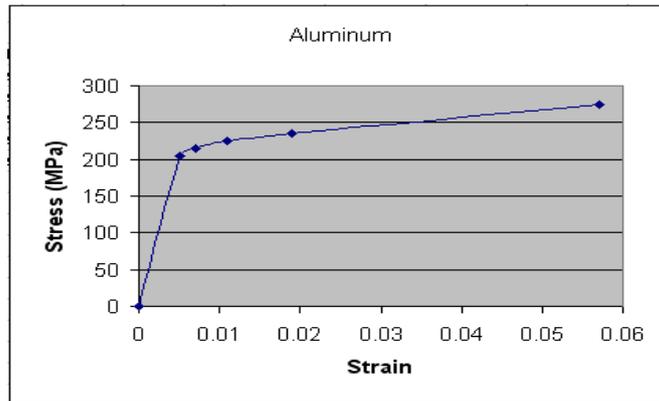


Figure 3. Engineering Stress-Strain diagram of Aluminium Alloy

A nonlinear elasto-plastic material model was used to describe the material behaviour of aluminium wheel in the analysis. The engineering stress-strain curve of the aluminium alloy is plotted in Figure 3. The striker used in this analysis was modelled as an elastic material using steel material properties.

Table 2. Mechanical Properties of Steel

Elasticity Modulus	Poisson's ratio	Density	Yield stress	Ultimate stress
210 GPa	0.3	7.85E-09 tonne/mm ³	250 MPa	400 MPa

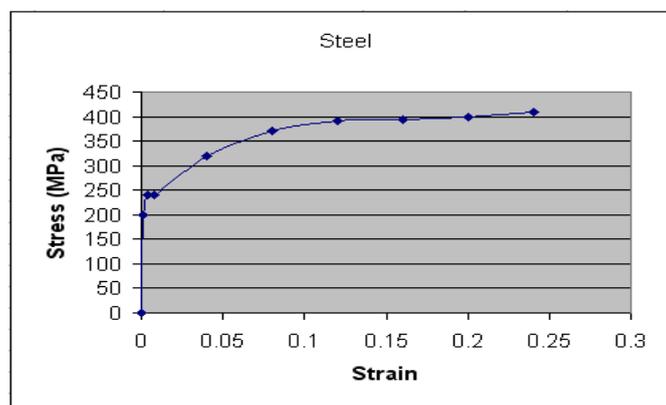


Figure 3. Engineering Stress-Strain diagram of Steel

IV. Finite Element Analysis

The purpose of this analysis is to predict the plastic strains induced during Impact testing. Modelling the mechanical response of impact test rim is extremely complex. Commercial finite element software LS-Dyna is utilized to perform 3-D impact analysis of wheel impact test. The numerical modelling of wheel impact test obeys the experimental procedure, which was described in the ISO 7141. The whole numerical model is an assembly consisting of three portions, namely a wheel, fixture and striker shown in Figure 4.

Uniform shapes and forms of elements play important role in the sensitivity of the results when using the finite element method. Therefore, the meshing of the wheel, fixture and impact striker models is mainly constructed by 3-D structural solid having 8- node finite element. It can tolerate irregular shapes without much loss of accuracy. The fixture model was generated based on the assumption. However, including every detail, it makes the model too complicated to be solved within a reasonable time limit. In order to simplify and reduce the overall size of the model, some of the features which are not essential in cornering, are either simplified. All degrees of freedom of the nodes on the mounting surface of the hub and bolt holes were fully constrained. The nodes at the surface between the wheel and striker are constrained to move together. The full model including rim, fixture and striker compose of **159757** elements. The mass of the striker is a variable

related to the maximum static wheel load as presented in Equation 1. The unit of mass is kilogram:

$$m = 0.6 m_w + 180 \quad \text{-----} \quad (1)$$

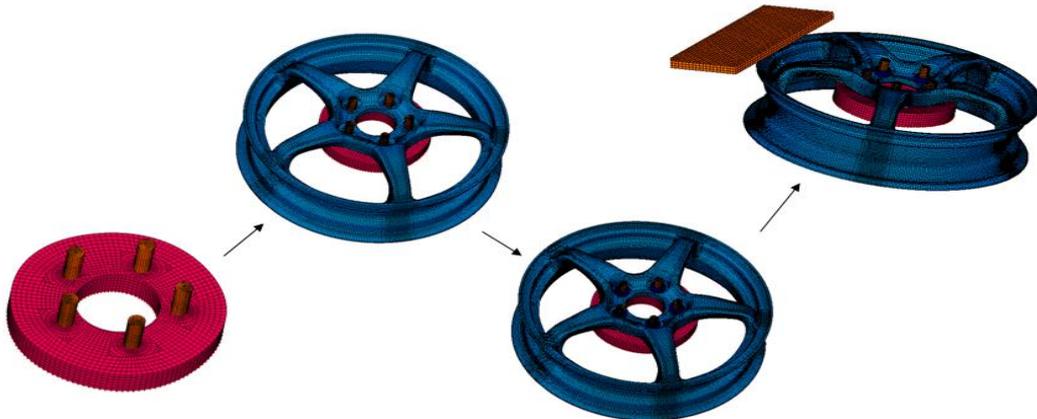


Figure 4. Finite element model of the Wheel Assembly with Striker

Where m is the mass of striker, and m_w is the maximum static wheel loading as specified by the wheel and/or vehicle manufacturer. Mass of the striker for the wheel–tire was determined to be 570 kg. The volume of the striker was adjusted so that the total mass of the striker is the same as that of the striker used in a real impact test. Striker dimensions are 15 mm in height, 125 mm in width and 375 mm in length. The striker was constrained in the horizontal direction to ensure that the striker could only be displaced vertically as in the impact test. For the purpose of reducing computational time, the initial dropping height, which represents the distance between the lower surface of the striker and the impact point on the rim flange, was modified from the prescribed value of 230 mm to 0, but with similar impact energy. The magnitude of the initial velocity of the striker prior to impact was calculated using the following equation and applying the energy conservation principle.

$$V = \sqrt{2gh} \quad \text{-----} \quad (2)$$

Where V is the initial impact velocity of the striker, g is the acceleration of gravity and $h = 0.23$ m is the initial height of the striker. The Boundary Conditions of the Alloy wheel for impact analysis is shown in figure 5.

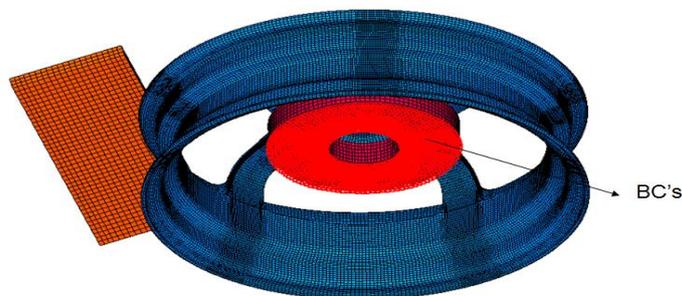


Figure 5. Boundary Conditions of Finite element model of the Wheel Assembly with Striker

Topology optimization is carried out by changing the rim thickness of the alloy wheel as shown in figure 6, until the value of the plastic strain is less than 4.0%.

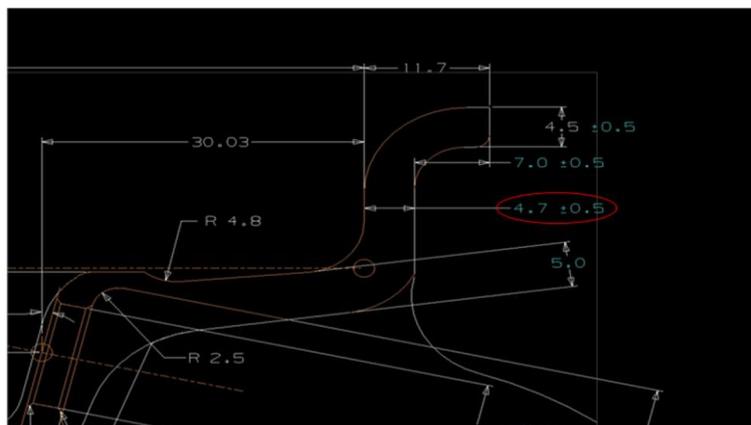
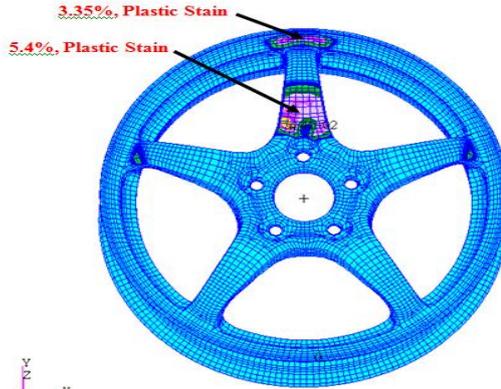


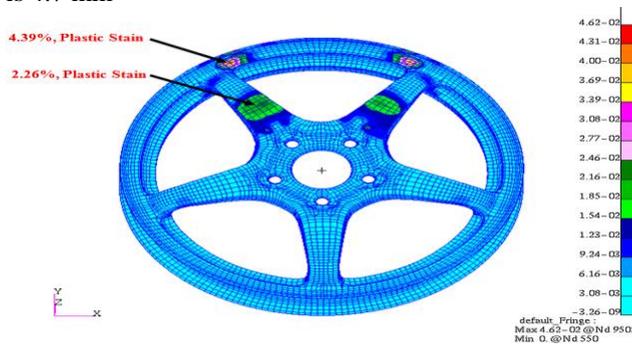
Figure 5. Thickness of the rim indicated in circled area

Several load cases have been performed by changing the thickness of the rim. Out of which, the following are the three load case studies where a change of plastic strain noticed drastically while post-processing.

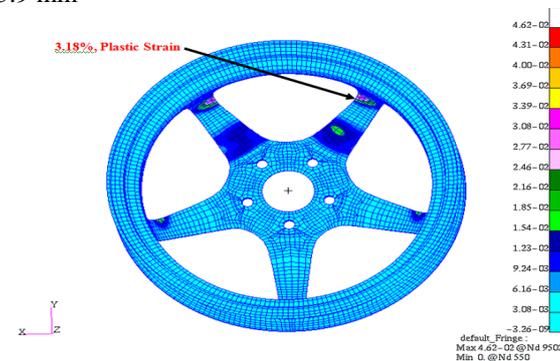
Case-1: The thickness of the rim is 3.5 mm



Case-2: The thickness of the rim is 4.7 mm



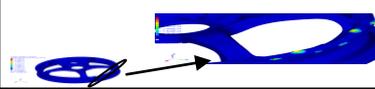
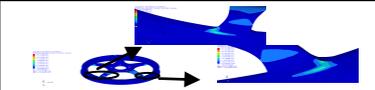
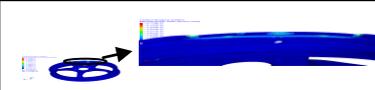
Case-3: The thickness of the rim is 5.9 mm



Failure Criteria: Cracks will appear if the plastic strains are higher than 4%

V. Summary

The Effective Plastic Strain for the three load cases are as following:

Load case with rim thickness of 5.9mm	Load case with rim thickness of 4.7mm	Load case with rim thickness of 3.5mm
		
		
Effective Plastic Strain for rim thickness of 5.9mm	Effective Plastic Strain for rim thickness of 4.7mm	Effective Plastic Strain for rim thickness of 3.5mm
3.2 %	4.4%	5.4%

VI. Conclusion

The response of wheel assembly during the impact test is a critical phenomenon. In this paper, a numerical study of impact test of the wheel assembly was performed using explicit finite element code. 3-D finite element analysis with a reasonable mesh size can reliably estimate the response. Such results will help to predict the locations, in which the failure may take place during impact test and improve the design of a wheel with required mechanical performance. Topology optimization is carried out using impact test on Aluminium alloy wheels by varying thickness of the rim and the results obtained are shown above. Since the standard fail value of plastic strain for standard wheel is 4.0%, the thickness of Cast Aluminium Alloy Wheel should be 5.9mm from the results obtained above which will perform satisfactorily.

References

- [1] Muhammet Cerit. Numerical simulation of dynamic side impact test for an Aluminium alloy wheel, Department of Mechanical Engineering, Sakarya University, Turkey. August-2010
- [2] Si-Young Kwak, Jie Cheng and Jeong-Kil Choi. Impact analysis of casting parts considering shrinkage cavity defect, Department of Virtual Engineering, School of Engineering, Korea University of Science & Technology, Daejeon, Korea. August-2010.
- [3] C. Bosi, G.L. Garagnani, R.Tovo. Fatigue Properties of a Aluminium Alloy for Rims of Car Wheels, Department of Ingegneria, University of Degli Studi Di Ferrara, Italy.
- [4] Mi Guoga, Liu Xiangyu, Wang Kuangfei, Fu Hengzhi. Numerical simulation of low pressure die-casting aluminium wheel, Department of Mechanical Engineering, Chengde Petroleum College, Chengde 067000, P. R. China. August-2008.
- [5] Mohd Izzat Faliqfarhan Bin Baharom. Simulation test of Automotive Alloy Wheel Using CAE Software, Department of Mechanical Engineering, University of Malaysia Pahang. October-2008
- [6] Cleginaldo Pereira de Carvalho, Herman Jacobus Cornelis Voorwald. Automotive Wheels – An Approach for Structural Analysis and Fatigue Life Prediction, Department of Materials Tecnology, Brasil. November-2001.