Study of wheel rim impact test using finite element analysis

H.B. Zainuddin[1], M.B. Ali[1,2,*]

[1] Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka,
Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia
[2] Centre for Advanced Research on Energy, Universiti Teknikal Malaysia Melaka,
Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia

*Corresponding e-mail: basri@utem.edu.my

ABSTRACT – This study measures the impact energy absorbed experienced by the wheel rim under dynamic loading. Computational simulation is time saving, and in contrast the wheel impact experiments involve high cost including the manpower. Alloy wheel is widely used due to its excellent performance and appearance. Wheel rim is modelled using CATIA and imported to ABAQUS for further finite element analysis. Yield strength affects the energy absorbing capacity of a material. Aluminium 6061-T1 is found to have the greatest energy absorption value compared to the other materials.

1. INTRODUCTION

Since the last two decades, vehicle weight has increase about 20%, constituently due to the engine size and safety additional feature [1]. Generally, every 10% of weight reduction allows the reduction of fuel consumption by 5%-7%. One of vital component in a vehicle is wheel. Traditional stone and wood wheel were inefficient with poor traction, low friction, harsh ride and poor load capacity [2]. The wheel nowadays concerns on the durability to endure harsh working condition, fabricated from lightweight material with low manufacturing cost, taking into account the safety requirement as well [3]. Sabri et al. [4] found that alloy wheel allows better performance compared to steel wheel. There are few suitable candidates material in wheel rim manufacturing, which are aluminium alloy [1,4], magnesium alloy [1,5] and steel alloy [5]. Economically, finite element analysis helps to minimize cost and time of experimentation.

Previous study [1,3] has investigated on the 13 degree lateral wheel impact test. However, there are quite few studies on the 90 degree vertical wheel impact. For that reason, this work focuses on the 90 degree vertical impact test using finite element analysis. Impact test is carried out under dynamic loading. Focusing on alloy metallic rim, the objective is to study the energy absorbed by wheel rim with different materials. Different material is expected to give different energy absorbing ability.

2. METHODOLOGY

Reverse engineering has been implemented to get the wheel rim dimension. The wheel 3-dimensional model is done by using CATIA. Modelling of striker and the finite element analysis is executed in the ABAQUS software. The diameter of the wheel rim is 416 mm with 180 mm width. The striker is in rectangular shapes as in the model assembly in Figure 1. The material assigned in the analysis model is isotropic and homogenous. The material used in this analysis is Aluminium (Al) 6061-T1, Magnesium (Mg) AM60 and Stainless Steel (SS) 304L. The properties for the materials are tabulated in Table 1.
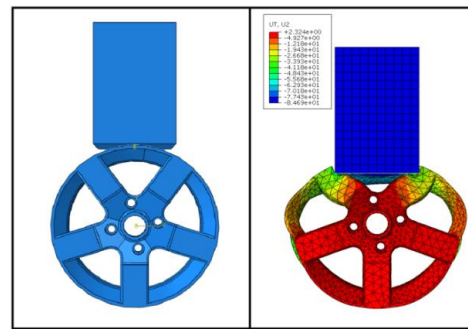

Figure 1 Model assembly; undeformed and deformed.

Table 1 Material properties.

Component	Material	Young Modulus, E (GPa)	Poisson Ratio, v	Density (kg/m^3)	Yield Strength (MPa)
Striker	Steel	206.9	0.30	8000	-
Rim 1	Aluminium 6061-T1	70	0.33	2700	270
Rim 2	Magnesium AM60	45	0.35	1790	130
Rim 3	Stainless Steel 304L	193	0.25	8000	172

The striker has a simple shaped, but the wheel rim has irregular geometry. Therefore suitable mesh for the striker is hexahedral element and the rim is tetrahedral element [3]. The mesh size used for the wheel rim is varies; 20 mm, 15 mm and 10 mm. Wheel rim is fixed at the four holes, similar to the operating conditions on a vehicle. The striker is set to be displaced in vertical direction with velocity of 22222.2 mm/s, related to the velocity during collision [6].

3. RESULTS AND DISCUSSION

The measurements were taken at the location where the striker experienced the highest reaction force. Figure 2, Figure 3 and Figure 4 shows the load-displacement curve for all the computational analysis. The mesh size does not affect the results of the analysis. Same pattern is obtained even though different mesh size is assigned. Particularly, if the mesh size is smaller, a more detail analysis is performed by the finite element software and it will increase the number of nodes used.
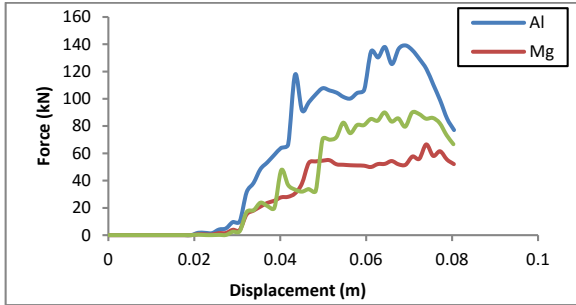

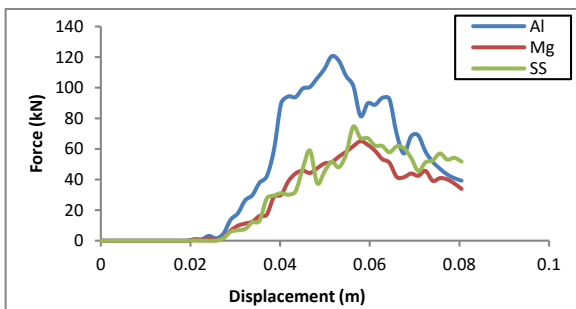Figure 2 Load-displacement curves for 20mm mesh size.

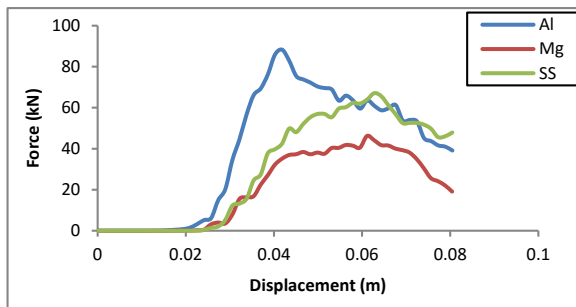Figure 3 Load-displacement curves for 15mm mesh size.

Figure 4 Load-displacement curves for 10mm mesh size.

Based on the results, it is observed that if the size of the mesh increased, the peak load of the model increased. The materials behave in similar pattern although different mesh size is assigned. It is observed that the aluminium is having the highest peak load in every case. The energy absorbed for the materials are calculated from the area under the load-displacement curve as shown in Figure 5. Comparing these three materials, Aluminium 6061-T1 absorbed greater impact energy compared to the other two materials, while Magnesium AM60 is found to absorb the least impact energy.
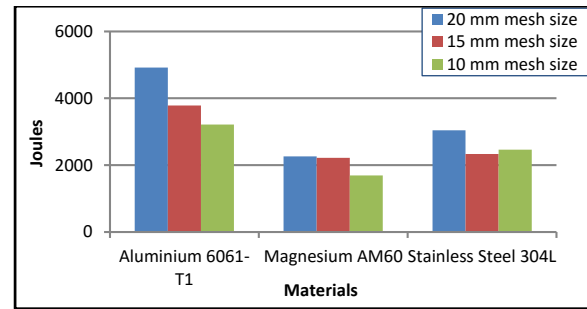

Figure 5 Energy absorbed for different materials.

Referring to the material properties in Table 1, Aluminium 6061-T1 has higher yield strength than the Magnesium AM60 and Stainless Steel 304L, with a percentage different of 51.85% and 36.30% respectively. Due to this property, the aluminium possessed greater strength to collapse and thus greater force is produced for the aluminium wheel rim to deform. This explains the high peak load as well as the high energy absorption [7]. High energy absorption capacity represents good structure reliability as well as having the ability to endure the operating condition during services.

4. CONCLUSIONS

It is found that smaller mesh size produced lower reaction force and lower energy absorbed. In general, the result in a finite element analysis is influenced by the size of mesh applied to the model. Different material shows different energy absorbing capacity. Aluminium 6061-T1 has been found to possess highest energy absorbed. Meanwhile, magnesium AM60 has the least energy absorbed. Hence, it is suggested that the Aluminium 6061-T1 is more reliable and safer to be implemented as wheel rim material.

REFERENCES

[1] S.K. Bisoyi and M.B. Ashok Kumar, "Analysis and Development Characterisation of Automotive Wheels," *Int. J. Mag. Eng. Technol. Manag. Res.*, vol. 2, pp. 667–671, 2015.

[2] R.N. Jazar, *Vehicle Dynamics: Theory and Application*. Springer Science & Business Media, 2008.

[3] C.-L. Chang and S.-H. Yang, "Simulation of wheel impact test using finite element method," *Eng. Fail. Anal.*, vol. 16, no. 5, pp. 1711–1719, 2009.

[4] M. Sabri, M. Rezal, A. Mu, K. Shahril, and J. Ihsan, "Deformation Behaviour Analysis Of Car Wheel Rim Under Different Loading Using Finite Element Method," *Int. J. Eng. Technol.*, vol. 5, no. 3, pp. 181–184, 2015.

[5] K.V. Rao and T. Dharmaraju, "Analysis of Wheel Rim Using Finite Element Method," *Int. J. Eng. Res. Technol.*, vol. 3, no. 1, pp. 1259–1263, 2014.

[6] OUED/ITF, "'Malaysia,'" in *Road Safety Annual Report 2015*, Paris: OECD Publishing, 2015.

[7] G. Lu and T. Yu, *Energy Absorption of Structures and Materials*. Woodhead Publishing Ltd and CRC Press LLC, 2003.