International Journal of Automotive and Mechanical Engineering (IJAME)
ISSN: 2229-8649 (Print); ISSN: 2180-1606 (Online);
Volume 13, Issue 3 pp. 3574 - 3583, December 2016

DOI: https://doi.org/10.15282/ijame.13.3.2016.4.0294

Correlation between vertical wheel impact energy with lateral wheel impact energy: A finite element analysis approach

H. Zainuddin[1*], M.B. Ali[1,2*], N.B. Muhammad Said[1*] and K.A. Zakaria[1,2*]

[1]Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka,
Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia
[*]Email: basri@utem.edu.my, hikmah_zainuddin@yahoo.com,
nurlaelamuhammadsaid@yahoo.com, kamarul@utem.edu.my
Phone: +6062346900
[2]Centre for Advanced Research on Energy, Universiti Teknikal Malaysia Melaka,
Hang Tuah Jaya, 76100 Durian Tunggal, Melaka, Malaysia

ABSTRACT

A dynamic loading impact test was performed on a wheel rim using the Finite Element method. It is a convenient, practical and more economic to use computational simulation compared to a real experiment which may involve high costs in order to provide enough specimens for repeated experiments. The study's aim is to investigate the correlation between the impact energy absorbed in 13 degree and 90 degree impact tests using three different materials, namely Aluminium 6061-T6, Magnesium AM60 and Stainless Steel 304L. For the wheel rim, the elasto-plastic condition was applied. The striker is set to move downwards with a velocity of -22222.2 mm/s and a mass of 144 kg. The mesh sizes implemented in this work are 10 mm, 7 mm and 5 mm. It is found that the peak load in the 90 degree impact test is greater than the 13 degree impact test. Due to dissimilar wheel orientation and contact area during the impact, the energy absorbed obtained in the 13 degree impact test is lower than the 90 degree impact test. Aluminium shows the best results compared to the other material with a percentage of correlation of 24.40 %. Whereas, Stainless Steel recorded the highest value of deviation percentage compared to the other materials. The result shows that this can be the basis to study the parameters for wheel changes such as the use of new materials, and the thickness, size and pattern of spokes.

Keywords: Impact; finite element analysis; wheel rim.

INTRODUCTION

The wheel is one of the crucial components of a vehicle. It consists of a rubber tyre and metallic rim. An excellent wheel should be able to endure harsh working conditions during operation such as shocks and force experienced due to potholes or bumpy roads, as well as protecting other components from damage due to collision [1]. These days, there are various rim designs with stylish appearance made of lightweight material, yet its safety features must not be neglected [2]. It is beneficial to have a light weight wheel rim considering the overall performance of a vehicle in terms of the overall weight of the wheel and the rotational inertia of the wheel which goes up with more weight, causing even more work for the brakes [3]. In order for a wheel to work at its top condition, repetition of experiment is essential during the development stage. A wheel impact experimentation needs high expenses. The use of computational simulation helps in minimizing the experimental cost and reduces the time to perform the test. The usage of a lightweight rim contributes to the reduction of a vehicle's weight, eventually helping to reduce fuel consumption. Generally, 5% to 7% of fuel consumption can be saved with every 10% of weight loss [4]. Typically, rims are made of alloy due to its light weight and stylish appearance [5]. The alloy wheel comes in many designs to suit customers' desire. It is possible to create quite exotic shapes and styles using the modern casting technique, whilst still retaining adequate strength [6]. There are two types of wheel impact tests which are the 90 degree and 13 degree impact tests used to fix wheels on the hub and bolt holes [7, 8]. Expenses are high as these two impact tests are done using real parts. In the product development stage, there is a need for test repetition. In the product development stage, the specimens will normally be damaged after the impact test. Considered as a single shot device, the cost of providing dozens of specimens to be tested will be expensive. For a practical and economical solution, the usage of Computer Aided Engineering (CAE) is very convenient. This study will make use of the FEA approach to simulate an impact test on wheel rims that are subjected to dynamic loading. In this paper, the work involves both the 90 degree and 13 degree impact tests, with the absence of inflated tyres. The objective is to investigate the correlation between the impact energy absorbed in the two cases. The simulation is performed under dynamic loading with three different types of materials. Various materials used for the wheel rim may contribute different absorbed energy values. Different wheel impact test orientations are predicted to have correlation in terms of energy absorption values.

METHODS AND MATERIALS

Structural Modelling

Basically, the models involved in the analysis are the striker, wheel and support jig. The structural modelling is made separately based on the part's complexity. The study makes use of CATIA and Abaqus/Explicit software. Complex parts such as the wheel rim and support jig are made in CATIA and then imported to the Abaqus/Explicit workbench, meanwhile the striker is made in the Abaqus/Explicit workbench. This wheel rim is readily available, while the spoke wheel is frequently used by previous researchers [2, 7, 9-13]. Dimensions were measured and recorded according to the actual rim to be used for modelling purposes. The rim has 5 spokes together with 4 bolt holes. This rim design is chosen primarily with the intention of introducing this study. Further work should revolve around this model as it was set to be the fundamental model. Research work involving this model can be varied by changing the parameters such as the base materials, thickness, size and pattern of spoke. The rim used has a 416mm diameter and 180mm width. The dimension of the striker is 300 mm × 200 mm × 300 mm, with a mass of 144 kg. The striker is set to move downwards with a velocity of -22222 mm/s. The same striker modelling is used for all impact test cases. Upon completing the modelling, the assembly is made for both parts. This is dissimilar to the 90 degree impact test, as there is a need for wheel support for the 13 degree impact test. The support is made to place the wheel rim onto it so as to get a better view of the 13 degree angle. Specifically, each impact test is assembled with different setup conditions. In every assembly, there is a 0.4 mm gap in between the impact striker and the wheel rim so as to offer a zero amplitude before the striker hits the wheel rim [14]. The models for both the 90 degree and 13 degree impact tests are as shown in Figure 1. The model assembly shows that the degree of impact test is indicated by the position of the wheel rims. In the 90 degree impact test, the wheel rim is placed upright, but for the 13 degree impact test, the wheel is placed horizontally with a 13 degree angle from the planar surface. For the boundary condition, the rim is fixed at the four holes to prevent translation and rotation of the rim in all directions, as shown in Figure 2.

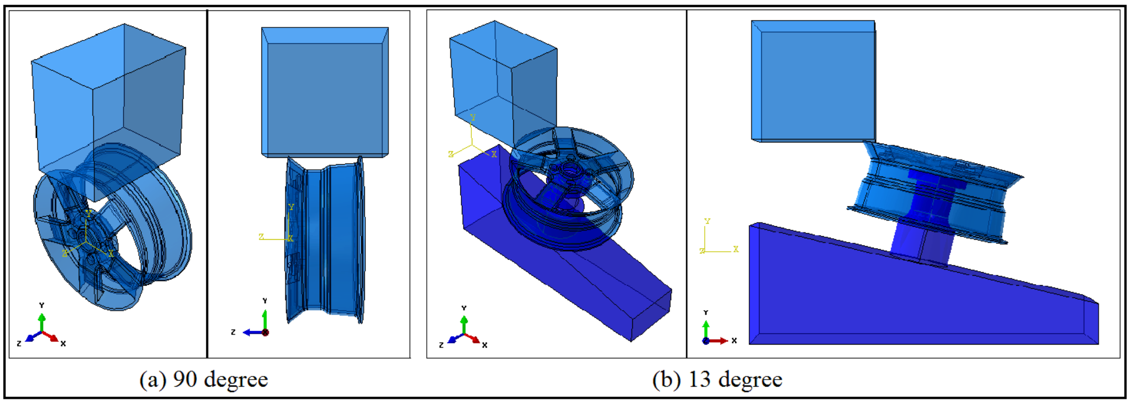

Figure 1. Model assembly for the 90 degree impact test and 13 degree impact test.

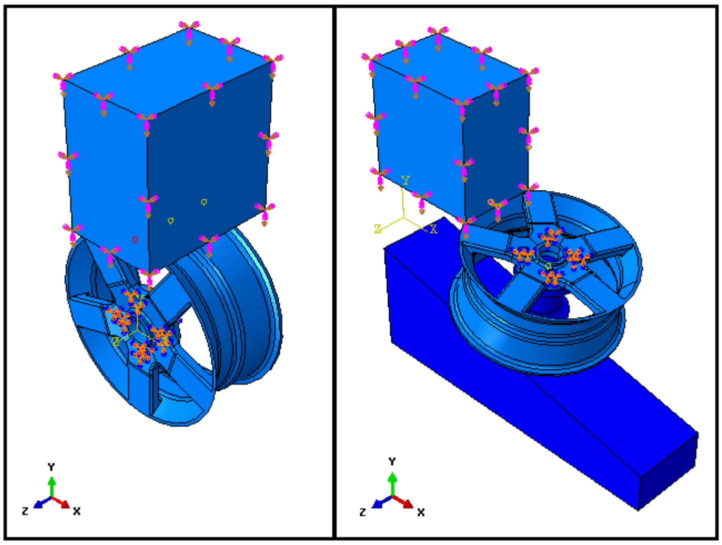

Figure 2. Boundary conditions applied in the simulation.

Material Properties

The material assigned is isotropic and homogenous. The wheel rim is modelled as elasto-plastic, while the striker and test jig use elastic material [8]. Steel material is assigned to the striker. Each type of impact test analyses the three different wheel materials; Aluminium (Al) 6061-T6, Magnesium (Mg) AM60 and Stainless Steel (SS) 304L. The materials' properties are tabulated in Table 1. Throughout the impact simulation, the parts possessed solid and homogenous properties. No material is described to the wheel support as it is excluded in the analysis.

Table 1. Material properties [11, 15-17].

Part	Material	Young Modulus, E (GPa)	Poisson Ratio, *v*	Density (kg/m^3)	Yield Strength (MPa)	Ultimate Strength (MPa)	Weight (kg)
Striker	Steel	206.9	0.30	8000	-	-	144
Rim 1	Aluminium 6061-T6	70	0.33	2700	270	310	6.81
Rim 2	Magnesium AM60	45	0.35	1790	130	220	4.52
Rim 3	Stainless Steel 304L	193	0.25	8000	172	482	20.18

Finite Element Analysis

The wheel and test jig were developed with tetrahedral elements, whereas the striker was created with hexahedral elements [2]. This is due to the shape of the parts, as the striker has a simple shape but the wheel rim has an irregular geometrical shape. The mesh sizes for the wheel rim are 10 mm, 7 mm and 5 mm. Different mesh models are used to assure that its size is accurate enough and can converge well [8, 18]. In addition, the work is expected to show the effect of mesh sensitivity on the simulation results [19]. The mesh applied to the wheel rim varies for each impact test; 10 mm, 7 mm and 5 mm, as illustrated in Figure 3.

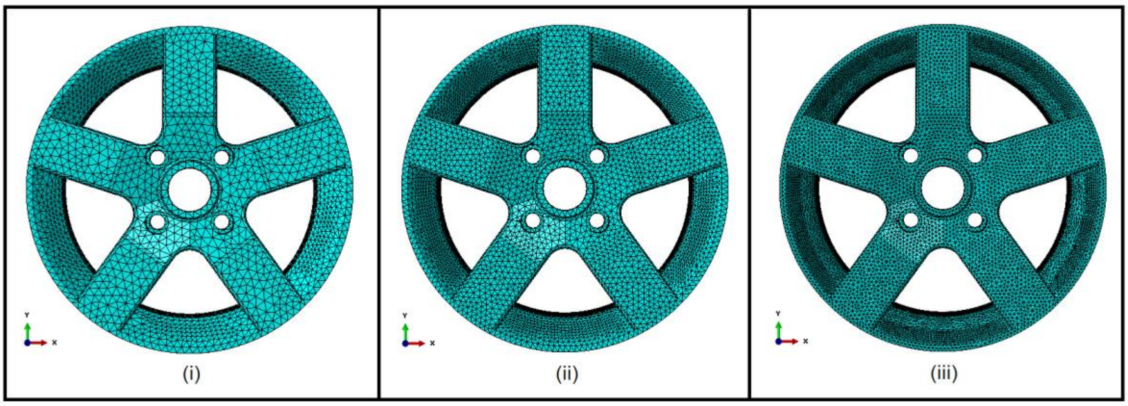

Figure 3. Different mesh sizes; (i) 10 mm, (ii) 7 mm, and (iii) 5 mm.

Table 2. Mesh data used in the simulation.

Mesh size	Element type	Number of elements	Number of nodes
10 mm	C3D10M	29478	56482
7 mm	C3D10M	70366	126000
5 mm	C3D10M	195855	323614

The mesh data for the rim is gathered in Table 2. The mesh data shows that the smaller the size of the mesh, the greater the number of nodes and elements of a particular model. For all tests, the striker is set to move in a vertical direction with a velocity of

22222.2 mm/s equal to 80 km/h as it is related to velocity during collision [20]. The bolt holes are fully constrained in all impact tests.

RESULTS AND DISCUSSION

Measurements were obtained at the node of the striker that possessed the greatest reaction force. Each impact case has identified different positions of reaction force on the striker. The position depends on the contact between the striker and wheel rim during the impacting event. Figure 1 shows that the contact regions differ for each case. The nodes showing greatest reaction force for both impact cases are as presented in Figure 4. Based on the overall result, the striker shows the maximum value of reaction force, especially at the particular nodes. That particular area comes in contact with the rim during the impact simulation, thus experiencing the greatest reaction force.

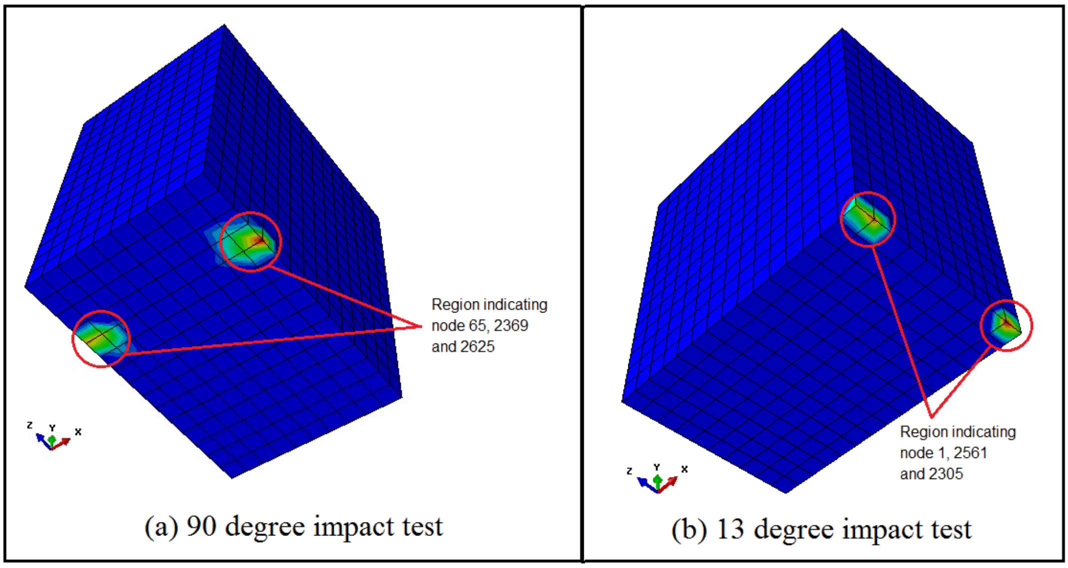

Figure 4. Node regions selected in both impact test analyses.

The value of the reaction force from each node was extracted and the greatest value was used in the analysis. The impact energy is calculated from the area under the load-displacement curves obtained. The load-displacement curves for the 90 degree impact test are as presented in

Figure 5, meanwhile the curves for the 13 degree impact test are as presented in Figure 6. All the graphs are in irregular pattern. In this case, the wheel in the 90 degree impact test is fixed in an upright position. The impact appears to take place at two contact surfaces. Meanwhile, the wheel in the 13 degree impact test was placed at 13 degrees from the planar surface, resulting in only a small vibration at the wheel edge. The peak load of the 90 degree impact test is greater than the 13 degree impact test. In other words, the peak force experienced in the impact test depends on the contact area during the impacting event. Hence, a larger area of contact in the impacting event, the greater the reaction force recorded by the striker. All of the force values recorded in all cases showed positive values. The peak load obtained in the 90 degree impact test is in between 40 kN to 90 kN, whereas for the 13 degree impact test, the peak load is in between 25 kN to 40 kN. According to the results, the decrement of mesh size causes the peak load of the model to decrease.

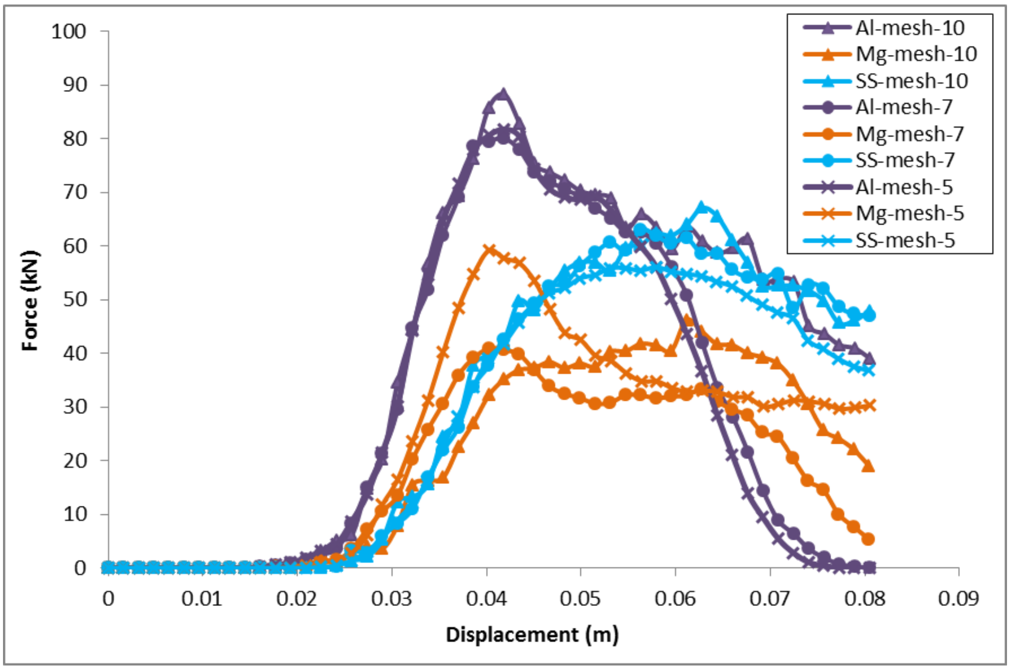

Figure 5. Force-displacement curves for the 90 degree impact test.

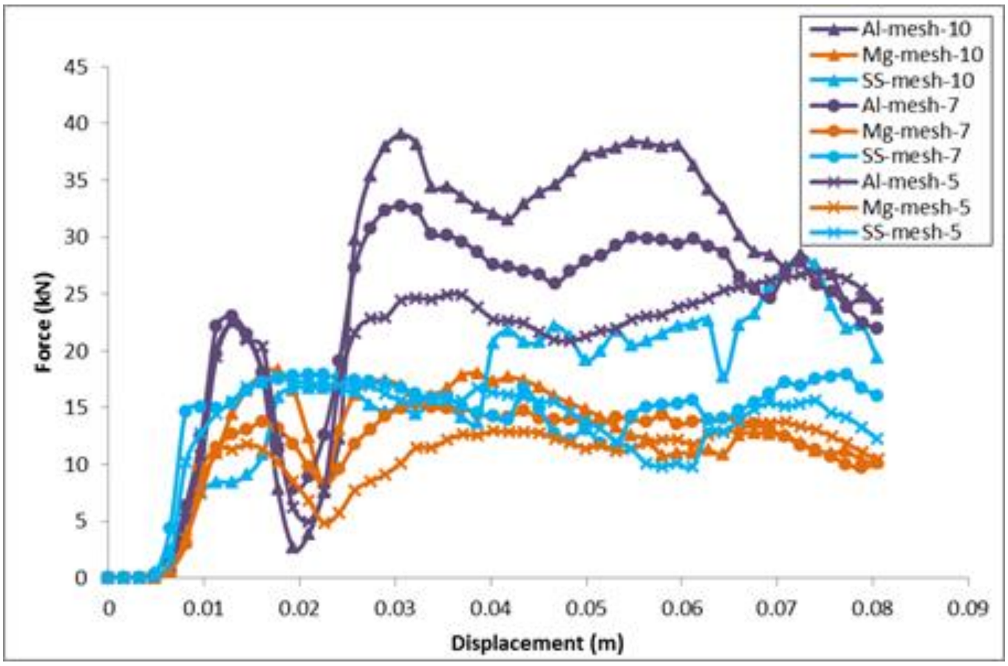

Figure 6. Force-displacement curves for the 13 degree impact test.

Correspondingly, the number of elements increases when the mesh size becomes smaller. Under this circumstance, the energy absorbed increases when the number of element decreases. This is because bigger mesh sizes are stiffer that the fine meshes [19]. The energy absorbed for all simulations are calculated from the total area of the load-displacement curves. The calculated energy absorbed is gathered in Table 3. The energy absorbed is also presented in a graphical form as shown in

Figure 7. In the beginning, five mesh sizes were considered in the analysis as shown in Figure 8. The purpose of setting up different mesh sizes is to study the mesh convergence and ensure that the model mesh is accurate enough [2, 8, 18, 20-22]. In this paper, only three different mesh sizes were used. Since the results suggest that the energy absorbed response is almost convergent for the mesh sizes 7 mm and 5 mm; the graphs are approximately to be linear.

Table 3. Energy absorbed and percentage difference for all cases.

Mesh Size	Material	Energy Absorbed (J)		Percentage Different
		90 degree	13 degree	
10 mm	Aluminium 6061-T6	3214.8	2048.6	35.16 %
	Magnesium AM60	1698.1	1010.5	40.36 %
	Stainless Steel 304L	2463.6	1349.5	45.22 %
7 mm	Aluminium 6061-T6	2412.0	1823.4	24.40 %
	Magnesium AM60	1469.1	935.8	36.30 %
	Stainless Steel 304L	2420.4	1153.6	52.34 %
5 mm	Aluminium 6061-T6	2339.2	1569.7	32.90 %
	Magnesium AM60	1454.0	813.4	44.06 %
	Stainless Steel 304L	2229.8	1077.2	51.69 %

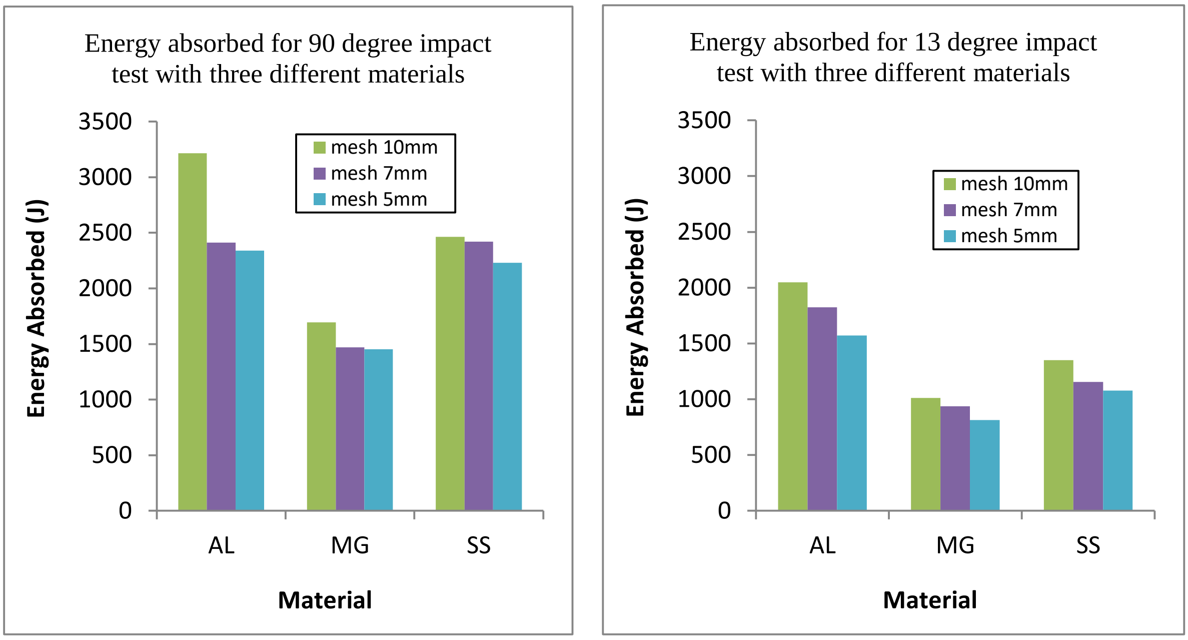

Figure 7. Energy absorbed in both impact tests.

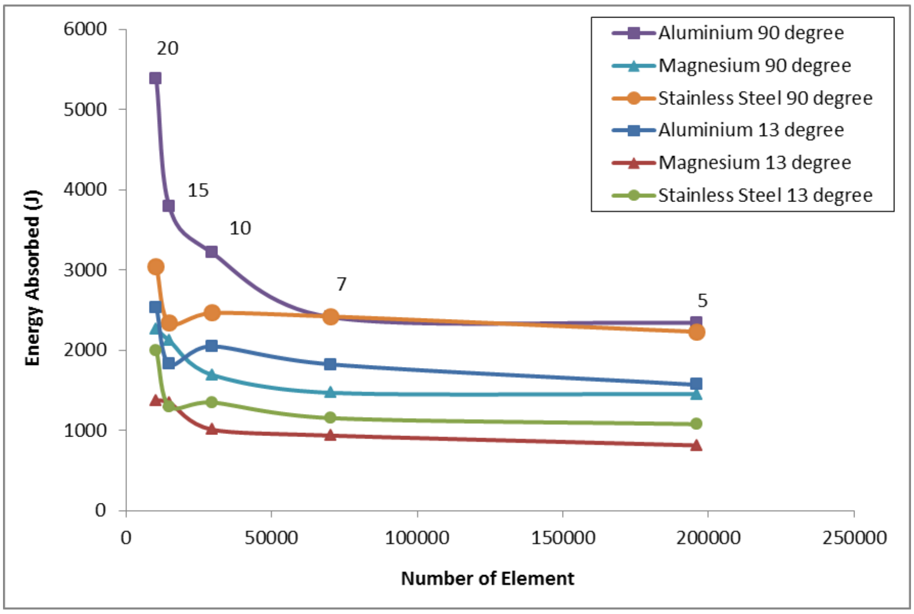

Figure 8. Energy absorbed in the 90 degree impact test and 13 degree impact test corresponding with the number of elements.

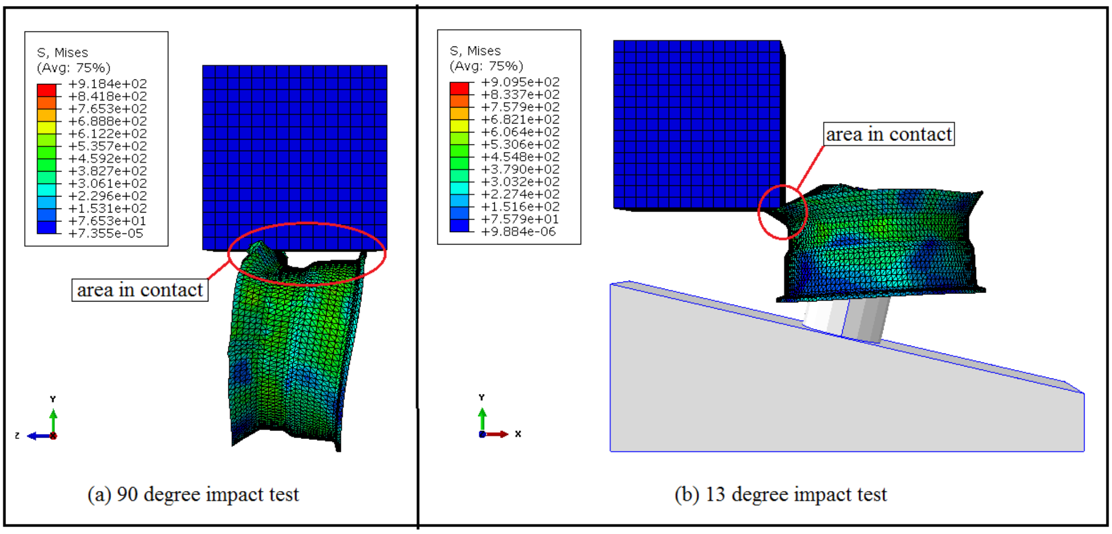

Figure 9. Wheel deformation in both impact tests.

Besides, the energy absorbed decreases as the mesh size becomes smaller. The pattern for all types of material is similar, even when the mesh size assigned to them is different. For all cases, Aluminium possesses the highest values of reaction forces, followed by Stainless Steel and Magnesium. It was found that Aluminium has the highest energy absorption value, while Magnesium has the lowest energy absorption value. The energy absorption results are highly affected by the yield strength of the material [21, 22]. In this work, Aluminium has a higher value of yield strength compared to the other two materials; Magnesium and Stainless Steel. This explains why the wheel rim made of Aluminium needs greater strength and produces a higher reaction force to collapse. The condition shows that Aluminium possesses excellent structure reliability. Looking at the weight of the material itself, the wheel rim made of Magnesium contains the lowest weight, yet its capability to absorb impact energy is weak. On the other hand, the weight of the Stainless Steel rim is the heaviest compared to the other materials. This makes Aluminium the most practical and economical material to be used as the wheel rim. Based on previous studies, Aluminium is more suitable to be used in commercial vehicles compared to Magnesium due to its good mechanical properties, durability and acceptable price [3, 4].

Comparing the 90 degree and 13 degree impact tests and disregarding the different mesh sizes applied, the energy absorbed in the 90 degree impact test recorded a higher energy absorbed than the 13 degree impact test. Since the orientation of the wheel rim is dissimilar, the contact between the striker and the rim during the impacting event is different for each impact case. Figure 9 shows the deformations, which for each impact case is not the same. It is observed that the areas of contact and impact occurring in both tests are not similar. In the 90 degree impact test, the contact area is most likely to cover the whole circumference of the wheel rim. This is in contrast with the 13 degree impact test, where the contact area in the impacting event occurred at the edge of the wheel rim. The contact area of the 13 degree impact test is smaller than the 90 degree impact test. This phenomenon explains why the energy absorbed obtained in the 13 degree impact test is lower than the 90 degree impact test.

CONCLUSIONS

It appears that the peak force of the 90 degree impact test is superior to the 13 degree impact test. From the entire test, Aluminium has the highest value of reaction force, followed by Stainless Steel and Magnesium. In terms of energy absorption, Aluminium had recorded the greatest energy absorption value, while Magnesium has the lowest energy absorption value. This complies with the material's property as aluminium has higher yield strength than the other materials, thus a high force is needed for the material to collapse. Different wheel assemblies and orientations have shown different results of energy absorption capacity. The area of contact between the striker and wheel rim plays an important role in affecting the energy absorbed. The correlation for both impact tests shows that Aluminium has the best percentage of deviation at 24.40 %, even though dissimilar mesh sizes were used throughout the simulation. From the results of this study, it shows that the results can be used as a basis platform to study the parameters for wheel changes such as new material, thickness, size and pattern of spokes. Consequently, more study will provide better solutions for industrial problems involving the production of real parts.

ACKNOWLEDGEMENTS

The authors would like to thank to the reviewer community who greatly help to produce this manuscript. Special thanks to financial support under grant of FRGS/2/2013/TK01/FKM/03/3/F00173 (UTeM).

REFERENCES

[1] Ramakrishna K. Automobile engineering: PHI Learning Pvt. Ltd.; 2012.

[2] Chang C-L, Yang S-H. Simulation of wheel impact test using finite element method. Engineering Failure Analysis. 2009;16:1711-9.

[3] Meghashyam P, Naidu SG, Baba NS. Design and Analysis of Wheel Rim using CATIA & ANSYS. International Journal of Application or Innovation in Engineering & Management. 2013;2.
[4] Bisoyi SK AKM. Analysis and development characterisation of automotive wheels. International Journal & Magazine of Engineering. 2015;2:667–71.
[5] McMahon. M. Trade of motor mechanic: Wheels and tyres. In: SOLAS 2013 Module 7. 2013:1–40.
[6] Guide to alloy wheels manufacture and use. 2015.
[7] Ishikawa S, Sakai Y, Nosaka N. Application of Impact Analysis for Aluminum Wheel with Inflated Tire. SIMULIA Community Conference2014.
[8] Chauhan MR, Kotwal G, Majge A. Numerical simulation of tire and wheel assembly impact test using finite element method. SAE Technical Paper; 2015.
[9] Akbulut H. On optimization of a car rim using finite element method. Finite Elements in Analysis and Design. 2003;39:433-43.
[10] Siribodhi P, S T. Hybrid experimental-numerical simulation of impact structure of aluminum wheels. Journal of Research in Engineering and Technology. 2007;4:109–24.
[11] Chang C, Yang S. Finite element simulation of wheel impact test. Journal of Achievements in Materials and Manufacturing Engineering. 2008;28:167-70.
[12] Ekhar VIS OA. Design and performance analysis of alloy wheels using CATIA ANSYS modeling tool. International Journal of Scientific Engineering and Technology Research. 2014;3(43):8789–93.
[13] Sivakrishna V, Balabashker J. Impact Analysis of Aluminum Alloy Wheel. International Journal & Magazine of Engineering,Technology, Management and Research. Impact Analysis of Aluminum Alloy Wheel;1:608–18.
[14] Ali MBA. Evaluating Charpy impact signals using power spectrum densities: a finite element method approach. International Journal of Mechanical and Materials Engineering. 2010;6:92-101.
[15] Mariappan D, Shankar M, Pratap A. Simulation and experimental validation of automotive components. Proceedings of the IMAC-XXVII. Orlando, Florida USA: Society for Experimental Mechanics Inc.; 2009.
[16] Data A. NADCA product specification standards for die castings. Arlington Heights, NADCA. 2003.
[17] General Properties of Steels. 7 December, 2015.
[18] Kamal M, Rahman M. Finite Element-based fatigue behaviour of springs in automobile suspension. International Journal of Automotive and Mechanical Engineering. 2014;10:1910-9.
[19] Ramon-Villalonga L, Enderich T. Advanced Simulation Techniques for Low Speed Vehicle Impacts. Sixth LS-DYNA Anwenderforum, Frankenthal. 2007:25-36.
[20] OI. Road Safety Annual Report 2015. Paris; 2015.
[21] More ST, Bindu R. Effect of Mesh Size on Finite Element Analysis of Plate Structure. International Journal of Engineering Science and Innovative Technology. 2015;4:181-5.
[22] Kankariya N, Sayyad F. Numerical Simulation of Bumper Impact Analysis and To Improve Design for Crash Worthiness.