Jurnal Teknologi

Full paper

Dynamic Energy Absorption Performances of Rain Forest Vehicle (RFV) under Frontal Impact

M. S. Othman, Z. Ahmad*

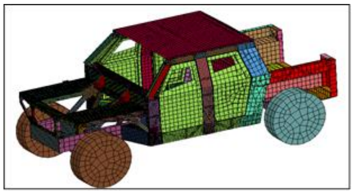
Computational Solid Mechanics Laboratory (CSMLab), Faculty of Mechanical Engineering, Universiti Teknologi Malaysia, 81310 Johor Bahru, Malaysia

*Corresponding author: azaini@fkm.utm.my

Article history

Received :23 October 2013
Received in revised form :
14 December 2013
Accepted :10 January 2014

Graphical abstract



Abstract

This paper treats the crash analysis and energy absorption response of Rain Forest Vehicle (RFV) subjected to frontal impact scenario namely impacting rigid wall and column. Dynamic computer simulation techniques validated by experimental testing are used to carry out a crash analysis of such vehicle. The study aims at quantifying the energy absorption capability of frontal section of RFV under impact loading, for variations in the load transfer paths and geometry of the crashworthy components. It is evident that the proposed design of the RFV frontal section are desirable as primary impact energy mitigation due to its ability to withstand and absorb impact loads effectively. Furthermore, it is found that the impact energy transmitted to the survival room may feasibly be minimized in these two impact events. The primary outcome of this study is design recommendation for enhancing the level of safety of the off-road vehicle where impact loading is expected.

Keywords: Crash; rain forest vehicle; impact loading; energy absorption; finite element

Abstrak

Kajian ini menganalisis pelanggaran dan penyerapan tenaga terhadap Kenderaan Hutan (RFV) pada impak hadapan iaitu memberi kesan dinding tegar dan lajur. Teknik simulasi dinamik pengkomputeran diujiguna dalam eksperimen untuk menjalankan analisis kemalangan kenderaan itu. Kajian ini bertujuan untuk mengukur keupayaan penyerapan tenaga bahagian hadapan RFV dengan berbeban, untuk variasi di dalam laluan pemindahan beban dan geometri komponen perlindungan. Adalah terbukti bahawa reka bentuk yang dicadangkan pada bahagian hadapan RFV adalah wajar kerana kesan utama pada pengurangan tenaga yang dapat menahan dan menyerap kesan bahan berbeban. Tambahan pula, didapati kesan tenaga yang dihantar ke ruang kenderaan yang mungkin dapat dikurangkan dengan kesan muatan yang diberikan. Dapatan utama kajian ini adalah cadangan reka bentuk untuk meningkatkan tahap keselamatan kenderaan di luar jalan raya dengan kesan muatan yang dijangkakan.

Kata kunci: Pelanggaran; kenderaan hutan; kesan muatan; penyerapan tenaga; unsur terhingga

■1.0 INTRODUCTION

Rain Forest Vehicle (RFV) is defined as an outdoor vehicle that can operate in the rural area and military particularly in the forest. This vehicle consists of four wheels and has a great stability when operating on the uneven terrain and sloping condition. Based on the condition of the forest and safety condition, the frontal part of the RFV must be able to sustain the crash phenomenon. For controlling the crush response of the vehicle and ensuring acceptable deformation, a survivable space is the primary goal in designing such a vehicle. The main energy management components, such as longitudinal rails that provide the major portion of the energy absorption, undergo large plastic deformations. Controlling the crash response of the vehicle without any encroachment into survival space are the main reasons in the vehicle safety [1].

Numerous established works have been published in studying the theoretical prediction on the mechanics of thin-walled tubular structures [1-8]. Various types of thin-walled tubes have been frequently used as energy absorbers in the automotive industry to mitigate the adverse effects of impact and hence protect the vehicle structure under consideration when subjected to different loading conditions. More recently, by considering various important parameters influencing the crashworthiness performance namely strain rate, material and geometry nonlinearity, the study continues comprehensively to improve crash worthiness capability either using experimental or numerical techniques. In general, the crush response of vehicle tubular components involves three main collapsible modes which are axial collapse mode, bending mode and mixed mode. Hsu and Jones [4] have research on the energy absorption capacity of two different materials. Such study has concluded that aluminum tubes absorb less energy compared to stainless steel tubes.

However, stainless and mild steels have less efficient according to dimensionless energy absorption effectiveness. Witteman [5] has also explained in-depth on the energy absorption capacity of the inherent structure obtained from experimental and simulation results. Moreover, the study has suggested a new model for improvising the crashworthiness of the vehicle. In different study, it is evident that by having too many side or corner of the structure, it will improve the energy absorption yet will limit until certain extent [9-11]. Nowadays, lightweight materials have widely been used in automotive technology due to decreasing the fuel consumption [12-19]. In contrast, the requirement of lightweight structure is not the main goal of this present study. Thus, the material used for the RFV is fully mild steel.

For decades, many researchers have extensively employed finite element method to improve vehicle safety in the event of impact. Some useful studies regarding impact loading of full vehicles have been carried out in recent times. For instance, Gursel *et al*. [12] have compared two car models of Dodge Intrepid and Ford Taurus and replicated the crash tests by using finite element method. This method aims to minimize the number of crash tests required while designing a vehicle. Zaouk *et al*. [20] have evaluated impacting of Chevrolet C-1500 to a rigid barrier by using commercial nonlinear finite element code LS-DYNA. In similar approach, the similar code has also been used by Cheng *et al*. [9] to investigate the effects of applying different crash scenarios to Honda Accord DX. For vehicle impacting rigid pole scenarios, Liu [15] has simulated the crash analyses of an off-road utility vehicle and identified the energy absorption and acceleration of human dummies.

The present study presents the crash analysis and energy absorption response of Rain Forest Vehicle (RFV) subjected to frontal impact scenario namely impacting rigid wall and column. Dynamic computer simulation techniques validated by experimental testing are used to carry out a crash analysis of such vehicle. The study aims at quantifying the energy absorption capability of frontal section of RFV under impact loading, for variations in the load transfer paths and geometry of the crashworthy components. Toward this end, the proposed design of the RFV frontal section may be advantageous as primary impact energy mitigation due to its ability to withstand and absorb impact loads effectively. Furthermore, it is found that the impact energy transmitted to the survival room may feasibly be minimized in these two impact events. The primary outcome of this study is design recommendation for enhancing the level of safety of the off-road vehicle where impact loading is expected.

■2.0 EXPERIMENTAL STUDIES AND FINITE ELEMENT MODEL VALIDATION

2.1 Experimental Studies

This section describes the experimental studies used to validate the FE model under axial loading. The validated FE model was then used to treat the full crash analysis of RFV using detailed finite element model. This approach is considered acceptable as the main goal of the present study is to study the impact response of RFV under frontal axial loading and evaluate the influence of the important controlling components of frontal sections. Also, it provided reasonable results for RFV by using the same material model of the validated FE model. It should be noted that it is a formidable task to perform a full scale crash test of RFV hence results from quasi static testing were used to validate the FE model of the thin-walled components. In order to further consider dynamic impact loading, the associated strain rate effects were taken into account by using the Cowper-Symonds parameters for inclusion in the FE model, as established in a previous study [4].

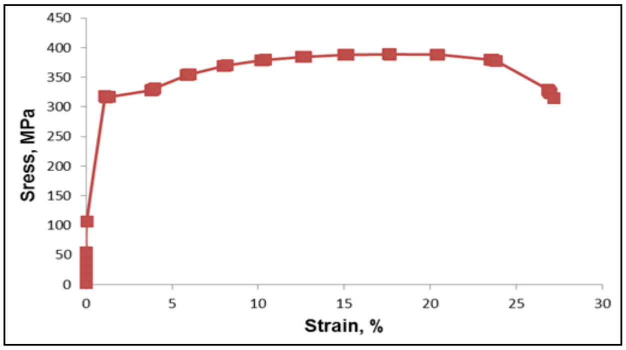


Figure 1 Engineering stress-strain curve of the mild steel material

The tube specimens were manufactured from mild steel. The material properties of the tube were extracted from the standard tensile test in accordance with ASTM A370. Figure 1 shows the engineering stress-strain curve for the tube material. For the subsequent test, the longitudinal component was quasi statically compressed to observe the crush response up to 130 mm displacement. The experimental test was used to validate the numerical results in terms of deformation modes and load-displacement curves. From the experiment, the component has failed in form of symmetric or axial whereas from the simulation, the FE model has deformed in form of axial yet not really stable. Figure 2 depicts the deformation modes obtained from the numerical and experimental results.

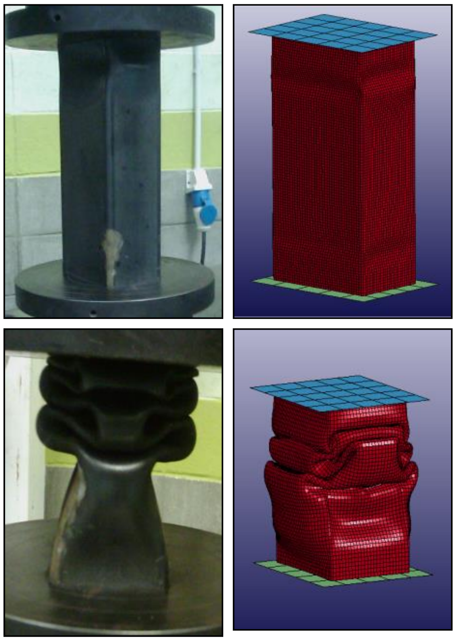


Figure 2 Deformation modes obtained from experimental and numerical results

Nonetheless, the load-displacement curves show a good agreement between experimental results and numerical results as shown in Figure 3. It is well-known that the first peak of simulation result is always higher than that of experimental results due to the presence of the perfect model in the simulation. However, the tested component test has some imperfections on the tube wall that may influence the crush response of the overall performance. Overall, the mean load is about similar for both results. From these results, it is deemed that the FE model is an accurate and reliable model for subsequent analysis for RFV simulation.

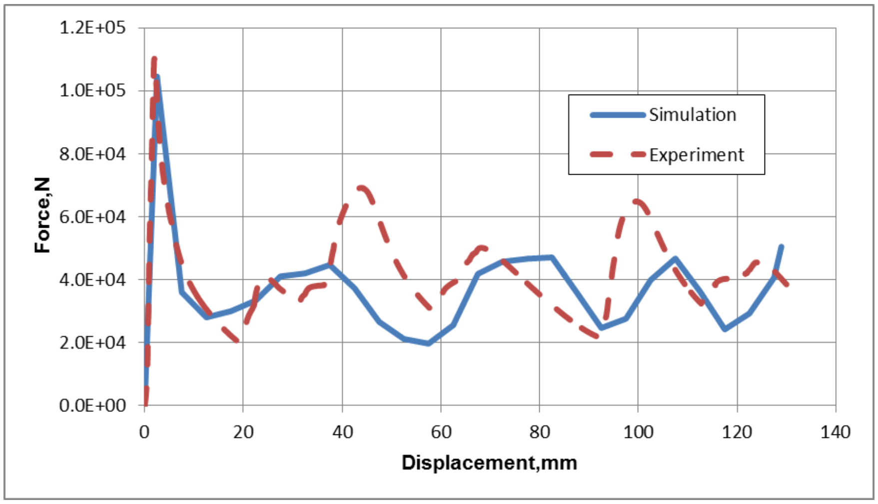


Figure 3 Load-displacement curves obtained from experiment and simulation

2.2 Description of FE Model

2.2.1 FE Model of Frontal Sections

All computer modeling in the present study was conducted using the nonlinear finite element (FE) code LS-DYNA 971. A model of the tube components was developed using quadrilateral Belytschko-Tsay shell elements with 5 integration points through the thickness which suitably model the buckling response of the tubes. For the efficient crashworthy design of RFV, the structural component with the highest energy absorption capacity must be identified. It is noteworthy that the frontal longitudinal member plays an important role in absorbing impact energy. Using various assemblies of frontal longitudinal section, dynamic simulation was carried out to evaluate the energy absorption capacity under an initial velocity of 13 m/s.

The structure was impacted onto the rigid wall to represent the crash event. Five different FE models have been simulated in this analysis. Model *A* is the original design. The middle part of Model *A* was improved by adding an additional part as a supporter to prevent the bending occurred for Model *B*. In different way, Model *C* was also improved by adding an additional part in the longitudinal direction and Model *D* is the improved model of Model *C* by adding another part in the middle section. Model *E* was improved by removing the additional part. For illustration, Figure 4 shows the various models introduced for the longitudinal sections.

Figure 5 shows load-displacement curves for various simulated model. The load increases drastically and drops onset of the initial peak load. All the model design exhibits the same characteristic due to similar crash scenario. In reaching the initial peak load, the structure experienced folding process. Hereafter, the bending process has taken place up to 70 mm displacement, thus resulting in the reduction in load till the maximum deformation. The higher force will result to the weakening of the structure leading to the deformation of irregular fold. Upon the increase of the load, the structure tends to be weakened as the energy absorption and stiffness decrease. As a result, it will produce bending due to the presence of the unstable load. As can be seen from Figure 5, Model *C* has the highest initial peak load compared to the others. This analysis is only valid for the axial impact and may differ for the oblique impact. Table 1 tabulates the energy absorption of each model.

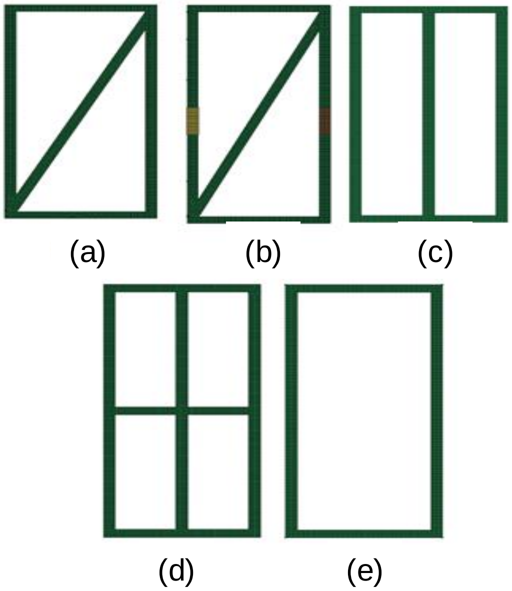


Figure 4 FE model of frontal longitudinal section (a) Model A (b) Model B (c) Model C (d) Model D (e) Model E

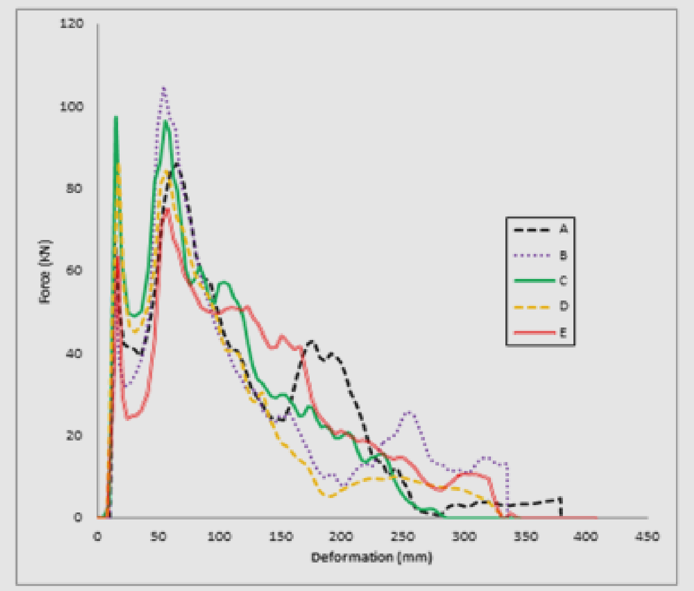


Figure 5 Load-displacement curve of different design of longitudinal frontal section of RFV

Table 1 Energy absorption capacity of different design of longitudinal frontal section of RFV

Design	Initial peak load (kN)	Energy absorbed at 50ms (kJ)
A	67.5	34.3
B	58.5	35.8
C	93.8	54.3
D	86.5	47.0
E	63.7	33.8

2.2.2 FE Model of RFV

From Table 1, it is obvious that the energy absorption capacity is great for Design C. However, from the crash energy management point of view, the crashworthy design has to compromise between the energy absorption capacity and initial peak load. As such, Design B was chosen for the subsequent analysis for a full model of RFV. Figure 6 depicts the finite element model of RFV which consists of 278 different parts. The RFV model has a total length of 4960 mm, 2200 mm width, 1800 mm height and total mass of the body is 2900 kg. The FE model was meshed by using 4 nodes Belytschko-Tsay shell element.

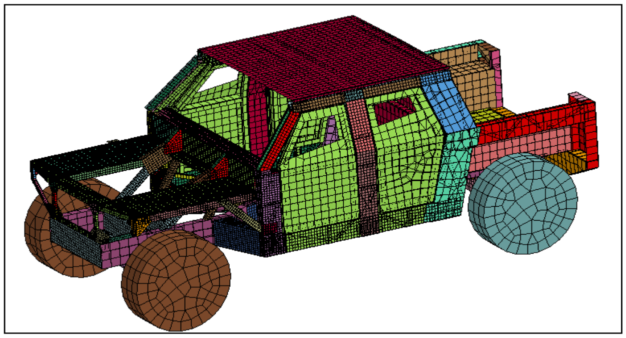


Figure 6 FE model of RFV

Overall, this model consists of 84199 shell elements and 85738 nodes. The material model was defined by a piecewise linear plasticity and Blatz-KO rubber only for the tires. The connections between the parts were modelled with nodal constraints namely constrained nodal rigid bodies and spot welds. The contact condition between the parts of the RFV was simulated using an automatic single surface contact with the static and dynamic coefficients of 0.3 and 0.2, respectively. This simulation was terminated at 60 ms with an initial velocity of 15 m/s (54 km/h) for both crash conditions. Parts A, B, C and D as shown Figure 7 represent the longitudinal members of RFV that contributes in dissipating the impact load. Parts A, B, C and D denote a cross column, an end longitudinal column, a frontal longitudinal column and a chassis, respectively. There are two crash scenarios that have presently studied:

(a) Rigid barrier
The rigid wall is defined by a planar shell element with constrained all nodes with a boundary condition to restrict the movement in all directions.

(b) Rigid pole
The rigid pole is defined by a solid cylinder by constraining all the nodes at the bottom. The diameter of the pole is 800 mm and is placed at the centre of the impact. This crash scenario represents the vehicle impacting a tree.

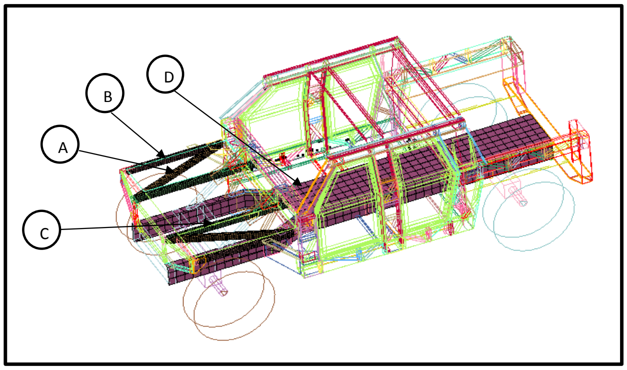


Figure 7 Main component of longitudinal parts

■3.0 CRUSH RESPONSE AND ENERGY ABSORPTION CAPACITY

3.1 Full Frontal Impact on a Rigid Wall

Figure 8 shows the deformation of the RFV model at crushing period of 0, 20, 4 and 60 ms. From crashworthiness point of view, the deformation of the crashworthy structure must take place in the frontal portion only by having a progressive crushing as evident in the present study. The safety compartment has to sufficiently remain stiff to withstand the impact energy transferred in the crash events. It is noteworthy that the frontal parts start to crumple at a crash period of 20 ms as shown in Figure 8. Furthermore, at a crash period of 40 ms, the right and left sides of the vehicle have influenced the whole structure to crush upwards under bending mode particularly at the frontal section. Most importantly, the flange of the section has started to buckle locally due to the initiation of plastic hinge mechanism. The distribution of Von Misses stress with impact duration of 60 ms is shown in Figure 9. As expected, the maximum stress occurs at critical points where the deformation of folding and global bending could be seen. Part D has deformed in a progressive crushing mode followed by more excessive bending mode upon impact duration of 40 ms. Further improvement needs to be conducted to improvise this model.

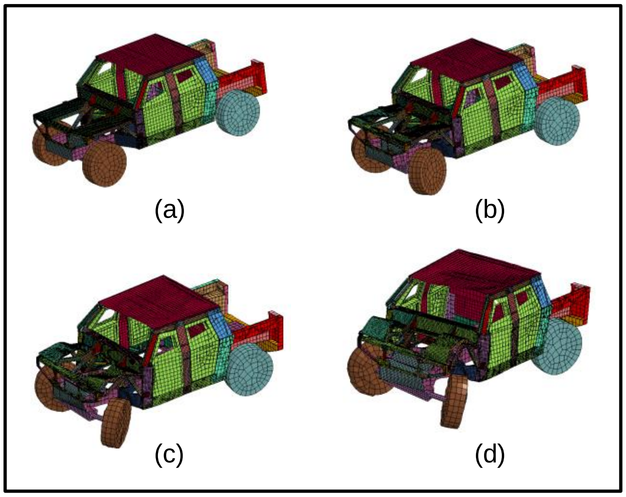


Figure 8 Deformation profiles of RFV during the frontal impact on a rigid wall (a) 0 ms (b) 20 ms (c) 40 ms (d) 60 ms

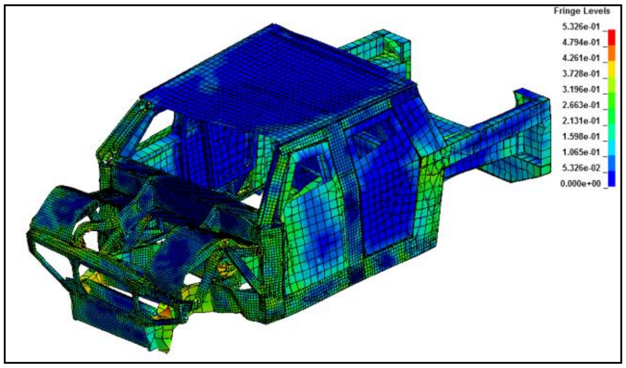


Figure 9 The distribution of stress at 60 ms according to Von-Mises theory for a rigid wall impact

Figure 10 illustrates the internal energy of frontal longitudinal members up to the maximum impact duration. In details, Part D has the highest energy absorption of 56 kJ followed by C, A and B in which are 31.6 kJ, 23.3 kJ and 13.5 kJ, respectively. From the numerical results, the maximum crash energy was almost fully absorbed by the frontal part of the vehicle and was partly transferred to other sections. From the trend depicted, Part D exhibits the highest energy gradient below 20 ms due to the initiation process of the deformational structure. The energy absorption capacity continues to increase yet with a lower energy gradient due to the bending process. For this type of crash phenomenon, a chassis must absorb the highest impact energy since it is purposely designed to absorb the most impact energy by means while maintaining the survival space.

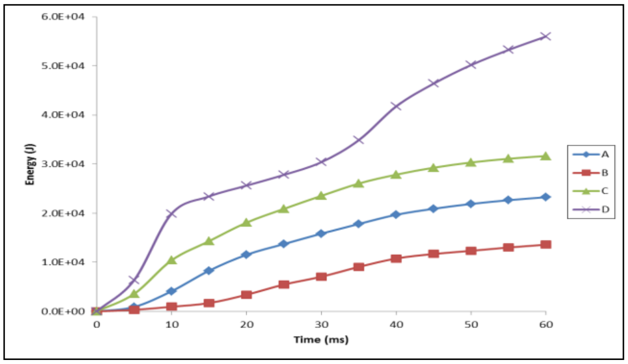


Figure 10 The internal energy dissipated by different parts under rigid wall impact

3.2 Full Frontal Impact on a Rigid Column

For the rigid pole impact, the RFV model was impacted onto the stationary rigid pole with an initial velocity of 15 m/s. Figure 11 shows the deformation of RFV at 0, 20, 40, and 60 ms. In this simulation, it is obvious that the safety compartment has less deformation compared to rigid wall impact case. In particular, the A pillar, toe-pan and dashboard do not deform thus showing the deformation only taken place by frontal section. Furthermore, Part C (frontal longitudinal columns) exhibits higher deformation by progressive deformation and followed by bending compared to the other parts, thus resulting in the highest energy absorption capacity. As a result, this model is capable of withstanding impact load from the rigid pole while maintaining the survival room.

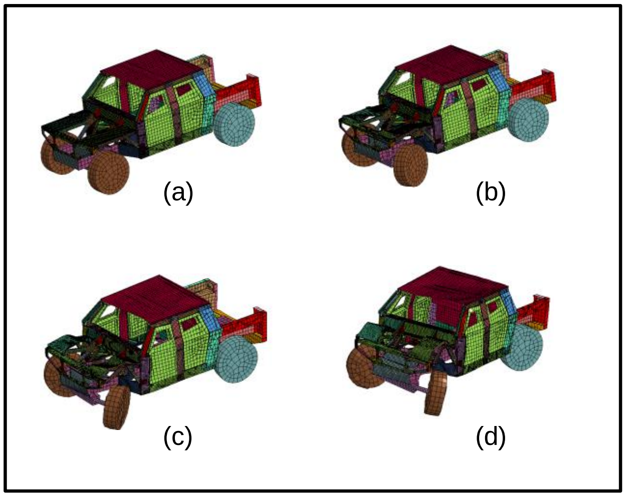


Figure 11 Deformation profiles of RFV during the frontal impact on a rigid pole (a) 0 ms (b) 20 ms (c) 40 ms (d) 60 ms

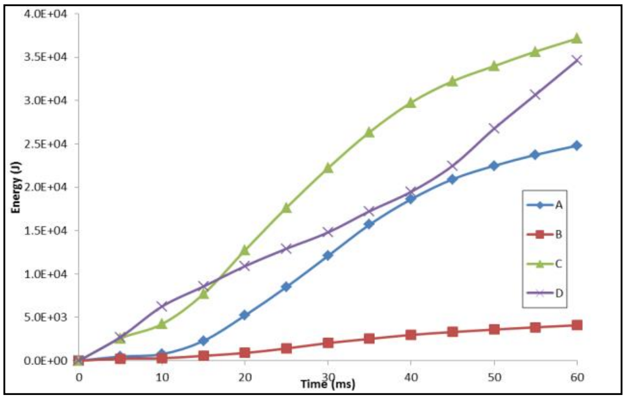


Figure 12 The internal energy dissipated by different parts under rigid wall impact

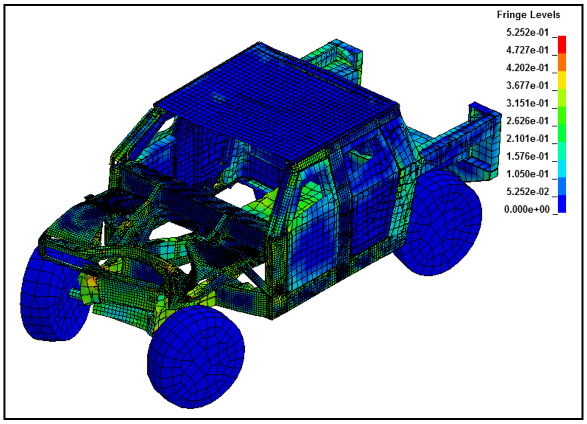


Figure 13 The distribution of stress at 60 ms according to Von-Mises theory for a rigid pole impact

Figure 12 shows the energy absorbed by the different parts under frontal impact loading. Since Part C is the nearest point to the impact zone, a higher energy absorption capacity of 37.1 kJ is shown and followed by Parts D, A and B. In addition to this, Part C crumples more compared to the others. These structures deform under buckling mode onset of reaching a critical value and initiate a plastic hinge mechanism. The moment distribution changes as the additional hinges form. Part B has the lowest energy absorption capacity due to the long distance from impact zone and deforms accordingly to the pole circumferences. Moreover, Part D (chassis) also deformed in a crumple mode. Nonetheless, the impact energy has been transferred to the middle of the compartment which could jeopardize the survival room. Due to this weak structure of Part D, it has to be reinforced accordingly (Figure 13).

■4.0 CONCLUSION AND DESIGN INFORMATION

This paper has examined the crush response and energy absorption of existing RFV model under impact loads on the rigid wall and rigid pole. Throughout this simulation result of the full frontal impact, it shows that Part D as the main crashworthy component or the chassis of the vehicle has experienced in non-progressive crushing which is global bending and buckling modes resulting in excessive deformation. The present results shows that the highest energy absorption is about 56 kJ at impact duration of 60 ms. The chassis stiffness and strength of the body is insufficient and need to be improvised. In the rigid pole impact case, Part C or frontal longitudinal has absorbed the highest energy absorption of about 37.1 kJ compared to others due to nearest crush zone. However, Part D has deformed at the passenger compartment and need to be reinforced accordingly. This study has shown that the proposed design of frontal section particularly the longitudinal columns appear to be advantageous in impact applications where both crash scenarios are expected. The new research information generated has merit in developing guidance for the design of RFV.

Acknowledgement

This project is supported by the Ministry of Higher Education (MOHE) Malaysia and Universiti Teknologi Malaysia Research Grant: No. Q.J130000.2624.01J75 and Q.J130000.2524.04H82. Appreciation and acknowledgement also goes to all Universiti Teknologi Malaysia (UTM) and Innopeak Sdn. Bhd. staff who involved in this project.

References

[1] Wierzbicki, T. 1983. Crushing Analysis of Plate Intersections. StructuralCrashworthiness, edited by Jones, N., and Wierzbicki, T., Butterworth Press.

[2] Abramowicz, W., and Jones, N. 1984. Dynamic Crushing of Square Tubes. *International Journal of Impact Engineering*. 2(2): 179–208.

[3] C. C. Yang. 2003. Dynamic Progressive Buckling of Square Tubes. Presented at the The 27thconference on Theoretical and Applied Mechanics, Taiwan.

[4] S. S. Hsu, N. Jones. 2004. Dynamic Axial Crushing of Aluminium Alloy 6063-T6 Circular Tubes. *Latin American Journal of Solids and Structures*. 1: 277–296.

[5] W. J. Witteman. 1999. Improved Vehicle Crashworthiness Design by Control of the Energy Absorption for Different Collision Situations. Technische Universiteit Eindhoven.

[6] Z. Fan, *et al.* 2011. Quasi-static Axial Compression of Thin-walled Tubes with Different Cross-sectional Shapes.

[7] F. Schneider and N. Jones, 2004. Impact of Thin-walled High-strength Steel Structural Sections. *Journal of Automobile Engineering*. 218.

[8] V. Tarigopula, *et al.* 2005. Axial Crushing of Thin-walled High-strength Steel Sections. *International Journal of Impact Engineering*. 32: 847–882.

[9] Cheng, Z. Q., J. G. Thacker, W. D. Pilkey, W. T. Hollowell, S. W. Reagan and E. M. Sieveka. 2001. Experiences in Reverse-engineering of a Finite Element Automobile Crash Model. Finite Elements in Analysis and Design. 37: 843–860.

[10] Jenefeldt, F. and R. Thomson. 2004. A Methodology to Assess Frontal Stiffness to Improve Crash Compatibility. *International Journal of Crashworthiness.* 9(5): 475–482.

[11] Ghannam, M. Y., M. Niesluchowski and P. M. Culkeen. 2002. Analysis of a Frontal Rail Structure in a Frontal Collision. SAE 01(0688).

[12] Gursel, K. T., Nane and S.N. 2010. Non-linear Finite Element Analyses of Automobiles and Their Elements in Crashes. *International Journal of Crashworthiness*. 15(6): 667–692.

[13] Hong, S.-W., C.-K. Park, Pradeep Mohan, R. M. Morgan, C.-D. Kan, K. Lee, S. Park and H. Bae. 2008. A Study of the IIHS Frontal Pole Impact Test. SAE International 0507(01).

[14] Huiwen Hu, W.-J. Lu and Z. Lu. 2011. Impact Crash Analyses of an Off-road Utility Vehicle–Part II: Simulation of Frontal Pole, Pole Side, Rear Barrier And Rollover Impact Crashes. *International Journal of Crashworthiness.* 17(2): 163–172.

[15] Liu, Y. 2004. Crashworthiness Analysis of Finite Element Truck Chassis Model Using LS-DYNA. 11th International LS DYNA Users Conferences.

[16] Matsumoto, André Takashi, Driemeier, Larissa, Alves and Marcílio. 2012. Performance of Polymeric Reinforcements in Vehicle Structures Submitted to Frontal Impact. *International Journal of Crashworthiness:* 1–18.

[17] Paul Du Bois, Clifford C. Chou, Bahig B. Fileta, Tawfik B. Khalil, Albert I. King, Hikmat F. Mahmood, Harold J. Mertz and J. Wismans. 2004. Vehicle Crashworthiness and Occupant Protection. Southfield, Michigan, Automotive Applications Committee American Iron and Steel Institute.

[18] R, L. K. and V. K. Banthia. 2009. Numerical Simulation for Crashworthiness of Frontal Rail Structure Using Explicit Finite Element Code. SAE 26(60).

[19] Witteman, W. J. 1999. Improved Vehicle Crashworthiness Design by Control of the Energy Absorption for Different Collision Situations, Technische Universiteit Eindhoven.

[20] Zaouk, A. K., N. E. Bedewi, C.-D. Kan and D. Marzougui. 1997. Development and Evaluation of a C-1500 Pick-up Truck Model for Roadside Hardware Impact Simulation.