RESEARCH PROGRESS IN MECHANICAL AND MANUFACTURING ENGINEERING VOL. 4 NO. 2
(2023) 137-142

© Universiti Tun Hussein Onn Malaysia Publisher's Office

RPMME

Homepage: http://penerbit.uthm.edu.my/periodicals/index.php/rpmme
e-ISSN : 2773-4765

Investigation of Crash Impact Toward the Highway Impact Attenuator Structure Using Finite Element Analysis

Awgku Azrol Nur Afiq Awgku Aji[1]*, Al Emran Ismail[1]

[1]Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia, Batu Pahat, 86400 Johor, MALAYSIA

*Corresponding Author Designation

DOI: https://doi.org/10.30880/rpmme.2023.04.02.016
Received 10 Sept 2023; Accepted 26 Nov 2023; Available online 15 December 2023

Abstract: This study aims to investigate and evaluate the effectiveness of an impact attenuator designed for installation on roads. The goal is to identify the impact attenuator design that can best absorb kinetic energy from crash impacts. The study proposes the use of computational crash simulations to assess the safety performance of the impact attenuator, reducing the costs associated with development and testing of new safety designs. The analysis begins with studying the behavior of impact attenuator structures and accurately modeling them in numerical simulations using ANSYS Workbench. The simulations involve crashing the impact attenuator with a rigid object at various speeds to observe the resulting deformation and energy absorption. The review is divided into three categories: simulation tests of the existing model, simulation tests of an improved version of the previous model, and a comparison between the two simulation tests. The outcomes of these tests can then be examined and analyzed.

Keywords: Crash Impact Simulation, ANSYS Workbench, Impact Attenuator, Finite Element Method (FEM)

1. Introduction

Impact attenuators, also known as crash cushions, are road safety devices that absorb the kinetic energy of colliding vehicles to bring them to a safe stop. They are typically installed in areas such as exits, road diversions, and toll plazas where unprotected spaces pose a significant risk to vehicles and occupants [1]. The effectiveness of impact attenuators is crucial in preventing severe injuries and fatalities. To assess their performance during the design phase, crash analysis is necessary. Finite element analysis (FEA) using software like ANSYS is a commonly employed method for conducting crash simulations [3]. In this study, the impact attenuator on a highway is analyzed using ANSYS Explicit Workbench software, which allows for crash simulations to evaluate its performance [4].

The crash analysis using FEA saves a lot of time and resources compared to an actual crash test on the actual vehicle in evaluating the impact attenuator. However, FEA is usually done using high-performance computers to keep computational time minimum. Even with high-performance computers,

---

*Corresponding author: emran@uthm.edu.my

crash simulation using FEA usually takes hours even days to complete the simulation. When a high-performance computer is not available, static analysis can be done to estimate the crash analysis result, such as done by [5]. However, the static analysis does not consider the dynamic behavior of the model, which makes the static analysis results to be not accurate.

This study focuses on the development, design, and simulation of a highway impact attenuator. The impact attenuator is modeled using SolidWorks software, and its behavior and energy absorption during impact are analyzed using ANSYS Workbench software. The proposed simulation method enables efficient crash simulations, including the determination of impact attenuator parameters, within a relatively short timeframe [6]. This method is particularly suitable for use during the design stage, allowing for a parametric study of the impact attenuator. The paper first demonstrates the problem formulation and modeling of the impact attenuator, followed by a comprehensive discussion of the simulation results obtained from the developed model.

2. Materials and Methods

In this phase, a parameter of an existing impact attenuator product was selected, considering its structure and characteristics to achieve good results for the project. The selected product's geometry was sketched using the identified parameters, allowing for a clear visualization of the impact attenuator structure before proceeding. The geometry modeling and design were then performed using Solidworks 2020 software to create the selected impact attenuator's structure and characteristics. After completing the modeling design, the impact attenuator model was subjected to dynamics simulation to simulate its behavior. Numerical simulation using ANSYS software was conducted to evaluate the model's performance by simulating crash impacts with a specific object. The results of the crash simulation provided insights for potential improvements and enhancing the model's performance. Factors influencing crash performance, such as structural design, materials, and product capabilities, were identified to guide the improvement process. A new design for the impact attenuator model was developed, followed by dynamics simulation and numerical simulation to assess its capabilities. Finally, the crash simulation results of the new model were obtained for further analysis and reference in improving its performance. Aluminium alloy is a popular choice for in-car body designs due to its corrosion resistance, ductility, high electrical conductivity, and lightweight nature. It offers a superior weight-to-weight ratio compared to steel, making it suitable for car body materials. In the context of impact attenuators, structural steel and rubber are commonly used materials. Table 3.4 provides properties of aluminium alloy, rubber, and structural steel specifically for lightweight vehicles and impact attenuators.

Table 2.1: Materials properties

Properties	Aluminium Alloy	Structural Steel	Natural Rubber
Density (kgm^{-3})	2770	7850	1000
Young’s Modulus (MPa)	71	210	-
Poisson’s Ratio	0.33	0.3	0.495
Bulk Modulus (GPa)	69.6	16.7	-
Shear Modulus (GPa)	26.7	76.9	-
Yield Strength (MPa)	280	250	30
Tangent Modulus (Pa)	0.330	0.361	-

The study involves geometry modeling, sketching, and tentative design of an impact attenuator, which is shown in Figure 2.1. The CAD implementation using Solidworks 2021 software was used to construct the design before importing it into ANSYS Workbench. Previous research and analysis served as references for geometry, facilitating the validation process. The pre-processing stage involved an interactive procedure prior to mesh generation. Table 2.2 presents the parameters of an existing impact attenuator product, which served as a basis for comparison. Additionally, two design variations of impact attenuators, labeled as attenuator A and attenuator B, were created, with their respective parameters outlined. In this project, a simple simulation model of impact attenuator crash would be developed. The situation considered for modelling is given in figure 2.2. The vehicle is modeled as a rigid body with mass M. During the crash, the vehicle will hit the impact attenuator with constant velocity V and deform. Table 2.2 shows the variables considered for the rigid body.

Table 2.2: Parameters of TRACC Crash Cushion

Parameters	Definitions		
Impact Attenuator	TRACC Crash Cushion	A	B
Track	690 mm	690 mm	690 mm
Wheelbase	1350 mm	1350 mm	1350 mm
Height	400 mm	400 mm	400 mm
Materials	Structural Steel Rubber	Structural Steel	Structural Steel Natural Rubber
Impact shape	oval	Rectangle	Cylinder
Young's Modulus	2.0×10^{11} Pa	1.92×10^{11} Pa	2.0×10^{11} Pa
Poisson's Coefficient	0.3	0.27	0.9
Density	7850 kg/m^3	8235 kg/m^3	6852 kg/m^3

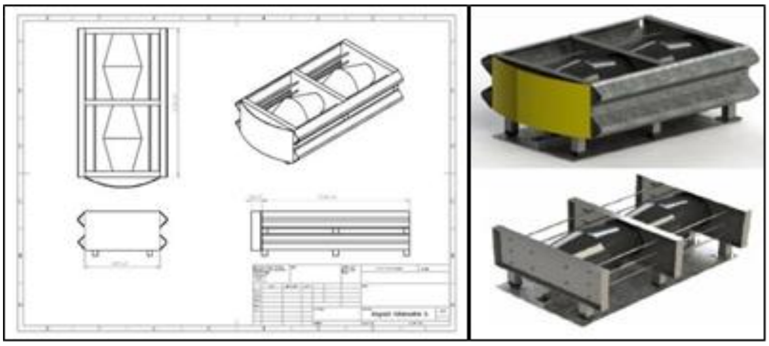

Figure 2.1: SolidWork drawing of TRACC Crash Cushion

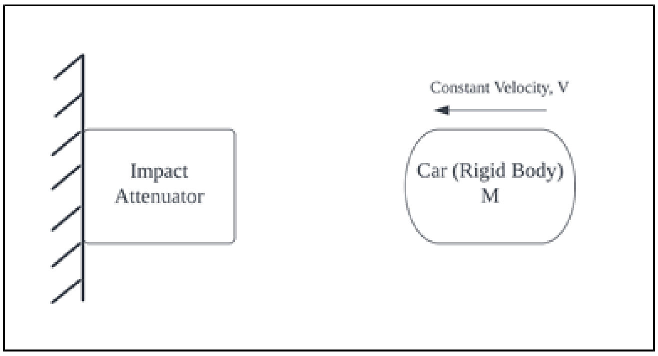

Figure 2.2: Simulation phenomenon under consideration

Table 3.3: Variables considered for rigid body

Variables	Definitions
Length	2822 mm
Width	1000 mm
Height	690 mm
Thickness	30 mm
Impact speed	110 km/s
Vehicle mass	300 kg
Study Time	0.001 s

3. Results and Discussion

This part of the study discusses the main results collected from the frontal crash of the lightweight vehicle against the impact attenuators. This crash simulation was analyzed for the TRACC Crash Cushion (validated model) and other two created designs which were Impact attenuator A and Impact attenuator B. Each simulation model was subjected to the same conditions and the results presented in terms of total deformation, equivalent Von-Mises stress and equivalent elastic strain obtained from the simulation. Figure 3.1 shows total deformation, figure 3.2 shows equivalent plastic strain and figure 3.3 maximum stress of impact attenuator A.

The impact attenuator models have a length of 1.350 meters and maximum displacement at 110 km/h for TRACC Crash Cushion, Impact attenuator A, and Impact attenuator B. Table 3.1 compares maximum deformation values, showing TRACC crash cushion experiencing high elastic deformation. Traumatic failure causes higher strain rates, affecting mechanical properties. A lightweight vehicle colliding with a TRACC crash cushion at 110km/h has the highest strain rate, indicating increased material elongation when yielding stress exceeds its critical value. This indicates that the material's mechanical properties are affected.

Designers use von Mises stress, a component of plasticity theory, to assess a design's ability to withstand specific load conditions. It is particularly useful for ductile materials like metals, as stresses exceeding their yield point trigger plastic behavior, characterized by irrecoverable strain. Plasticity occurs when a material is loaded beyond its yield strength, resulting in permanent deformation. Geometry nonlinearity is present due to high loads. The TRACC crash cushion has a maximum von Mises stress of 3.7345 MPa at 110 km/h, while Impact Attenuator A and B have a maximum stress of 3.9233 and 3.4426 MPa, respectively.

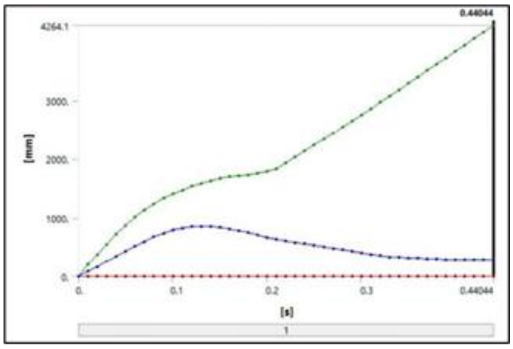

Figure 3.1: Graph total deformation (m) vs time (t)

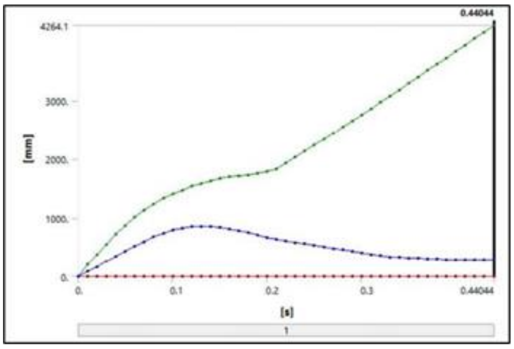

Figure 3.2: Graph equivalent plastic strain (m/m) vs time (s)

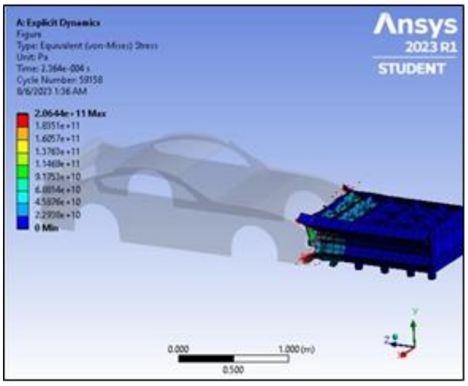

Figure 3.3: Maximum stress for Impact attenuator A.

4. Conclusion

This project successfully achieved its objectives by simulating a frontal crash against an impact attenuator using SolidWorks software. The lightweight vehicle and impact attenuator were designed using ANSYS Workbench version 19.1, and the deformation and failure behavior of the impact attenuator were investigated using Explicit Dynamic Analysis in ANSYS software. The results were compared to the TRACC Crash Cushion, an impact attenuator's previous design model, to provide a comprehensive analysis of the frontal crash simulation. The analysis reveals that the deformation of impact attenuators increased with rubber material compared to metal. The least deformation was observed in the impact cylinder shape, while the highest deformation was recorded in the impact oval shape at 0.70688 meters. The properties of the material and impact shape could affect deformation changes. The higher impact elasticity material used in the attenuator causes plastic deformation and absorbs the impact when subjected to a crash against lightweight materials. The equivalent von Mises Stress for all impact attenuators exceeds the yield strength of the metal structure, with the maximum stress increase varying depending on the materials used. The plastic strain also increased with impact speed, with the highest plastic strain at TRACC crash cushion at 3.3324 meters.

Crash simulation analysis generates results without destructive testing, allowing for optimization of design models before a real car prototype is produced. This method allows for problem-solving before spending time and money on actual crash tests, as tests can be conducted quickly and inexpensively using a computer. To assess the impacts' ability to absorb kinetic energy, we can consider the deformation values as an indicator. A higher deformation implies a greater amount of energy absorbed by the impact attenuator. Comparing the deformations, TRACC crash cushion exhibits the highest deformation value, followed by Impact attenuator A and then Impact attenuator B. This suggests that TRACC crash cushion has the potential to absorb the most kinetic energy among the three impacts attenuator.

Table 3.1: Comparison of result for different impact attenuator models.

Models	Maximum Displacement (m)	Maximum Elastic Strain	Maximum Stress (MPa)
TRACC Crash Cushion	12161	0.26387	3.7345
Impact Attenuator A	4264.1	1.3928	3.9233
Impact Attenuator B	2543.5	1.3191	3.4426

Acknowledgement

The authors would also like to thank the Faculty of Mechanical and Manufacturing Engineering, Universiti Tun Hussein Onn Malaysia for its support.

References

[1] C. Itu and S. Vlase, "Impact Attenuator Design for Improvement of Racing Car Drivers' Safety,"

Symmetry (Basel), vol. 15, no. 1, p. 159, Jan. 2023, doi: 10.3390/sym15010159.

[2] E. Özyurt, "Design and optimization of impact attenuator for a Formula SAE racing car," *Sigma Journal of Engineering and Natural Sciences – Sigma Mühendislik ve Fen Bilimleri Dergisi*, 2022, doi: 10.14744/sigma.2022.00041.

[3] A. Vettorello, G. A. Campo, G. Goldoni, and M. Giacalone, "Numerical-experimental correlation of dynamic test of a honeycomb impact attenuator for a formula sae vehicle," *Metals (Basel)*, vol. 10, no. 5, May 2020, doi: 10.3390/met10050652.

[4] L. W. Prasetya *et al.*, "Crashworthy Examination of a Newly Proposed Impact Attenuator Design: Experimental Testing and Numerical Analysis," *Modelling and Simulation in Engineering*, vol. 2021, 2021, doi: 10.1155/2021/5001060.

[5] S. Boria, S. Pettinari, F. Giannoni, and G. Cosimi, "Analytical and numerical analysis of composite impact attenuators," *Compos Struct*, vol. 156, pp. 348–355, Nov. 2016, doi: 10.1016/j.compstruct.2015.09.032.

[6] C. I. Montaleza, J. W. Gallegos, M. E. Amaya, and O. Arteaga, "Design of an impact attenuator for passive safety on roads," in *Materials Today: Proceedings*, 2021, vol. 49, pp. 2073–2080. doi: 10.1016/j.matpr.2021.08.309.