Design and Analysis of Impact Attenuator: A Review

Naman Agrawal[1], Janmit Raj[2], Gaurav Saxena[3]
[1]Research Scholar, [2]Research Scholar, [3]Assistant Professor, Department of Automobile Engineering, Rustam Ji Institute of Technology, Tekanpur, Gwalior (M. P.)- 475005, India

Abstract

This paper focuses on review of a work carried out by researchers in the field of design and analysis of impact attenuator to understand the behaviour of crushing during vehicle impact. The geometric structure and the material chosen by researchers for impact attenuator is first briefly summarized followed by a description of the FEA analysis and Experimental test method applied. The review is divided into three categories experimental test, simulation test and comparison of experimental and simulation test. This review paper contains the work of past researchers published between 2000 and 2015.

Keywords*: Impact Attenuator, Drop Test, FEA.*

I. INTRODUCTION

Vehicle safety is one of the major research areas in automotive engineering. The car industry is developing new passive and active safety systems and techniques to increase the safety of vehicle occupants [10]. Increased interest on vehicle safety has led to a comprehensive research of the crash response of vehicle from several points of view namely, simulation and experimental test [6]. In order to ensure the driver's safety in case of high-speed crashes, special impact structures are designed to absorb the race car's kinetic energy and limit the decelerations acting on the human body [9]. An impact attenuator, which is also known as a crash cushion or crash attenuator, is a device that is used to reduce the damage done vehicle structure during collision of vehicle [12-13].

II. BRIEF DESCRIPTION TO IMPACT ATTENAUTOR

The impact attenuator is a crucial part of the vehicle and is the most important structure of the race car because it acts as a safety barrier between the driver and the impacted surface [15]. The impact attenuator is designed to absorb the vehicle kinetic energy in the form of even deformation and keeps the force level sufficiently low [2-3]. If the deformation is uneven then the driver might suffer injury by experiencing spikes in G's. The energy absorption normally takes place by extensive crushing and crumbling of the structure [1]. The impact attenuator provides a load path for transverse and vertical load in the event of off-centred and off-axis impacts [4]. The crashworthiness of impact attenuator is assessed for total energy absorbed as well as specific absorbed energy. The distinctive characteristic of impact attenuator is that the rate of energy dissipation is concentrated over a narrow zone, while the reminders of the structure experience a rigid body motion [7-8]. The schematic view impact attenuator is shown below in Fig. 1.

The impact attenuator is mounted on the bulk head of the chassis. The design of impact attenuators should be of lightweight materials, which may contribute to improving the acceleration performance and fuel economy of the vehicle [5], [17]. Impact attenuators are generally made up of aluminium, honeycomb, nomex, carbon composite, kevlar, aluminium foil or a combination of these materials as they provide maximum protection to the driver. The structure of impact attenuator is of various types viz. tube and plate type, honeycomb structured, truncated trapezoidal shaped and sandwich structured [14-15], [19].

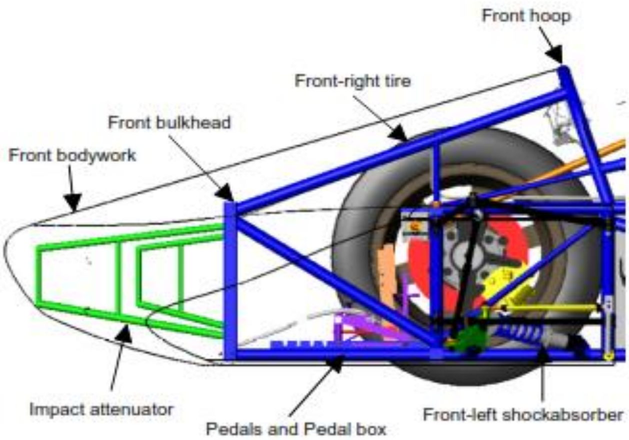


Fig.1: Schematic of Front Part of FSAE Vehicle.

III. PREVIOUS RESEARCHES

A. Previous Work Done on Experimental Test of Impact Attenuator

The dynamic test is done to determine the energy absorbing capacity of impact attenuator during crushing [3]. The most commonly used method for impact testing is drop test [1]. The drop test is done by dropping an object of known mass from a known height onto the full-scale impact attenuator while measuring the deformation of the specimen [4]. To test an attenuator dropping a weight from a calculated height is an easy and effective method [2]. So in that context Abrahamson Chad et al. [1] tested two specimens of impact attenuator designed according to SAE 2010 formula rules, by drop weight test method. The dimension of first specimen was (10 x 8 x 5 inches) while the dimension of second specimen was (10 x 10 x 4 inches). Both the specimens were made of Plascore PCGA-XR1-5.2-1/4-P-3003 aluminium honeycomb. The test was performed using 661 lb of impact mass. The impact mass was dropped from 8.3 ft. height with impact velocity of above 23ft/sec. The peak deceleration and average deceleration of first specimen was 46.88 G's and 17.86 G's, while of the same of the second specimen was 36.2 G's and 14 G's. The tested impact attenuator by them is shown in Fig. 2.

Fig. 2: Impact Attenuator Before and After Testing.

Further the team of researchers Kumar Devender et al. [2] selected the elliptical shaped impact attenuator made of Aluminium 6063 T6 material for performing experimental test. The simulation was done in finite element analysis software package LS-DYNA. While the experimental test was done by impact testing. The impact test was conducted considering face to face direct collision. The simulation result showed average deceleration was 18.8G on the other hand the average deceleration recorded from experimental drop testing was 13.15G. Both the average deceleration where under formula SAE rule of 20G. The impact attenuator designed by them is shown below in Fig. 3.

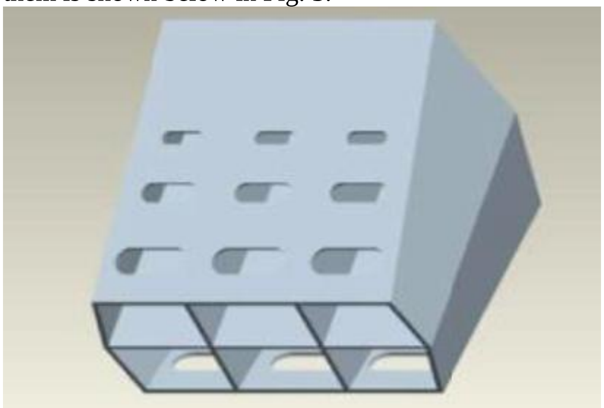


Fig. 3: Elliptical Shaped Impact Attenuator.

An exact year later Singhal Arpit and Subramanium S. Vignesh [3] in their work tested the plate and tube type impact attenuator by drop weight test analysis method. A two stage impact attenuator was built for testing. The impact attenuator was designed to absorb 7350 J of energy. The drop test was conducted from a drop height of 5 m with 150 kg of dropping mass. The result of analysis shown that the sheet metal shell was creating problems in deformation.

In the recent work Sengupta Akash et al. [4] analysed the truncated trapezoidal shaped impact attenuator by conducting compressive testing. The design of the impact attenuator was done in CAD modeling software CATIA V5R20. The designed impact attenuator was fabricated using aluminium sheets. The thickness of the aluminium sheet was 2 mm while the anti-intrusion plate was consists of solid steel of 1.5mm. The result shown that there was considerable deformation of impact attenuator. The midway crushing of impact attenuator in shown in Fig. 4.

Fig. 4: Midway Crushing of Impact Attenuator.

B. Previous Work Done on Finite Element Analysis of Impact Attenuator.

The development and testing costs of a new safety design can be reduced by the use of computational crash simulations [10-11]. The safety behaviour of a vehicle can be determined by simulating it in actual conditions [5], [7]. The FEA is useful tool for determining the crushing behaviour of impact attenuator [12-13]. Several researchers have conducted a variety of simulation test to investigate the behaviour of impact attenuator upon impact.

The researcher, Williams T. D. et al. [5] investigated the impact behavior of the frontal part of the Caterham 7 sports car. Detailed simulations were carried out using finite element vehicle model to replicate a rigid barrier test in finite element analysis code Oasys LS-DYNA3D. This provided an understanding of the overall impact event as well as individual component contributions. As a result, excellent correlation was achieved between test and simulation results.

Further, Zarei Hamidreza et al. [6], designed an optimum filled crash absorber design. This involved axial and oblique experimental crash tests of the tube. Correlation was carried out with LS-DYNA giving satisfactory outcome. Multi-design optimization (MDO) technique has been applied to maximize absorption of energy and specific energy absorption of square, rectangular and circular tubes. The table 1 below gives the impact on honeycomb filled aluminium tubes. On the other hand, Rising David et al. [7], analyzed the frontal impact of a Formula SAE vehicle. The aim of this exercise was to evaluate the risk of injury to the driver during frontal crash scenario. More so, the analysis emphasizes the importance of having a good headrest design to absorb the head's impact energy. It was also suggested that to minimize driver's injury, tubes should be securely mounted at a sensible distance from the leg of the driver.

Table I : Impact on Honeycomb Filled Aluminium Tubes.

Test No.	Impact Velocity[m^{-1}]	Impact angle [°]	First Peak load [kN]	Mean Crush Load [kN]
F1	7.49	0	80.2	41
F2	8.69	0	79.3	40
F3	9.75	0	78.1	39
F4	7.75	5	47.3	31
F5	5.1	10	48.6	31

In another study Enomoto Hiroshi et al. [8], tested the strength of three different impact attenuator a space frame structure with steel pipes (SSF), a monocoque structure with aluminum (AM) and a monocoque structure with Carbon Fiber Reinforced Plastic (CFRPM) by performing finite element analysis. From the comparison of results, the study revealed, that among the three impact attenuators CFRPM were absorbing higher energy and the crush length were the best in the three tested models. The force and displacement curve of one of CFRPM impact attenuator is shown below in Fig. 5.

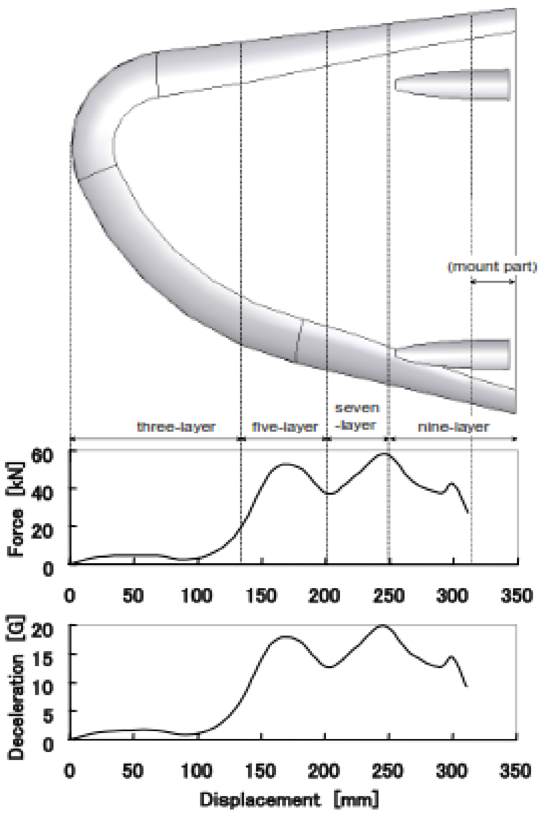


Fig. 5: Force and Displacement Curve of CFRPM Impact Attenuator.

Afterwards Heimbs S. et al. [9], investigated the crash behavior of the nose cone of F1 racing car impact structure. They developed finite element models for dynamic simulations with LSDYNA with emphasis placed on the composite material modeling. In terms of crushing mechanism, energy absorbed as well as deceleration levels, numerical results were compared to crash test data.

Later Belingardi Giovanni and Obradovic Jovan [10], in their work demonstrated the comparison of impact attenuator, as the independent structure and the complete assembly with car body. The complete geometric model of vehicle structure was built in 3-D modeling software CATIA according to design requirement of SAE 2008 rules. The simulation of the crash event was done in Radioss Code. The study revealed that both structure (Fig. 6) showed same crushing behaviour and have same energy absorption rate in the first half of the simulation while at the end of simulation the impact attenuator linked to the frame structure deformed more in comparison to independent structure. A year later, in the further work Belingardi G. et al. [11] showed that carbon fiber composites perform very well in crash. This research involved designing an impact attenuator for a Formula SAE car. The crash behaviour of the brittle composite material was studied using a simplified analytical model and finite element model. LS-DYNA was used to predict the crush pattern as well as stiffness of the Impact attenuator.

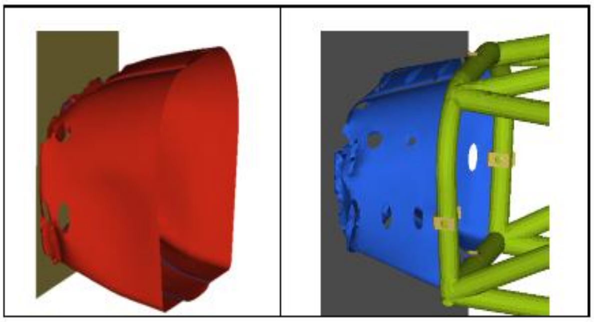


Fig. 6: Position of Two Structures During Impact at 14ms of Crash Event.

In another study Velea N. Marian et al [12], in their work presented the finite element analysis of ExpaAsym multi-layered cellular structured impact attenuator. During analysis the impact was made by a concentrated mass of 300 kg with the velocity of 7 m/s. The study concluded that the initial value of the impact force was close to the average load value. In the recent work Jain Mayank and Kalia Ved Aman [13], proposed the use cuboidal shaped impact attenuator on passenger vehicle. The geometric modeling of the impact attenuator was done in 3-D modeling software SolidWorks and the simulation is carried out in finite element software ANSYS. In analysis two materials namely, Galvanised Iron sheet (GI sheet) and Aluminium 2024 sheet with thickness of 1 mm were considered. The results obtained from the analysis shows maximum magnitude of peak acceleration, peak deceleration and velocity of impact is under acceptable limit and less than 20G. The Fig. 7 below shows the displacement vs time plot of impact attenuator analysed by them.

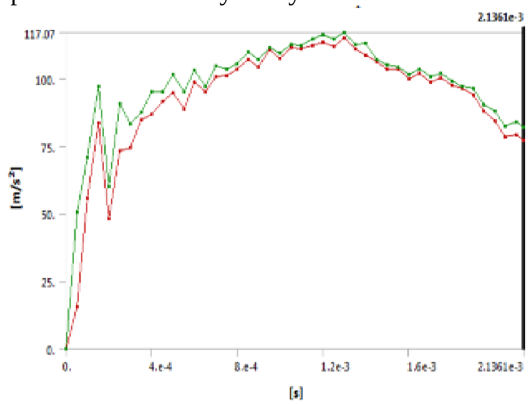
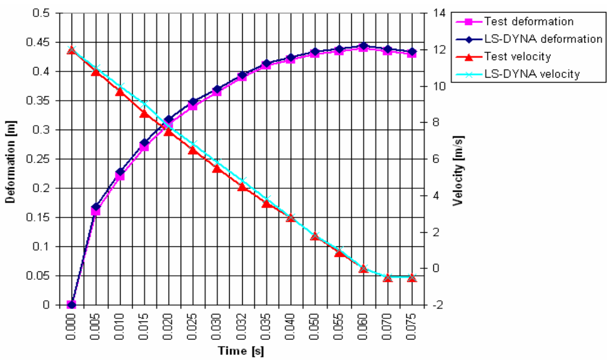


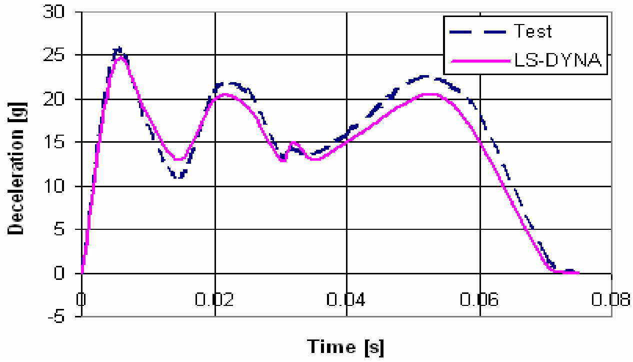
Fig. 7: Deceleration vs Time plot.

C. Previous Work Done on Simulation and Crash Test of Impact Attenuator.

The development time and cost of any product in real time is high. This can be reduced to a great extent by certain computer simulated software's [15], [19]. To reduce the development and testing costs of a new safety design, computational crash simulations for early evaluation of safety behaviour under vehicle impact test can be done [17], [20]. The simulated results however can't be used directly into real life without any validation with experimental results [18]. The simulation test gives an idea of how the impact attenuators would deform under collision [16], [21]. The comparative analysis confirms the quality of the methodology and approach used for the design of impact attenuator. In one of the major study Simonetta Boria et al. [14] Performed experimental and numerical test on thin-walled crash-box, made of aluminium sandwich material to investigate the energy absorbing capacity. The crash-tests were performed for a frontal impact at the velocity of 12 m/s; during the impact the load-shortening diagram, the deceleration and the energy absorbed by the structure were measured Based on the crash test result a finite element model is then developed using the non-linear, explicit dynamic code LS-DYNA simulating the condition of actual crash test. The result of both the test were compared (Fig. 8), showing the same trend.



(a)



(b)

Fig. 8: (A) Deformation and Velocity Versus Time and (B) Deceleration Versus Time.

The researcher Munusamy Raguraman and Barton C. David [15] carried out experimental and numerical simulations of the axial quasi-static and dynamic crushing of empty and honeycomb-filled tubes for impact attenuator. The Aluminium (6063 T6 alloy) tubes filled with Nomex_RHRH-10 honeycomb were employed in the research work. The finite element code RADIOSS was used for performing numerical simulation of the FE model of impact attenuator. The article reported that honeycomb-filled tubes were around 30% heavier absorbing 10% more energy than the empty tubes while the specific energy absorption (SEA) was found to be 15% less than empty tubes. Furthermore, the predicted failure modes are very comparable with the observed test behaviour for both empty and honeycomb-filled tubes under both quasi-static and dynamic axial loading. The crushing behaviour in quasi-static and simulation test is shown in Fig. 9. Later Babu Ajeet P.K. et al [16] examined the chassis and impact attenuator of a vehicle designed on the basis of SUPRA SAE INDIA 2011 rulebook. The impact attenuator used for testing was made of aluminium honeycomb structure. The author performed structural analysis in FEA software ANSYS for evaluating longitudinal acceleration, lateral acceleration, braking condition, braking with one circuit failure, bending stiffness and torsional stiffness. The design of impact attenuator was validated by conducting drop test. The result of test shown that the average deceleration is 11.55G which is under the limit design limit of SUPRA SAE of 20G.

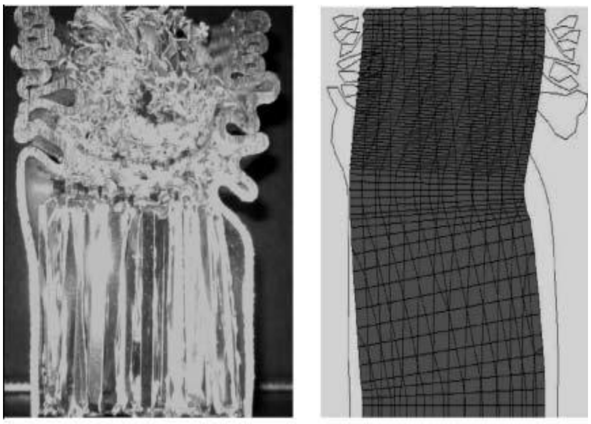


Fig. 9: Crushing Behaviour in Quasi-Static and Simulation Test.

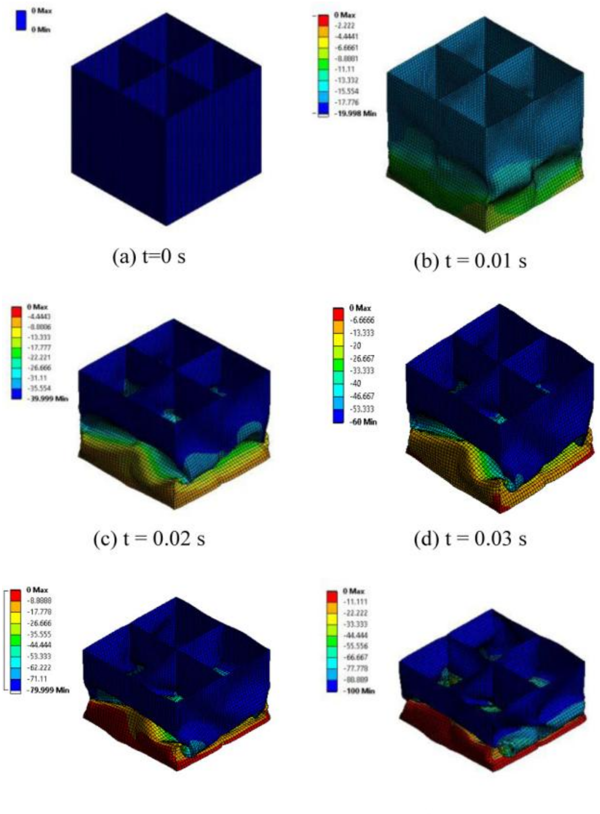
In the next work Boria Simonetta [17] determined the crush behaviour of CFRP sandwich structured impact attenuator of formula ford racing car. The experimental test was done by using weight drop test and performed FEA using the non-linear dynamic code LS-DYNA. For experimental test an impact mass of 413 kg with initial velocity of about 8.6 m/s made to collide with impact attenuator. While in simulation the boundary condition of real life was imposed on FE modal. The result of both the analysis were found identical.

In the further research work Sharavan Lalith C. et al. [18] compared three materials viz. Aluminium alloy foils 6082 T6, Balsa wood, Long Fiber Reinforced Thermoplastic Composite (LFRT) by performing FEA, Compression test and Drop test for impact attenuator. The designing of the impact attenuator was done according to SUPRA SAE 2014 rulebook. Sandwich construction along with honeycomb core is taken for analysis. The geometric modeling of impact attenuator was done in PRO-E and the FEA is performed in ANSYS software tool package imposing appropriate boundary condition. By comparing the results obtained from the three test, the researcher proposed the use of aluminium for impact attenuator, as aluminium is absorbing the maximum amount of energy. The team of researchers Boria S. et al. [19] analyzed a thin walled truncated square pyramid shaped composite impact attenuator for energy absorbed in a single crush cycle. The plain weave prepreg GG200 with a DT806R epoxy resin was used as composite material and the same is modeled as MAT54 element and analyzed in finite element software LS-DYNA. Furthermore, considering the crushing dynamics the researcher considered four different phenomena viz. fronds bending, axial splitting, crack propagation and friction between the annular wedge and the fronds and between the fronds and the platen of the press. The analysis revealed that the crushing force tends to decrease with increasing angle of inclination.

Table II: Comparison Between Analytical And Numerical Results in Terms of Mean Load and final Crushing.

Case	Geometrical parameters			P [kN]			δ [mm]		
	b_2 [mm]	θ [°]	t [mm]	An	Num	Error [%]	An	Num	Error [%]
1	200	0	1.5	24.3	22.6	7.5	65.9	70.9	7.1
2	200	0	2	64.8	59.7	8.5	24.7	26.8	7.8
3	200	0	2.5	117.6	110.3	6.6	13.6	14.5	6.2
4	165	5	1.5	16.2	14.8	9.4	98.7	108.3	8.9
5	165	5	2	25.1	23.3	7.7	63.6	68.6	7.3
6	165	5	2.5	41.6	39.1	6.4	38.5	40.9	5.9
7	130	10	1.5	13.6	12.7	7.1	118.0	125.8	6.2
8	130	10	2	23.7	21.4	9.7	67.4	74.7	9.7
9	130	10	2.5	34.0	31.6	7.6	47.0	50.7	7.3
10	93	15	1.5	11.9	11.5	3.5	133.4	138.8	3.9
11	93	15	2	20.1	18.3	9.8	79.4	87.5	9.3
12	93	15	2.5	28.4	26.5	7.2	56.3	60.3	6.6

In the further research work Rooppakhun Supakit et al. [20] performed simulation of thin wall type impact attenuator in FEA software ANSYS-LS DYNA code and further verified the results of FE results with experimental testing of the same. The analysis of thin-wall squared section of 1×1 simple cell, 2×2, and 3×3 multiple cells with various thickness as 1.2, 1.6 and 2 mm were done. The result of analysis show that the energy absorption capacity of simple section is lowered than the energy absorption capacity of multi cell sections and the increased thickness of wall contribute in the energy absorption capacity of the impact attenuator. The comparison of simulation and experiment results revealed that the maximum and mean force displayed were identical. The deformation of impact attenuator in each time step is shown in Fig. 10.



(a) t=0 s (b) t = 0.01 s (c) t = 0.02 s (d) t = 0.03 s (e) t = 0.04 s (f) t = 0.05 s

Fig. 10: Deformation Shape in Each Time Step.

In the more recent study Boria S. et al. [21] presented lightweight design of impact attenuator by conducting numerical and experimental analysis of a composite impact attenuator. The geometry of the analysed impact attenuator is similar to square frusta and manufactured by lamination of prepreg sheets in carbon fibres and epoxy matrix. The dynamic numerical analysis was done in explicit finite element code LS-DYNA and the experimental test was done by instrumented drop weight test machine. The finite element model consists of shell and solid elements. The result of the study shown that the composite material was able to absorb kinetic energy during impact by maintaining values near to 20 g as required by Formula SAE technical regulations, through a progressive brittle fragmentation. Furthermore, the force-displacement curves of solid modeling, shell modeling and experiment analysis were superimposed, with small differences in the force values and in the final crushing distance.

IV. CONCLUSION

This work has provided a comprehensive literature review of the previous research work carried out in past years on impact attenuator. An effort has been made to comprise all the important contributions and highlighting the most pertinent literature available for investigating the impact attenuator. The conclusion from the current literature survey are as follows: -

1. The review clearly indicate that impact attenuator reduces the damage caused to the vehicle structure and preventing driver from being injured during collision. The crushing of impact attenuator should absorb kinetic energy whilst transmitting a low load uniformly to the rest of the vehicle [14].

2. Through the review it is apparent that the most of the design based researches emphasis on weight reduction of impact attenuator. Weight of impact attenuator has considerable effect on the overall weight distribution of the vehicle [15]. Further, the material chosen for impact attenuator should have high energy absorbing capacity within an acceptance level of deformation [6].

3. The review of previous research on comparative analysis of experimental and simulation test of the Impact Attenuator shows the same trend of crushing characteristic of Impact Attenuator. Hence simulation based analysis are more economical and viable for research work [14], [19].

ACKNOWLEDGMENT

The review presented in this work is by no means complete but it gives a comprehensive representation of different methodology adopted for the testing of impact attenuator. The author wishes to apologize for the unintentional exclusions of missing references and would appreciate receiving comments and pointers to other relevant literature for a future update.

REFERENCES

[1] Abrahamson Chad, Bruns Bill, Hammond Joseph & Lutter Josh (2010). "Formula SAE Impact Attenuator Testing," pp. 35-46.

[2] Kumar Devender and Khanna Naman, "Drop Test Analysis of Impact Attenuator for Formula SAE Car," International Journal of Scientific and Research Publications, Vol. 02 (10), pp. 01-04, 2012.

[3] Singhal Arpit and Subramanium S. Vignesh, "Cost Effective & Innovative Impact Attenuator for Formula SAE Car with Drop Test Analysis," International Journal of Scientific and Research Publications, Vol. 03 (03), pp. 01-04, 2013.

[4] Sengupta Akash, Sharma Apurba Sharma and Chakraborty Amritabha, "Manufacturing an Impact Attenuator for Formula Prototype Cars," IPASJ International Journal of Mechanical Engineering, Vol. 02 (11), pp. 11-13, 2014.

[5] T. D. Williams, A. Pennington and D. C. Barton, "The frontal impact response of a space frame chassis sports car," Proceedings of the Institution of Mechanical Engineers, Journal of Automobile Engineering UK: University of Leeds, Vol. 214, pp. 865-873, 2000.

[6] H. Zarei and M. Kroger, "Optimum honeycomb filled crash absorber design," Germany: University of Hannover, Materials & Design, pp. 193-204, 2006.

[7] D. Rising, J. Kane, N. Vernon, J. Adkins, C. Hoff and J. Brelin-Fornari, "Analysis of a Frontal Impact of a Formula SAE Vehicle," USA: Kettrering University, 2006.

[8] E. Hiroshi, M Yusuke, M Hiroshi, H. Eiji, K Satoshi, Y Koetsu and U Naoki, "Development of CFRP Monocoque Front Impact Attenuator for FSAE with VaRTM," SAE Technical Paper Kanazawa, University, KADO Corporation, pp. 01-08, 2007.

[9] Heimbs S., F. Strobl, P. Middendorf, S. Gardener, B. Eddington and J. Key, "Crash Simulation of an F1 Racing Car Front Impact Structure," UK: A proceeding of 7th European LS-DYNA Conference, 2009.

[10] Belingardi Giovanni and Obradovic Jovan, "Design of the Impact Attenuator for a Formula Student Racing Car," Journal of the Serbian Society for Computational Mechanics, Vol. 4 (01), pp. 52-65, 2010.

[11] G. Belingardi, S. Boria and J. Obradovic, "Lightweight design and crash analysis of composite frontal impact energy absorbing structures," 16th International Conference on Composite Structures, 2011.

[12] Velea N. Marian and Lache Simona, "NOVEL IMPACT ATTENUATOR" 5Th International Conference Computational Mechanics and Virtual Engineering, COMEC 2013, Brasov, Romania, pp. 47-51, 2013.

[13] Jain Mayank and Ved Kalia Aman, "Study and Design of Impact Attenuator for Passenger Vehicle" International Journal of Scientific and Research Publications, Volume 4, Issue 11, pp. 01-04, 2014.

[14] Simonetta Boria and Giuseppe Forasassi, "Crash Anaysis of an Impact Attenuator for Racing Car in Sandwich Material," Department of Mechanical, University of Pisa, Italy, 2008.

[15] M. Raguraman and B. C. David, "Lightweight impact crash attenuators for a small Formula SAE race car," International Journal of Crashworthiness, pp. 223-234, 2010.

[16] Babu Ajeet P.K., Saraf M.R. and Vora K.C., "Design, Analysis and Testing of the Primary Structure of a Race Car for Supra SAEINDIA Competition," SAE International, 2012.

[17] Boria Simonetta, "Design Solutions to Improve CFRP Crash-Box Impact Efficiency for Racing Applications," Advanced Composite Materials for Automotive Applications, pp. 205-226, 2013.

[18] Sharavan Lalith C., Immanuel Joshuaraj K., Varun A.K., C. Sudir and Krishna S., "Design and Experimental Analysis of an Impact Attenuator," Proceedings of 12th IRF International Conference, Chennai, India, pp. 62-66, 2014.

[19] Boria S., Pettinari S., Giannoni F. and Cosimi G., "Analytical and numerical analysis of composite impact attenuators," Science Direct at Composite Structures, pp. 01-08, 2015.

[20] Rooppakhun Supakit, Boonporm Pornporm and Puangcha-um Worawat, "Design and Analysis of Impact Attenuator for Student Formula," SAE Technical Paper, 2015.

[21] Boria S., Obradovic J. and Belingardi G., "Experimental and numerical investigations of the impact behaviour of composite frontal crash structures," Composites Part B, pp. 01-12, 2015.