Design and Analysis of Automobile Front Bracket for Impact Strength

[1]Ms. Neha K. Attarde, [2]Dr. S.P. Mogal

[1]P.G Student, [2]Head of Mechanical Department
[1]Mechanical Design Engineering
[1]NDMVPS'S KBT COE. Nashik. Maharashtra. India.

***Abstract-* The Automobile market in India has been boom sector for over decades of year. The Automobile industry in India has tremendously growth due to demand of the products from the lower models to the highly luxurious models. Hence forth in this competitive sector, the industries are marketing their product by enhancing several new safety measures and luxuries. From all of them, front impact safety is one of the crucial Topic. In this Paper, the automobile front bracket is being optimize for material replacement so as to increase the impact strength and to reduce the weight. The Front Bracket is made of Heavy Structural Steel whose weight is comparatively high. In this project, we can replace the front chassis of automobile with the material which provides better or similar impact strength but can effectively reduce the weight of vehicle. The Structural Steel is replaced by the Polymer matrix composites of E-Glass and S-glass Fibers. The Simulation of Ansys is used to determine the feasibility of research. The polymer material is finalized for experimental validation. The experimental test is carried out for both Charpy Izod impact test.**

***Key words*: Automobile Bracket, Composite Material, FEA analysis, Impact and Drop Test.**

I. INTRODUCTION

A chassis is made of solid structural steel of heavy weight and impactful strength so as to withstand the impacts and body and passenger weight. Hence in this project an attempt is made to design the front chassis of an automobile so as to reduce the weight of chassis and thereby reducing the weight of vehicle. Reducing the weight of vehicle directly impacts the economy of vehicle. Reducing the weight also impacts that the strength should be more as that of solid material. In this case, the solid structural steel is replaced by light weight-high strength composite material. The material is especially composed so as to reduce the weight of previous material and also to increase the strength of material. In the Current study, the front bracket of automobile is used as case study for experimentation and simulation. Front bracket of chassis holds the engine weight, and other parts of the vehicle, also sustain the front collision.

II. PROBLEM STATEMENT

"To Design and Optimize the front bracket of automobile chassis to withstand the impact force and replacing it by using the composite material"

The main aim of this work to redesign the front section of the chassis so as to verify the impact behavior of the chassis made of steel as well as the chassis made of composite materials. The design parameter in the process is the impact strength of material. The purpose of this work is to evaluate the possibility and feasibility of redesigning a metal chassis part in composite materials. The part is to be chosen from a variety of existing parts, and the choice is to be motivated and based on the findings of the composite study. Since the choice of component is limited to chassis part.

The Objectives of this study are,
- Design of Bracket using Optimized Polymer Materiial
- Determining and comparing the impact analysis of steel and Polymer using simulation.
- Experimental Validation of the results that are obtained via simulation.

III. LITERATURE REVIEW

The Backbone Chassis of Tata Vehicle is used. The weight of chassis is optimized by changing the material of the chassis. In this research the optimization is done for implementation of heavy weight vehicle chassis. [1]. The Materials of carbon epoxy and glass carbon epoxy is used to design the front rail of car sub frame. In this project, the conventional material used for front sub frame rails in car, steel is replaced with the composite materials Carbon Epoxy and Glass Carbon. 3D model of the sub frame rail is done in Creo 2.0. Impact analysis is done in Solid works for all the materials to compare the displacements and stresses at different speeds 80km/hr, 100km/hr and 120km/hr. [2]. The author explains the design of automobile chasis using the material carbon fiber and glass carbon for car. Model is prepared in the creo software and FEA is done so as to prove the results in simulation. Through the analysis oflightweight materials, the carbon fiber composite is selected as the material of the bumper beam instead of steel in order to achieve the lightweight design. [3]. The Frontal Collision of car impact is studied. The Computational Dynamics analysis is done to [4]. The Author introduces the material of thermoplastics. The Car bumper which is made of steel is studies and simulated in the software. The material is replaced with the thermoplastic polymers. [5]. In this Study, Design and Impact Analysis on front Bumper Beam Crash box for a sedan car using glass fiber reinforced polymer is done. The Material of bumper used her is ANSI Simple grade steel. This material is replaced using Glass and Carbon Fiber [6]. The car front beam is selected for the case study. The model of beam is prepared in creo software. Initially static analysis was done on the chassis using different materials, and then the modal analysis was done to obtain the mode shapes of the chassis. This analysis is done using three materials which is steel, carbon epoxy and e-glass [7]. Crash simulation in ANSYS ls-dyna to explore the crash performance of composite and metallic materials. The crash simulation of a full frontal impact of the model at a velocity of 35 mph with a rigid wall is carried out and analyzed. The analysis is done for steel, aluminum, E Glass and Carbon Fiber [8]. The Impact Behavior of Materials is determined by various testing devices. In many cases, using composites is more efficient. The research is done for replacement of steel with composites. This is possible by replacing conventional metal alloys with composite materials. Even if the composite material costs may be higher, the reduction in the number of parts in an assembly and the savings in fuel costs make them more profitable. [9] This paper shows the Experimental Investigation of Fiber Reinforced Composite Materials under Impact Load. In this paper, glass fiber reinforced composite material specimen is fabricated and further experimentally tested under impact and tensile loading conditions. [10].

IV.SELECTION OF COMPONENT

The chassis selected is of Mahindra Jeep 2012 which is a mini Suv type of five seated vehicle.

Material	:	**Structural Steel**
Application	:	Mahindra MM540 Jeep
Type	:	Passenger Vehicle
Kerb Weight	:	1330 kg.
Max. Weight	:	1770 kg.
Type of Chassis	:	Ladder Type

Table 1

MATERIAL PROPERTIES OF SELECTED COMPONENT

Material and Properties	Structural Steel
Ultimate Tensile Strength (MPa)	1015
Ultimate Yield Strength (MPa)	979
Young's Modulus (MPa)	$2e^5$
Poisson's Ratio	0.29
Density (kg/m^3)	7850

V.APPROACH METHOLOGY

Composite Material:

Composite materials are flexible and complicated shapes are easily generated. Composites are light in weight and can be used as replacement for steel due to high stiffness and strength ratio. Composites are of three types, polymer matrix, ceramic matrix and metal matrix. The concept used in this study is polymer matrix composites. The mostly used polymers with help of which the bracket is designed using S-Glass, E-Glass Epoxy, Carbon fiber and Kevlar.

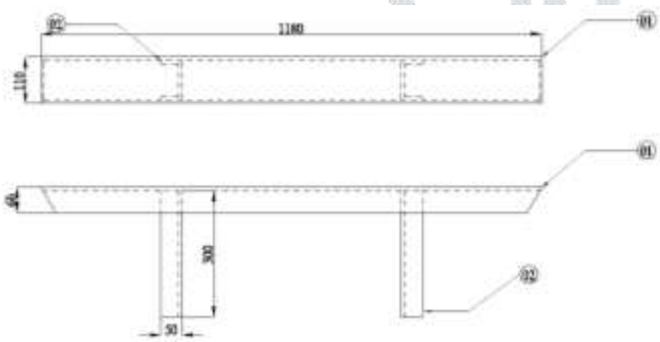
In the current Study, E-glass fibre polymer is used as as approach of material of replacement in place of steel. The material is durable and possess good strength to weight ratio. The Properties of material as listed below.

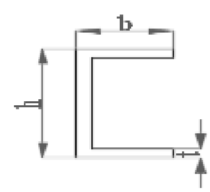
Table 2
PROPERTIES OF POLYMER MATRIX COMPOSITES

Material and Properties	E-Glass Fiber	S-Glass
Ultimate Tensile Strength (MPa)	767	4587
Ultimate Yield Strength (MPa)	300	3250
Young's Modulus (MPa)	3.69e5	$8.69e^5$
Poisson's Ratio	0.3	0.28
Density (kg/m^3)	1900	2480
Energy Absorption (J/Kg)	53	49

VI. MODELLING AND SIMULATION

The chassis front section is modelled in Creo 2.0 software. The terminology of chasis is as shown below with all the dimensions.





	h	b	t
Channel 01	110	50	8
Channel 02	94	50	8

Figure 1: Terminology of Chassis

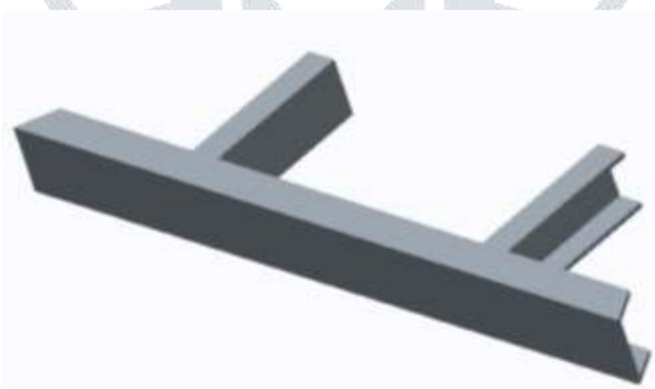


Figure 2: Simulation View of Front Section of Chassis

VII. FINITE ELEMENT ANALYSIS

The Finite element analysis is carried out using Ansys 18.1 element type Software. The Model is subjected to different impact loads which subjected on front section. The FEA is done under Von misses Criteria for Maximum stress distribution and maximum strain energy for Steel as well as composites of E-glass fibre.

a). Pre-Analysis Setting :

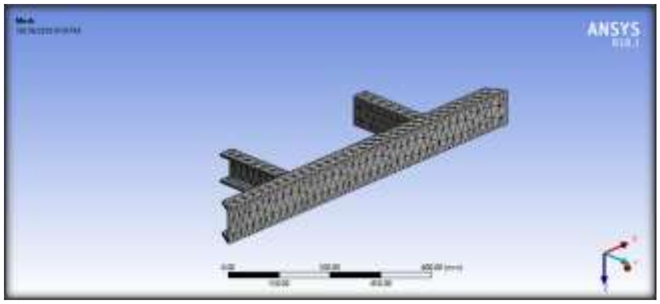


Figure 3 Meshing of Bracket

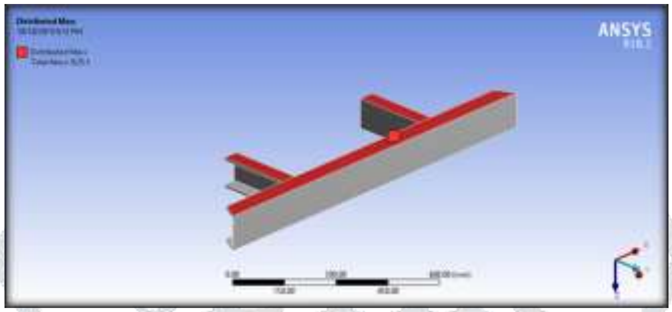


Figure 4 Mass Suppressed upon the Bracket.

The mass acting on the Front bracket is the kerb weight of the engine side of vehicle which is approximately equal to 250 kg. Hence selecting the acting mass on the front side as 250Kg

.

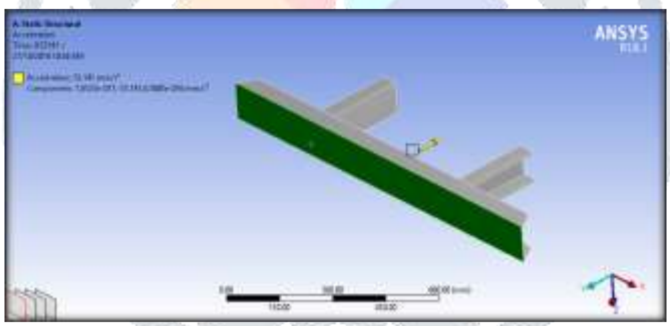


Figure 5 Forces impacting with acceleration

The Body is subjected to Force along with the acceleration acting on the Front side of the bracket as shown in the figure above. This acceleration makes it as the impactful force acting on the bracket.

b). Parameters of Analysis :

Consider the body hitting the beam with different acceleration for time limit travelling at rate for 60 second at a distance from 15m. Hence, from Equation, $\boldsymbol{v^2-u^2=2as}$, the acceleration for different impact can be determined which can be substituted in FEA. Here, V stands for velocity in m/s, a stands for the acceleration in mm/s^2, and s signifies the distance of impact in m.

Table 3
LEVEL AND PARAMETERS OF IMPACT FORCE

Level	Force (KN)	Speed (Kmph)	Velocity (m/s)	Acceleration (m/s^2)
1	0.5	50	13.885	6.426441
2		60	16.662	9.254075
3		70	19.439	12.59582
4		80	22.216	16.45169
5		90	24.993	20.82167
6		100	27.77	25.70576

c). *Maximum FEA Results for Structural Steel :*

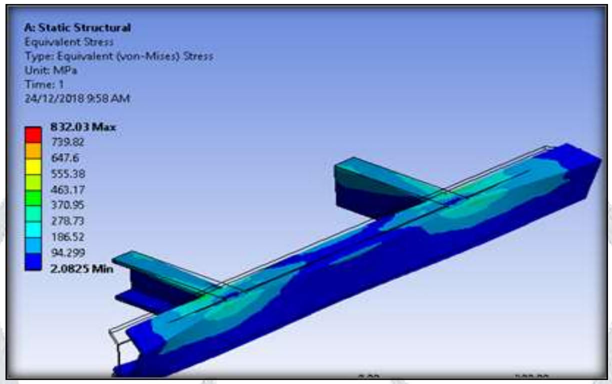


Figure 6 Maximum Stress Produced in Steel Panel for Level 6 Impact

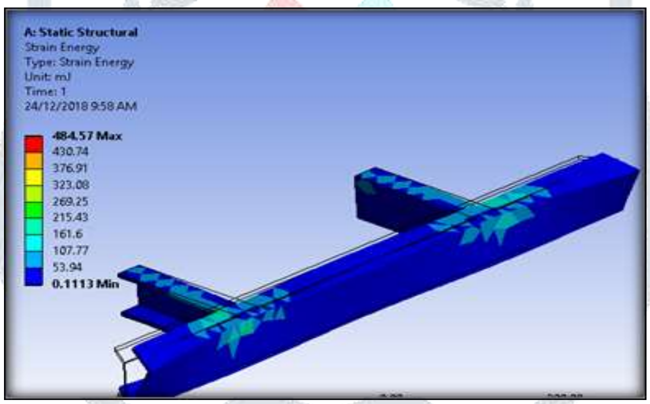


Figure 7 Maximum Strain Energy in Steel Panel for Level 6 Impact

d). *Maximum FEA Results for E-glass Fiber :*

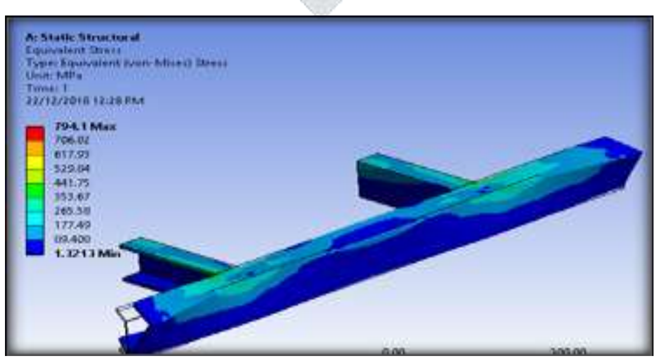


Figure 8 Maximum Stress Produced in E-Glass for Level 6 Impact

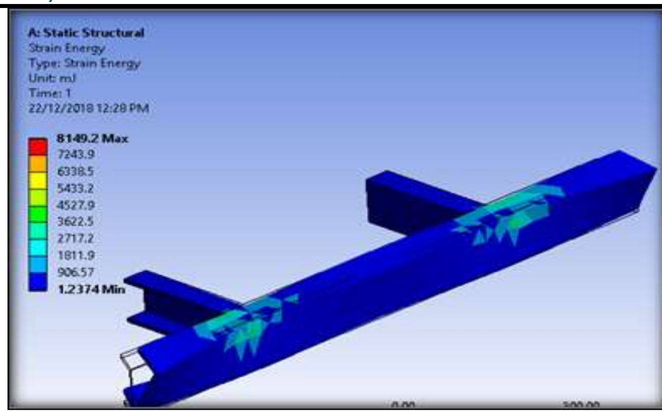


Figure 9 Maximum Strain Energy in E-Glass Panel for Level 6 Impact

e). Maximum FEA Results for S-glass Fiber :

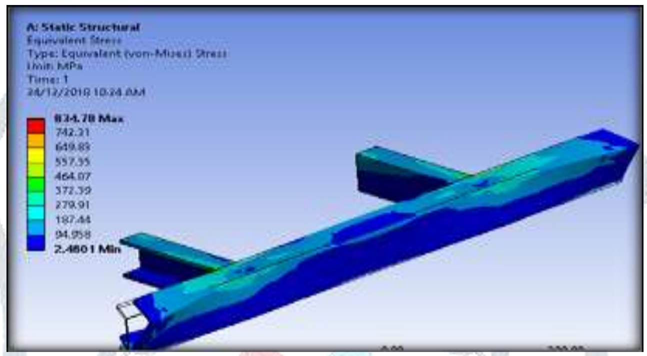


Figure 10 Maximum Stress Produced in S-Glass for Level 6 Impact

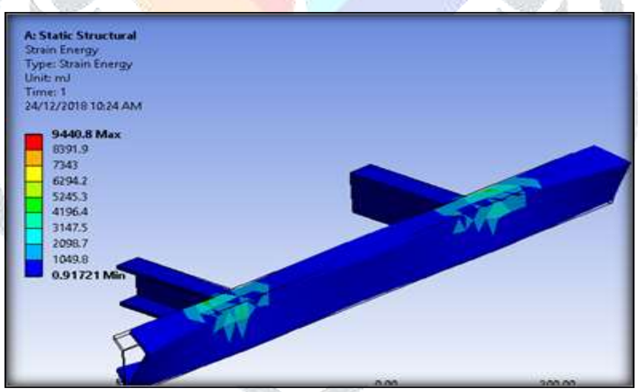


Figure 10 Maximum Strain Energy Produced in S-Glass for Level 6 Impact

f). Response Analysis for Different Impact Conditions :

Table 4
RESPONSE ANALYSIS OF STRESS DISTRIBUTION IN N/mm^2

Level	Struct. Steel	E-Glass Fibre	S-Glass Fibre
1	170.99	162.24	170.11
2	267.92	254.59	267.57
3	382.5	364.41	382.78
4	514.71	490.78	575.72
5	664.56	634.02	666.39
6	832.03	794.1	834.78

Table 5
RESPONSE ANALYSIS OF STRAIN ENERGY IN MJ

Level	Struct. Steel	E-Glass Fibre	S-Glass Fibre
1	20.85	345.58	397.07
2	50.75	847.93	977.07
3	102.98	1725.3	1993
4	185.98	3121.5	3610.8
5	309.5	5200	6021.2
6	484.57	8149.2	9440.8

Table 6
RESULTS FOR TOTAL DEFORMATION IN MM

Level	Struct. Steel	E-Glass Fiber	S-glass Fiber
1	0.54	1.01	1.06
2	0.85	1.25	1.62
3	1.225	1.94	2.23
4	1.6568	2.63	3.04
5	2.1455	2.65	3.95
6	2.6917	3.24	4.68

g). Material Finalization:

From Table 4 and 5, it can be observed that all the results for panel of composite can be used as replacement of panel made of steel. Hence considering the suitability for stress and strain energy produced, composite material can be used.

VIII. EXPERIMENTAL INVESTIGATION

The Experimentation Setup is done by preparing the setup for Drop test. The drop setup is prepared to conduct impact test from a weight falling on work piece at a height of building.

Part 01: Drop Test

a). Schematic Setup:

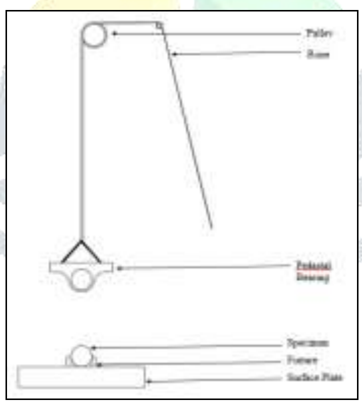


Figure 11 Schematic View of Experimental Setup

b). Requirement of Setup:

Figure 12 Component of Setup

c). Size of Specimen:

d = 40mm
l = 80 mm

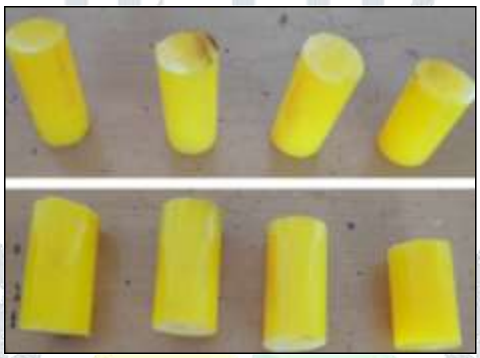


Figure 13 Sample Specimen for Four Different Velocities

d). Parameters for Experimental Calculations :

Some of Assumptions made during test are,

- The weight of Bearing is 5kg.
- The Weight is hanged up by Nylon rope.
- The Height of Building is 22 Meters.
- The falling velocity of weight is calculated by Eqn,
  $V = ((mgh)/t)^{1/2}$

V = Velocity in m/s
m = mass of object falling = 5Kg
g = Gravitational Force 9.81
t = Time required for falling of object.

e). Measurement of Deformation:

Figure 14 Impact of Weight on Specimen

Figure 15 Dial Indicator for Deformation

Table 7
EXPERIMENTAL RESULTS FOR DEFLECTION (mm)

Height h (m)	Time T (sec)	Velocity (m/s)	Deformation (mm) (approx.)
8	2.1	13.66957	0.3
10	2.42	13.86914	1.53
12	2.5	15.34405	1.83
14	2.6	16.25163	2.56
16	2.75	19.39325	2.82
18	2.95	19.45214	2.89

Part 02: Izod Test:

a). Schematic Setup:

Figure 16 Izod Impact Test Setup

Table 8
EXPERIMENTAL RESULTS FOR DEFLECTION (mm)

Drop Measure (J)	Velocity (m/s)	Deflection (mm) (approx.)
100	13	1.05
200	15	1.89
300	17	2.44
400	19	2.89

IX. RESULTS AND DISCUSSION

- From Simulation. It can be seen that the maximum stress, and the deformation are slightly greater than to that of Structural Steel.
- But, as the difference is not so more, so can be tolerated. With respect to the amount of Strain energy absorbed, the composites are better that steel.
- Hence absorption of energy leads to absorption of shocks to the materials and the humans in the vehicle,

Table 9
Comparison of simualtion for composite and steel

Material	Max. Stress (MPa)	Strain Energy (Mj)	Mass (Kg)
E-Glass	795	8149	5.39
Struct. Steel	833	485	22.27

- With the help of Simulation, we can find out that, the stress and deformation are within the permissible limit of transformation.
- On the other hand, the weight of panel can be reduced up to 60% by implementation of Composite materials.

Table 10
Experimental validation for deformation in mm

Velocity (m/s)	Simulation	Exp.	Difference	Average
13	1.01	1.53	0.4 %	0.6 %
16	1.25	2.56	0.5 %	
19	1.94	2.89	0.89 %	

- The percentage difference in the calculations, are about 0.6% which is next to the acceptable limit.
- Hence we can conclude that, the simulation are results are beneficial without going forward for experimentation and saving the cost of experimentation.

X.CONCLUSIONS

In this paper FEA of all the materials are carried, including the composites and steel, from which it can conclude that E-Glass Fiber gives 60% better results than steel with respect to maximum displacement and maximum stress distribution. Due to the strengths and stiffness of composite material, the results obtain are in favor E-Glass fiber.

Use of E-Glass fiber is also optimal as it reduced the cumulative weight of spring up to 60%. As the weight of one pannel is by 60%, hence we can say that, there will enough decreases in weight of vehicle. This reduction in weight of vehicle can lead to economic performance of vehicle in terms of power and fuel consumption.

Hence, in this way we can optimize the material and shape of chassis of automobile by replacing it with polymer matrix composite.

ACKNOWLEDGMENT

The report is outcome of guidance, moral support and devotion bestowed on me throughout my work. For this I acknowledge and express my profound sense of gratitude and thanks to everybody who have been a source of inspiration during the experimentation. First and foremost I offer my sincere phrases of thanks with innate humility to **Dr. S. P. Mogal** (H.O.D) Mechanical Engineering Department, for providing help whenever needed.

The consistent guidance and support provided by **Dr. S.P. Mogal** is very thankfully acknowledged and appreciated for the key role played by him in providing me with his precious ideas, suggestions, help and moral support that enabled me in shaping the experimental work.

REFERENCES

[1] M. Ravi Chandra, "Modelling and Structural analysis of heavy vehicle chassis made of polymeric composite material by three different cross sections", International Journal of Modern Engineering Research (IJMER), Vol.2, Issue.4, pp-2594-2600, July-Aug. 2012.

[2] Guguloth Ravi, "Design and evaluation of composite carfront subframe rails and crash injury response by carbon epoxy and glass carbon", Volume 3 Issue 10, March 2017.

[3] Tie Wang and Yonggang Li, "Design and analysis of automotive carbon fiber composite bumper beam based on finite element analysis", Advances in Mechanical Engineering Vol. 7(6) 1–12, 2015.

[4] Diogo Montalvão and Magnus Moore, "Computational Analysis of a Car Chassis Frame under a Frontal Collision, International Journal of Engineering Research & Technology (IJERT) Vol. 3 Issue 2, February – 2014.

[5] S Jeyanthi and J Janci Rani, "High velocity Impact Analysis of the Thermoplastic Bumpers in Automobiles", Journal of Scientific & Industrial Research Vol. 73, pp. 66-68, January 2014.

[6] Pooja Sreerama1 et.al. "Design and Impact Analysis on front Bumper beam Crash box for a sedan car using glass fiber reinforced polymer", International Journal of Computational Science, Mathematics and Engineering Volume-3-Issue-11-November-2016.

[7] Md Azhar Kazmi, "Modeling and analysis of automobile chassis using composite materials", International Research Journal of Engineering and Technology (IRJET), Volume: 04 Issue: 08, Aug -2017.

[8] Tony Punnoose Valayil, et.al. "Crash simulation in ansys ls-dyna to explore the crash performance of composite and metallic materials", International Journal of Scientific & Engineering Research, Volume 4, Issue 8, August 2013.

[9] Chircor Mihael et.al. "The Impact Behaviour of Composite Materials", Proceedings of the 3rd International Conference on Maritime and Naval Science and Engineering.

[10] Sravani Koppula, "Experimental Investigation of Fibre Reinforced Composite Materials under Impact Load", IOP Conf. Series: Materials Science and Engineering, 2018.