International Journal of Advanced Trends in Engineering and Technology (IJATET)
Impact Factor: 5.665, ISSN (Online): 2456 - 4664
(www.dvpublication.com) Volume 2, Issue 2, 2017

CRASH ANALYSIS OF LIGHT MOTOR VEHICLE BUMPER SYSTEM

Alok Kumar Garje* & Ravikant B Mulage**
Assistant Professor, Mechanical Department, B V Raju Institute of Technology, Narsapur, Telangana

Cite This Article: Alok Kumar Garje & Ravikant B Mulage, "Crash Analysis of Light Motor Vehicle Bumper System", International Journal of Advanced Trends in Engineering and Technology, Volume 2, Issue 2, Page Number 238-242, 2017.

Abstract:

Automotive industry is a very huge domain; enormous amount of research is going on in this domain. Crash analysis is also a part of the research done in this domain. Bumper system of a car is very important system in the automotive industry, many research are going on the bumper system in order to ensure the safety of the passengers in the car. In a collision, the bumper system will be the first system which makes contact with impact or either in front collision or rear collision condition.. This paper presents the study of the crashworthiness of the bumper system made up of various materials with various cross sections. The various materials considered are Aluminium, Carbon fiber reinforced with Poly Etherimide (PEI) and Carbon fiber reinforced with Acrylontrile Butadine Styren (ABS). The various Cross-section of the Bumper Beam are C Section, Hat Section and Double Hat Section. The Impact Conditions are as per the "Federal Motor Vehicle Safety Standards (FMVSS)". The stress distribution, deflection and energy absorbed by the Bumper Beam for various materials is compared along with the cross-sections Finite element method is used to perform the crash analysis. In this study existing Bumper Beam is implemented and it is meshed with Hypermesh 12.0 software as preprocessor and commercial finite element package ABAQUS 6.13 is used for performing crash analysis. The linear Quadrilateral and linear Triangular Shell Elements was used in the meshing process.

Key Words: Finite Element Method, Crash Analysis, Bumper System & Crashworthiness.

1. Introduction:

In recent few years vehicles have changed substantially based on design and other various practical elements. In modern days constantly developing nations requires light weight motor vehicles with increased efficiency. In order to meet these requirements vehicle manufacturers are using structures made up of composite fibers to reduce the weight of the vehicle. Although these vehicles provide lighter weight, protection of the passengers must be ensured. All automobile vehicles have Bumper System, which has to absorb not only the impact forces generated during a collision but also has to provide the safety of the passengers. The Bumper System is an assembly of Plastic Fascia, Energy Absorber and Bumper Beam. The Bumper System can be considered as a structural component. Some of functions of the Bumper System are:

- ✓ To absorb the impact forces generated during a collision of two vehicles.
- ✓ To protect all the component of the vehicle against sudden impact collision.
- ✓ To protect the passengers from injuries caused due impact.

By solving the equation of motion, the dynamic response of the simple structures like beam, bars, cylindrical shells etc.., Can be obtained. Since the Bumper System is assemblage of many components, it is almost impossible to obtain analytical solution to the equation of motion. To overcome this difficulty, finite element method is used.

Automotive industry is a very huge domain; enormous amount of research is going on in this domain. Crash Analysis is a very important part of the research done in this domain. Here an attempt is made to study the crashworthiness of the existing Bumper System. Whenever a collision between two vehicles occurs, Bumper system will be the first component which will receive the impact forces. Hence the Bumper System should be capable to absorb more impact forces and transfer fewer amounts of impact forces to the other components and to ensure the safety of the passengers travelling in the vehicle. This study deals with the amount of energy absorbed by the Bumper System.

The purpose of obtaining the crashworthiness of the Bumper System is to determine the amount of energy absorbed by the material used in manufacturing. This paper deals with a Bumper Beam that is already made up of steel and the energy absorbed by the bumper beam with suggested materials along with the cross-section is compared. Linear Quadrilateral and Linear Triangular Shell Elements was used in the meshing process because some of the critical point or area in the geometry needs to have a small meshing size in order to give an accurate model for the 3D-elements.

Many researchers carried out study on Bumper System, Hosseinzadeh RM and Marzbanrad JM et al. [1] has made an attempt to analyse the structure, shape and impact condition of a automotive bumper system made up of glass mat thermoplastic (GMT) using the FEA software LS-DYNA. The simulations were performed according to the standards E.C.E United Nations Agreement (Regulation no 42, 1994). The results obtained for the GMT Material is than compared with the conventionally used materials such as steel and aluminium. The GMT material has evolved as more impact energy absorber than the steel and aluminium, but it failed due to the use of dense material which has increased the weight of the bumper system and some manufacturing difficulties were seen. Evans D and Morgan T [2] have suggested that, as automobile manufacturing companies are keen on improving the design of the existing bumper system by giving a new style, it has to discover fresh solutions which should reduce the cost requirement and ensure the safety of the passengers. Hence they have suggested the use of Expanded Polypropylene (EPP) foam techniques. Butler M et.al [3] stated that it is necessary to increase the impact energy absorbing capacity of the light weight vehicles by using various new techniques along with the materials of the components which are associated with the energy absorbing unit. The crashworthiness of these components is mainly associated with the combination of material properties and the geometry. The material properties for the chosen material should have high yield strength. These increasing demands have led to growing interest in utilization of high strength stainless steels. Witteman WJ [4], automotive manufacturing companies are trying to reduce the overhang, greater sweeps and building of compact bumper system. In this paper, manufacturers trends linked with bumper energy absorbers is reviewed and a potentially in shape energy absorber is replaced with the existing model with reduction in the size. The energy absorber made up of Electron-Transfer Polymer (ETP) is replaced with Expanded Polypropylene (EPP) foam. The simulation made for the EPP energy absorber was according to the Federal Motor Vehicle Safety Standard Part No. 581. The simulation of EPP energy absorber results were compared with actually experimental data for ETP. This paper deals with a Bumper Beam that collides with a rigid wall according to the Federal Motor Vehicle Safety Standard (FMVSS). Shell element has been used for analysis. This element has better and more disciplined meshing in comparison with other elements and has the capability of gaining more accurate results with the same meshing containing the related 3-dimensional elements.

2. Bumper Beam:

In this article, a existing Bumper Beam has been studied. This Bumper Beam is made up of steel and it's cross-section is C type. In the assembly of the Bumper System, a Plastic Fascia and a Energy Absorber made up of honeycomb structure can be joined by the use of bolts and rivets to the Bumper Beam. The Fig1 shows the CAD model of Bumper Beam.

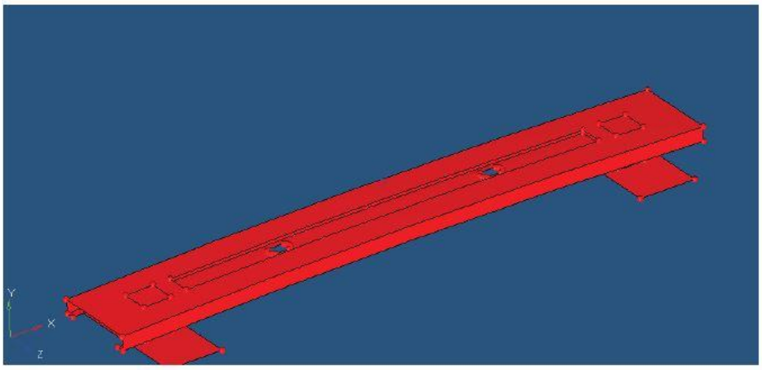

Figure 1: View of CAD model of Bumper Beam.

The material properties used for Bumper Beam are as mentioned in Table1.

Table 1: Material properties for Bumper Beam

Materials	Young's Modulus E GPa	Yield strength σ_Y MPa	Poisson Ratio	Density (Kg/m^3)
Steel	207	700	.30	7860
Aluminium	70	480	.33	2710
CFRwith PEI	31	230	.44	1480
CFR with ABS	31	200	.39	1050

3. Finite Element Analysis:

3.1 Basic Concept of FEM: The finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It is also referred to as finite element analysis (FEA). FEM subdivides a large problem into smaller, simpler, parts, called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. FEM then uses variational methods from the calculus of variations to approximate a solution by minimizing an associated error function.

3.2 A General Procedure for FEA: There are three main steps, namely: preprocessing, solution and post processing. In preprocessing the given problem is divided in finite numbers of nodes and elements. Basically, FEM splits the given domain into number of small finite elements and then each finite element is individually solved at a time. Shape function is assumed to represent the physical characteristic of an element. To obtain solution of an element, an approximate constant function is considered. Shape function is responsible determination of field variable for each element, relating to the nodes at the boundary of each elements. Then matrix equation for each element is prepared. Each element matrix is coupled together to create global stiffness matrix by equating the displacements at the intersecting nodes between the corresponding elements. The total number of nodes multiplies by the number of freedom per node, represents the size of global stiffness matrix. Then boundary condition is applied on the structure. In solution phase, the global stiffness matrix is then sent to the solver FE package to solve for unknowns. The solver of the FE package then solves the problem based upon the give inputs. It solves the set of linear and non-linear algebraic equation at a time and generates the results. such as how much displacement taken at different nodes, what is the temperature at different nodes, in case of heat transfer problems. In post processing phase, Readable results are obtained. The results which generated in the solution phase is obtained in the form of graphs and contours. Displacement or temperature contours can be generated by interpolation of the nodal values and plotting.

Crash Analysis of Finite Element Model: The development of Finite Element (FE) model was conducted using the ABAQUS software simulation system. Existing IGES Bumper Beam model was exported into the HYPERMESH for manual meshing then it was exported into the ABAQUS simulation system. The linear Quadrilateral and linear Triangular Shell Elements was used in the meshing process because some of the critical point or area in the geometry needs to have a small meshing size in order to give an accurate model for the 3D-elements. Fig 2 shows the finite element mesh with 70096 elements and 36046 nodes. The Bumper Beam is made to collide with a fixed rigid wall. Meshing of the Bumper System is shown in Fig 2.

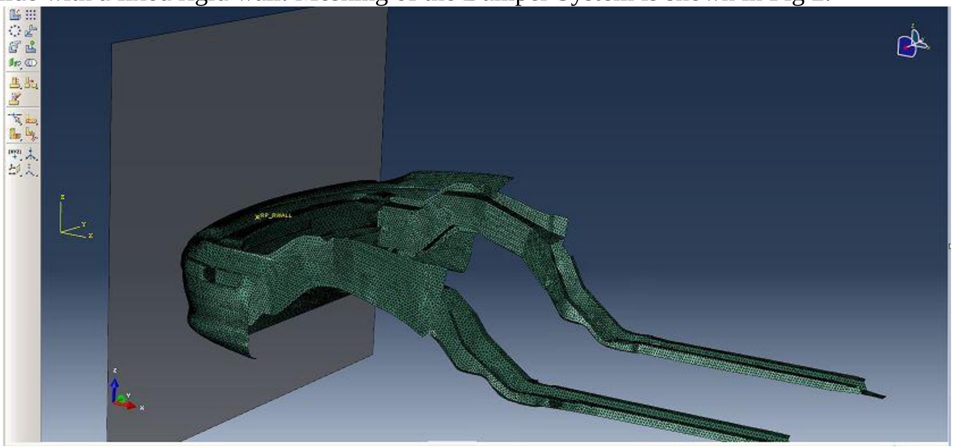

Figure 2: Meshing in Hypermesh.

4. Results:

Figure 3 to Figure 11 relates Stress Distribution, Deflection and Energy Absorption for Bumper Beam For Aluminium, CFR with PEI and CFR with ABS material and C Section, Hat Section and Double Hat Section.

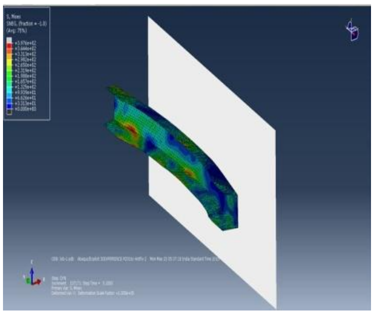

Figure 3: Stress Distribution for C Section of Aluminium material

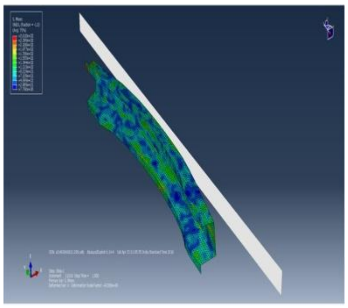

Figure 4: Stress Distribution for Hat Section of Aluminium material

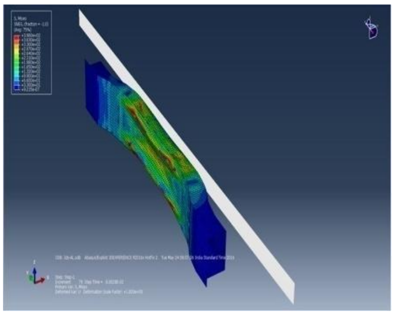

Figure 5. Stress Distribution for Double Hat Section of Aluminium material

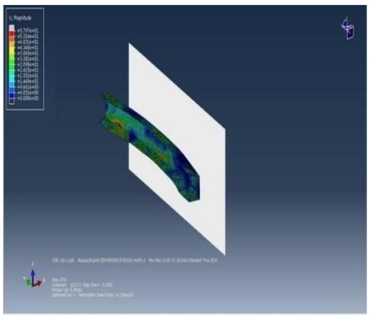

Figure 6. Max. Deflection For C Section of Aluminium material

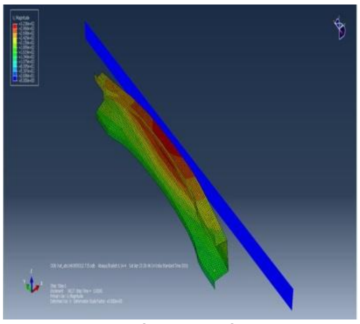

Figure 7. Max. Deflection for Hat Section of Aluminium material

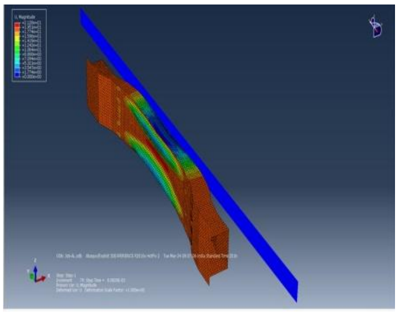

Figure 8. Max Deflection for Double Hat Section of Aluminium material

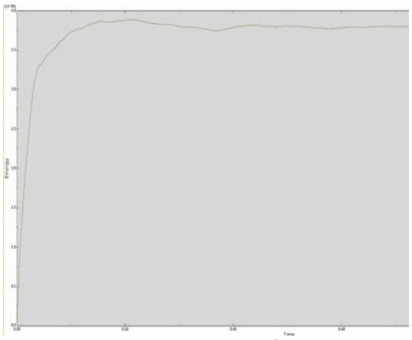

Figure 9. Energy Absorbed for C section of Aluminium material

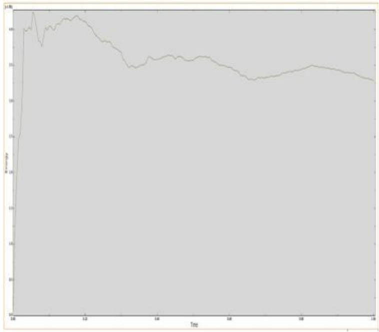

Figure 10. Energy Absorbed for Hat Section of Aluminium material.

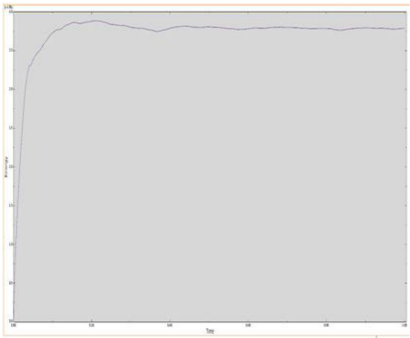

Figure 11. Energy absorbed for Double Hat Section of Aluminium material.

5. Conclusion:

In this study, an attempt is made to compare the stress distribution, deflection and energy absorbed by the bumper beam of various materials along with various cross-sections using finite element method. The stress distribution, deflection and energy absorbed by the existing bumper beam made up steel material is determined and then the material is replaced with Aluminium, CFR with PEI and CFR with ABS along with cross-sections C Section, Hat Section and Double Hat Section. Finally the results of all the materials and cross-sections are compared to suggest the new material along with the cross-section to replace the existing Bumper Beam made up of Steel material. The comparisons of all the results are as shown in the Table 2.

Table 2: Comparisons of all the results

Material	Max. Stress	Max Deflection	Max Energy absorbed
For C Section			
Steel	$4.980e^{02}$	$2.354e^{02}$	$2.5e^{06}$
Aluminium	$3.976e^{02}$	$5.797e^{01}$	$3.8e^{06}$
CFR with ABS	$1.05e^{02}$	$1.151e^{02}$	2e06
CFR with PEI	$3.2e^{01}$	$1.150e^{02}$	$2.5e^{06}$
For Hat Section			
Aluminium	$2.61e^{02}$	$4.081e^{02}$	$4e^{06}$
CFR with ABS	$9.25e^{01}$	$3.23e^{02}$	3e06
CFR with PEI	$3.2e^{01}$	$3.914e^{02}$	$3.5e^{06}$
For Double Hat Section			
Aluminium	$3.96e^{02}$	$2.12e^{01}$	$3.8e^{06}$
CFR with ABS	$1.043e^{02}$	$2.113e^{01}$	$3.8e^{06}$

CFR with PEI	$3.557e^{01}$	$2.09e^{01}$	$3.8e^{06}$

6. References:

1. Hosseinzadeh RM, Shokrieh M, and Lessard LB, "Parametric study of automotive composite bumper beams subjected to low-velocity impacts", Composite Structures 68 (2005) 419–427, Elsevier Ltd.
2. Evans D and Morgan T, "Engineering Thermoplastic Energy Absorbers for Bumpers", SAE Paper, 1999.
3. Butler M, Wycech J, Parfitt J, and Tan E, "Using Tero core Brand Structural Foam to Improve Bumper Beam Design", SAE Technical Paper, 2002,
4. Witteman WJ, "Improved Vehicle Crashworthiness Design by Control of the Energy Absorption for Different Collision Situations", Doctoral dissertation, Eindhoven University of Technology, 2000.
5. Strohl, Daniel. "Endura Front Bumper The bounce-back bumper that freed automotive styling", Hemmings Motor News, Burlington, Vermont July 2006
6. Sheshadri, A, "Design and analysis of a composite beam for side-impact protection of occupants in a sedan, "(Master's thesis), Wichita state University, Wichita.
7. Finite Element Method: An Introduction: Uday S. Dixit, Department of Mechanical Engineering, Indian Institute of Technology Guwahati-781 039, India
8. S. S. Bhvikatti. Finite element analysis. Copyright 2005, ISBN (13):978-81-224-2524-6
9. Practical Aspect of Finite Element Simulation, Hperworks, altair student guide.