

# Crash Analysis of Automotive Bumper Beam by Using Finite Element Analysis

Dadi Ganesh<sup>1</sup>, KV Siva Prasad<sup>2</sup>

Department of Mechanical Engineering, Gonna Institute of Information and Technology and Sciences, Visakhapatnam,  
India <sup>1</sup>

Department of Mechanical Engineering, Gonna Institute of Information and Technology and Sciences, Visakhapatnam,  
India <sup>2</sup>

**ABSTRACT:** Passenger vehicle bumper plays a vital role in case of an accident. Automotive industry is a very huge ground and research is still evolving. The main purpose of this thesis is to study the performance of bumper beams for low speed impact test. The bumper beam of a car is modelled and analysed with different aluminium and steel alloy materials and then the design is modified and improvised. In this study Frontal impact of car on the rigid barrier is considered. This study helps to improve the stability and energy absorption, displacement of the bumper beams for various most commonly used sections are compared in order to know the behaviour of impact.

Passenger automotive bumper beam design should be good enough in order to provide the safety of passengers and their life as well as car body, which should also be of less weight to improve the efficiency of the passenger's car. The test is simulated using finite element method by using HyperMesh and HyperCrash (RADIOSS solver).

**KEYWORDS:** Finite element method, HyperMesh, HyperCrash, Design, Safety, Conclusion

## I. INTRODUCTION

The development process of vehicles with reference to safety behaviour depends strongly on virtual testing and simulation like Finite Element Analysis (FAE). Thus, development work supported cost intensive prototype building and testing has been comprehensively reduced for the vehicle structure still as for the outside and interior trim. The dramatic shortening of the entire development time during the last years needs a far more front-loaded development process which has been realized by numerical simulation. The numerical simulation accompanying the design of a car could also be divided into three main phases – the concept, the series development and also the optimization phase. The concept phase deals the passive safety concept and its needed packaging space must be determined. The series development is finished by prototype testing which should confirm the virtual development in a perfect case. Optimization work should close the event before the car's launch.

The foremost important parameters including shape, material, thickness and impact condition was studied by Javad Marzbanrad for design and analysis of an automotive front bumper beam to enhance the crash worthiness design in low-velocity impact [1]. Han J. et al., Maximization of the Crushing Energy Absorption of Tubes [2], studied crushing energy absorption of square tubes. Solid finite elements and Four node shell in DYNA 3D program were used to model the tubes. A number of the results were compared with the available experimental data. It was found that Maximum energy was absorbed for the Axisymmetric crush mode. Willem Witteman et al, Adaptive frontal structure design to realize optimal deceleration pulses [4], discuss possibilities to style an adaptive vehicle structure which will change the stiffness real time for optimal energy absorption in several crash situations. Besides that all the energy which is absorbed is additionally important to manage intensity during the crash time, because the resulting crash pulse features a large influence on the injury level because of predetermined crash velocity. Bahig B. Fileta. et al., Design of vehicle structures for crash Energy Management [6], which provides an immense resource that has extinguished the author for literature on crash worthiness engineering within the automotive domain.

The main aim to perform crash analysis on two different design variants of automotive bumper beam and observe the crashworthiness of different shapes of bumper beam and also observe variation of stress value with respect to shape optimization.

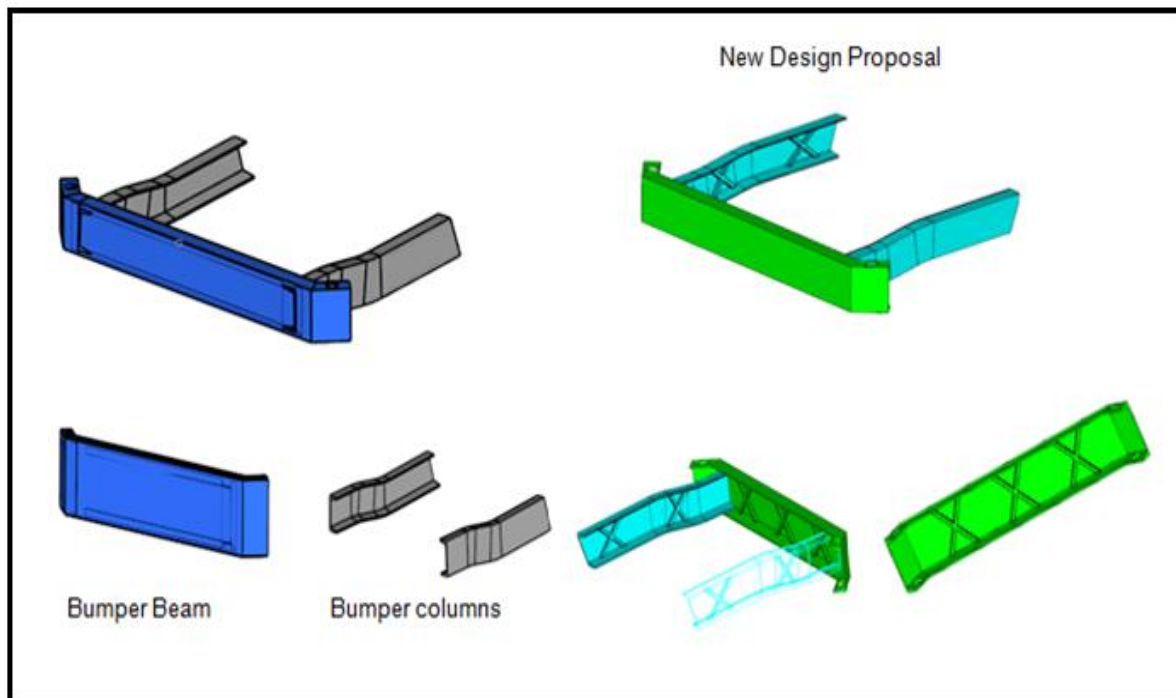
Paper is organized as follows. Section II describes the methodology. Section III describes about the experimental details. Section IV deals with the results and Finally, Section V presents the conclusion.

## II.METHODOLOGY

The experimentation work consists of design and analysis of front bumper of a car for its performance enhancement and effecting compliance to the quality practices within the Industry using Explicit Finite Element Code. So as to attain the main objective of the analysis of loads on the bumper beam of the passenger's car. In comparative manner.

Modelling of bumper beam of car – Catia V5 version it was one of the software application within the CAD/CAM/CAE category, along with HYPERMESH which is used to mesh the product, and HyperCrash (RADIOSS solver).

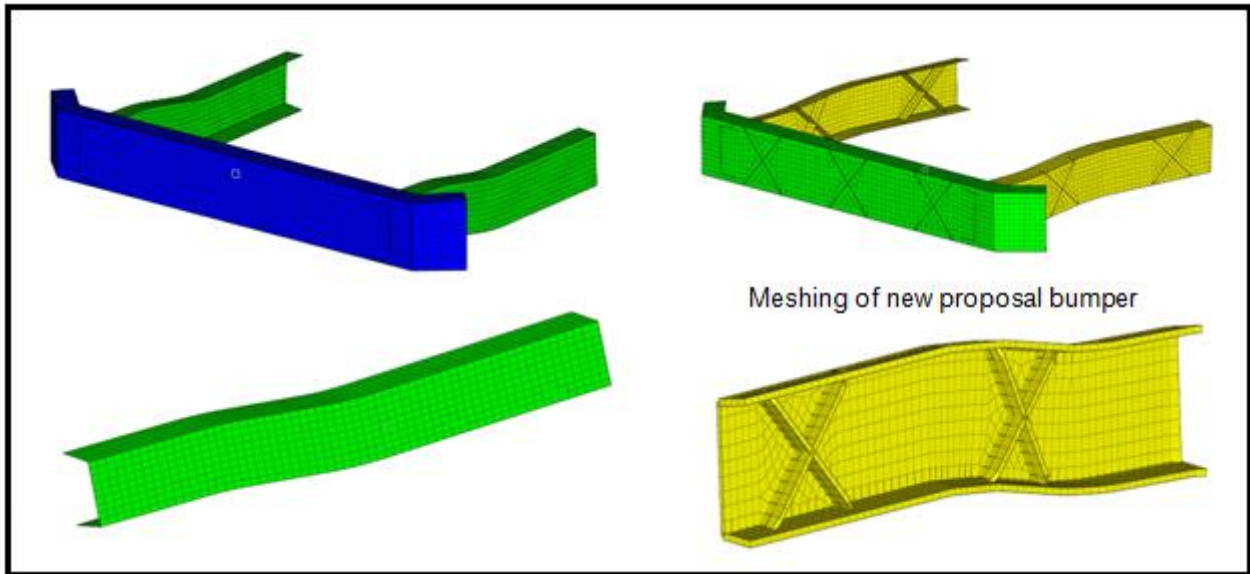
Modeling of the Crankshaft is done using CATIA. CATIA is powerful modeling software package which is one of the most effective parametric modeling and surface feature based modeling software.



The procedure followed in CATIA for modeling of the bumper, initially according to the passenger's car group norms the bumper beam along with the columns drawing the sketch and made that drawing into solid body. Care should be taken while construction the sketch. After modelling the bumper it is saved as CATIA file and IGES file format for meshing purpose. IGES means Initial Graphic Exchange Specification; this file can be easily imported to any software package. The CATIA part file can also be saved as STEP file.

The basis of Finite Element Analysis relies on the decomposition of the domain into a finite number of sub-domains (elements) that the systematic approximate solution is built by applying the variational or weighted residual methods. In effect, FEA reduces problem there to of a finite number of unknowns by dividing the domain into elements and by expressing the unknown field variable in terms of the assumed approximating functions within each and every element. These functions (also called interpolation functions) are defined in terms of the values of the sector variables at specific points, named as nodes. The finite element method could be a numerical procedure which will be wont to obtain solutions to an outsized class of engineering problems involving stress analysis, heat transfer, electro-magnetism, and fluid flow.

Meshing of Normal Design of bumper and new proposal as figured below:



Meshing has been carried out with Hypermesh tool with shell elements. Minimum element Size 1 mm used for proper mesh flow. In this model a uniform thickness of 3 mm is given to mesh as per CAD geometry. It is best practise in finite element method for sheet metals which thickness less than  $\leq 6$ mm mesh carried out with shell elements.

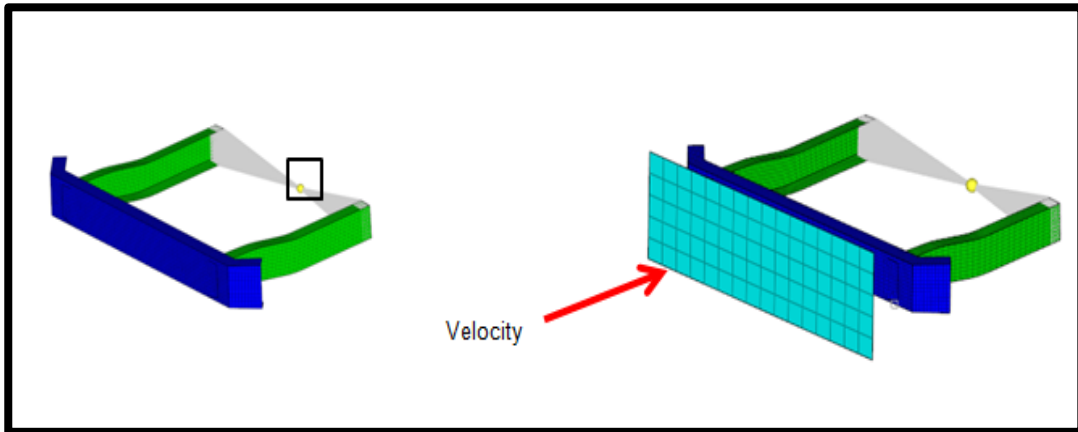
**Material Properties :**

Components	Material	Young’s modulus (Mpa)	Poisson’s Ratio	Density tonnes/mm <sup>3</sup> )
Beam & Columns	Steel BSK46	210E53	0.3	7.8E-9
Beam & Columns	Mild Steel DP800	210E53	0.3	7.5E-9

\*Note: 1. Material temperature used for analysis is room temperature 23 C

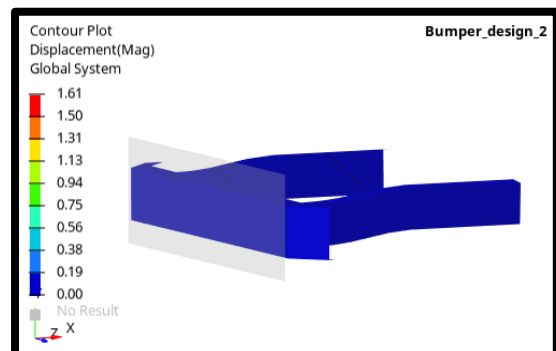
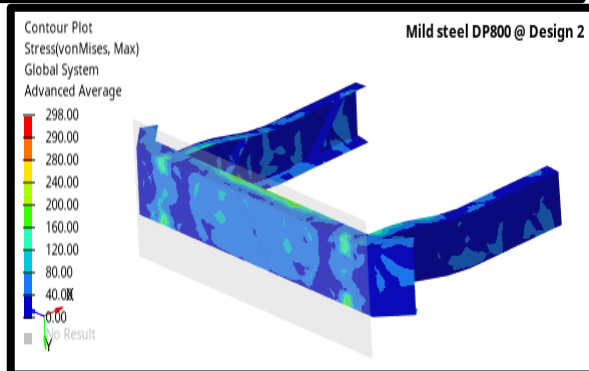
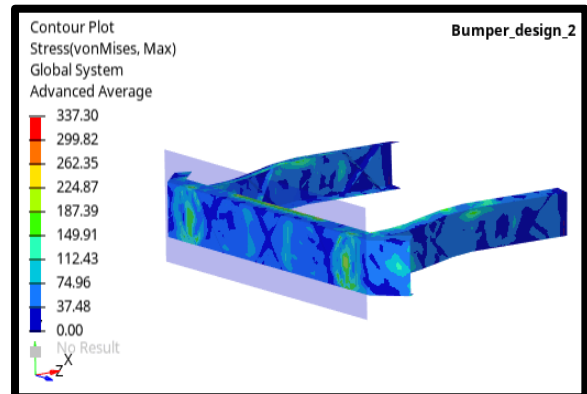
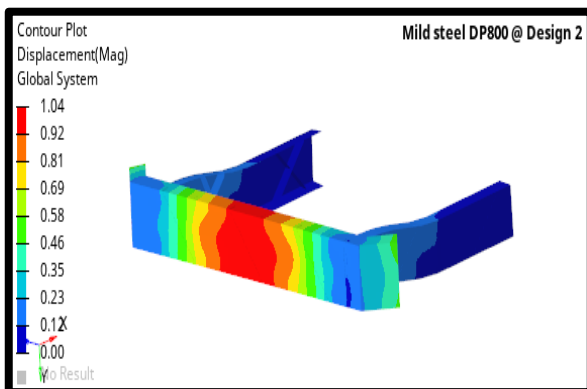
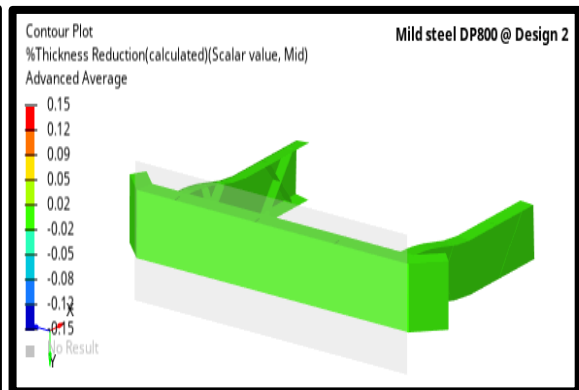
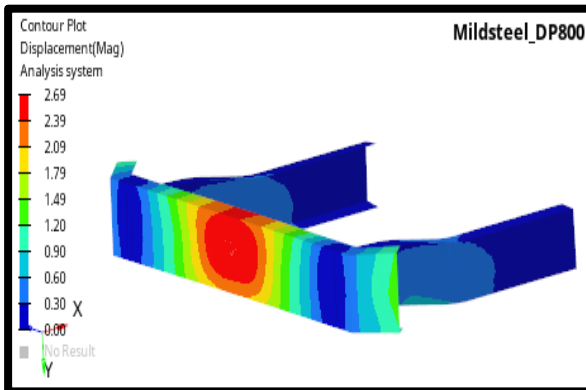
**III. LOADING AND BOUNDARY CONDITION**

The boundary conditions are given to the bumper Highlighted location has been constrained in all DOF. Bumper supported columns conneted with rigid elements, center node of rigid element is fixed in All Dof’s Initially, Rigid wall is created , on this rigid wall a velocity of 11 kmph is assigned to rigid wall and also mass of vehicle is assigned to the rigid wall and then Contacts has very crucial role in crash analysis. In this case slave is defined for bumper and master defined for rigid wall. Self contact is defined for between rigid wall and bumper beam cannot penetrate into the bumper.



#### IV.RESULTS

For a materials Mild steel DP800, Steel BSK46 for Design@ 1,Design@2





Steel BSK46	Displacement (mm)	Stress (MPa)	% thickness reduction
Design@1	2.72	406	0.18
Design@2	2.69	387	0.19

Mild Steel DP800	Displacement (mm)	Stress (MPa)	% thickness reduction
Design@1	1.61	337	0.11
Design@2	1.04	298	0.15

**V.CONCLUSION**

Lightweight and crashworthiness are contradictory targets in auto structural design; therefore, it is of great necessity to optimize their comprehensive performance of lightweight and crashworthiness based on the criteria such as energy absorption, peak stress value. In the process of size optimization in order to improve lightweight and crashworthiness performance, it is found that using multiple materials is of greater potential.

**REFERENCES**

[1] Javad Marzbanrad, et al., “Design and analysis of an automotive bumper beam in low-speed frontal crashes”, School of Automotive Engineering, Iran University of Science & Technology, 16846-13114 Narmak, Tehran, Iran. 2008

[2] Han.J, et al., “Maximization of the Crushing Energy Absorption of Tubes”, Structural Optimization 16, 37-46, Springer-Verlag 1998

[3] Kazuhiro Saitou. et al., “Design Optimization of vehicle structures for Crashworthiness using Equivalent mechanism approximations”, Journal of mechanical design, ASME, DETC2003/DAC-48751

[4] Willem Witteman, “Adaptive frontal structure design to achieve optimal deceleration pulses”, CIP-data library technical university, Eindhoven, ISBN 05-0243

[5] Marian Ostrowski. et al., “Feasibility study of an adaptive energy absorbing system for passenger vehicles”, Institute of Fundamental Technological Research, CMM-2005-Computer Methods in Mechanics, Poland

[6] Bahig B.Fileta. et al., “Design of vehicle structures for crash Energy Management”, Automotive Application Committee, unit of American Iron & Steel Institute,2004

[7]Paul Du Bois, et al., “Vehicle Crashworthiness and occupant protection”, Automotive Applications Committee, American Iron and Steel Institute, Michigan, 2004

[8] Nitin S. Gokhale, Sanjay S. Deshpande, Dr. Anand N.Thite, "Practical Finite Element Analysis", Finite To Infinite, India, 2007.

[9] “LS-DYNA KEYWORD USER’S MANUAL”, Livermore Software Technology Corporation.

[10] <http://www.nhtsa.gov/>