International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 05 Issue: 11 | Nov 2018 www.irjet.net p-ISSN: 2395-0072

Finite Element Analysis of Passenger Vehicle Bumper

Nayan G[1], Anil L[2], Ashish S[2], Sagar V[2]

[1]*University of Maryland, College Park*
[2]*IICAE, Indore*

Abstract - *Passenger vehicle bumper assembly plays very important role in case of an accident. Automotive industry is a very huge ground and research is still evolving. From this safety of passenger has to be focused on safety and comfortably. In this paper, the most important parameters of an automotive front bumper beam such as material, shape and impact condition are to be studied to improve the stiffness and strength. The strength of the bumper is investigated with energy absorption and impact force in maximum deflection situation. Similar bumper beams made of different materials are simulated to determine the deflection, impact force, stress distribution and energy-absorption behavior; these characteristics are compared with each other to find best choice of material. These designs had been studied by impact modeling using Finite Element Analysis software, LS DYNA to determine the energy absorption by analyzing the kinetic energy, displacement and impact force.*

Key Words: FEA, Bumper, Safety, LS-Dyna

1. INTRODUCTION

The development process of vehicles with regard to safety behaviour depends strongly on virtual testing and simulation like Finite Element Analysis (FAE). Thus, development work based on cost intensive prototype building and testing has been comprehensively reduced for the vehicle structure as well as for the exterior and interior trim. The dramatic shortening of the total development time during the last years needs a much more front-loaded development process which has been realized by numerical simulation. The numerical simulation accompanying the design of a car may be divided into three main phases – the concept, the series development and the optimization phase. During the concept phase the passive safety concept and its needed packaging space have to be determined. The series development is finished by prototype testing which should confirm the virtual development in an ideal case. Optimization work should close the development before the car's launch. Javad Marzbanrad. et al., studied the most important parameters including material, thickness, shape and impact condition for design and analysis of an automotive front bumper beam to improve the crashworthiness 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. Four node shell and solid finite elements in DYNA 3D program were used to model the tubes. Some 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 achieve optimal deceleration pulses [4], discuss possibilities to design an adaptive vehicle structure that can change the stiffness real time for optimal energy absorption in different crash situations. Besides that all the energy which is absorbed is also important to manage the intensity during the crash time, because the resulting crash pulse has a large influence on the injury level due to predetermined crash velocity. Bahig B.Fileta. et al., Design of vehicle structures for crash Energy Management [6], provides an immense resource that has quenched the author for literature on crashworthiness engineering in the automotive domain.

The aim of this work is to study front bumper of passenger car. Design modifications can be suggested or tried out on following basis:

- Performance related parameters of bumper
- Deformation/ Energy absorption capability
- Shape/ Size/ Thickness (Geometry)

The study will focus on modifying few of above stated parameters to suggest improvements in existing bumper of passenger car.

2. METHODOLOGY

The research work consists of design and analysis of front bumper of a passenger car for its performance enhancement and effecting compliance to the standard practices in the Industry using Explicit Finite Element Code. The main objective of this work is to determine energy absorption structure to absorb the impact energy during speeds 56 km/h. In order to achieve the main objective, the analysis of stress, strain, displacement for existing and our modified bumper then to study these two bumpers in comparative manner.

Modelling of Car Bumper by Creo – Software Creo/Engineer is new version of pro-e it is a software application within the CAD/CAD/CAM/CAE category, along with other similar products currently on the market. Creo/Engineer is a parametric, feature-based modelling architecture incorporated into a single database philosophy with advanced rule-based design capabilities.

Figure 1: CAD model of Bumper

The mid surface mesh tool enables direct extraction of a shell mesh from solid geometry and applies the thicknesses to the corresponding meshed output. Thin Solids mesh enables the automatic hexa and/or penta dominant mesh for thin plastic or sheet metal-type parts. Hypermesh was used to create mesh of the model.

Figure 2: Mesh model of Bumper

Two iterations were performed

1. Bumper thickness 3 mm
2. Bumper thickness 5 mm

Material Properties:

The steel mechanical properties were used for this analysis as shown in Table 1.

Table -1 Material description (Steel BSK46)

Parameters	**Values**
Density	7.8E-9 Tons/mm3
Elastic Modulus	2.1E5 MPa
Poisson's ratio	0.3
Yield Stress	460 MPa

Complete model:

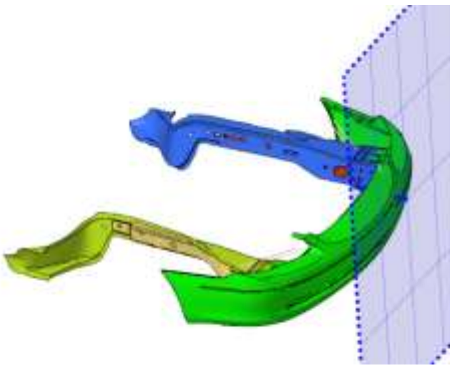

Figure 3: model setup of Bumper Impact

3. RESULTS

Figure 4-9 shows the difference in total deformation and strain of the baseline and modified design.

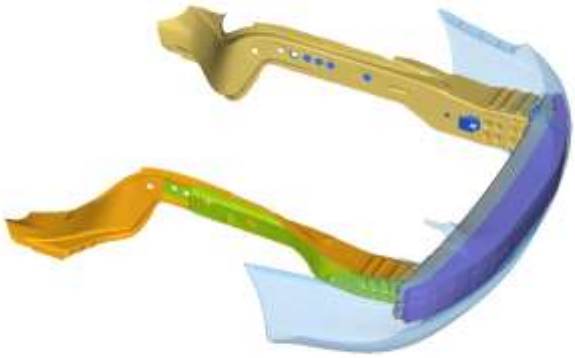

Figure 4: Bumper before Impact

Figure 5: Bumper after Impact

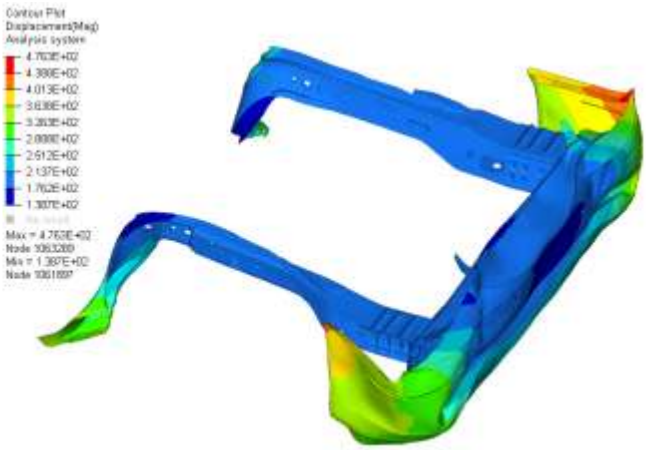

Figure 6: Bumper displacement Baseline Design

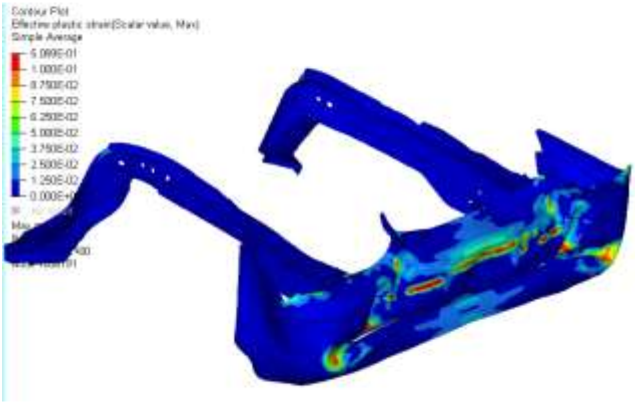

Figure 7: Bumper strain Baseline Design

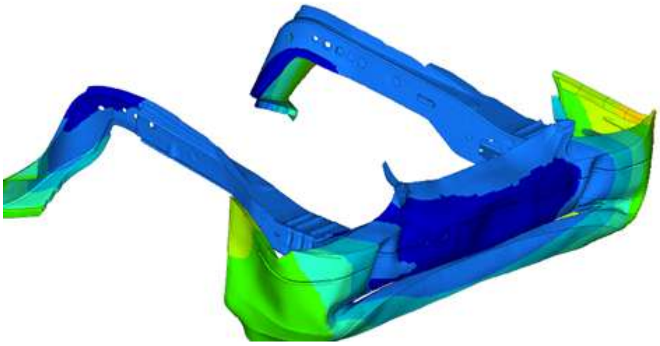

Figure 8: Bumper displacement modified Design

Figure 9: Bumper strain modified Design

4. CONCLUSIONS

Table 2 shows the overall improvements in deformation and strain of the bumper. It can be seen clearly that with modified design, the deformation and strain can be reduced significantly.

Table 2: Results Summary

	Maximum Strain	**Maximum Deformation**
Baseline Design	**50 %**	**476 mm**
Modified Design	**15 %**	**423 mm**

5. 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. Despande, 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/