International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 2 Issue 6, June - 2013

Crash Analysis Of Bumper Assembly With Solver To Improvise The Design For Impact Tests

F. B. Sayyad
Mechanical Engineering Department,
Genba Sopanrao Moze College of
Engineering, Pune, India.

Abhaysingh Diliprao Deshmukh
Mechanical Engineering Department,
Genba Sopanrao Moze College of
Engineering, Pune, India.

Abstract

A good design of car bumper must have optimized weight and must provide safety to passengers. Different countries have different performance standards for bumpers. Under the International safety regulations originally developed as European standards and now adopted by most countries outside North America, a car's safety systems must still function normally after a straight-on pendulum or moving-barrier impact of 4 km/h (2.5 mph) to the front and the rear, and to the front and rear corners of 2.5 km/h (1.6 mph) at 45.5 cm (18 in) above the ground with the vehicle loaded or unloaded.. Due to increasing competency now a day's tests are carried out at some higher velocities such as 10 m/s to increase the safety level of vehicle. Increased safety of vehicle helps to claim for larger insurance amount. Automotive development cycles are getting shorter by the day. With increasing competition in the marketplace, the OEM's and suppliers main challenge is to come up with time-efficient design solutions. Researchers are trying to improve many of existing designs using novel approaches. Many times there is conflicting performance and cost requirements, this puts additional challenge with R&D units to come up with a number of alternative design solutions in less time and cost compared to existing designs. These best solutions are best achieved in a CAE environment using some of the modern CAD and FEM tools. Such tools are capable of effecting quick changes in the design within virtual environment.

1. Introduction

Nowadays, in development of technology especially in engineering field make the engineers must be precise and showing careful attentions on what they produce. Here, we concentrate on automotive industry. One of the options to reduce energy consumption is weight reduction. However, the designer should be aware that in order to reduce the weight, the safety of the car passenger must not be sacrificed. In development of bumper systems for the automotive industry, iterative Finite Element (FE) simulations are normally used to find a bumper design that meets the requirements of crash performance. The crash performance of a bumper system is normally verified by results from standardized low speed crash tests based on common crash situations. Consequently, these crash load cases are also used in the FE simulations during the development process. The increasing legal and customer demands on passive safety of automobiles have to be fulfilled under the conditions of shortened development times and cost reductions. Today the design process of a car with regard to its crashworthiness function is driven by a virtual development.

In a case of a collision to the front or rear occurring at low speed, the bumper shall absorb the energy to prevent or reduce damage to the car. Consequently, the purpose of the bumper is not to be a structural component that actively contributes to occupant protection during front or rear collisions but more to protect components like the hood, lights and cooling

ISSN: 2278-0181
Vol. 2 Issue 6, June - 2013

system of the car. In bumper system development, iterative finite element (FE) crash simulations are most commonly used to find a candidate design that may meet the requirements stated by the manufacturers, by insurance companies and in legislations. Besides those requirements, considerations of weight and cost for manufacturing are also factors that are regarded. Here we deal with the plastic strain values of components in bumper assembly which are to be kept in permissible limit in both analysis by solver and actual testing.

2. Analysis of original model

We are provided with bumper assembly as shown in figure 1 whose component thicknesses are listed in the table 1.

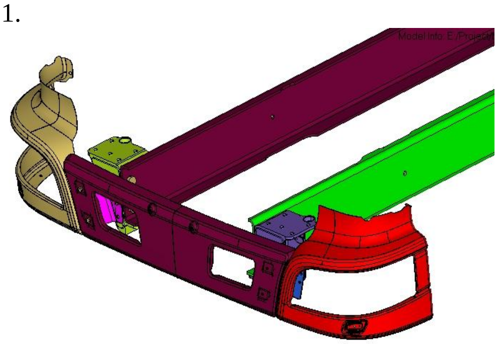

Figure 1. Assembly of bumper system to be analyzed

Table 1. Part details

Sr. no.	Part/ component name	Original model thickness
1	Front panel	01.60mm
2	Side panel	01.60mm
3	Bracket	04.00mm
4	Supporting bracket	12.00mm
5	Chassis parts	10.00mm

21. Preprocessing

The model consists of infinite number of points hence it should be discretized to some finite number of divisions on which analysis is to be carried out. So we mesh this model to divide it into finite number of divisions called as nodes and elements. We prefer 2d or shell mesh as the third dimension (thickness) of all the components is very small as compared to other two dimensions (length and width) Mesh size is selected by convergence criteria. After meshing the model appears as shown in figure 2.

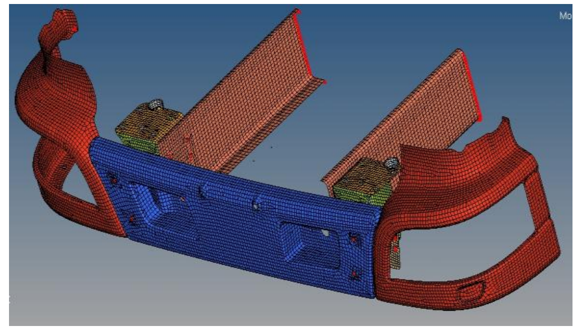

Figure 2. Meshed model of bumper assembly system

Once the meshing is done model is checked for quality and normals so that stress regions are properly defined after analysis. After meshing materials of respective parts are assigned to them by their material properties such as modulus of elasticity, poisons ratio, density of material, etc. . Here we are provided with Steel as a basic material whose properties are described in table 2 below.

Table 2 Material properties of Steel

Material	Modulus of elasticity	Density	Poisson's ratio
Steel	210 kN/mm^2	7.86e^{-6} kg/mm^3	0.29

This material is further classified as soft steel and hard steel from non linear stress strain curve data and assigned to the components as shown in table 3.

Table 3. Assignment of Materials to components

Sr. no.	Part/ component name	Material
1	Front panel	Soft steel
2	Side panel	Soft steel
3	Bracket	Hard steel
4	Supporting bracket	Hard steel
5	Chassis parts	Hard steel
6	Impactor	Rigid

Boundary conditions are reference for problem solving in analysis. This deals with constraining (fixing) the model, application of loads, giving proper contacts etc. . Here we are provided with the constrained conditions, mass of vehicle which is about 887 kg and velocity of impact which is 10m/sec. bolt connections are given by beams and proper constraints are applied. The constrained model appears as shown in figure 4

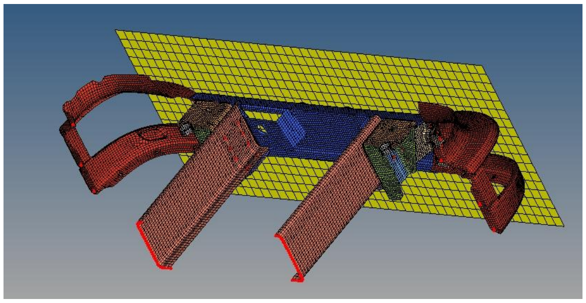

Fig. 4 Bumper assembly after preprocessing

The velocity is given to the impactor through contacts defined between bumper and impactor. The mass of vehicle here is considered during impact conditions. Contacts are defined via elements as shown in figure 5.

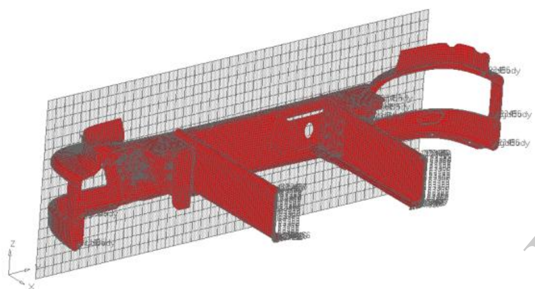

Figure.5. Image of contacts in the model

2.2. Solution stage

After pre processing model is further send for analysis. Here we use LS- Dyna solver for analysis purpose which is an explicit solver. The model in LS-Dyna solver appears as shown in figure 6.

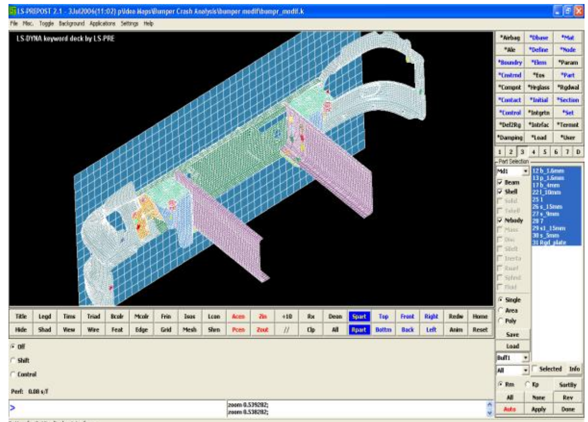

Figure 6. Bumper assembly in LS- Dyna Solver

2.3. Post processing

After carrying out analysis results are viewed. Our analysis is of nonlinear type. Here we deal with the effective plastic strain values for the components used in analysis. Our aim is to maintain the plastic strain values of components up to their prescribed tolerance values. Permissible values for plastic strain is 0.25 (25%) for soft Steel and 0.33 (0.33%) for hard steel. The plastic strain results for bumper original model are as shown below

1] Bumper Front panel

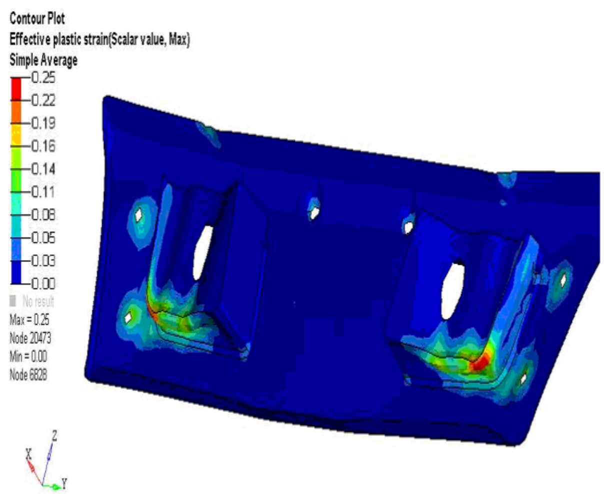

Figure 7. Contour plot of effective plastic strain values for Bumper Front panel.

From figure 7 the maximum plastic strain value for bumper front panel is 0.25 which is equal to permissible limit hence the design for Bumper frontal panel is safe.

2] Bumper Side panel

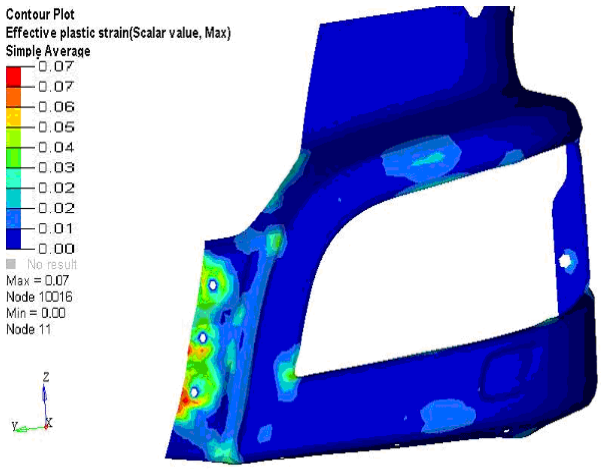

Figure 8. Contour plot of effective plastic strain values for Bumper Side panel.

From figure 8 the maximum plastic strain value for bumper Side panel is 0.07 which is less than permissible limit hence the design for Bumper side panel is safe.

3] Bumper Bracket

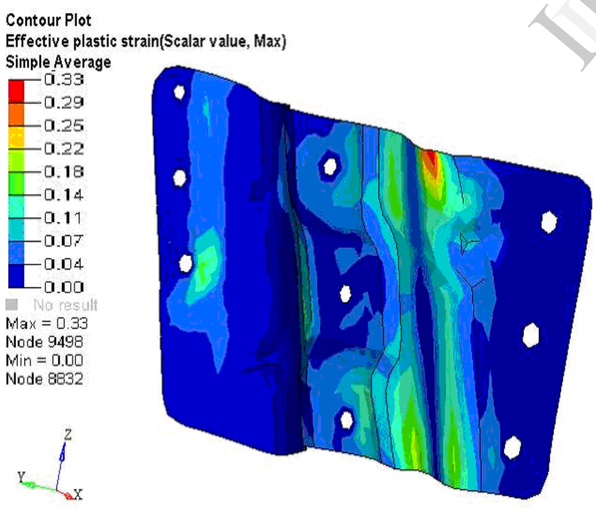

Figure 9. Contour plot of effective plastic strain values for Bumper Bracket

From figure 9 the maximum plastic strain value for bumper bracket is 0.33 which is more than permissible limit hence the design for Bumper side panel is not safe and the effective plastic strain value need to be brought within permissible range.

4] Bumper supporting Bracket

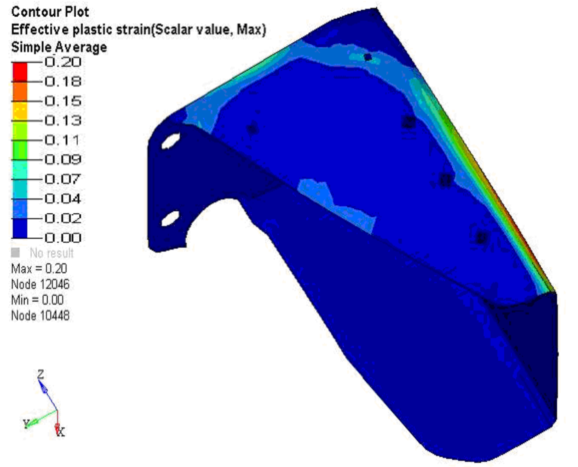

Figure 10. Contour plot of effective plastic strain values for supporting Bracket

From figure 10 the maximum plastic strain value for bumper supporting Bracket is 0.20 which is less than (0.33) permissible limit hence the design for Bumper supporting Bracket is safe.

5] Chassis section

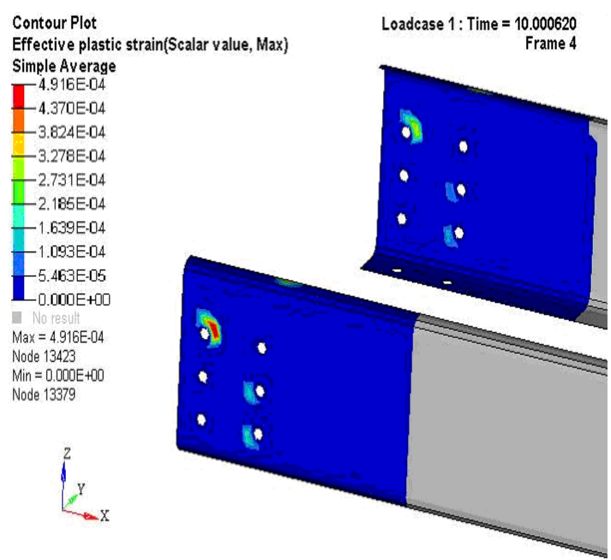

Figure 11. Contour plot of effective plastic strain values Chassis section

From figure 11 the maximum plastic strain value for bumper Chassis section is 0.00049 which is less than permissible limit hence the design for Bumper Chassis section is safe.

Table 4. Plastic Strain values for all the original model components in the assembly

Sr. no.	Part/ component name	Original model thickness	Original model Plastic strain
1	Front panel	1.6mm	0.25
2	Side panel	1.6mm	0.07
3	**Bracket**	**4.00mm**	**0.33**
4	Supporting Bracket	12.00mm	0.20
5	Chassis	10.00mm	0.0004

From table 4 we come to conclusion that Bumper bracket exceeds the effective permissible strain value hence some necessary changes should be made in assembly to bring the plastic strain values in the permissible limit so that design is safe.

2.4. Possible solutions

1] Change in design of components-: we can change the design of components to get the required results. This leads to redesigning the components by addition of ribs or change the geometry thus it increases the cost.

2] Change of material-: We can change the material of the components for proper stress distribution. We can go for some additional composite materials to avoid the design failure.

3] Change in thickness-: we can change the thickness of components to achieve the effective plastic strain values for the respective components. It is cost effective and time consuming way of modification.

From above we choose the third possible solution to achieve the plastic strain values and make the design safe.

3. Analysis of modified model

Hence the modified model with change in thickness of components is given in table 5.

Table 5. Thickness of components for bumper modified assembly

Sr. no.	Part/ component name	Modified model thickness
1	**Front panel**	**2.00mm**
2	Side panel	1.60mm
3	**Bracket**	**6.00mm**
4	Supporting Bracket	12.00mm
5	Chassis	10.00mm

The boundary conditions remain same for the modified model the plastic strain results achieved after the changes in thickness are as follows-:

1] Bumper Front panel-:

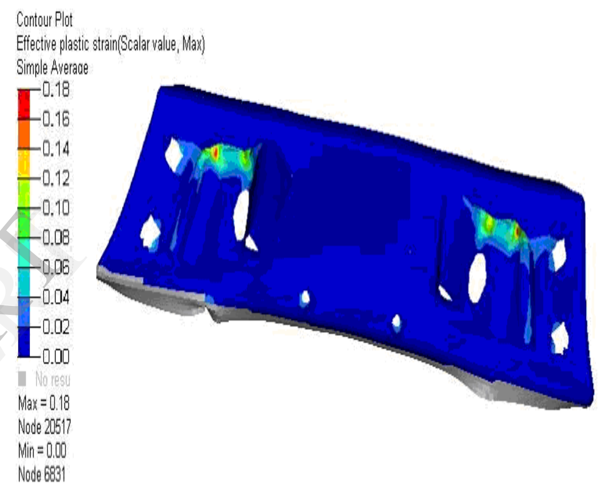

Figure 11. Contour plot of effective plastic strain values for Bumper Front panel

The maximum plastic strain value for bumper front panel is 0.18 which is less than permissible limit hence the design for Bumper frontal panel is safe.

2] Bumper Side panel-:

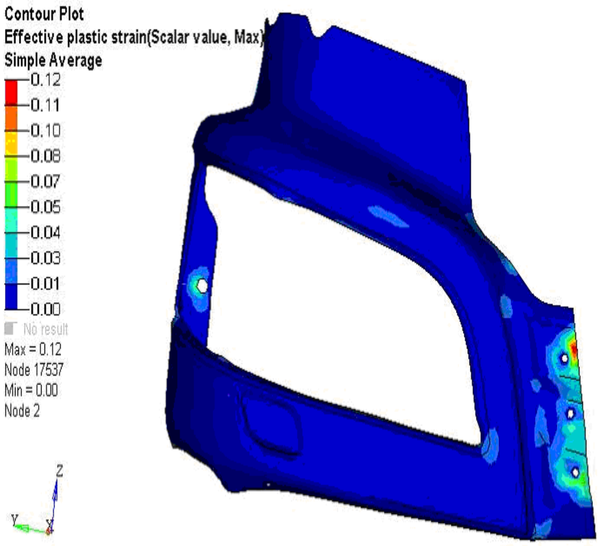

Figure 12. Contour plot of effective plastic strain values for Bumper Side panel

The maximum plastic strain value for bumper Side panel is 0.12 which is less than permissible limit hence the design for Bumper side panel is safe.

3] Bumper Bracket-:

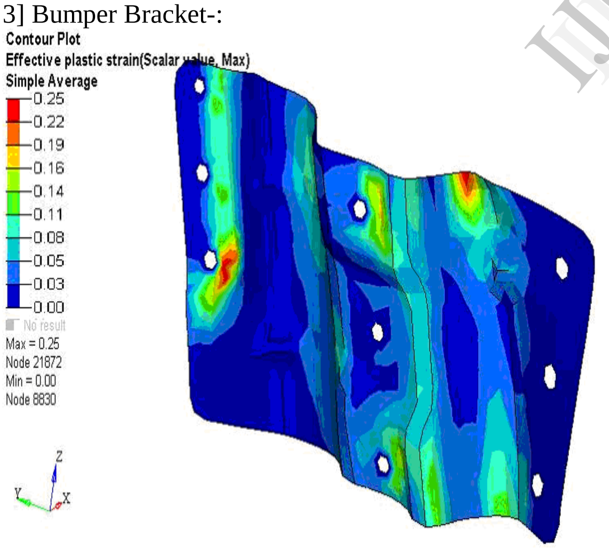

Figure 13. Contour plot of effective plastic strain values for Bumper Bracket

The maximum plastic strain value for bumper bracket is 0.25 which is equal to permissible limit hence the design for Bumper side panel is safe.

4] Bumper supporting Bracket-:

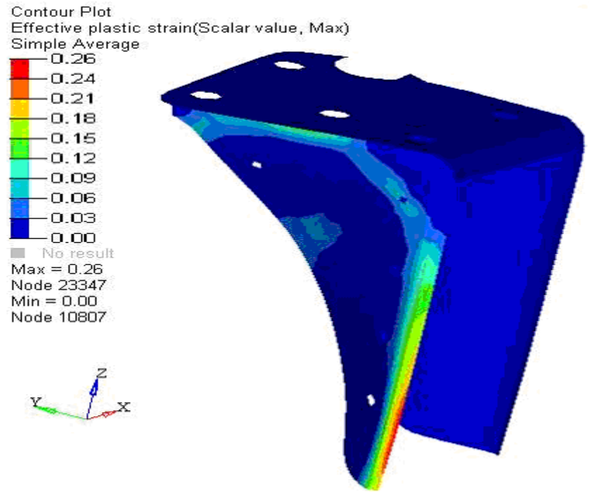

Figure 14. Contour plot of effective plastic strain values for supporting Bracket

The maximum plastic strain value for bumper supporting Bracket is 0.26 which is less than 0.33 permissible limits hence the design for Bumper supporting Bracket is safe.

5] Chassis section-:

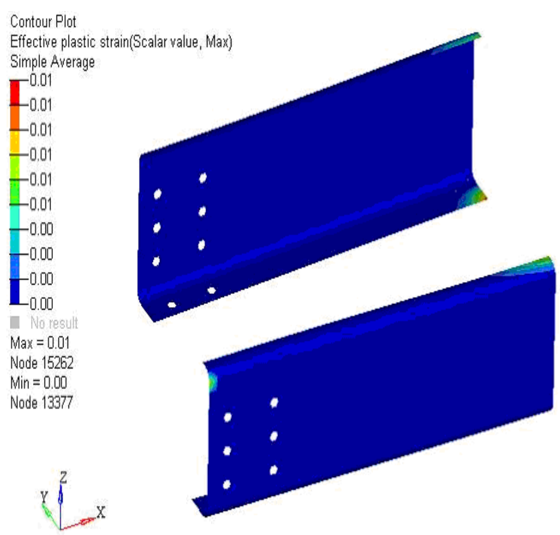

Figure 15. Contour plot of effective plastic strain values Chassis section

The maximum plastic strain value for bumper Chassis section is 0.01 which is less than permissible limit hence the design for Bumper Chassis section is safe.

Table 6. Plastic strain values for modified Bumper assembly.

Sr. no.	Part/ component name	Modified model thickness	Modified model Plastic strain values
1	Front panel	2.00mm	0.18
2	Side panel	1.60mm	0.12
3	Bracket	6.00mm	0.25
4	Supporting Bracket	12.00mm	0.26
5	Chassis	10.00mm	0.01

Permissible values for plastic strain is 0.25 (25%) for soft steel and 0.33 (33%) for hard Steel. Hence from table 6 in modified model all the components strain values are within permissible limit thus making the design safe.

Testing is carried out according to regulations. Figure 16 shows an image of test arrangement.

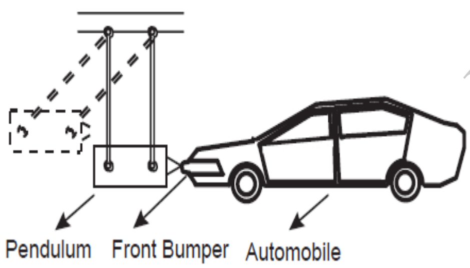

Figure 16. Bumper impact test arrangement

Table 7 shows comparison of experimental and analysis report referred from test report.

Table 7. Comparison of experimental and analysis result for modified geometry

Sr. No.	Part name	Strain values by analysis	Strain values by experimentation	% difference
1	Front panel	0.18	0.21	+3
2	Side panel	0.12	0.14	+2
3	Bracket	0.25	0.20	-5
4	Supporting Bracket	0.26	0.18	-8
5	Chassis	0.01	0.02	+1

From table 7 we found that analysis and experimental values are in close agreement.

4. Conclusion

From above study we come to the conclusion that the permissible strain values can be achieved by changing the thickness of bumper components. Changing the thickness is one of the cost effective way to get the assembly in safety zone as compared to others such as change in geometry or addition of ribs.

Permissible plastic strain value results show efficient energy absorption, thus making the component assembly safe. Hence running an analysis on explicit solver leads to cost effective way to solve the crash related problems prior to actual production.

It is observed that more kinetic-energy transfer from impactor to vehicle and less plastic strain energy dissipates with increasing the bumper thickness.

Increasing bumper thickness causes a rise in bumper rigidity increasing its strength. Consequently, it results in reduction in strain values.

10. References

[1] Javad Marzbanrad , Masoud Alijanpour , Mahdi Saeid Kiasat, "Design and analysis of an automotive bumper beam in low-speed frontal crashes", *Science Direct paper*, 2009, pp.902-911.

[2]Liquan Mei and C.A. Thole , "Data analysis for parallel car-crash simulation results and model optimization", *Science Direct paper*, 2007, pp. 329-337.

[3] Nader Abedrabbo and Robert Mayer, "Crash response of advanced high-strength steel tubes: Experiment and model", *Science Direct paper*, 2009, pp.1044-1057.

[4] F.Ince, H.S. Turkmen, Z. Mecitoglu, N. Uludag, I. Durgun, E. Altınok and H. Orenel , "A numerical and experimental study on the impact behavior of box structures", *Science Direct paper* , 2011, pp.1736-1741.

[5] Bryan C. Baker, Joseph M. Nolan, Brian O'Neill and Alexander P. Genetos , "Crash compatibility between cars and light trucks: Benefits of lowering front-end energy-absorbing structure in SUVs and pickups", *Science Direct paper*, 2007, pp.116-125.

[6] Tso-Liang Tenga, Fwu-An Changb, Yung-Sheng Liuc and Cheng-Ping Peng , "Analysis of dynamic response of vehicle occupant in frontal crash using multibody dynamics method", *Science Direct paper*, 2007, pp.1724-1736.

ISSN: 2278-0181
Vol. 2 Issue 6, June - 2013

[7] O.G. Lademo, T. Berstad, M. Eriksson, T. Trylan, T. Furu, O.S. Hopperstad and M. Langseth , "A model for process-based crash simulation*", Science Direct paper,* 2007, pp.376-388.

[8] Jovan Obradovic, Simonetta Boria and Giovanni Belingardi, "Lightweight design and crash analysis of composite frontal impact energy absorbing structures", *Science Direct paper,* 2011, pp.423-430.

[9] Ping Zhu, Yu Zhang, Guanlong Chen, "Metamodeling development for reliability-based design optimization of automotive body structure", *Science Direct paper,* 2011, pp.729-741.

[10] Xin Yang, Yong Xia, Qing Zhou, Pei-ChungWang and KathyWang , " Modeling of high strength steel joints bonded with toughened adhesive for vehicle crash simulations", *Science Direct paper,* 2012, pp.21-32.