

## Numerical Simulation of Bumper Impact Analysis and To Improve Design for Crash Worthiness

Niketa Kankariya , Dr. F.B.Sayyad

Department of Mechanical Engineering, Genba Sopanroa Moze College of engineering pune  
Department of Mechanical Engineering, Genba Sopanroa Moze College of engineering pune

---

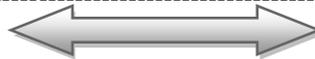
### ABSTRACT

*The major concern in design of bumper is their potentiality to bear impact loads. Therefore, bumper impact test is required to fulfill the safety requirement. 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. Explicit finite element method is used to investigate stress and effective plastic strain of bumper at impact. Design modification is applied to the bumper to improve its impact performance.*

**Keywords** - Bumper impact, finite element analysis, effective plastic strain.

---

Date of Submission: 23-May-2015



Date of Accepted: 05-June-2015

---

### I. INTRODUCTION

An automobile's bumper is the front-most or rear-most part, ostensibly designed to allow the car to sustain an impact without damage to the vehicle's safety systems. They are not capable of reducing injury to vehicle occupants in high-speed impacts, but are increasingly being designed to mitigate injury to pedestrians struck by cars. Front and rear bumpers became standard equipment on all cars in 1925. What were then simple metal beams attached to the front and rear of a car have evolved into complex, engineered components that are integral to the protection of the vehicle in low-speed collisions. Car accidents are happening every day. Most drivers are convinced that they can avoid such troublesome situations. Nevertheless, we must take into account the statistics – ten thousand dead and hundreds of thousands to million wounded each year. These numbers call for the necessity to improve the safety of automobiles during accidents. Automotive bumper system is one of the key systems in passenger cars. Bumper systems are designed to prevent or reduce physical damage to the front or rear ends of passenger motor vehicles in collision condition. They protect the hood, trunk, grill, fuel, exhaust and cooling system as well as safety related equipment such as parking lights, headlamps and taillights, etc. A good design of car bumper must provide safety for passengers and should have low weight. 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. In North America (FMSS: Federal Motor Vehicle Safety Standards) and Canada (CMVSS: Canadian Motor Vehicle Safety Standards), it should be meet 4KMPH pendulum and barrier impacts.

### II. IMPACT ANALYSIS BACKGROUND

In engineering practice, impacted systems present a complexity of geometry, stiffness, mass distributions, contact areas and impact angles that are impossible to analyze and design with the traditional impact dynamics methods. Therefore, using the knowledge of traditional method in finite element analysis will help to solve the engineering problems. Some traditional methods are:

#### 2.1 Stereomechanics

This method uses the conservation of energy and momentum laws along with the impulse-momentum law. For collisions where energy is dissipated, the method uses the coefficient of restitution. The coefficient of restitution ( $e$ ) is used to relate the effect between the pre and post-impact velocities as the result of energy dissipation and is given by

$$e = \frac{v_f}{v_i} \tag{1}$$

The coefficient of restitution is a dimensionless parameter ranging from 0 to 1 with  $e = 0$  representing a perfect plastic and  $e = 1$  representing rigid body or perfectly-elastic impact. The COR is not a material property, but varies with the material types impacted, impact velocities and surface geometry.

### 2.2 Stress-Wave Impact Approach

Stress wave theory addresses the phenomenon when a striker strikes elastic solid. At impact, a strain wave is initiated at the contact region and transverses or radiates throughout the solid at a velocity of  $\left(\frac{E}{\rho}\right)^{\frac{1}{2}}$ , the speed of sound in a solid where  $E$  is the Young’s modulus and  $\rho$  is the mass density of the struck material. As time progresses, these original waves contact the boundary surfaces of the solid and reflect inward. This generates standing or interfering strain waves, which produce the larger strains and stresses associated with impact condition than that of ordinary static loading.

### 2.3 Energy Method Approach

The basis for the method is that mechanical energy is conserved. The potential and kinetic energies of the striker are converted into strain energy, which is stored within the struck system or structure. The maximum deflections and stresses of the impacted system occur when the velocity of the striker becomes zero when all of the striker’s energy has transferred to the target. Equating the striker’s energy to the system’s strain energy yields the maximum deflection of the system.

$$\frac{1}{2} m v^2 = \frac{1}{2} k \delta_m^2 \tag{2}$$

$$\delta_m = \sqrt{\frac{m}{k}} v \tag{3}$$

Eq. (3) indicates that the dynamic deflection  $\delta_m$  of the mechanical system can be determined.

## III. FINITE ELEMENT ANALYSIS

A general-purpose commercial explicit finite element code, HyperMesh and LS-Dyna is applied to conduct the dynamic impact test simulations. The FEA model of tire and wheel used in this study is constructed based on the geometry. A full 3-D solid model consisting of six portions, namely front panel, side panel, bracket, supporting bracket, chassis parts is constructed for the dynamic impact test simulation. The schematic of an FEA model used in dynamic impact test simulations is shown in figure 1 and figure 2 respectively.

Table 1. Bumper Part list

Sr. no.	Part/ component name	Model Thickness
1	Front panel	1.60mm
2	Side panel	1.60mm
3	Bracket	4.00mm
4	Supporting bracket	12.00mm
5	Chassis parts	10.00mm

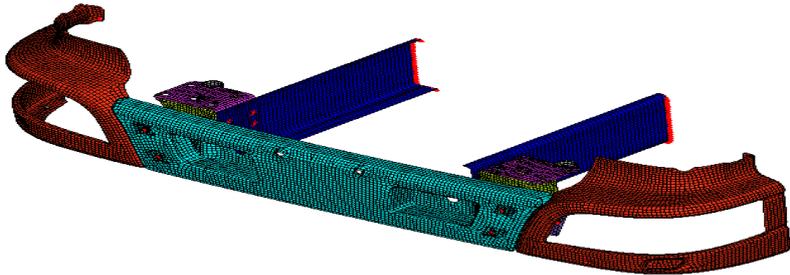


Figure 2. Finite Element Model of Bumper

3.1 Mesh Convergence

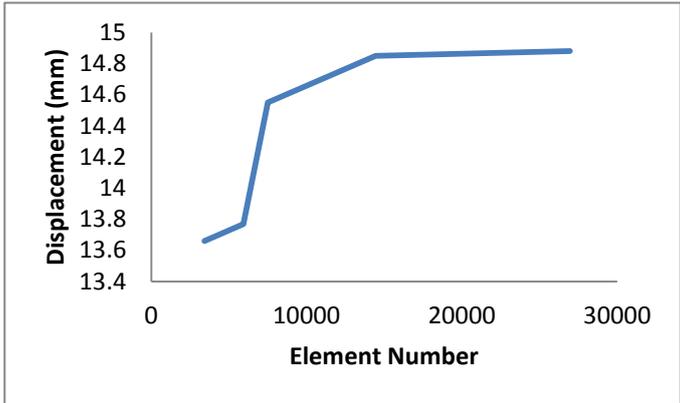


Figure 3. Maximum displacement at contact point for various mesh size

In order to assure the mesh model is accurate enough, a mesh convergence has been studied to ensure that the displacement at the contact point between the striker and bumper is convergent. In figure 3 the abscissa represents the element number of model. It shows that the maximum displacement response at the contact point is almost convergent for mesh size 12 mm. In this study, the simulation model for bumper is constructed by mesh size 12 mm. Bumper model is meshed with about 21068 nodes and 20382 elements.

3.2 Material Properties

The material properties of bumper that is widely used in automotive engineering industry was considered in the FE simulation. Mechanical properties of steel are given in table 2. A nonlinear elastic–plastic material model was used to describe the material behavior of bumper in the dynamic analyses. The true stress–strain curve of the soft steel and hard steel is plotted in figure 4. True stress–strain material data are required for input into the finite element model. The engineering stress–strain curve data were converted into the true stress–strain curve data and then imported into the FEA model. Figure 4 shows true stress–strain curve.

Table 2 Material Properties of Bumper

Modulus of Elasticity (N/mm <sup>2</sup> )	Poisson’s Ratio	Yield Strength (N/mm <sup>2</sup> )	Ultimate Strength (N/mm <sup>2</sup> )	Fracture Strain
210000	0.3	350	650	0.21

$$\sigma_{true} = \sigma_{engg} (1 + \epsilon_{engg}) \tag{4}$$

$$\epsilon_{true} = \ln(1 + \epsilon_{engg}) \tag{5}$$

Equation (4) and (5) are used to calculate true stress–strain values for steel.

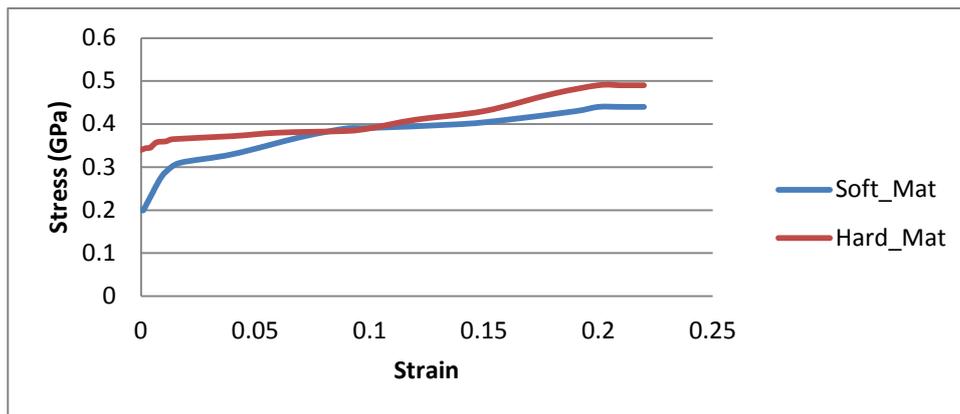


Figure 4. True Stress-Strain curve

### 3.3 Loading and Boundary Condition

To carry out impact analysis of bumper system within framework of FEA, this is low speed impact(4kmph) and as per standards bumper should withstand it. Appropriate contacts have been defined at appropriate locations between different parts. Fig. 5 shows loads and boundary conditions. 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 5. 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.

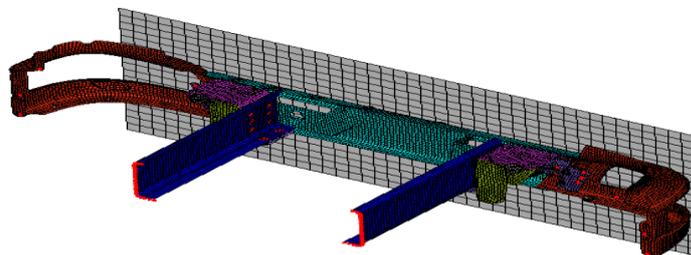


Figure 5. Constrained Bumper Model

The failure criterion to assess the bumper impact characteristics for the FEA simulation is on the basis of a fracture strain criterion. Such a criterion considers the failure of bumper when the calculated equivalent plastic strain exceeds the critical plastic strain of the material.

$$\bar{\epsilon}_p = \frac{\sqrt{2}}{3} \sqrt{(\epsilon_{p1} - \epsilon_{p2})^2 + (\epsilon_{p2} - \epsilon_{p3})^2 + (\epsilon_{p3} - \epsilon_{p1})^2} \quad (6)$$

Where, subscripts (1, 2, 3) indicate the three axes corresponding to the three principal directions. The critical plastic strain is assumed to be the percent elongation, that is, the fracture strain. The critical value is

$$\epsilon_c = 0.21 \text{ mm / mm} \quad (7)$$

## IV. RESULT AND DISCUSSION

### 4.1 Existing Bumper Model:

Explicit finite element analysis carried out for 100 millisecond and the results were recorded at 5 millisecond time interval. For the FE model of bumper makes instant contact with the impactor at a velocity of 10 mm/ms. The plastic strain results for bumper original model are as shown below

**4.1.1 Bumper Front Panel:**

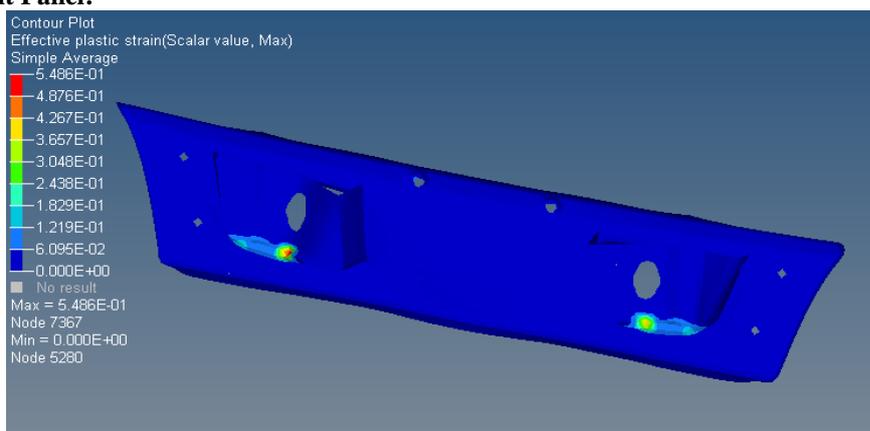


Figure 6. Contour plot of effective plastic strain values for Bumper Front panel.

From figure 6 the maximum effective plastic strain value for bumper front panel is 0.54 which is more than permissible limit hence the design for Bumper frontal panel is not safe.

**4.1.2. Bumper Side Panel:**

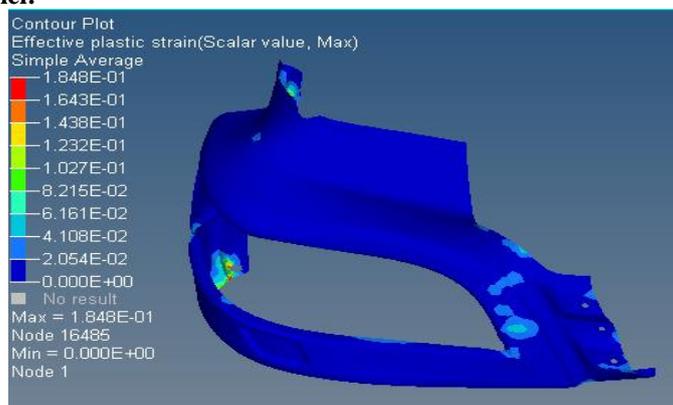


Figure 7. Contour plot of effective plastic strain values for Bumper Side panel.

From figure 7 the maximum plastic strain value for bumper Side panel is 0.18 which is less than permissible limit hence the design for Bumper side panel is safe.

**4.1.3. Bumper Bracket:**

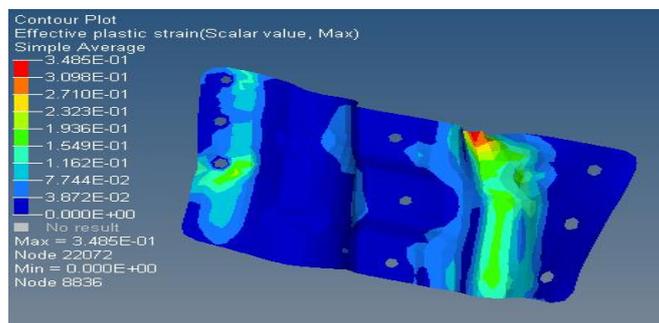


Figure 8. Contour plot of effective plastic strain values for Bumper Bracket

From figure 8 the maximum plastic strain value for bumper bracket is 0.34 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.

**4.1.4 Bumper Supporting Bracket:**

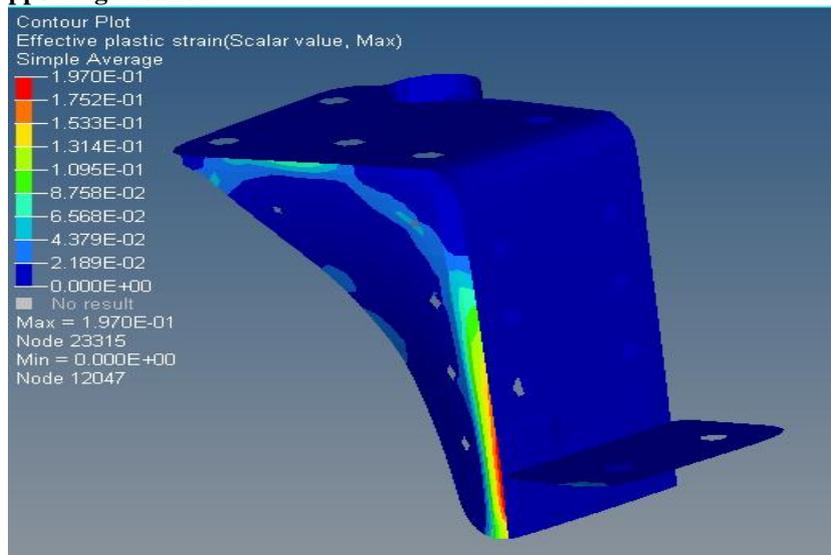


Figure 9. Contour plot of effective plastic strain values for supporting Bracket

From figure 9 the maximum plastic strain value for bumper supporting Bracket is 0.19 which is less than (0.22) permissible limit hence the design for Bumper supporting Bracket is safe.

**4.1.5 Chassis Section:**

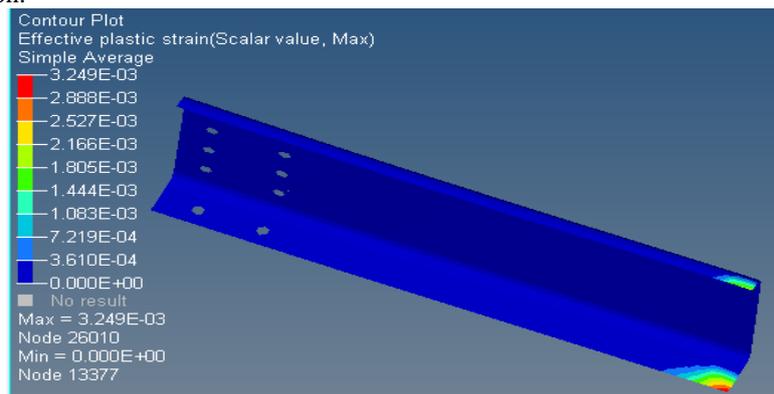


Figure 10. Contour plot of effective plastic strain values Chassis section

From figure 10 the maximum plastic strain value for bumper Chassis section is 0.0032 which is less than permissible limit hence the design for Bumper Chassis section is safe.

From above plots it has to conclude 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.

**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.
  - 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 first possible solution to achieve the plastic strain values and make the design safe.

4.2 Modified Bumper Model:

4.2.1 Bumper Front Panel

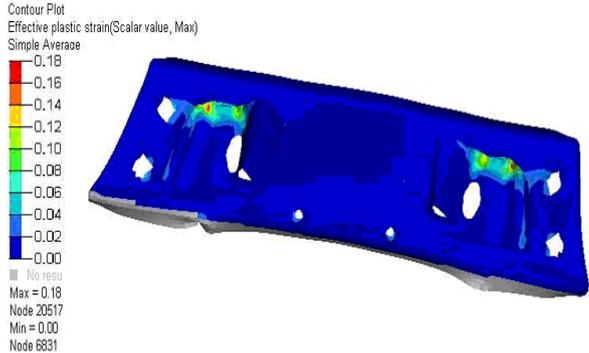


Figure 11. Contour plot of effective plastic strain values for Bumper Front panel

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.

4.2.2 Bumper Side panel:-

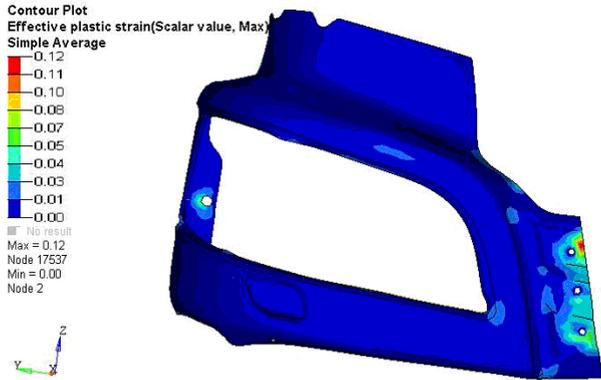


Figure 12. Contour plot of effective plastic strain values for Bumper Side panel

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.

4.2.3 Bumper Bracket:-

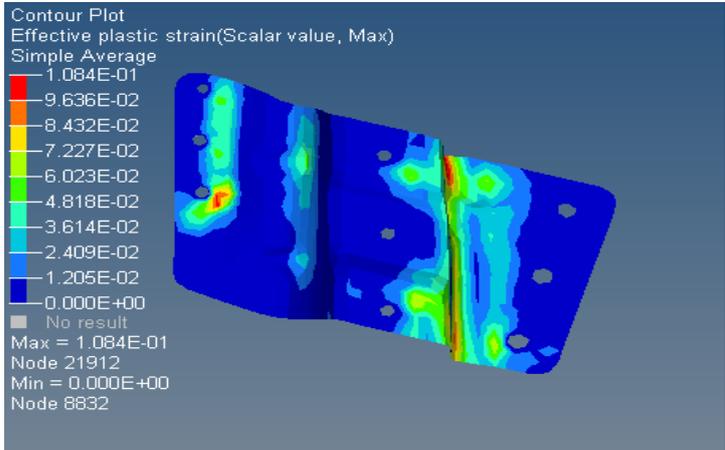


Figure 13. Contour plot of effective plastic strain values for Bumper Bracket

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.

4.2.4 Bumper supporting Bracket:-

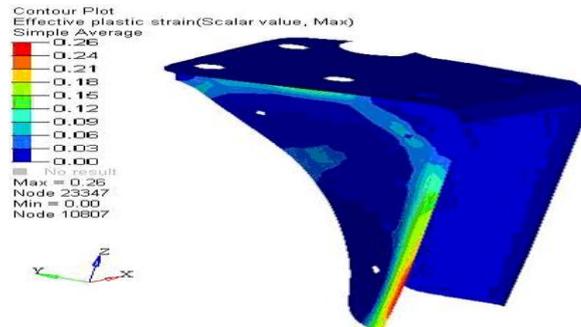


Figure 14. Contour plot of effective plastic strain values for supporting Bracket

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.

4.2.5 Chassis section:-

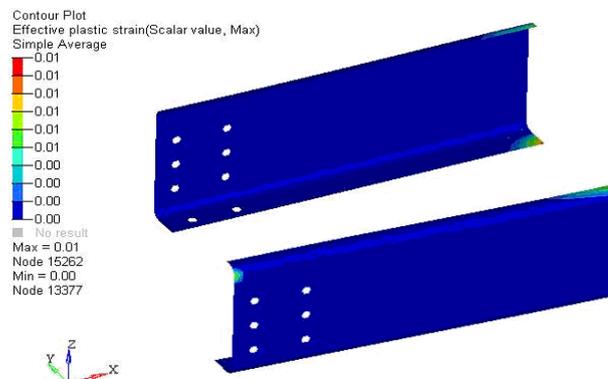


Figure 15. Contour plot of effective plastic strain values Chassis section

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.

V. CONCLUSION

From above study we come to the conclusion that the permissible strain values can be achieved by adding the stiffener to bumper components. Adding the stiffener is one of the cost effective way to get the assembly in safety zone as compared to others such as change of material or increase in thickness. 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. Adding stiffener to the bumper causes a rise in bumper rigidity increasing its strength. Consequently, it results in reduction in strain values.

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] Liqun 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. Altinok 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.

- [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.