Safety performance optimization of Vehicle Front Bumper using Finite Element Analysis

[1]Nikhil Dhanawade, [2]Mohan Patwari, [3]Mohit Bhagat, [4]Piyush Navle, [5]Meenu Parwez

[1,2,3,4]Student, [5]CAE Engineer
[1]Department of Mechanical Engineering, [5]Research and Development
[1234]Indore Institute of Science and Technology II, Indore, India, [5]IICAE

Abstract: **The bumper is one of the imperative members of vehicle safety components which are used to provide protection for passengers from front Impact. In case of impact, the bumper system will be the first which makes interaction with another collision object. Design of bumper is an important issue for the automotive makers, as it is difficult to conduct several tests. The aim of this project was to analyze and study the structure and material employed for car bumper. In this work, the most important variables like material and structures, shapes are studied for analysis of the bumper beam in order to improve the crashworthiness during a collision. In this research work, Finite Element Analysis was used to check the performance of the bumper system. LS-Dyna explicit solver was used to perform and analysis and Hypermesh was used as a preprocessor. Multiple iterations were performed with the material and design of the Front bumper beam to enhance the safety of the bumper system. Overall acceleration and resultant displacements were reported for every iteration. The final design is proposed with optimum safety behavior.**

Index Terms**: FEA, Hypermesh, LS-Dyna, Crashworthiness, Bumper.**

I. INTRODUCTION

In the current scenario, developing countries are focusing on lightweight motor vehicles with increased fuel efficiency. To meet these requirements, automobile makers are using different structures and materials to reduce the weight of the vehicle. Although these vehicles provide lighter weight, the safety of the occupants must be ensured. At times of frontal impact collisions during accidents, the front bumper is the first part to receive the collision, which is entitled to protect the car body and passenger to some extent. All types of vehicles have Bumper System, which has to absorb not only the impact forces generated during an impact but also has to provide the safety of the passengers. The Bumper System is an assembly of Plastic Fascia, Energy Absorber and Bumper Beam.

In order to guard the passengers through an accident, a structure has to be optimized based on its strength and stiffness. The structure should collapse in a well-defined deformation pattern and keep the accelerations under the limit. It is important to know how much energy is absorbed.

In this research work, an attempt is made to study the crashworthiness of the existing Bumper System. Whenever an impact between two vehicles occurs, Bumper system will be the first component which will receive the crash forces (Figure 1). 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 research deals with the amount of energy absorbed by the Bumper System and an overall reduction in vehicle acceleration.

Figure 1: Bumper system of a car

Many pieces of research were performed to characterize the crash resistance of different materials based solely on energy absorption. Composite bumper material was analyzed by Kleisner et al., in 2009 [1]. Bumper performance at low speed was analyzed by Heon et al., in 2003 [2].

Hosseinzadeh RM et al. [3] has made an attempt to analyze the structure, shape and impact condition of an automotive bumper system made up of glass mat thermoplastic (GMT) using the FEA software LS-DYNA. Evans D and Morgan T [4] have suggested the use of Expanded Polypropylene (EPP) foam techniques to manufacture bumper beam. Butler M et.al [5] stated that it is necessary to increase the impact energy absorbing capacity of the lightweight vehicles by using various new techniques along with the materials of the components which are associated with the energy-absorbing unit. These increasing demands have led to a growing interest in the utilization of high strength stainless steels. Witteman WJ [6], automotive manufacturing companies are trying to reduce the overhang, greater sweeps and building of the compact bumper system.
In this paper bumper beam materials and shapes were modified to achieve the optimal safety. Linear Quadrilateral and Linear Triangular Shell Elements were 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.

II. METHOD

Physical testing for crashworthiness of vehicle would be destructive and expensive in nature. Even testing on a reduced scale or on a selected component basis, it would still be an experimental approach to the design and validation of an optimized crashworthy bumper. FEA would provide a way to carry out several iterations in a minimized cost.

At first, the CAD model of the bumper system is prepared. The overall purpose of the bumper system is to absorb the impact energy and sustain the loads by impacting object. The key areas for modification are identified. The main task in this study is to design the bumper system with sufficient strength and optimizing it for a various design modification. The 3-Dimensional model of the bumper system is prepared. Different design modification is done and analysis is carried out using finite element analysis software named Hypermesh and Optistruct.

The FEA is a computerized simulation of a real-time mechanical system (geometry and loading environment). Works with the mathematical assumptions relating to the physical system. The elements are the discretized unit of the geometry converting infinite solution to a finite number. The elements are bounded by nodes for which the calculations are made.
This impact analysis is a case of transient dynamic analysis that is used to solve the dynamic response of a certain structure under time-dependent load. Time-varying load (like displacement), strain, stresses and force are analyzed in this method.
It responds to any combination of static, transient and harmonic loads.

Figure 2 shows the computer-aided design of the base model bumper system. It mainly consists of Beam and crush canes. Figure 3 shows the meshed model for the assembly. This bracket has been assigned to various design modification by adding stiffeners to the base design. FINITE ELEMENT ANALYSIS is carried out by using Hypermesh and LS-Dyna. The mass of the overall vehicle in the laden condition is lumped to the bumper and full rigid 56kmph impact scenario was simulated. The model setup of the baseline design is shown in figure 4.

Non-linear material was used to model the bumper assembly. MAT 24 available in LS-Dyna was utilized to capture the non-linear behavior of the material. The detailed properties are shown in Table 1.

Figure 2: CAD Model of bumper system

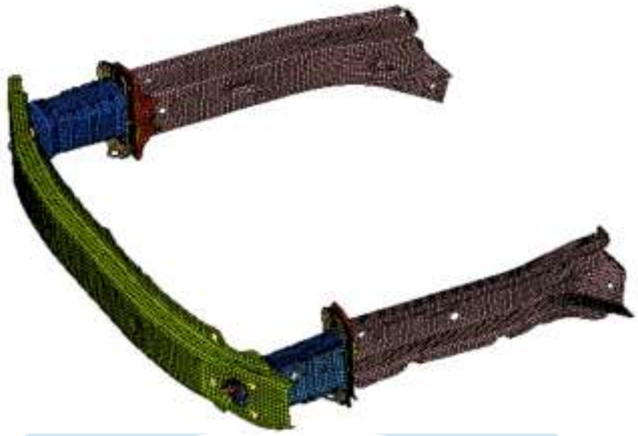


Figure 3: FEA Model of bumper system

Density	7.89e-009 tonn mm^{-3}
Youngs modulus	200000N mm^{-2}
Poisson's Ratio	0.3
Yield Strenght	270N mm^{-2}

Table 1: Mat 24 properties

The thickness of the different part of the assembly is provided according to the specification. The bumper is processed with loads to collide with a fixed rigid wall.

To provide an interface between the components of this impact simulation, the "Automatic-Surface-Surface" type contact is used. The Automatic-Single-surface contact is defined for the interface by creating sets of bumper beam, crush cans and brackets. The FEA model characteristics of each component in the original modelling are shown in Table 1

The simulation was carried out as a low-speed collision condition in which the bumper with a mass of 1200 kg collided with the rigid wall at a speed of 56 kmph as stipulated. The impact model was processed by LS-DYNA solver. Different iterations were made to improve the effective crashworthiness of the bumper. Iterations like material, thickness and structural were done in order to get the optimized result for the research.

Bumper Beam Material

The thickness selection plays a vital role in defining the impact performance of the vehicle. Proper selection can minimize the weight and can provide a fuel-efficient performance without harming the strength of the vehicle. Weight plays the key factor in deciding the fuel consumption of the vehicle. Around 10 per cent reduction in weight could result in 6-8% decrement in fuel consumption.

From the references, it was concluded that materials with different thicknesses can give different performances. So here in this simulation different thicknesses were assigned to the component to define the effect of thickness in impact performance of the crumble zone.

Thickness like 4mm, 2mm, 1.8mm. 1.2mm as shown in table 3 were assigned.

To objectify the importance of thickness in case of impact behavior, different specification of materials were assigned to the bumper system in different impact models. Mechanical specifications of used materials are listed in Table: 2, 3, 4 &5

Component	Old thickness	Update
Bumper	4mm	-
Crush-can	2.5mm	-
Crush-can support rail	2.5mm	-

Table 2: thickness for Base model)

Component	Base thickness	Update
Bumper	4mm	2.3mm
Crush-can	2.5mm	2.5mm
Crush-can support rail	2.5mm	2.5mm

Table 3: iterations 1

Component	Base thickness	Update
Bumper	4mm	2.3mm
Crush-can	2.5mm	2.5mm
Crush-can support rail	2.5mm	1.3mm

Table 4: iteration 2

Component	Base thickness	Update
Bumper	4mm	2.3mm
Crush-can	2.5mm	1.8mm
Crush-can support rail	2.5mm	1.3mm

Table 5: iteration 3

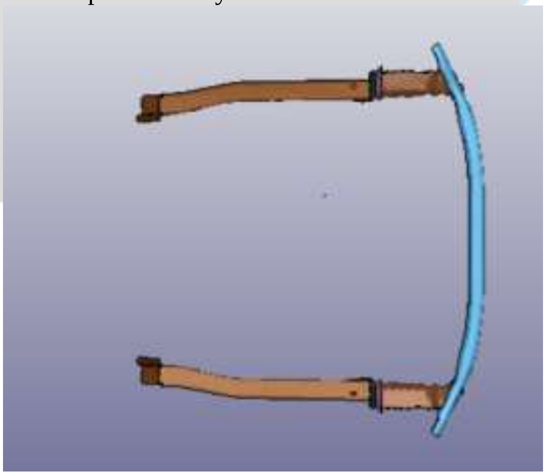
Different sets of thickness were tested resulting in substantial improvement, like the vehicle-weight, optimized stiffness, fuel efficiency. Such material, strengthening the structure could possibly help in gaining axial crushing and pure bending. Overall increasing the energy absorption and increasing the crash-worthiness of the vehicle.
According to the study of energy curves during a collision, it is observed that the kinetic energy decreases and the internal energy increases. This represents that the energy is absorbed by the bumper system. Then in the second phase, it is observed that the kinetic energy increases and the internal energy increase this indicates the spring-back of the vehicle after the collision. However total energy remains constant.

Now the energy curves along with deformation curves can be analyzed to study the impact behavior of used material.
Another parameter that can be analyzed is impact acceleration that can help to relate the impact force felt by the passenger during the course of a collision. The acceleration vs time graph is plotted for a certain node to study the differences for impact behavior in different materials.

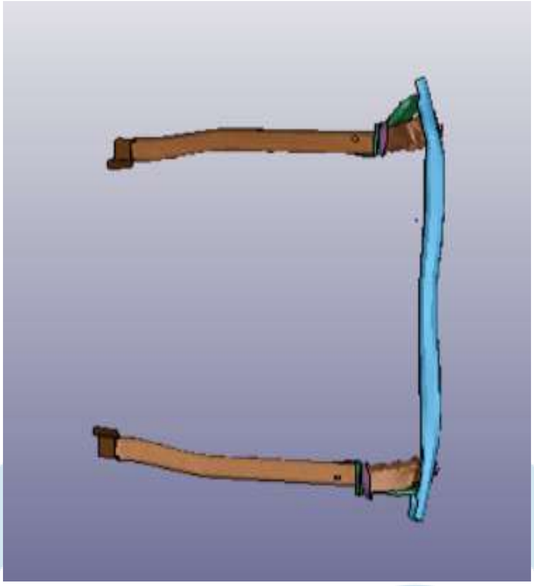
It was observed that the rate of deformation decreases with an increase in thickness, but the value of acceleration increases with an increase in thickness.
The increase in acceleration can increase the impact force felt by the passenger, so it is not necessary to choose a thicker material. The objective is to conclude a lightweight without scarfing the impact behavior. So choosing a thickness with less deformation and less acceleration will be the possible way to gain an optimized result.

III. RESULTS

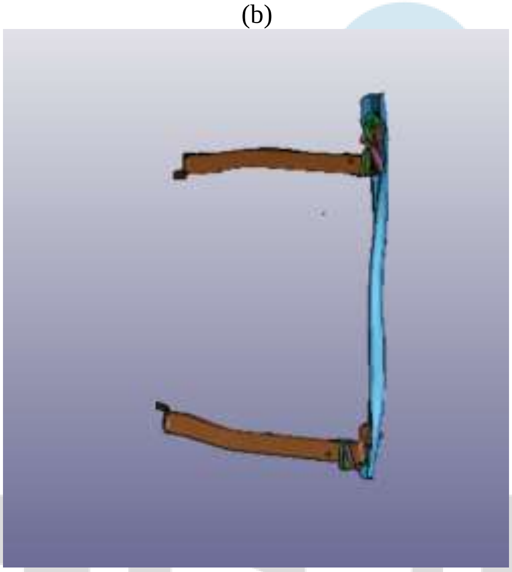
1. Base model simulation
   The bases model impact simulation of the bumper assembly



(a)

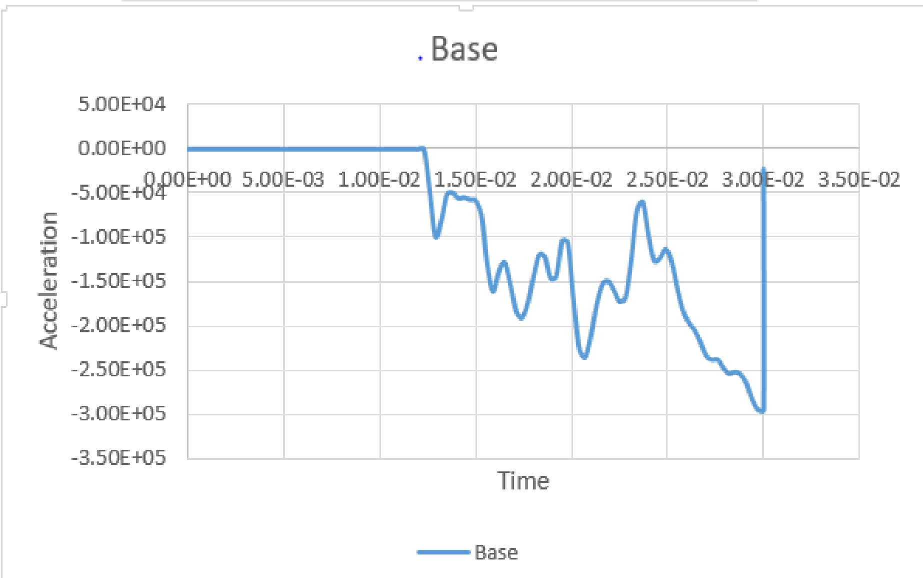


(b)



(c)

Figure 4 : Deformstion of bumper system of design iteration 1 (a) Initial condition, (b) at middile stage and (c) Final deformed stage

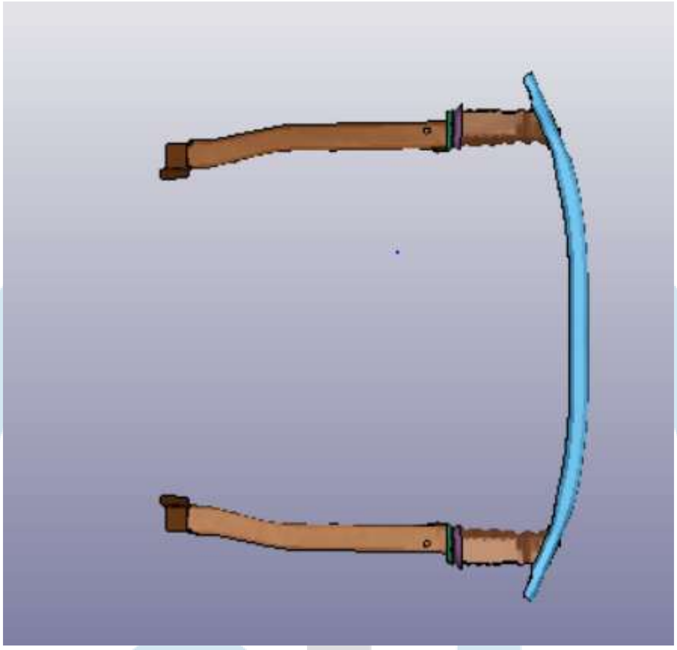


Graph 1 : acceleration –time curve

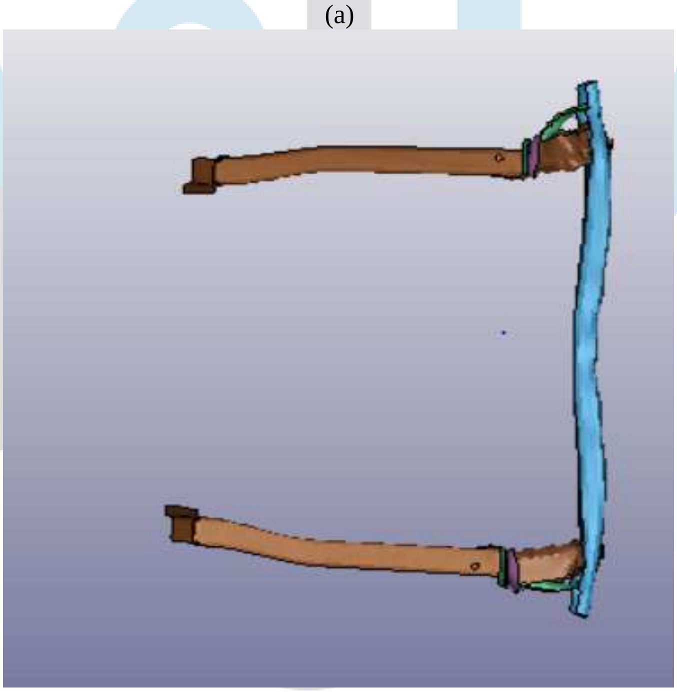
It was observed that the peak of acceleration was very high, so to control the peak value iteration were made. From the simulation it was necessary to increase the axial crushing and pure bending. Overall increasing the energy absorption and increasing the crash-worthiness of the vehicle.

2. Iteration 1

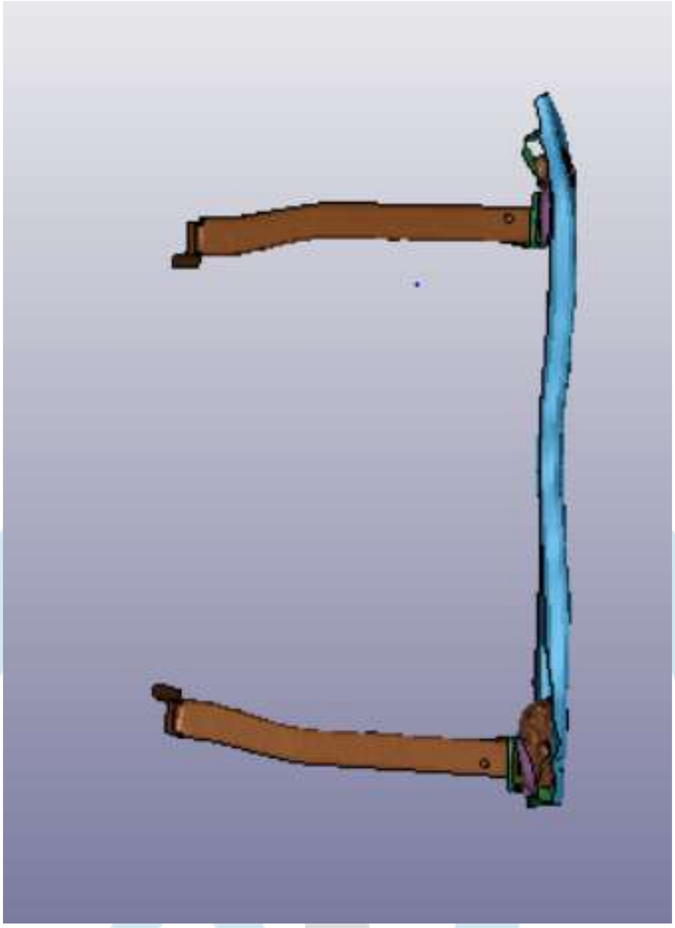
Here the thickness of bumper was changed from 4mm to 2.3mm in order to absorb more energy.



(a)

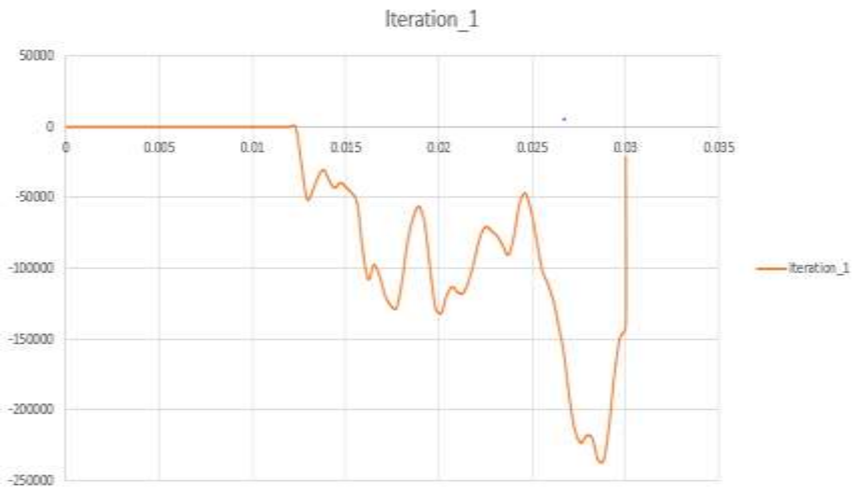


(b)



(c)

Figure 5: Deformstion of bumper system of design iteration 1 (a) Initial condition, (b) at middile stage and (c) Final deformed stage

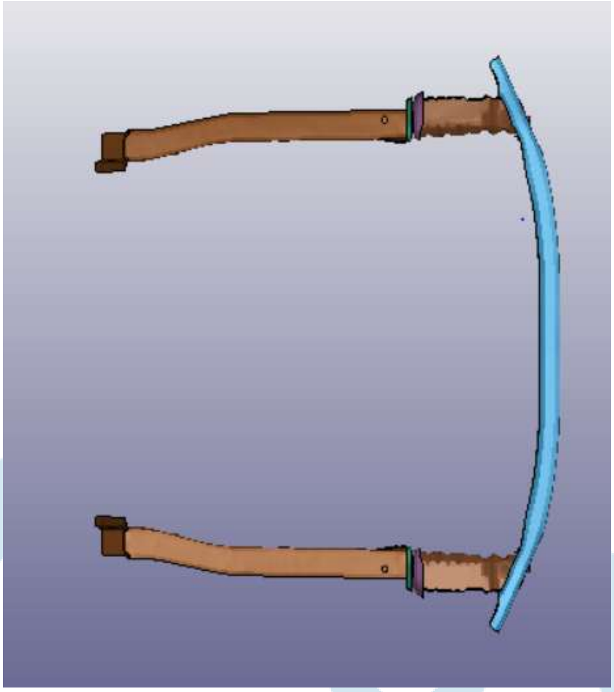


Graph 2: Acceleration- time curve

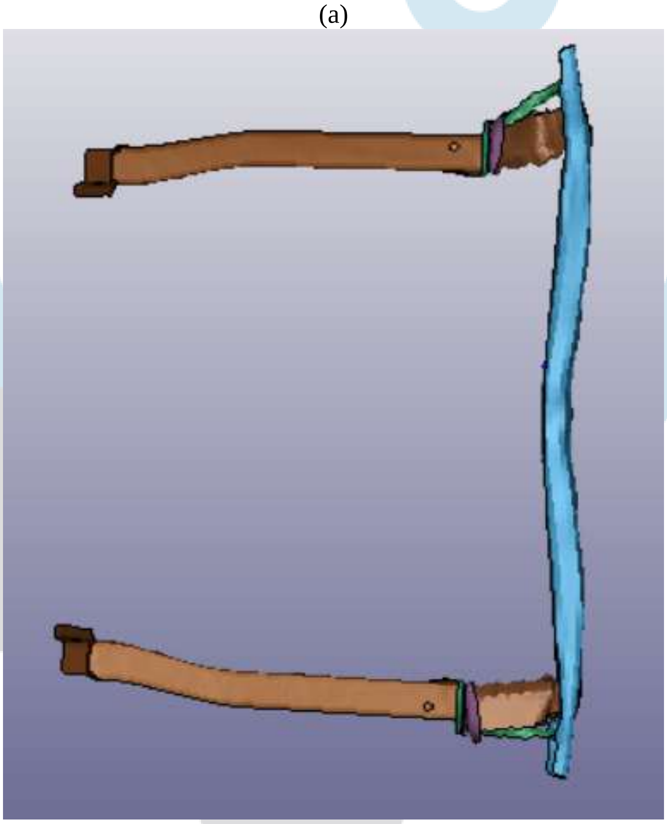
Simulation result showed a varied difference in the acceleration peak but it was felt that there were chances to increase the axial crushing of the crash cans .

3. Iteration 2

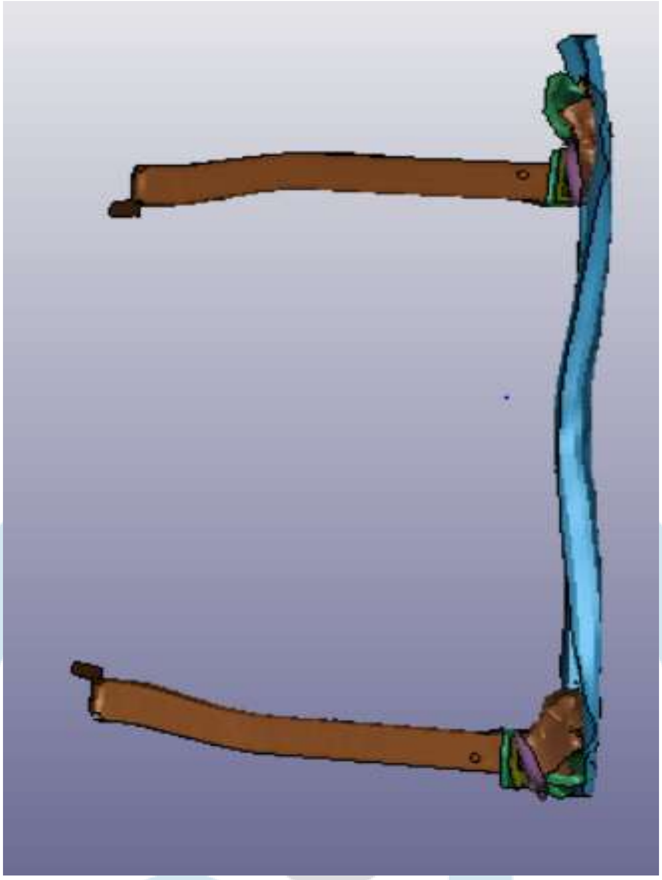
In iteration 2 the crush-can supportrail thickness was updated from 2.5mm to 1.29 mm in order to increase the axial crushing.



(a)

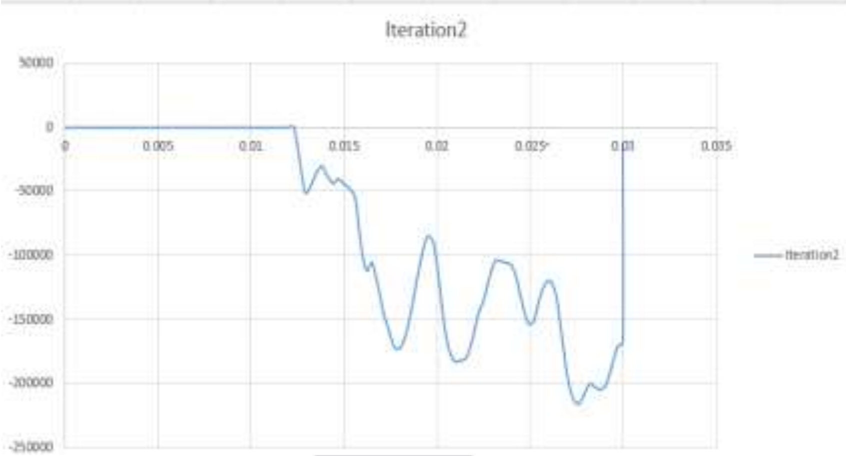


(b)



(c)

Figure 6: Deformstion of bumper system of design iteration 1 (a) Initial condition, (b) at middile stage and (c) Final deformed stage

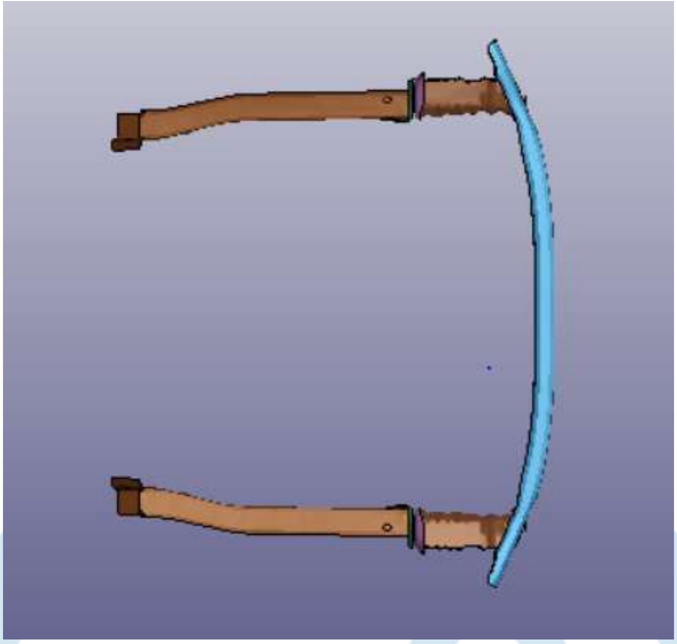


Graph :3 Accelration-time curve

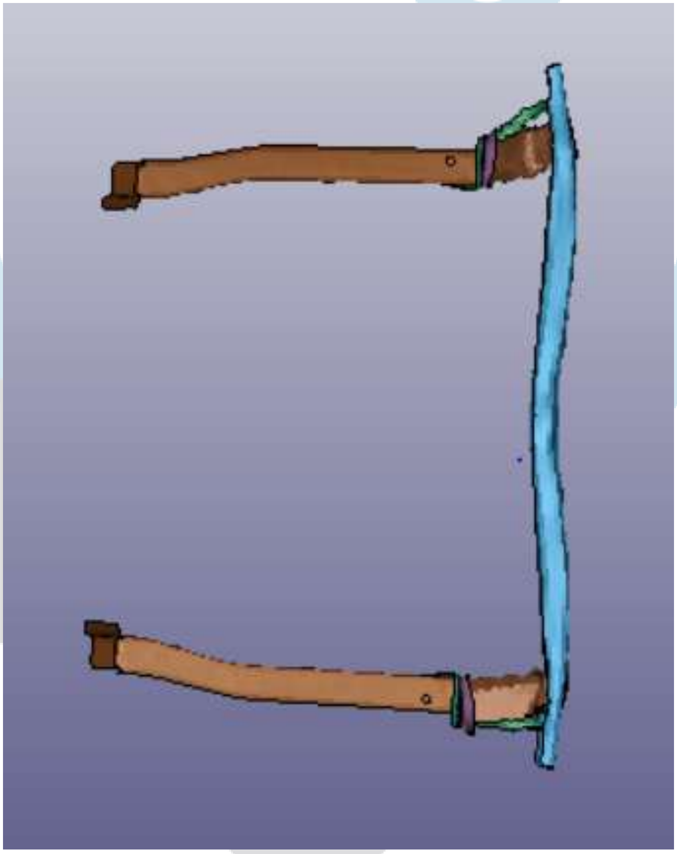
Simulation result shows us the decreament in the peak valvue of acceleration but it was realised that, the crush cans were not properly showing the axial crushing so the thickness of crush can was updated.

4. Iteration 3

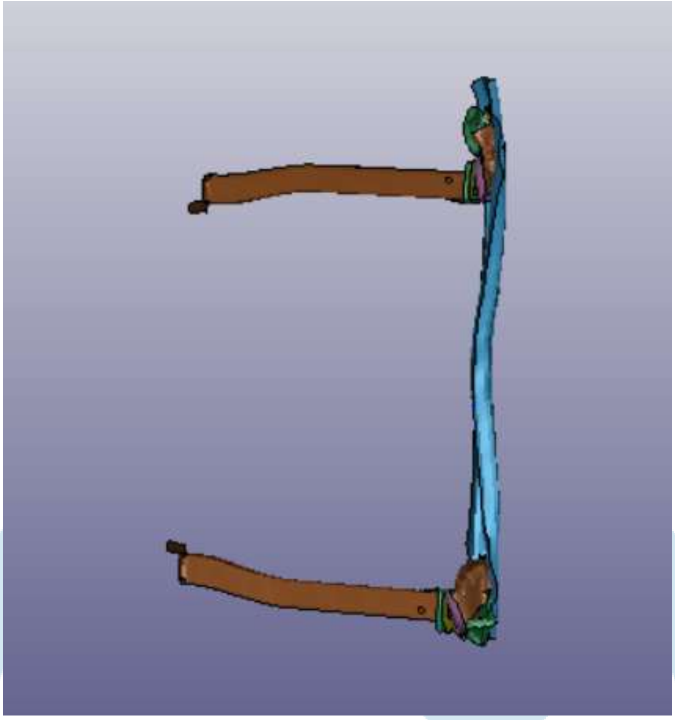
So from the previous iterations it was clear that the chances of increasing the crash worthiness was from iterating the thickness of crush cans. So in this iteration 3 the thickness of crush cans were updated from 2mm to 1.8mm.



(a)

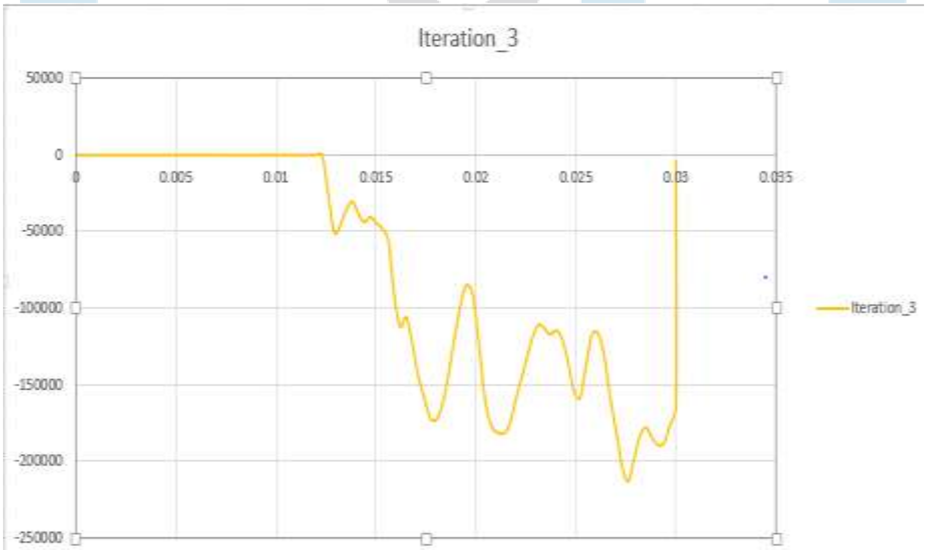


(b)



(c)

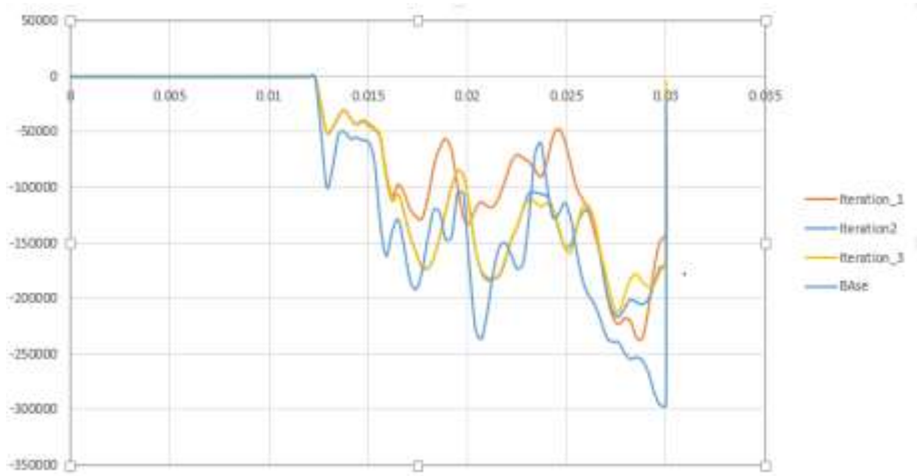
Figure7 : Deformstion of bumper system of design iteration 1 (a) Initial condition, (b) at middile stage and (c) Final deformed stage



Graph :4 Accelration-time curve

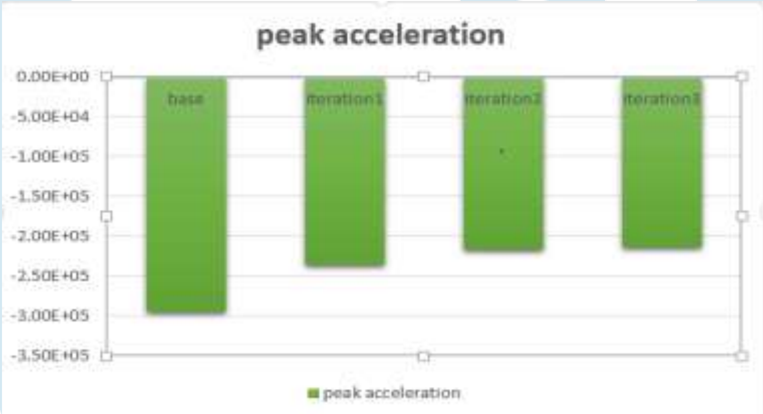
IV. CONCLUSION

After comparing the results obtained from different iterations, the acceleration-time curve was plotted in ls pre post and then the graphs were compared. It was found that the iteration 3 shows us the least value of acceleration peak , thus it would help in getting least impact force at the time of crash .



Graph: 5 Acceleration-time curve comparison

So from changing the thickness distribution of the bumper assembly, the increase in crashworthiness performance of the bumper system is achieved. Even making it light weight would make the vehicle safer and fuel economic.



Graph 6: Difference in peak acceleration

REFERENCES

[1] V. Kleisner, R. Zemck, "Analysis of composite car bumper reinforcement," applied and computational mechanics. vol. 3, pp. 287-296, 2009.

[2] Heon Y. Kim, S. B. Kim, and al. "A Study on the Design of Crash Zone for RCAR Low-Speed Impact." Symposium, pp. 131-135, 2003.

[3] 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.

[4] Evans D and Morgan T, "Engineering Thermoplastic Energy Absorbers for Bumpers", SAE Paper, 1999.

[5] Butler M, Wycech J, Parfitt J, and Tan E, "Using Tero core Brand Structural Foam to Improve Bumper Beam Design", SAE Technical Paper, 2002,

[6] Witteman WJ, "Improved Vehicle Crashworthiness Design by Control of the Energy Absorption for Different Collision Situations", Doctoral dissertation, Eindhoven University of Technology, 2000.