Bumper Design Enhancement through Crash Analysis

Katkar A.D.¹, Dr. Bagi J.S.²,
¹ M.E. CAD CAM CAE, KIT’s College of Engineering, Kolhapur, India
² Prof. Dept. of Production Engineering, KIT’s College of Engineering, Kolhapur, India

ABSTRACT

Bumper beam is one of the key structural parts, which plays an important role in the frontal crashes of automobile. In this paper, the most important design parameter like thickness is studied for design and analysis of an automotive front bumper beam to improve the crashworthiness design in low-velocity impact. The simulation of original bumper under condition impact is carried out according to the low-speed standard of automotive stated in E.C.E. United Nations Agreement, Regulation no. 42, 1994. In this research, a front bumper beam made of three different thicknesses: 4mm, 4.5mm and 5mm are studied by impact analysis to determine the deflection and plastic strain induced in the bumper beam. The mentioned characteristics are compared to each other to find best choice of Bumper thickness.

Keywords— crash, optimization, simulation

1. INTRODUCTION

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 [7]. Bumper systems are designed to prevent or reduce physical damage to the front or rear ends of passenger motor vehicles in collision condition. A good design of car bumper must provide safety for passengers and should have low weight [2].

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.

Efforts were taken to model the bumper as similar as possible to reality. Thus, the computer aided design (CAD) data of the bumper was imported directly into Hypermesh (pre-processor) and meshed in order to make a precise model. The modelling was done according to the conditions stated in E.C.E. United Nations Agreement, Regulation no. 42, 1994 [8]. Fig. 1 shows the schematic diagram of a low-speed impact test [2]. According to these conditions, the car should be placed on a flat surface with released gear and brake and impacted both from front and side directions [10]. Since the real low-velocity test stated in the agreement requires laboratory equipment, simplifications were assumed to make finite element modelling possible. The consequence of adopting this concept, however, is that when the bumper is impacted by a stiff object, such kind may happen in a parking accident or in the legislative low-speed impact pendulum test [2].

2. CRASHWORTHINESS

Crashworthiness is a measure of the vehicle’s structural ability to plastically deform and yet maintain a sufficient survival space for its occupants in crashes involving reasonable deceleration loads.

Structural design for crashworthiness seeks to tone down two adverse effects of a crash i.e.

- Rapid deceleration of the occupant compartment
- Crush of the occupant compartment survival space

Figure 1 Low Speed impact test.
In case of severe crash, the speed of a vehicle often decreases from its travel speed to zero in a hundred thousandths of a second. One important way to minimize the injury consequences of this abrupt change in velocity is to extend the amount of time necessary to slow the vehicle down – the less abrupt the change in velocity, the lower the crash forces on the occupant. The front end of vehicles is designed to crumple in a controlled manner in a collision to give their occupants the necessary additional time to safely decelerate in a crash. The controlled crush or crumple of the front-end, a safety positive feature, is totally different from the crush or collapse of the actual occupant compartment – which is to be avoided. At a minimum, partial collapse of the structural cage, which surrounds the occupant, allows vehicle parts (e.g., the engine or steering mechanism) to “intrude” into the occupant space and strike the occupant causing injury. In extremely severe collisions, the occupant compartment may suffer a tragic collapse, and allow the occupant to be crushed.

So the structure of a vehicle should be designed to efficiently absorb the kinetic energy in case of vehicle crash. To improve the crashworthiness of a vehicle, the load carrying capacity and collapse mode has to be estimated at the initial stage of design. As a conservative design, collapse behaviour has to be such that the front part of a car deforms severely with absorption of most impact energy while there is little deflection toward the passenger room.

The following requirements needed for crashworthiness are as follows:

- The vehicle structure should yield a deceleration pulse that satisfies the following requirements for a range of occupant sizes, ages and crash speeds for both genders
- Deformable, yet stiff, front structure with crumple zones to absorb the crash kinetic energy resulting from frontal collisions by plastic deformation and prevents intrusion into the occupant compartment, especially in case of offset crashes and collisions with narrow objects such as trees, short vehicle front ends presents a challenging task to the crashworthiness engineer.
- Deformable rear structure to maintain integrity of the rear passenger compartment and protect the fuel tank.
- Properly designed side structures and doors to minimize intrusion in side impact and prevent doors from opening due to crash loads.
- Strong roof structure for rollover protection.
- Properly designed restraint systems that work in harmony with the vehicle structure to provide the occupant with optimal ride down and protection in different interior spaces and trims.

In spite of the tremendous progress achieved in crashworthiness simulations of vehicle structures from components to full-scale vehicles, using the latest techniques in computational mechanics and super computers, final crashworthiness assessment still relies on laboratory tests. This is especially true in vehicle certification.

3. FRONT BUMPER

In the most automobile accidents, the first crash part is bumper system [7], which comprises three main parts: bumper skin, foam and bumper beam assembly. As a major crash safety structural part of bumper system, bumper beam assembly has an important influence on the frontal crash of automobile, and its performance is directly related to protective effect for occupants and car body [7].

When traveling automobile has an impact on easily deformed barriers such as trees, poles, etc., the damaged condition is determined by the static stiffness performance of bumper beam [7]. According to three-point bending test, static press simulation model of bumper beam is established and the obtained peak force can be used for evaluating static stiffness performance [7]. The absorber is fixed to an installation bench, and a constant displacement load produced by a rigid indenter is applied to the middle of the bumper beam.

When automobile is in a low speed state (speed is usually less than 10 km/h) such as starting, stopping and reversing, etc. [7], the front and rear of it have an impact on fixed barriers, and the damaged condition of car body parts (such as fender, radiator support and engine hood, etc.) and maintenance costs are determined by the low speed performance of bumper beam. The low speed crash simulation model of bumper beam was established, bumper beam assembly is fixed on the rigid wall [7], then driving an impactor with the mass of 1200 kg to impact the middle of bumper beam at the speed of 4 km/s.

In the frontal crash of automobile at a moderate speed (10~20 km/s), it is generally desirable that bumper beam assembly can absorb all of the collision energy to minimize the damage of car body; moreover, when crash speed exceeds 20 km/h, the bumper beam cannot crack so that it can provide a stable power transmission to ensure continuous deformation of engine cabin [7].
The aim of this work is to study front bumper of one of the existing passenger car in Indian market. 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 have focused on modifying few of above stated parameters to suggest improvements in existing bumper of passenger car present in Indian markets. Based on observations, design improvements have been suggested. Modified front bumper design was tested using FEM software for deflection and Plastic strain. Results of modified bumper have been compared against existing design.

FEM is backbone of today’s automotive industry. In recent times FE analysis is widely used to validate the complex designs like bumper. Use of FEA not only reduces product development time but also saves lot of cost. Hence, this work proposes FE analysis of bumper to validate the design modifications in from bumper of car.

4. VEHICLE SAFETY STANDARDS

The National Highway Traffic Safety Administration (NHTSA) strives to establish test procedures in regulatory requirements that lead to improvements in real world safety, often in connection with performance standards. According to the Law, the entire vehicle must meet and pass certain safety tests before they are sold but legislation provides a minimum statutory standard of occupant safety for new cars. To make the standard, experimental crash tests are conducted under the governing norms of the National Highway Traffic Safety Administration (NHTSA) and Insurance Institute of highway Safety (IIHS), two US government bodies, and EU Directive of the European Union.

The below standards that has helped in reducing causalities in case of full frontal crashes are

- Federal motor vehicle safety standard (FMVSS) 208
- EU Directive 96/79 EC
- United Nations - E/ECE/TRANS505 Addendum 41, Regulation No.42 [10]
- TP-581-01 [9]

4.1 UNITED NATIONS - E/ECE/TRANS505

The purpose of this test is to simulate frontal and rearward low speed impact conditions with another vehicle [10]. The test shall make it possible to verify whether the protective devices of the vehicle meet the requirements of this Regulation.

The test area shall be large enough to accommodate the impactor (striker) propulsion system and to permit after-impact displacement of the vehicle impacted and installation of the test equipment. The vehicle shall be placed on a horizontal and level rigid smooth surface [10].

The vehicle shall be at rest. The front wheels shall be in the straight-ahead position. The tyres shall be inflated to the pressure recommended by the vehicle manufacturer [10]. The brakes shall be disengaged and the transmission control shall be in neutral position. Vehicles equipped with hydro pneumatic, hydraulic or pneumatic suspension or a device for automatic levelling according to load shall be tested in the normal running conditions specified by the manufacturer [10].

The impactor shall be of rigid construction, the impact contour being of hardened steel. The impacting surface shall conform to the diagram in the figure. The effective mass shall be equal to the mass corresponding to the “unladen weight” of the vehicle to be tested [10]. With plane A of the impactor vertical, the reference line shall be horizontal. The first contact of the impactor with the vehicle shall be by the impact contour on the protective device. The reference height is 445 mm. The impactor may either be secured to a carriage (moving barrier) or form part of a pendulum [10].

Longitudinal impact test:

- This test consists of two impacts on the front surface and two impacts on the rear surface of the vehicle. On each surface one impact is made with the vehicle under "unladen weight" [10].
- For the impacts on the front and rear surfaces, the choice of location of the impactor for the first impact is free, but for the second impact the median plane of the impactor shall be at a distance at least 300 mm from the first, provided that during these impacts the extremities of the impactor do not pass outside a zone defined by two planes parallel to the longitudinal median plane and passing through the corners of the vehicle [10].
- The impactor shown in the figure should be placed so that plane A is vertical and the reference line is horizontal at the reference height of 445 mm [10].
- The vehicle should be aligned so that a point between the vehicle corners touches, but does not move, the impactor, the longitudinal median plane of the vehicle being perpendicular to plane A of the impactor [10].
- The vehicle should be impacted at a speed of 4 km/h [10].
5. MODEL CONSTRUCTION AND SOLUTION
TECHNIQUES

5.1 CAD MODELLING FRONT BUMPER

The CAD modelling of the Front Bumper was carried out using CATIA V5.

Baseline Bumper design was 4.0 mm which is assigned to FE model.

5.2 FINITE ELEMENT MODELING

The Finite Element model of Front Bumper structure is created using Hypermesh-11.0. Here the surface model of the Front Bumper is imported to the Hypermesh environment. Using SHELL element with an element size of 5 creates the finite element model of the Front Bumper structure. The ratio of surface area to the wall thickness is less than 5 and hence SHELL elements are preferred for creating the FE model of the Front Bumper structure which is as shown in the Figures 3 to 5. In this case a finer mesh is preferred as to obtain better accuracy of the results. The thickness of the Baseline Bumper design was 4.0 mm which is assigned to FE model.

Figure 3 Finite Element model of Front Bumper

Figure 4 Finite Element model of Structure.

Figure 5 Finite Element model of Baseline Car

Here the analysis involves the understanding of the minute parameters of the original model like dimensions, material properties and structural characteristics. The overall goal of the model is to be
computationally effective and therefore the front portion of the vehicle where the front bumper is located and its neighbouring parts were meshed with fine mesh. Main area of interest is in the crashworthiness and the displacement of the front bumper structure.

The FE model of the Front Bumper is connected to mounting brackets through bolts. The bolting is represented by rigid links. Then the mounting bracket is arc welded to base structure using shell elements. These connections are shown in the figure 4. Thus the bumper is connected to base structure through mounting brackets and load path is developed. The impact force gets transferred from bumper to base structure via mounting brackets. All these assembly task are done in the Hypermesh environment.

After assembling the front bumper structure, next step is to assign the material to the front bumper structure. LS-DYNA is having a wide range of material and equation of state model, each with a unique number of history variables. From the material library material model named MAT_PIECEWISE_LINEAR_PLASTICITY [8] is used for the Front bumper structure. A brief description of this particular material model is given below:

**MAT_PIECEWISE_LINEAR_PLASTICITY (Mat type 24)**

This is an Elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency [8]. The stress-strain relation for this material is defined with stress-vs.-strain points. With these material models, failure based on a plastic strain or a minimum time step size can be defined. In the FE model, this material is primarily used for the front bumper.

<table>
<thead>
<tr>
<th>Material properties / parameters</th>
<th>Units / values used</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length unit</td>
<td>Millimetre</td>
</tr>
<tr>
<td>Time unit</td>
<td>Seconds</td>
</tr>
<tr>
<td>Mass unit</td>
<td>Tonne</td>
</tr>
<tr>
<td>Force unit</td>
<td>Newton</td>
</tr>
<tr>
<td>Young’s modulus of Plastic</td>
<td>2500 N/mm²</td>
</tr>
<tr>
<td>Density of Plastic</td>
<td>1.05e-09 t/mm³</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.4</td>
</tr>
<tr>
<td>Acceptable Plastic Strain (%)</td>
<td>15%</td>
</tr>
<tr>
<td>Young’s modulus of steel</td>
<td>210e+03 N/mm²</td>
</tr>
<tr>
<td>Density of steel</td>
<td>7.89e-09 t/mm³</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.3</td>
</tr>
<tr>
<td>Velocity equivalent at 10 kmph</td>
<td>2.7mm/sec</td>
</tr>
</tbody>
</table>

Table 1 Material properties of Plastic and Steel assigned for section shell

5.3 DEFINING RIGID WALL

In this simulation the barrier is constructed ‘rigid’ with all degrees of freedom constrained at each node. The rigid wall is created using shell element. The material assigned for the rigid wall is MAT_RIGID [8] (Mat type-20). The parts made from this material model are considered to belong to a rigid body. This material model provides a convenient way of turning one or more parts comprised of beams, shells or solid elements in to a rigid body. Rigid bodies do not undergo any deformation. Figure 6 shows the Rigid wall and bumper structure.

5.4 BOUNDARY CONDITIONS

Boundary conditions includes defining contacts, constraints etc. In this case contact between rigid wall and bumper are defined. Contact type used in LS-DYNA for crashworthiness application is CONTACT_AUTOMATIC_SINGLE_SURFACE. In LS-DYNA [8], a contact is defined by identifying what locations are to be checked for potential penetration of a ‘slave’ node through a ‘master’ segment. As presently implemented, one surface of the interface is identified as a master surface and the other as slave. By using number of different algorithm, a search for penetration is carried out during the execution of solution.

In general, an input for the contact-impact algorithm, only the slave surface is defined and each node in the surface is checked each time step to ensure that it does not penetrate through the surface. Internal logic identifies a master segment for each slave node and a slave segment for each master node and updates this information every time step as the slave and master nodes slide along their respective surfaces.

6. SIMULATION & RESULT DISCUSSION

6.1 SIMULATION OF FRONT BUMPER – BASELINE DESIGN

In this simulation the finite element model of Baseline Bumper design is crashed against a rigid
The Bumper is impacted at a velocity of 2.7 m/s. The Figure 7 to 8 shows the setup for the simulation of front Bumper impact test. The simulation of this crash event was run for a minimum of 100 milliseconds using explicit finite element solver LS-DYNA Program Manager. The simulation of this crash event can be viewed or post-processed using LS-POST or by means of Altair Hyperview/Hypergraph.

**RESULT DISCUSSION – BASELINE DESIGN (4MM THICK)**

The below figures show displacement, and plastic strain of crash simulation of frontal impact of Baseline Bumper into a rigid wall. The Figures 9 shows Displacement Plot of crash simulation of frontal impact of Baseline Bumper into a rigid wall. The maximum displacement is 103.1 mm. Here we can see that the Bumper has been completely collapsed.

The Figures 10 shows Plastic strain Plot of crash simulation of frontal impact of Baseline Bumper into a rigid wall. The maximum plastic strain observed is 19%. This is above the acceptable breaking plastic strain of 15%.

**6.3 SIMULATION & RESULT DISCUSSION - DESIGN ITERATION#1**

As the analysis results for Bumper are still above the acceptable limits, the thickness of the Bumper is now increased to 4.5 mm and similar frontal impact analysis is carried out for new design.

The below figures show displacement and plastic strain of crash simulation of frontal impact of Baseline Bumper into a rigid wall. The Figures 11 shows Displacement Plot of crash simulation of frontal impact of Design Iteration#1 Bumper into a rigid wall. The maximum displacement is 95.4 mm. Here we can see that the Bumper displacement is lower than the Baseline design.

The Figures 12 shows Plastic strain Plot of crash simulation of frontal impact of Design Iteration#1 Bumper into a rigid wall. The maximum Plastic strain observed is 17%. Here also we can see that the plastic strain value for Design Iteration#1 Bumper is lower than the Baseline design but it is still above the acceptable breaking plastic strain of 15%.
6.4 SIMULATION & RESULT DISCUSSION - DESIGN ITERATION#2

To improve the Crashworthiness performance of the Front Bumper, its thickness is further increased to 5.0 mm and similar Frontal Impact analysis is carried out for new design.

The below Figures shows Displacement and Plastic strain of crash Simulation of frontal impact of Baseline Bumper into a rigid wall. The Figures 13 shows Displacement Plot of crash Simulation of frontal impact of Design Iteration#2 Bumper into a rigid wall. The maximum displacement is 86.2 mm. Here we can see that the Bumper displacement is lower than the Baseline design.

The Figures 14 shows Plastic Strain Plot of crash Simulation of frontal impact of Design Iteration#2 Bumper into a rigid wall. The maximum Plastic strain observed is 14 %. Here also we can see that the plastic strain value for Design Iteration#2 Bumper is lower than the Baseline design and it is below the acceptable breaking plastic strain of 15%. Hence Design Iteration#2 Bumper design is passing the analysis criteria and the bumper design is structurally adequate.

6.5 ANALYSIS RESULT COMPARISON

The maximum displacement of Baseline bumper design is 103.1 mm. Here we can see that the Bumper has been completely collapsed.

The maximum displacement of Design Iteration#1 is 95.4 mm. Here we can see that the Bumper displacement is lower than the Baseline design.

The maximum displacement of Design Iteration#2 is 86.2 mm. Here we can see that the Bumper displacement is lower than the Baseline design.

The maximum Plastic strain observed in the Baseline bumper design is 19 %. This is above the acceptable breaking plastic strain of 15%.

The maximum Plastic strain observed in the Design Iteration#1 is 17 %. Here we can see that the plastic strain value for Design Iteration#1 Bumper is lower than the Baseline design but it is still above the acceptable breaking plastic strain of 15%.

The maximum Plastic strain observed in the Design Iteration#2 is 14 %. Here also we can see that the plastic strain value for Design Iteration#2 Bumper is lower than the Baseline design and it is also below the acceptable breaking plastic strain of 15%. Hence Design
iteration#2 Bumper design is passing the analysis criteria and the bumper design is structurally adequate.

7. CONCLUSIONS

The slow speed frontal impact analysis has been carried out on the Baseline Bumper design. Based on the analysis results two more design iterations are analysed. From the Non-linear finite element analysis following conclusion were drawn:

- The maximum displacement of Baseline bumper design with 4.0 mm thickness is 103.1 mm.
- The maximum displacement of Design Iteration#1 with 4.5 mm thickness is 95.4 mm.
- The maximum displacement of Design Iteration#2 with 5.0 mm thickness is 86.2 mm. For Design Iteration#2, Bumper displacement is lower than the Baseline design.
- The maximum Plastic strain observed in the Baseline bumper design is 19 %. This is above the acceptable breaking plastic strain of 15%.
- The maximum Plastic strain observed in the Design Iteration#1 is 17 %. The plastic strain value for Design Iteration#1 Bumper is lower than the Baseline design but it is still above the acceptable breaking plastic strain of 15%.
- The maximum Plastic strain observed in the Design Iteration#2 is 14 %. Here also we can see that the plastic strain value for Design Iteration#2 Bumper is lower than the Baseline design and it is also below the acceptable breaking plastic strain of 15%. Hence Design iteration#2 Bumper design is passing the analysis criteria and the bumper design is structurally adequate.

So it may be concluded that, with increase in thickness, load carrying capacity of the bumper has increases due to increased stiffness. This has resulted in reduced Bumper deformation which in turn resulted into reduced plastic strain.

REFERENCES:


