© 2018 IJSRST | Volume 4 | Issue 5 | Print ISSN: 2395-6011 | Online ISSN: 2395-602X
Themed Section: Science and Technology

Design and Analysis of a Car Bumper to Mitigate Injuries to Occupants

Bilal Abdullah Baig*, Shehzad Ahmed, Jawwad Lodhi, Mohammad Zubair, Aamir Khan
Assistant Professor, Department of Mechanical Engineering, Anjuman College of Engineering and Technology, Nagpur, Maharashtra, India

ABSTRACT

Car accidents are happening every day. 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 which helps to protect the vehicle during impacts A bumper is a shield made of steel, aluminum, rubber, or plastic that is mounted on the front and rear of a passenger car. The goal of this paper is to design a bumper with minimum weight. The bumper model was made on software which is SOLID WORKS. The crash test was conducted on software LS-DYNA. It is widely used by the automotive industry to analyze vehicle design. It accurately predicts a car's behaviour in collision.

Keywords: Bumper, Solid Works, Crash test, LS-DYNA, Displacement

I. INTRODUCTION

E Car accidents are happening every day. 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 which helps to protect the vehicle during impacts A bumper is a shield made of steel, aluminium, rubber, or plastic that is mounted on the front and rear of a passenger car. When a low speed collision occurs, the bumper system absorbs the shock to prevent or reduce damage to the car. The car bumper is designed to prevent or reduce physical damage to the front and rear ends of passenger motor vehicles in low -speed collisions. Bumper are one of the key structures in passenger cars for which careful design and manufacturing should be considered in order to achieve good impact behaviour. The bumper beam is the main structure for absorbing the energy of collisions..

Passenger cars are a major mode of transport in the developed as well as in the developing countries. Therefore the accidents caused due to passenger cars are also significantly on the rise. In all types of crash accidents, about 30 % of the total numbers of accidents are frontal crash case. Therefore, measures to improve passenger vehicle passive safety performance in crash to reduce injury and death of passengers during a crash to the maximum has become an important subject of research.

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.

II. LITERATURE SURVEY

The simulation of vehicle crashes by using computer software has become an indispensible tool for shortening automobile development time and lowering costs. It also has huge impact on the crashworthiness of an automobile .This work reports on the simulated crash test of an automobile. The objective of this work is to simulate a frontal impact crash of an automobile and validate the results. The aim is also to alter some of the materials of the components with a view to reduce the forces experienced during the crash. Computer models were used to test the crash characteristics of the vehicle in the crash. The model used here was that of a Chevrolet C1500 pick-up truck. The software used for the simulation is LS-DYNA. It is widely used by the automotive industry to analyze vehicle designs. It accurately predicts a car's behaviour in a collision.

The finite element method is comprised of three major phases: (1) *pre-processing*, in which the analyst develops a finite element mesh to divide the subject geometry into sub domains for mathematical analysis, and applies material properties and boundary conditions, solution, during which the program derives the governing matrix equations from the model and solves for the primary quantities, and (3) *post-processing*, in which the analyst checks the validity of the solution, examines the values of primary quantities, and derives and examines additional quantities.[1]

It is important in the study of impacts to distinguish between the two different types of impacts that occur, elastic and plastic impacts. In an elastic impact a negligible amount of energy is lost between the two impacting bodies, for example, the collision of two billiard balls. A plastic impact involves a significant amount of energy dissipated in the collision. An impact between two vehicles or between one vehicle and a rigid body, where the vehicles crumple on impact, is an example of an elasto-plastic impact. The impacting phenomenon between an impactor and the front bumper in a low-speed full crash could be very complicated, since transient and nonlinear analyses are involved. But, in designing the front bumper, automobile manufacturers insist that the bumper system should not have any material crash or failure. Therefore, up to that point, the total energy is conserved throughout the impact duration. Since the impactor is assumed to be rigid and the bumper beam was made of metallic and composite material and shock absorber is a relatively low stiffness material, the distribution of the impact load is irregular along the contact area and over the contact region of the bumper, the bumper beam subjected to the impact load undergoes a constant deformation.[2]

Today's plastic auto bumpers and fascia systems are aesthetically pleasing, while offering advantages to both designers and drivers. The majority of modern plastic car bumper system fascias are made of thermoplastic olefins (TPOs), polycarbonates, polyesters, polypropylene, polyurethanes, polyamides, or blends of these with, for instance, glass fibers, for strength and structural rigidity.

The use of plastic in auto bumpers and fascias gives designers a tremendous amount of freedom when it comes to styling a prototype vehicle, or improving an existing model. Plastic bumpers contain reinforcements that allow them to be as impact-resistant as metals while being less expensive to replace than their metal equivalents.

Modeling of car bumper is done with help of Pro-e software and dimensions are selected from one of car bumper. As the impact is more for the front portion of bumper only outer dimensions of car bumper has been considered for modeling, Slots provided in middle of car bumper is used for reducing drag effect in car bumper. [3]

In this paper, a domestic A-class car bumper as the object of study. CAE model was established by using Hyper Mesh software, and the finite element method is applied to solve the calculation of the model. The bumper were studied in C-NCAP and E.C.E R42 two different standards. The bumper is optimized by changing the structure size of the way. Appropriate to increase the thickness of the metal bar bumper can improve its safety performance, while its weight changed little.

Today, the finite element analysis method has become the main method of the research on vehicle collision, so this article will be in accordance with the steps of the finite element method study, starting from establishing the CAE model, to obtained the solution and analysis the method by using LS-DYNA and Hyper Mesh software and analysis, finally put forward the optimization suggestions to, provides the reference for bumper impact study.[4]

For occupant safety, Crash test analysis provides vital information to design safety elements for vehicles. This paper focuses on the 40% offset bumper beam crash test analysis with obstacles based on the objectives and comparable safety performances by Euro NCAP. The bumper is fixed with the internal and external crash boxes to absorb energy developed. The model built using CATIA and discretized with HYPERMESH while the simulation of crash done in LSPREPOST and ANSYS LS-DYNA as a solver. Result illustrates the von Misses stress behavior of the bumper beam system to assist for safety design of the vehicle components.

The bumper is fixed to the chassis with crash boxes. Indentations in the crash boxes behind the bumper are designed especially to absorb the energy released at the moment of collision. The more energy is used to deform the beams, the less of it remains to deform and damage other elements. [5]

Box-shaped bumper beams mounted on vehicles serve as shock absorbers in a potential crash. In this study, their optimal shape design is investigated. The objective is to maximize the crashworthiness of the beam. The crash phenomenon in standard tests is simulated in which the vehicle hits a deformable barrier with 40% offset by 64 km/h speed. The bumper beam and the brackets supporting the beam are modelled as deformable bodies in full detail. For the rest of the car, a lumped parameter model is developed. The crash event is simulated using explicit finite element method.

The types of obstacles that bumper beams endure during frontal impact are countless. However, they can be categorized into three major divisions: full frontal collision, offset frontal collision, and pole frontal collision. The harshest one among the three scenarios is the pole; however it is also the rarest among them. The second harshest one is the offset impact. The majority of the frontal collisions happen at an offset with varying percentages. In this study, considering the severity and frequency of the three major frontal crash scenarios, the bumper beam is optimized for collisions with a 40% offset in accordance with European New Car Assessment Program (Euro NCAP), IIHS, ANCAP standard tests. The objective of this study is to develop a methodology to find the globally optimum shape or near globally optimum shapes for the cross-sectional profile of a hollow bumper beam to maximize its crash performance under the loading conditions in Euro NCAP tests. [6]

III. ANALYSIS

The cad model is generated with the help of cad software solid works. After the cad modeling, the chapter further deals with the analysis of the generated model with the help of analysis software Nastran.

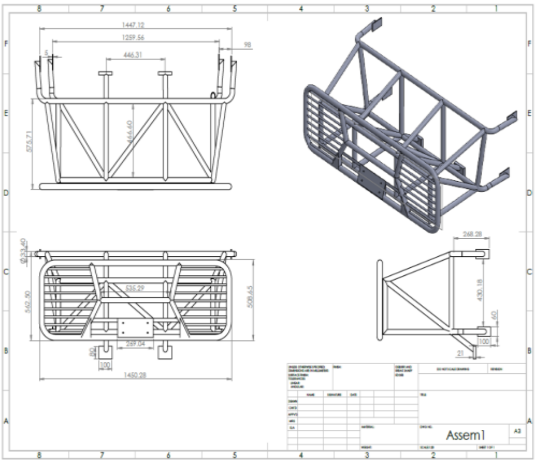

Figure 1. Solid Modeling and 2D views of bumper modified design

igure 2. describes the finite element bumper car assembly in different 2D views with different nodal structure.

A. Crash Analysis

A crash-test is a form of destructive testing usually performed in order to ensure safe design standards in crashworthiness and crash compatibility for automobiles or related components. To test the cars safety performance under various conditions and during varied types of crashes, vehicle manufacturers crash test their cars from different angles, different sides and with different objects, including other vehicles. The most common types of crash tests are listed below.

- ✓ Front impact test
- ✓ Front offset crash test
- ✓ Side impact test
- ✓ oll over test

B. Method of Analysis (LS-DYNA)

Crash-testing requires a number of the test vehicle to be destroyed during the course of the tests and is also time consuming and uneconomical. One new recent trend that is gaining vast popularity is computer simulated crash-testing. Here instead of a real vehicle, a FE (Finite Element) model of the vehicle is generated and is used to carry out the different tests that were carried out before using actual vehicles.

There are several software packages that are equipped to handle the crash-testing of vehicles, but one of the most popular is from Livermore Software Technology Corporation called LS-DYNA.

With LS-DYNA, automotive companies and their suppliers can test car designs without having to tool or experimentally test a prototype, thus saving time and expense. While the package continues to contain more and more possibilities for the calculation of many complex, real world problems, its origins and core competency lie in highly nonlinear transient dynamic finite element analysis (FEA) using explicit time integration. The application of LS-DYNA covers a wide range of industries.

The FE model was then used to simulate crash test. The FE software used here to carry out the simulation was LS-DYNA. One of the tests carried out was the Frontal-offset crash at 40 mph. Before the simulation could be carried out, several other preprocessing conditions have to be specified. The test results were verified using results from actual crash-test reports. Present runtimes on high-end workstations for LS-DYNA vehicle models are still measured in days, while multi-body run-times are typically less than 1h, even for the most complex models.

C. 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). After meshing the model appears as shown in figure.

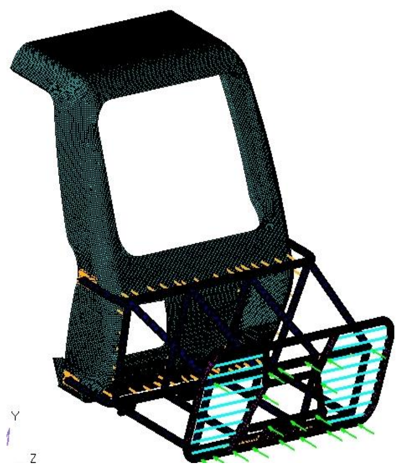

Figure3. Meshed model of bumper assembly system

The car model is constrained as shown in the fig above. The assembly is provided with the initial velocity of 16666 in negative x direction so as it hits the rigid wall.

D. 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.

IV. RESULTS

The model was solved to obtained results at various thicknesses: 1mm, 2mm, and 2.5mm respectively.

A. For 1mm

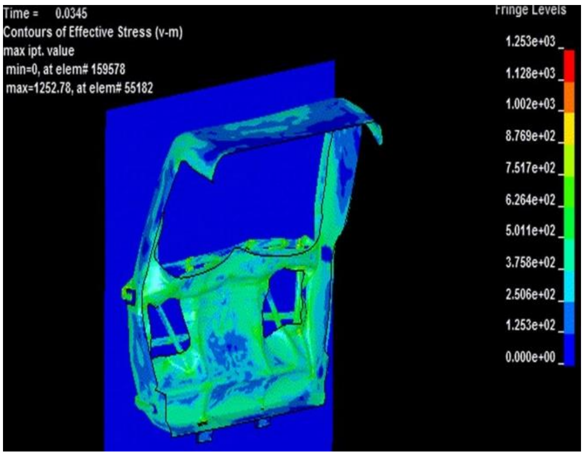

Figure 4

Figure above shows the effective stress developed in the bumper and the car because of the applied boundary condition for 1 mm thickness since the results of crash analysis obtained was very critical as the bumper was totally crushed, we decided to increase the thickness to 2.5mm.

B. For 2.5 mm

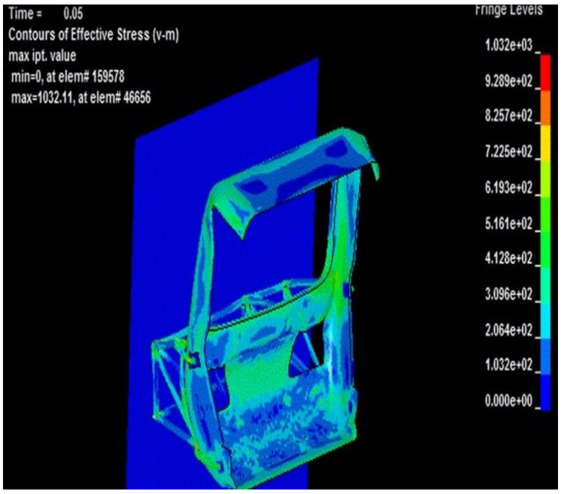

Figure 5.

Though the bumper performed better than the 1mm thick bumper, it is still not suitable as it did not undergo large deformation and thus did not absorb enough energy.

C. **For 2 mm**

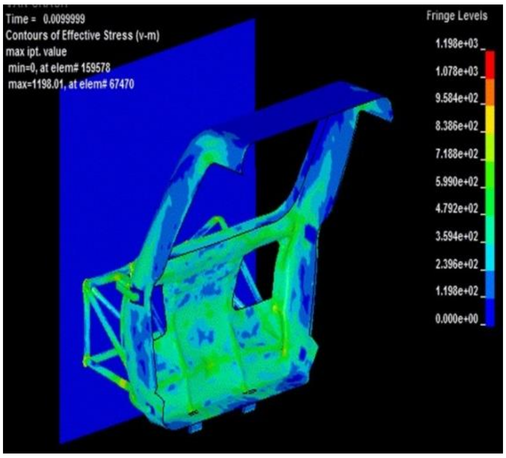

Figure 6.

The bumper with 2mm thickness performed optimally, it did not get crushed totally and also was not too very stiff so that the energy is transmitted to the occupant.

V. CONCLUSION

1mm, 2mm and 2.5mm are the thickness of the bumper tube for which crash analysis is carried out. This analysis is carried out in explicit environment, the event is transient dynamic in nature. The software used to perform the analysis is LS-Dyna. As can be seen in the result, 1mm thick design is not sufficiently stiff to absorb the frontal impact and thus it easily yields plastically without absorbing any significant impact energy and thus transferring it directly to the vehicle which can be termed as failure of bumper. The second design with 2.5 mm thick design is rigid in nature, thus it does not yield much and transfers the impact energy directly to vehicle as like a rigid link. The third design is 2.0mm thick, in this event the bumper deforms permanently but during the process it absorbs lots of impact energy which significantly reduces the effect of frontal collision with rigid wall and will be helpful in reducing the fatalities.

VI. REFERENCES

[1]. "Simulation of Vehicular Frontal Crash-Test"TejasagarAmbati, K.V.N.S. Srikanth& P. Veeraraju Dept. Mechanical Engineering, Godavari Institute of Engg& Tech., Rajahmundry, India.

[2]. "Design and analysis of an automotive bumper beam in low-speed frontal crashes" JavadMarzbanrad, MasoudAlijanpour, School of Automotive Engineering, Iran University of Science & Technology, 16846-13114 Narmak, Tehran, IranMahdiSaeidKiasat, Department of Marine Technology, Amirkabir University of Technology, 15914 Tehran, Iran

[3]. "Impact Analysis on Car Bumper by varying speeds using Materials ABS Plastic and Polyether Imide by Finite Element Analysis software Solid works"Pradeep Kumar Uddandapu, Assistant Professor ,Dept of Mechanical Engineering, K.S.R.M College of Engg, Kadapa, Andhra Pradesh, India

[4]. Research On The Crash Safety Of The Car Bumper Base On The Different Standards"Q. H. Ma, C. Y. Zhang , S. Y. Han and Z. T. Qin,ShanghaiUnivEngnSci, college of automotive engineering,Shanghai 201620, Peoples R China

[5]. "CRASH TEST FOR 40% OFFSET FRONTAL BUMPER CAR ANALYSIS USING CAE"ChandanD.Chaudhari, Anand P. Joshi,Dept. of Mech. Engg., DattaMeghe College of Engg., Airoli, Navi Mumbai SainathA.Waghmare, Dept. of Mech. Engg., LokmanyaTilak College of Engg., Koparkhairane, Navi Mumbai

[6]. "Shape Optimization Of Bumper Beams Under High-Velocity Impact Loads" NiyaziTanlak , Fazil O. Sonmez, Department of Mechanical Engineering, Bogazici University, Bebek 34342, Istanbul, Turkey, MahmutSenaltunOyak-Renault Oto. Fab., Bursa, Turkey