

A numerical analysis of front beam of an automobile by using Ansys APDL

J. Paul Chandra Kumar¹, N. Rajesh Kumar

Department of Mechanical Engineering, Jeppiaar Engineering College, Rajiv Gandhi Salai,
Chennai – 600119, India

*Corresponding author: E-Mail:paulie1980@gmail.com

ABSTRACT

Bumper systems in vehicles are important to assist and defend passengers throughout frontal and rear collisions. To satisfy the protection necessities, to scale back the element weight and to extend the fuel potency, testing and development of bumper system is indispensable. Typical testing and design procedures are dearly-won and time intense. This paper tries a numerical modeling and FE simulation of the impact and also crashworthiness of the bumper material in the course of low velocity impact testing. The design conception uses a simplified dynamic model in terms of quasi-static methodology. The performance of the bumper structures by experimental testing is compared with finite element method (FEM) by quasi-static methodology and conjointly simulated via ANSYS12.0 /LS-DYNA to achieve the most effective design.

KEY WORDS: Impact Testing, Bumper Beams, Quasi-static, Ansys APDL, FEA.

1. INTRODUCTION TO FEA

Finite Element Analysis (FEA) is a method for modeling endless structures without the employment of Partial Differential Equations (PDE). FEA approximates continuous assemblies with a series of discretized systems. FE simulations are commonly used to arrive at a bumper design that meets the boundaries of crash performance.

The crash testing of a bumper system is mostly substantiated by results from standardized low speed crash tests supported by common crash situations. Consequently, the crash load cases also are utilized in the Finite element simulations throughout the event method.

However, lack of information for the automobile under development implies that simplified models should be used as an outline of the automobile in Finite Element Analysis.

Present-day simplified models of the automobile cause inaccuracy of the design albeit the bumper system is designed during in a correct manner. Because of the recent performance improvement of computers and software package, numerical analysis like the Finite Element technique are nowadays exercised as a method of the many industrial development progressions.

A bumper system is a armor most typically fabricated from steel or aluminium mounted to the front and rear end of an automobile. During a case of a collision to the front or rear happening at low speed, the bumper absorbs the energy to prevent damage to the automobile. Subsequently, the aim of the bumper isn't to be a structural element that inertly contributes to occupant safety throughout front or rear collisions nonetheless to defend the elements like hood, lights and cooling arrangement of the vehicle.

In bumper system development, reiterative finite Element crash analysis are most typically accustomed to find a candidate design which will meet the necessities expressed by the makers, by insurance firms and in legislations. Besides those necessities, issues of weight and price for producing are also factors that are mirrored. Within the Finite Element simulations, the FE mesh of the bumper system is coupled to a simplified model of the automobile. For analyzing the bumpers, testing ways ought to gratify the federal automotive standards.

During this check, a pendulum impacts the bumper with the mass of the automobile with the speed of at least 8.0 kmph. The result of collision and also the impact on design of bumpers will be analyzed in FE model by using ANSYS/LS-DYNA.

Presently, with the event of the vehicle technology, increasingly a lot of more lightweight advisement materials just like the Glass Mat Thermoplastic (GMT) are smirched to the auto body. GMT delivers a high strength to weight magnitude relation, chemical, corrosion resistance and wonderful impact properties at each low and high temperature. Compared to metals, GMT proposes larger design flexibility, lesser tooling prices, and opportunities for improvement. During this method the GMT bumper beam was examined in ANSYS and competed with experimental results.

2. EXPERIMENTAL METHODOLOGY

Bumpers were assessed in accordance with the legislative law of FMVSS. The bumper fabricated from GMT materials was squashed in a research laboratory that is certified by ISO9000.

Testing support for this project was directed employing a pendulum impact apparatus delineated by SAE J1980.

The normal mass of the automobile is 1300 kilogram and also the average mass of the thermo plastic bumper is 6 kilogram. The velocity of the pendulum is 8Km/Hr and the mass of pendulum is 1130 kilogram. A typical FMVSS apparatus head impacts at a velocity of eight km/hr.

Design Methodology: First, a Computer Aided Design program is employed to outline the pure mathematics of the candidate bumper system. It's common that the primary pure mathematics candidate comes out from former styles for automobile models almost like the present automobile that's under development.

The second step in design methodology is to use a finite element mesh on the model. With the mesh, an outline of the component characteristics of the elements that builds up the bumper arrangement follows. During this step, the mesh of the bumper model is combined to a basic model of the automobile and preliminary conditions, boundary environments and constraints are applied.

A FE solver is then applied to unravel the defined problem. State variables like stresses, strains and displacements are stored throughout the advancement of the simulation. An appropriate explicit solver is often utilized in the crash simulations once the period time is brief, and a high degree of geometrical and material nonlinearities are existent.

The last step in the design and development phase is that the post process stage where the obtained results of the simulation are explored. If the solution of the present design is insufficient, changes in the model are done and also the steps are reiterated till the planning of the bumper is appropriate.

We assume the Impact nature of the Bumper System to the Spring-mass system as shown in Fig. 1.

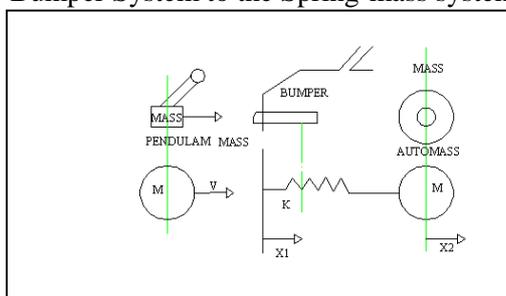


Figure.1. Equivalent spring mass system of a bumper model

Quasi-Static Impact: Feasibly one among the foremost elementary queries of this kind of impact is whether or not a transient, dynamic analysis is important. Since the pendulum method is non-linear, geometric behavior can be vital.

When considering structural FEA it's fascinating to attain the most accurate answer at the smallest cost. Deciphering a completely non-linear, dynamic problem will add difficulty to the model and CPU. Thus it's better to examine for correct approximations.

A 2.22 m/s pendulum bumper impact event will be checked out as a quasi-static event, instead of a completely non-linear dynamic event.

To assess this issue a straightforward spring-mass model is used. Here the pendulum is designed as a rigid body of "Mass, M" with associated "Initial speed, V". Similarly, the automobile is designed as a rigid solid of "Equivalent mass, M". As the mass of the bumper is tiny with relation to the automobile, it's neglected during this analysis. Nevertheless, the stiffness of the bumper is enclosed within the bumper model as a spring of "Stiffness, KB". This stiffness will be quantified either through an experiment or analytical analysis by including the "Displacement, y" of the bumper beam under "Load, F" from the pendulum. Thus the stiffness KB merely becomes

$$K_B = \frac{\text{Load}}{\text{Deflection}} = \frac{F}{y} \quad (1)$$

The governing differential equations for dynamic system represented in Fig.1 are

$$M\ddot{x}_1 + K_B(x_1 - x_2) = 0 \quad (2)$$

$$M\ddot{x}_2 + K_B(x_1 - x_2) = 0 \quad (3)$$

and the initial conditions are

$$x_1(0) = 0 \quad (4)$$

$$\dot{x}_1(0) = V \quad (5)$$

$$x_2(0) = 0 \quad (6)$$

$$\dot{x}_2(0) = 0 \quad (7)$$

Solution of this set of synchronic linear equations results to

$$x_1 = \frac{V}{2}t + \frac{V}{2\sqrt{2}\omega} \sin \sqrt{2}\omega t \quad (8)$$

$$x_2 = \frac{V}{2}t + \frac{V}{2\sqrt{2}\omega} \sin \sqrt{2}\omega t \quad (9)$$

where "Time, t" and "Angular velocity, ω ",

$$\omega = \sqrt{\frac{K_B}{M}} \quad (10)$$

A number of important conclusions may be drawn from equations (8) and (9).

Associate expression for the overall force exerted by the pendulum on the bumper for equation (11) may be derived as

$$t \leq \frac{\pi}{\sqrt{2}\omega} \quad (11)$$

$$F = K_B(x_1 - x_2) \quad (12)$$

$$F = V \sqrt{\frac{K_B M}{2}} \sin \sqrt{2}\omega t \quad (13)$$

$$F_{\max} = V \sqrt{\frac{K_B M}{2}} \quad (14)$$

Adding together, the impact event terminates once the force goes to zero which "Total impact period, τ_S " may be expressed from equation (13).

$$\tau_S = \frac{\pi}{\sqrt{2}\omega} \quad (15)$$

The information contained in equations (14) and (15) is vital from variety of standpoints. The equation (14) outlines a structural load in terms of the mass and speed of pendulum. Such loads are needed for application of elaborate finite element analyses.

Moreover, the variation of this structural load will currently be clearly outlined for various impact velocities and automobile masses.

This is often essential for the thought of impact response at totally different velocities and design for application on vehicles of varied sizes.

Because the impact event takes place, the pendulum speed bit by bit decreases from V to 0 whereas the car speed will increase.

As a result the bumper doesn't absorb all the K.E. of the pendulum. Instead, some of the K.E. is transferred to K.E. of the automobile. The "Maximum energy absorbed, U" by the bumper throughout the impact may be outlined as

$$U = \frac{1}{2} K_B [(x_1 - x_2)_{\max}]^2 \quad (16)$$

$$U = \frac{1}{2} \frac{F_{\max}^2}{K_B} = \frac{1}{4} MV^2 \quad (17)$$

If the bumper is be similar to as a simply supported beam, then "First natural frequency, f_1 " of that beam is

$$f_1 = \frac{9.81}{2\pi} \sqrt{\frac{EI}{\mu L^4}} \quad (18)$$

and the corresponding natural time period is

$$\tau_1 = \frac{1}{f_1} = \frac{2\pi}{9.81} \sqrt{\frac{\mu L^4}{EI}} \quad (19)$$

where "Mass per unit length, μ " "Span, L" of the beam, "Second moment of inertia, I" and "Material Young's modulus, E" of the cross section. Equation (15) may be used to categorize the period of impact between maximum load and inertia as

$$\tau_2 = \frac{\pi \sqrt{M}}{2\sqrt{2} \sqrt{K_B}} \quad (20)$$

To conclude, K_B is the spring constant of the beam,

$$K_B = \frac{48EI}{L^3} \quad (21)$$

$$\frac{\tau_1}{\tau_i} = 3.97 \sqrt{\frac{m}{M}} \quad (22)$$

Since the quantitative relation m/M can invariably be an awfully tiny variety, the fundamental natural period of the bumper are abundantly smaller than the pendulum impact time and application of the basic principles of impact response permits the inertia effects of the bumper to be unheeded during this downside.

3. METHODS AND MATERIALS USED IN FINITE ELEMENT MODELING

Finite Element Modeling: Work Confined by, the bumper system of an automobile is mounted stiffly at 2 supports and a pendulum consisting the mass of an automobile goes certain a low-speed collision with the car bumper. Low speed collisions is outlined as one in which the stricken vehicle speed modification is a smaller amount than 5 mph (8.0 km/h). Throughout the method of bumper model development, a FE model of the bumper system constrained at 2 specific positions serves as an associate effectual tool to match totally different bumper design with supported material properties in an affordable and time saving manner.

This simulation method leads to a final design and development of the bumper system that's factory-made during a few prototypes. The simulation model is made up in ANSYS LS-DYNA which is both a solver and a post processor. Within the FE simulations inside this work, explicit time integration is used. The bumper system is meshed with SOLID164.

Each element is outlined by eight nodes and uses viscous hourglass management and reduced integration for quicker element formulation. The FE model for the bumper consists of 140 8-noded solid elements. The dimension of each element is averaged to 50 millimeter. Nodes at a space of a 100 millimeter from each ends of the bumper are constrained in all degrees of freedom. Fig.2 shows the meshed model of the simulation with the bumper constrained at 2 supports in all degrees of freedom. Mechanical properties are implemented as shown in Table I.

Table.1. Mechanical Properties of the Bumper model

Material	Young's Modulus, E (Gpa)	Poisson Ratio	Yield Strength (Mpa)	Density (Kg/m ³)
GMT	12	0.41	230	1280

The Pendulum is outlined by a rigid component with a mass of 1130 kilo with its movement restricted to only 1 direction is modeled and meshed. Bilinear Isotropic properties of GMT are defined to outline the Bumper and steel for the pendulum.

APDL Coding: ANSYS Parametric Design Language (APDL) was used to encode the complete system. It offers a singular combination of power and simple use to supply comprehensive interface selections and suppleness. Some part of the code implemented in this work is listed below.

TB, BISO, 2

TBDATA, 1, 230E6 ! Yield stress (Pa)

TBDATA, 2, 0 ! Tangent modulus (Pa)

EDMP, RIGI, 1, 5, 7

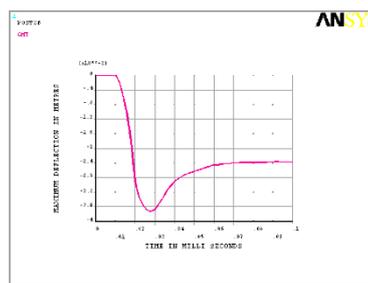
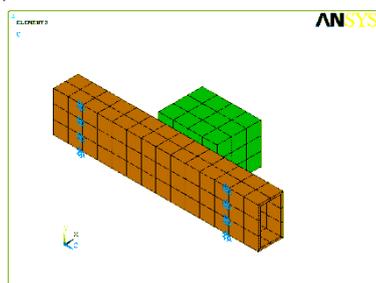


Figure.2. Meshed view of bumper and pendulum **Figure.3. Deflection of bumper w.r.t time**

Automatic single surface contact is used here during this simulation with the gap of 25 millimeter between the front bumper surface and the striking pendulum as the initial condition.

The pendulum is allowed to hit the bumper at a velocity of 8.0 km/h.

EDCGEN, ASSC, , , 0.1, 0.1, 0, 0, 0, , , , 0, 10000000, 0, 0

EDPV, VGEN, 1, -2.222, 0, 0

An Impact analysis is performed using Ansys explicit dynamics for 0.1 second impact duration. A node at the middle of the bumper is stained for Time-History post process. Fig.3 plots the displacement of the bumper for GMT vs. time planned on a graph for the chosen node in ANSYS Time-History post processor.

NSEL, S, NODE, 359

CM, SCALE, NODE

EDHIST, SCALE

Fig.4 shows a Von Mises model with Bilinear isotropic Material properties of GMT at the location of maximum deflection. The figure conjointly shows an association of the position of the bumper after deflection from its initial position. It's clear from the analysis that the utmost deflection of the bumper is 0.03732 millimeter under the subsequent test conditions.

Using APDL programs, numerous impact load cases can be simulated in ANSYS LS-DYNA. The load of the pendulum was increased from 0 to 1700 kilo and discrete impact tests are modeled and simulated. Results show the utmost deflection under each loads lasted just for a fraction of second.

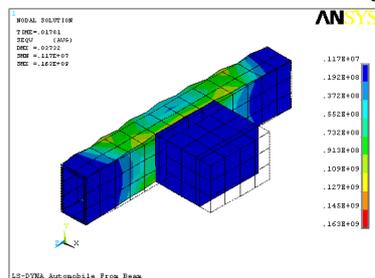


Figure.4. Von mises stress of the bumper beam

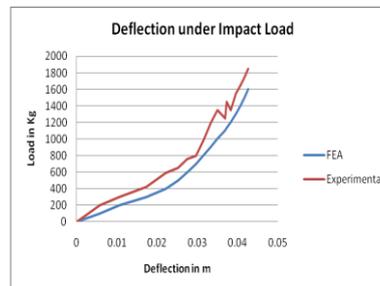


Figure.5. Comparison of deflection by FEA and Experimental methods

Fig.5 shows that valuation of FEA with physical testing on the thermoplastic well correlate. Results also indicate the FEA technique used within the bumper model were splendid and boundary conditions assumed are also nearer to the experimental results.

4. CONCLUSION

As the growth of thermoplastic bumpers in automotive system increases, there's a desire to accurately investigate the prevalence timely on the planning stage itself. Elaborate finite element analysis of bumper design states the prediction of bumper employment one step nearer.

These analysis correlate well to actual experimental routine, providing the assumptions created in conducting the FE analysis. This paper confirms a way for getting correct forecast of bumper functioning before the production process. An Ansys APDL code was programmed to mimic the bumpers with quasistatic methodology.

The efficient dynamic model will assist to conserve time and manufacturing cost of developing bumper systems for vehicles.

REFERENCES

- Abdi M, Karami Mohammadi A, Numerical Simulation and Active Vibration Control of Piezoelectric Smart Structures, International Review of Mechanical Engineering, 3, 2009, 175-181.
- Arsad SS, Noh SNSS, Ayob A, Ahmad Z, Abdul-Latif A, Kamsah N, Tamin M.N, Deformation Response of Steel Sheet Metal under Transverse Impact Loading, International Review of Mechanical Engineering, 6, 2012, 680-684.
- Federal Automotive standards, part 581, bumper standard, September 1 -1990.
- Gokula S, Hopper Stad O.S, and Lademo O, Offset impact behavior of bumper beam longitudinal systems, International Journal of Crashworthiness, 11, 2006, 317–336.
- Kachare PS, Bimleshkumar, Effective Location and Packing Ratio of a Particle Impact Damper for a Mild Steel Cantilever Beam under Transient and Forced Vibration, International Review of Mechanical Engineering, 5, 2011, 1226-1234.
- Murray AD, Rusch KC, Ford Escort, Thermoplastic Bumper, SAE paper 840218.
- Nimmer RP, Moran H, and Tryson G.R, Impact response of polymeric structure, Comparison of analysis and experiment proceedings. Society of plastics Engineers annual technical meeting. New Orleans. 1984, 565.