International Journal of Engineering and Technical Research (IJETR)
ISSN: 2321-0869 (O) 2454-4698 (P), Volume-3, Issue-7, July 2015

Deploying FEA for simulating the phenomenon of impact while improving the Design of Bumper

Ajay D. Katore, Prof. Sachin Jain

***Abstract*— Nowadays, in development of technology especially in engineering field make the engineers must be precise and showing careful attentions on what they produce. Here, we concentrate on automotive industry. The greatest demand facing the automotive industry has been to provide safer vehicles with high fuel efficiency at minimum cost. Current automotive vehicle structures have one fundamental handicap, a short crumple zone for crash energy absorption. One of the options to reduce energy consumption is weight reduction. However, the designer should be aware that in order to reduce the weight, the safety of the car passenger must not be sacrificed. 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. Consequently, these crash load cases are also used in the FE simulations during the development process. The increasing legal and customer demands on passive safety of automobiles have to be fulfilled under the conditions of shortened development times and cost reductions**

***Index Terms*— Automotive bumpers, Bumper, Bumper analysis, Bumper design, Crash energy Absorption, Development of bumper system.**

I. Introduction

In automobiles a 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.

Effect of materials and their properties on bumper beam

1.1 Modulus of Elasticity

Mechanical specifications of the isotropic and metallic materials are illustrated in Table to study the effect of elastic modulus on bumper impact behavior, three mentioned alloys metals with different modulus of elasticity are selected where they have equal yield strength. The impactor collides to the bumper perpendicularly with 4 km/h velocityThe deflection was measured at the nodes located in the middle of the bumper horizontally. Point of center of impact was assumed 445 mm above ground in this simulation according to the low-velocity impact standard for passenger cars, which gives a fixed value where most collisions occur.

Ajay D. Katore, NRIIST Bhopal
Prof. Sachin Jain, NRIIST Bhopal

Material	E (GPa)	ν	S_y (MPa)	ρ (kg/m^3)
Commercial steel bare-CS	207	0.3	190	7860
Aluminum 3105-H18	68.9	0.33	193	2720
Magnesium AZ31B	450	0.35	180	1740
PEP	1.2	0.4	27	900

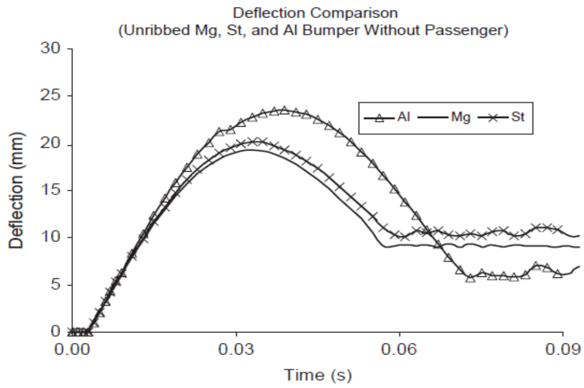


Table 1.1 Material properties of the models of bumpers.

1.2 Yield Strength-

The effect of yield strength on impact behavior is studied with three different specifications on aluminum alloys. All phenomena are attributed to the yield strength of aluminum. For different aluminum bumpers, difference between vehicle and impactor velocities after impact increases by increasing the yield strength. According to these figures, the velocity of impactor is not reduced to zero. The major reason is plastic deformation that occurs in the bumper and holders.

Material	E (GPa)	ν	S_y (MPa)	ρ (kg/m^3)
Aluminum 3105-H18	68.9	0.33	193	2720
Aluminum 2219-T31	73.1	0.33	248	2840
Aluminum 2024-T86	72.4	0.33	440	2780
Steel bare/EG-HF 80Y100T	207	0.3	584	7860

Table 1.2.1 Material properties for Aluminium and Steel material.

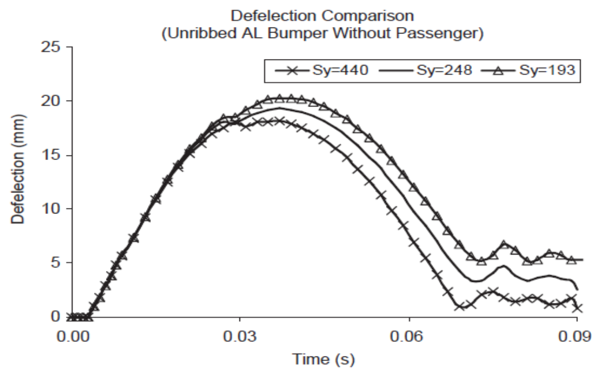


Fig. 1.2.1 Various aluminum bumper deflections.

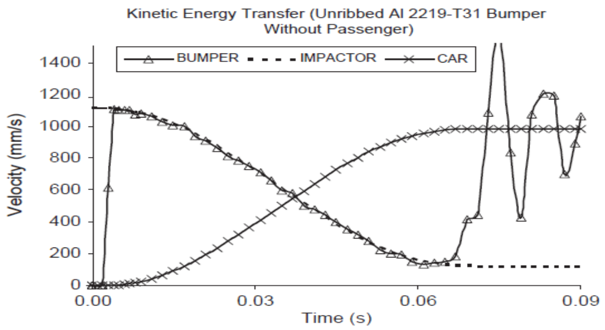


Fig. 1.2.2Kinetic energy transfer in aluminum 2219-T31 bumper.

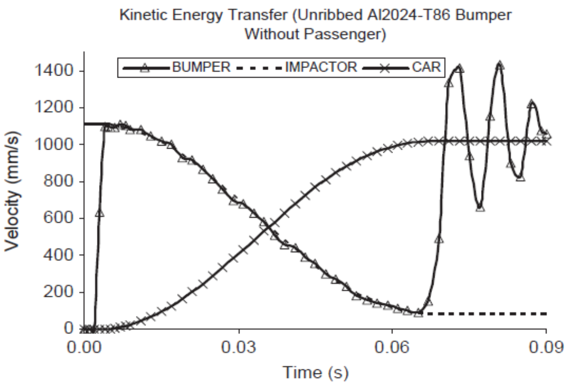


Fig. 1.2.3 Kinetic energy transfer in aluminum 2024-T86 bumper.

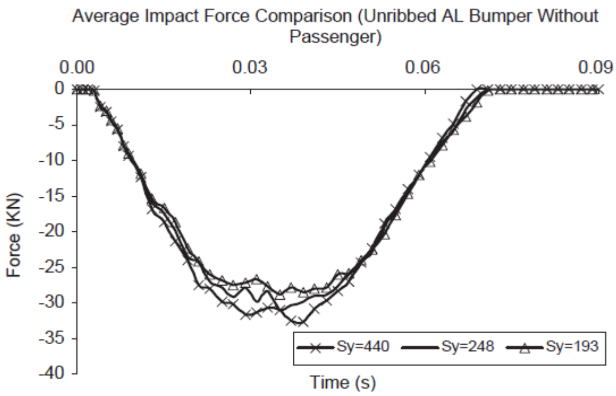


Fig.1.2.4 Impact forces in aluminum bumpers.

II. EFFECT OF THICKNESS PARAMETER ON BUMPER

Different bumper beam thickness made of high-strength steel (Bare/EG-HF 80Y100T) with 584 MPa yield strength were chosen to determine the effect of impact behavior.

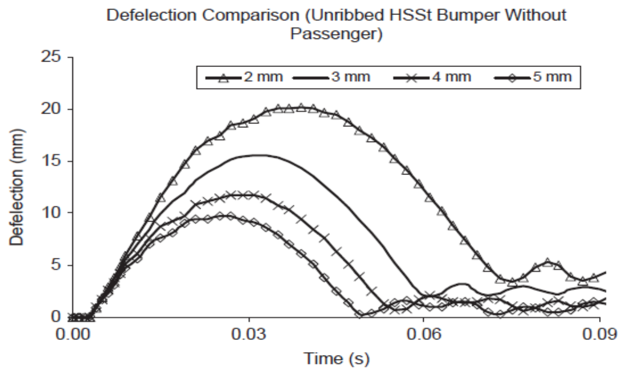


Fig. 2.1 Effect of thickness on bumper deflection.

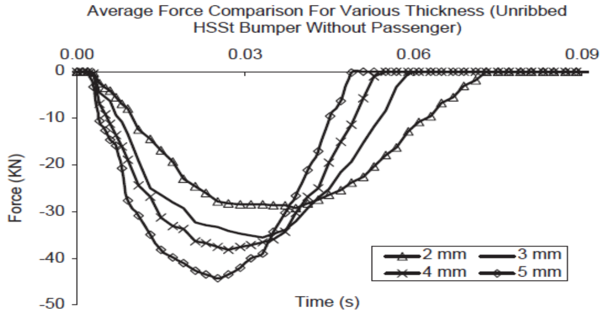


Fig. 2.2 Effect of thickness on impact force.

The separation point and the maximum deflection point take place with a delay in thicker bumper. The study of impact forces on bumper with various thicknesses shows that the impact force enhances following increasing the bumper thickness as illustrated in Fig. 2.2. So, the acceleration rate of the car increases very fast, since this force applies in short-time interval. By investigation of kinetic-energy diagram, it is observed that more kinetic-energy transfer from impactor to vehicle and less plastic strain energy dissipates with increasing the bumper thickness.

III. EFFECT OF RIBS ON BUMPER

The ribs are strengthening plates of average thickness 4 mm, mainly placed along the vertical and horizontal direction of bumper beam as shown in Fig.3.1, for preventing deflection of lateral surfaces and thus creating a rigid structure. To study the effect of ribs on impact behavior, high-strength steel (Bare/EG-HF 80Y100T) with 584 MPa yield strength is chosen. Fig.3.2 clearly shows how ribs can reduce deflections: 19% comparing conditions of bumper with-ribs and without-ribs. As shown in this figure, this decrease is also noticeable in separation time of the without-ribbed bumper after a time of 0.054 s, due to lower rigidity of the structure.

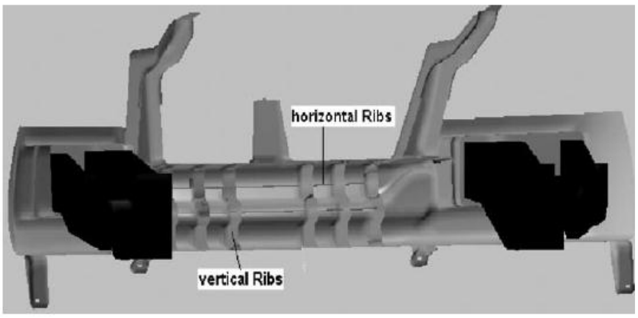


Fig. 3.1 Ribs in vertical and horizontal direction.

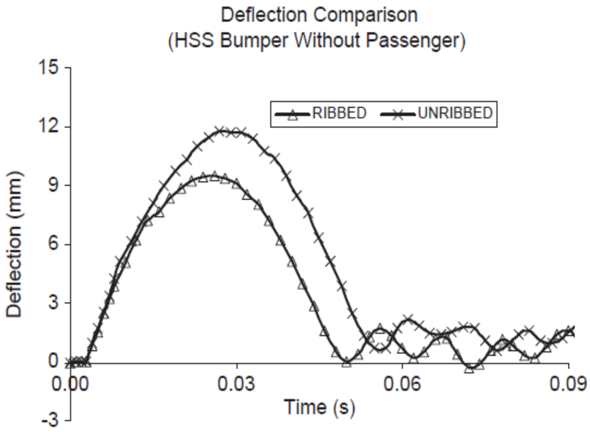


Fig. 3.2 Deflections in two case studies of bumpers

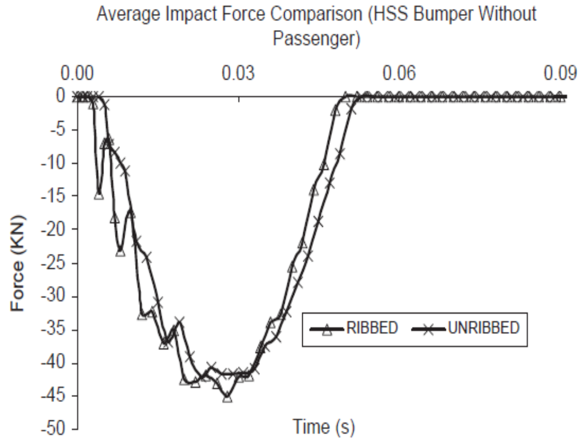


Fig. 3.3. Impact force in two case studies of bumpers.

In addition, it is observed from Fig. 3.3 that ribbed bumper has a stronger impact force than un-ribbed one. Augmentation of maximum impact force is 7%. This phenomenon increases the rigidity of the bumper structure and grows impact force. Careful attention of the impact velocities represents that the ribs do not have an influence on vehicle and impactor velocities. Here, it is comprehended that finding an un-ribbed structure with the same speed decelerating behavior as the ribbed bumper is a very reasonable replacement solution and should be precisely focused due to the advantage of ease of manufacturing, however; the ribs have an effect on impact behavior.

IV. PASSENGER EFFECT

The presence of passengers on impact behavior with mentioned steel is investigated by considering the passenger's weight in the mass point elements. For simplification, the effect of distribution of passengers was ignored here. In fact, the presence and absent of passengers investigated in this study as in the standards also recommend three passengers added to driver. The impact force with and without passengers is calculated and shown in Fig. 4.1. It shows that the impact force is increased up to 12% by existing passengers. So car's kinetic energy decreases comparing with the case of without passenger.

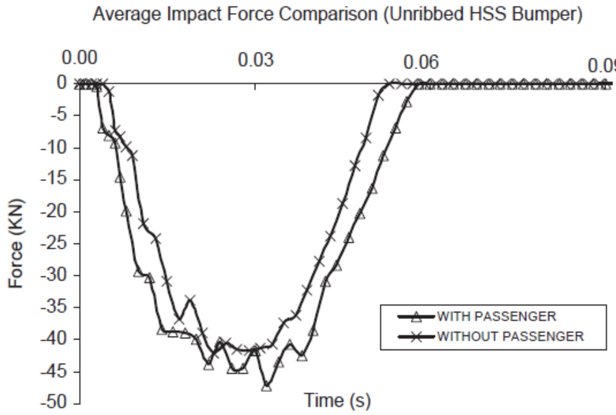


Fig.4.1 Impact force in two case studies of bumpers for passenger effect.

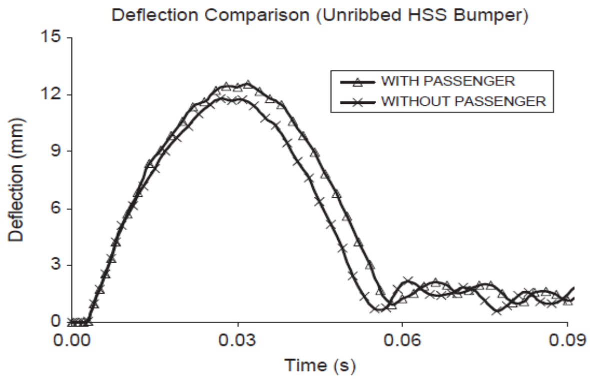


Fig.4.2 Deflections in two case studies of bumper.

V. INTRODUCTION TO FINITE ELEMENT METHOD

The finite element method (FEM), sometimes referred to as finite element analysis (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. The boundary conditions are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field. Depending on the type of physical problem being analyzed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity to name only a few.

4.1 A general procedure for finite element analysis

1. Preprocessing

The preprocessing step is, quite generally, described as defining the model and includes

- Define the geometric domain of the problem.
- Define the element type(s) to be used.
- Define the material properties of the elements.
- Define the geometric properties of the elements (length, area, and the like).
- Define the element connectivity's (mesh the model).
- Define the physical constraints (boundary conditions).
- Define the loadings.
- The preprocessing (model definition) step is critical.

2. Solution

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s). The computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow. As it is not uncommon for a finite element model to be represented by tens of thousands of equations, special solution techniques are used to reduce data storage requirements and computation time. For static, linear problems, a wave front solver, based on Gauss elimination is commonly used.

3. Post-processing

Analysis and evaluation of the solution results is referred to as postprocessing. Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution. Examples of operations that can be accomplished include-:

- Sort element stresses in order of magnitude.
- Check equilibrium.
- Calculate factors of safety.
- Plot deformed structural shape
- Animate dynamic model behavior.
- Produce color-coded temperature plots.
- While solution data can be manipulated many ways in post-processing.

VI. INTRODUCTION TO ANALYSIS

Analysis is done by selecting appropriate solver and carrying out the operations in various stages to obtain solution. The analysis of bumper can be done by using explicit solvers like LS- Dyna, Abaqus, etc. whichever suitable. Particularly analysis is carried out in three stages by performing various operations in software.

1. Stage-I

In this stage .igs file is imported to the meshing software like Hypermesh. The CAD data of the bumper structure is imported and the surfaces were created and meshed. Since the average thickness of bumper is much smaller than the other dimensions of the part, the best element for meshing is the shell element. Some various choices of impact elements can be considered like implicit and explicit model. Here, nonlinear explicit impact modelling elements are used for analysis.

2. Stage-II

After meshing is completed we apply boundary conditions. These boundary conditions are the reference points for calculating the results of analysis. In short we here go for the preparation of deck. Here we apply define and apply various loads. Different loadsteps are created which are to be applied during analysis. Here surrounding effect is been taken into consideration while applying loads. Elements are defined by their properties. Material properties such as density, modulus of elasticity, Poisson's ratio etc. is assigned to the elements. Here proper arrangements are made so that we can run the analysis in solver software.

3. Stage-III

Meshed and boundary condition applied model is imported to the solver. Analysis process starts after applying run in the solver software. Software first calculates the deflection with respect to the boundary conditions applied. Then on the basis of deflection it calculates strain. Once the strain is calculated we know modulus of elasticity then we can calculate the stress values. Results are viewed and accordingly modifications are suggested. Modifications are suggested according to high stress regions obtained. If the stresses are beyond the permissible limits then changes such as change in material, change in thickness of component or addition of ribs etc are made according to suitability.

VII. TESTING METHODS

1. FULL-WRAP FRONTAL COLLISION TEST:-

Dummies are placed in both the driver's and passenger's seats and the vehicle is made to collide with a concrete barrier at a rate of 55 km/h. Actual collisions of this type tend to occur at speeds lower than that of this test. The dummies are then checked for injuries to the head, neck, chest and legs, the vehicle is checked for damage, and the results are used to evaluate the degree of passenger protection in 5 levels.

Fig. 12.1 Full- Wrap frontal collision test image

2. OFFSET FRONTAL COLLISION TEST:

The dummies are placed in the driver's and front passenger's seats and the test vehicle is made to collide head-on with an aluminum honeycomb, on the driver's side (at an offset of 40%). The dummies are checked for injuries to the head, neck, chest and legs, the vehicle is checked for damage and deformation, and the results are used to evaluate the degree of passenger protection in 5 levels.

Fig. 12.2 Offset frontal collision test image

3. SIDE COLLISION TEST:-

Among the passenger injuries which occur in automobile collisions, side collisions cause the most damage next to frontal collisions. In this test, a truck with a weight of 950 kg is made to collide at a speed of 55 km/h with the side of a stationary test vehicle with a dummy in the driver's seat. The dummy is checked for injuries to the head, chest, abdomen, and waist, and the results are used to evaluate the degree of passenger protection in 5 levels. 5

Fig. 12.3 Side collision test image

VIII. IMPACT PRINCIPLE

An isometric view of impact layout is shown in Fig. 13.1 to study the impact principle on Bumper assembly.

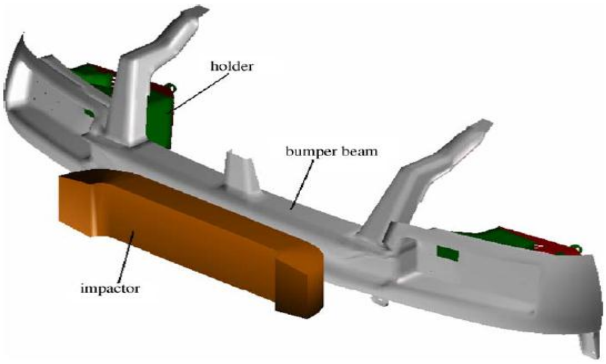


Fig.13.1 Isometric view of Impact Layout

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 is made of metallic 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 d_{max}. A principle of energy conservation in the elastic impact is used; the kinetic energy before impact is conserved and converted to elastic energy and the kinetic energy of the impactor and the automobile at its maximum deflection, i.e.,

$$\frac{1}{2}m_A v_A^2 = \frac{1}{2}K_{eq}\delta_{\max}^2 + \frac{1}{2}m_A v_0^2 + \frac{1}{2}m_B v_0^2 \quad (1)$$

Where m_A is the mass of the impactor, m_B the mass of vehicle, v_A the velocity of the impactor before impact and v_0 the final velocity of the impactor and vehicle in maximum deflection point. K_{eq} the equivalent impact stiffness of a bumper and is obtained by the relationship of displacement and reaction forces from beam analysis. An important consideration of momentum is that it can be neither created nor destroyed. Thus, the momentum before an impact is equal to the momentum after the impact. At the moment of its maximum deflection, a principle of momentum conservation before and after impact can be ex- pressed as follows:

$$m_A v_A = (m_A + m_B)v_0 \quad (2)$$

From equations (1) and (2) the maximum deflection d_{max} is obtained as follows:

$$\delta_{\max}^2 = \frac{1}{K_{eq}}\frac{m_A m_B}{m_A + m_B}v_A^2 \quad (3)$$

After separation point, energy and momentum conservation equations can be expressed as follows:

$$\frac{1}{2}m_A v_A^2 = \frac{1}{2}m_A v_{A2}^2 + \frac{1}{2}m_B v_{B2}^2 \quad (4)$$

$$m_A v_A = m_A v_{A2} + m_B v_{B2} \quad (5)$$

The Eq. (6) can be used to find the energy dissipated, E_D, during an impact. This is found by subtracting the kinetic energy of the two masses after impact, and the kinetic energy of the impactor before impact.

$$E_{\text{Plastic}} = \frac{1}{2}m_A v_A + \frac{1}{2}m_B v_B - \frac{1}{2}m_{A2} v_{A2} - \frac{1}{2}m_{B2} v_{B2}$$

System formulation -:

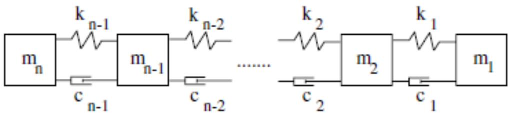


Figure 13.2 One dimensional mass spring damper model.

The time-invariant mass spring damper model with *n*-degrees of freedom (DOF) shown in Figure 13.2 above can be described in continuous time by a second order differential equation

$$m\ddot{x} + c\dot{x} + kx = bf \quad (1)$$

where *m*, *c* and *k* represents *n* * *n* mass, damping and stiffness matrices respectively. *X* is a *n* * *1* vector of displacements, and $\dot{x}$ and $\ddot{x}$ are both vectors of the same size with velocities and accelerations. The matrix *b* is the *n* *r input matrix and *f* is an *r* * *1* vector of input excitations. By introducing the state vector *X* as $[x\ \dot{x}]^T$ the system can be written in the first order matrix form as

$$\dot{X} = AX + Bf$$
$$y = CX \quad (2)$$

Where $A_{N\times N}, B_{N\times r}, C_{M\times N}$ are the time-invariant continuous time system matrices where *N* = *2n* and *M* denotes the number of outputs *y*. Multiplying (2) with e^{-At} and integrating yields

$$X(t) = e^{At}X(0) + \int_0^t e^{A(t-\tau)}Bf(\tau)d\tau \quad (3)$$

Equation (3) is the analytical solution to the continuous model.

IX. BUMPER ASSEMBLY ANALYSIS

9.1 ORIGINAL MODEL ANALYSIS

We are provided with bumper assembly as shown in fig.14.1 whose component thickness are listed in the table

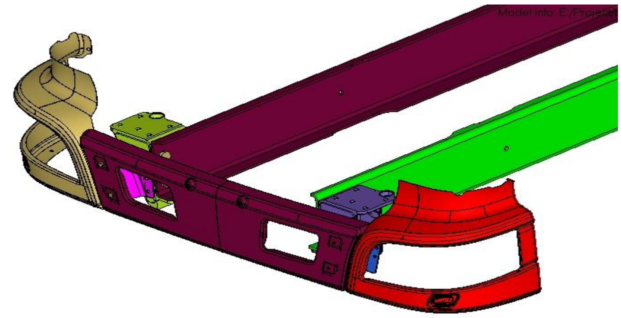


Fig. 9.1 Assembly of bumper system to be analyzed

Table no.9.1 part details-:

Sr. no.	Part name	Original model thickness
1	Front panel	01.60mm
2	Side panel	01.60mm
3	Bracket	04.00mm
4	Supporting Bracket	12.00mm
5	Chassis parts	10.00mm

9.2 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) Mesh size is selected by convergence criteria. After meshing the model appears as shown in fig.9.1 and 9.2 The meshed model is then checked for quality of mesh.

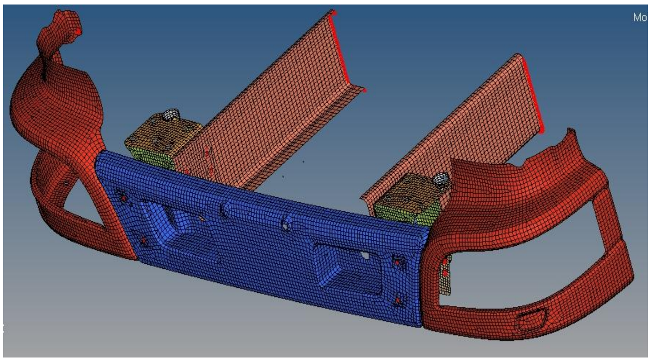


Fig. 9.2.1 meshed model of bumper system assembly (front view)

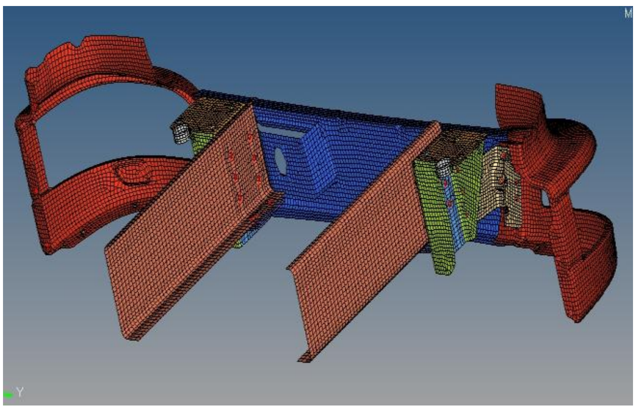


Fig. 9.2.2 meshed model of bumper system assembly (rear view)

Once the meshing is done model is checked for quality and normals so that stress regions are properly defined after analysis. After meshing materials of respective parts are assigned to them by their material properties such as modulus of elasticity, poisons ratio, density of material, etc. . Here we are provided with Steel as a basic material whose properties are described in table below.

Table 9.2.1 Material properties of Steel

Material	Modulus of elasticity	Density	Poissons ratio
Steel	210 kN/mm^2	7.86e^-6 kg/mm^3	0.29

This material is further classified as soft steel and hard steel and assigned to the components as shown in table 9.3

Table 9.2.2 Assignment of Materials to components-:

Sr. no.	Part name	Material
1	Front panel	Soft steel
2	Side panel	Soft steel
3	Bracket	Hard steel
4	Supporting Bracket	Hard steel
5	Chassis parts	Hard steel
6	Impactor	Rigid

The values and curves of stress strain plots of both the material is as shown below.

Table 9.2.3 Soft steel stress strain curve data-:

STRESS (X)	STRAIN (Y)
0.001	0.2
0.002	0.21
0.003	0.22
0.005	0.24
0.008	0.27
0.1	0.391
0.14	0.4
0.16	0.41
0.19	0.43
0.2	0.44
0.21	0.44
0.22	0.44

Table 9.2.4 Hard steel stress strain curve data-:

STRESS (X)	STRAIN (Y)
0.001	0.342
0.002	0.344
0.003	0.345
0.005	0.35
0.007	0.358
0.011	0.36
0.1	0.39
0.12	0.41
0.15	0.43
0.18	0.47
0.2	0.49
0.21	0.49
0.22	0.49

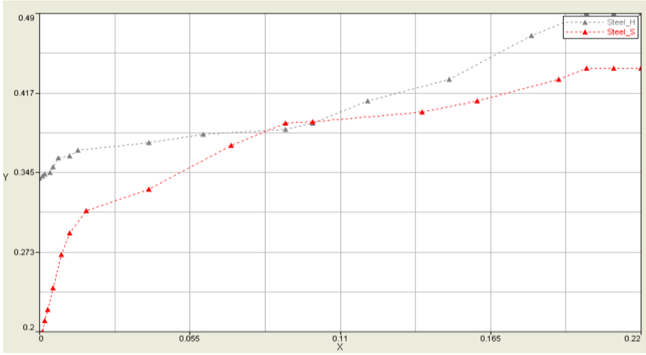


Fig. 9.2.2 Comparison curve between hard and soft steel

Figure 14.1.1.3 shows the comparison curve of hard steel and soft steel through their non- linear curves. 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 887kg 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 fig. 9.2.3

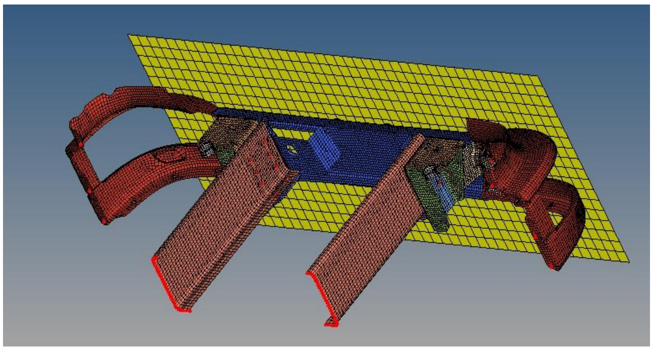


Fig. 9.2.3 bumper assembly after preprocessing

The velocity is given to the impactor through proper contacts defined between bumper and impactor. The mass of vehicle here is considered during impact conditions. Contacts are defined via elements as shown in fig. 14.1.1.5

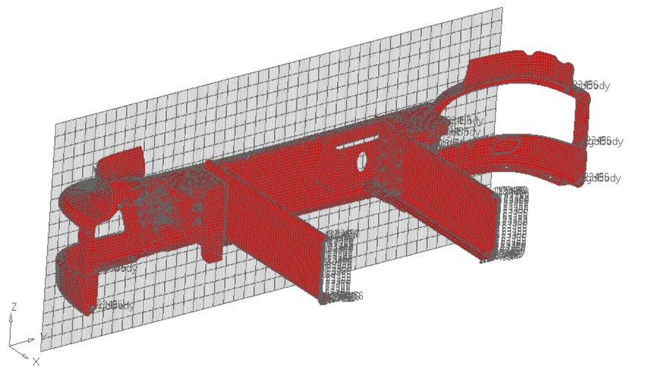
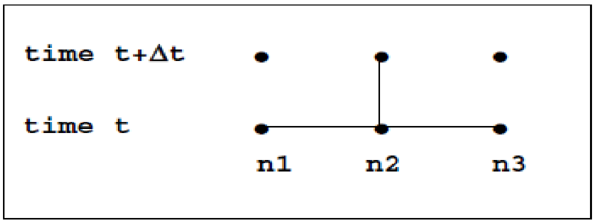


Fig. 9.2.4 Image of contacts in the model

9.3 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. Explicit refers to the numerical method used to represent and solve the time derivatives in the momentum and energy equations. The following figure presents a graphical description of



explicit time integration. The displacement of node n2 at time level t+Δt is equal to known values of the displacement at nodes n1, n2, and n3 at time level t. Systems of explicit algebraic equations are written for all the nodes in the mesh at time level t+Δt. Each equation is solved in-turn for the unknown node point displacements. Explicit methods are computational fast but are conditionally stable. The time step, Δt, must be less than a critical value or computational errors will grow resulting in a bad solution. The time step must be less than the length of time it takes a signal traveling at the speed of sound in the material to traverse the distance between the node points.

The model in LS-Dyna solver appears as shown in fig. 9.3.1

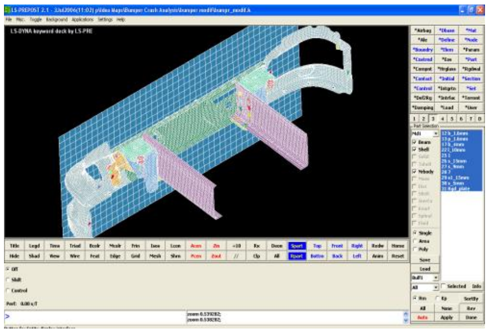


Fig.9.3.1 Bumper assembly in LS- Dyna Solver

REFERENCES

[1] Javad Marzbanrad , Masoud Alijanpour , Mahdi Saeid Kiasat (2009),"***Design and analysis of an automotive bumper beam in low-speed frontal crashes***" Science Direct paper pp.902-911.

[2] Liquan Mei and C.A. Thole (2007) "***Data analysis for parallel car-crash simulation results and model optimization***" Science Direct paper pp. 329-337.

[3] Nader Abedrabbo and Robert Mayer (2009) "***Crash response of advanced high-strength steel tubes: Experiment and model***" Science Direct paper pp.1044-1057

[4] F.Ince, H.S. Turkmen, Z. Mecitoglu, N. Uludag, I. Durgun, E. Altınok and H. Orenel (2011) "***A numerical and experimental study on the impact behavior of box structures.***" Science Direct paper pp.1736-1741

[5] Bryan C. Baker, Joseph M. Nolan, Brian O'Neill and Alexander P. Genetos (2007) "***Crash compatibility between cars and light trucks: Benefits of lowering front-end energy-absorbing structure in SUVs and pickups***" Science Direct paper pp.116-125

[6] Tso-Liang Tenga, Fwu-An Changb, Yung-Sheng Liuc and Cheng-Ping Peng (2007) " ***Analysis of dynamic response of vehicle occupant in frontal crash using multibody dynamics method***" Science Direct paper pp.1724-1736

[7] O.G. Lademo, T. Berstad, M. Eriksson, T. Trylan, T. Furu, O.S. Hopperstad and M. Langseth (2007) "***A model for process-based crash simulation***" Science Direct pp.376-388

[8] Jovan Obradovic, Simonetta Boria and Giovanni Belingardi (2011) "***Lightweight design and crash analysis of composite frontal impact energy absorbing structures***." Science Direct pp.423-430

[9] Ping Zhu, Yu Zhang, Guanlong Chen (2011) "***Metamodeling development for reliability-based design optimization of automotive body structure."*** Science Direct paper pp.729-741

[10] Xin Yang, Yong Xia, Qing Zhou, Pei-ChungWang and KathyWang (2012) " ***Modeling of high strength steel joints bonded with toughened adhesive for vehicle crash simulations***" Science Direct paper pp.21-32.