Optimization of Impact attenuator design using Finite element method for Enhanced Vehicle Safety during collision

Vikas Shrivastava[*], Dr. Amit Telang [*]

[*] Department of Mechanical Engineering, MANIT Bhopal

Abstract: Vehicle collision also known as crash or vehicle accident is a dynamic phenomenon and occurs when a vehicle collides with another vehicle, or any other stationary or moving object. It may result in injury, death and property damage.

Crash box, with which a car is equipped at the front end of its front side frame, is one of the most important automotive parts for crash energy absorption. In case of frontal crash accident, for example, crash box is expected to be collapsed with absorbing crash energy prior to the other body parts so that the damage of the main cabin frame is minimized and passengers are saved. Conventionally, a crash box is equipped with several ditches called as ―crash beads, so that those crash beads may initiate buckling deformation and make the crash box easily collapse. Recently, it has been strictly required to satisfy both reduction of body weight and improvement of crash worthiness in the design and thus, regarding crash box, it is required to ensure high energy absorption using sheet as thin as possible.

In this Paper, attention is focused upon finding an optimum cross sectional shape of a crash box to ensure high capability for energy absorption with the use of finite element analysis. At First a mechanism through which a body part absorbs crash energy in axial collapse was clarified and then, the influence of cross sectional shape of the part on energy absorption was quantitatively revealed. Finally, a new design scheme of cross sectional shape of a crash box was proposed. And further, the Co-relation between the FEA plots and the theory is being done.

Index Terms- Crash box, Impact Attenuator, Optimization, validation

I. INTRODUCTION

Vehicle crash is because of many reasons, be it a mechanical failure or human error. Various safety techniques are now being adopted by automobile manufactures to reduce /avoid the effect of crash. Crash safety features are classified as;

- Active Safety
- Passive Safety

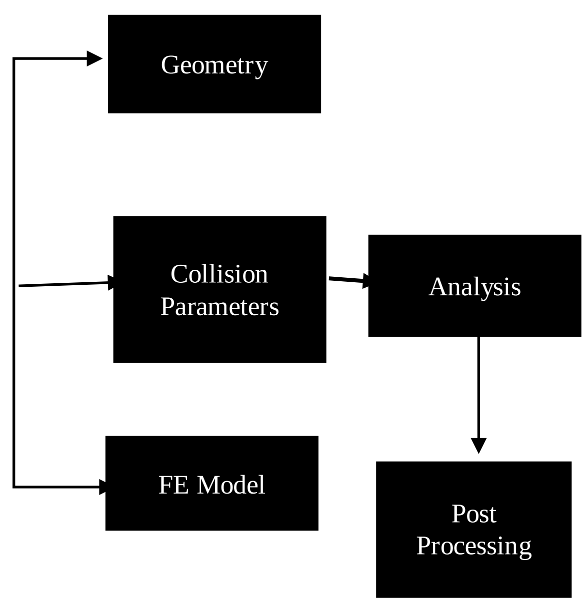
In the automotive sector the term active safety (or primary safety) refers to safety systems that are active prior to an accident. While that of with passive safety (or secondary safety), which are active during an accident. For e.g. seat belts, deformation zones and air-bags, etc.

An impact attenuator, also known as Crash box or crash cushion is a passive safety device intended to reduce the damage to structure of the vehicles. Impact attenuators are usually placed in front of fixed structures. They are designed to absorb the colliding vehicle's kinetic energy by collapsing thus absorbing Strain energy from the impact through controlled deformation.

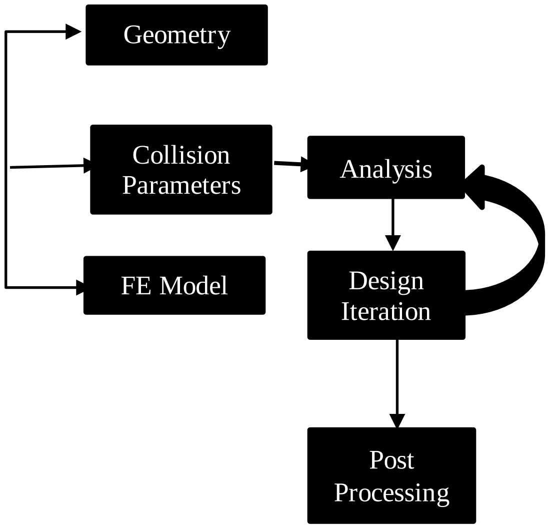
II. RESEARCH METHODOLOGY

Research methodology adopted here is divided into three steps;

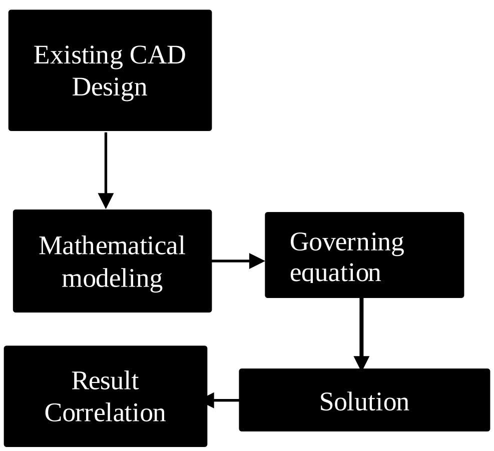
1. Existing design Study



2. Optimization



3. Result Validation



III. DESIGN

Crash analysis is one of the biggest concerns for the automobile industry. With the increase in allowable weights, the growing traffic, and increasing trend of rash driving, it has been a great concern for the industry to design a crash box that can prevent major loss to life and property, with more emphasis being on the life of the occupants of the vehicle. There has been a considerable amount of research already being done to optimize the crash box design, various iterations has been carried out with different materials, cross sections etc. Here, a basic design of crash box is first analyzed here and then various iterations has been made to optimize it further. To optimize the crash box design, a crash analysis of a vehicle template is done to find the approximate deformation at the test speed as specified.

Load Determination:

Estimation of Impact force for a perfectly inelastic collision

Energy Transferred, (DE) = $\frac{1}{2}*[\{(m1*m2)/(m1+m2)\}*(u2-u1)^2]$

Where, m1 and m2 are the two colliding masses with velocities u2 and u1 respectively. Since both m1 and m2 are two vehicles with similar masses and the vehicle (m2) is at rest,

=> m1=m2& u2=0

=>**DE = 1/4 *m1*{u1}2**

Now, Force = DE/t, Where ‘t’ is impact time.

=>**F = ¼ * m1*$\{u1\}^2$/t**

Mass of the vehicle =900 Kg

Mass of Four passengers = 250 Kg

Total Mass (m1) =1150 Kg

Maximum Speed of Vehicle, u1 =90 KMPH = 25m/s

In most crash time t is of the order of 0.1s.

=>F = ¼ * 1150*$\{25\}^2$/0.1= 1796 KN.

The Design Factor of Safety, FSd was taken as 1.5. This relatively high value is taken to account for the uncertainty in the nature of forces.

=> F = 1.5 * 1796 = 2695 KN.

Hence for design purposes force is taken to be 2695 KN.

Also, design output is plastic deformation of the vehicle frontal structure.

IV. ANALYSIS

TEST NO 01: TWO POINT TEST

Analysis parameters

- Mass of vehicle + passenger = 1150 kg.
- Avg. Vehicle speed = 54 KMPH (15m/s)
- Software used = ANSYS 13.0

ISSN 2250-3153

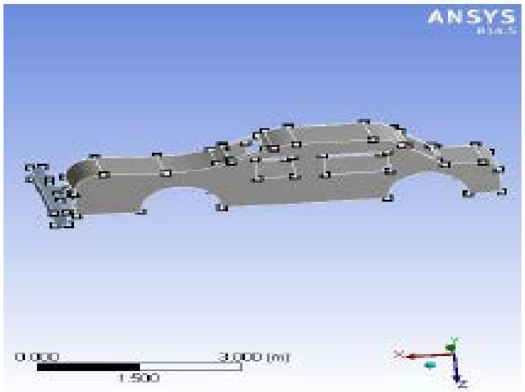


Figure 1: CAD Model

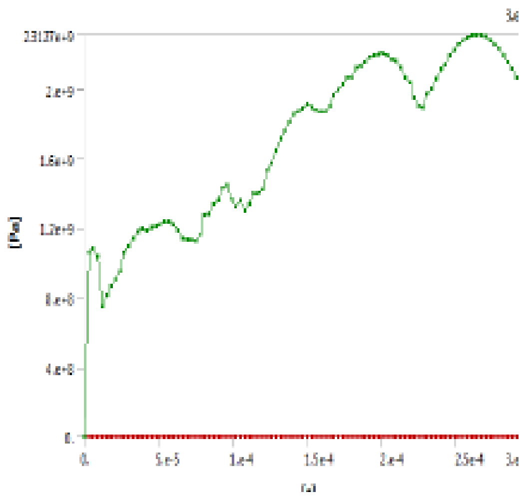


Figure 2: Stress-time variation

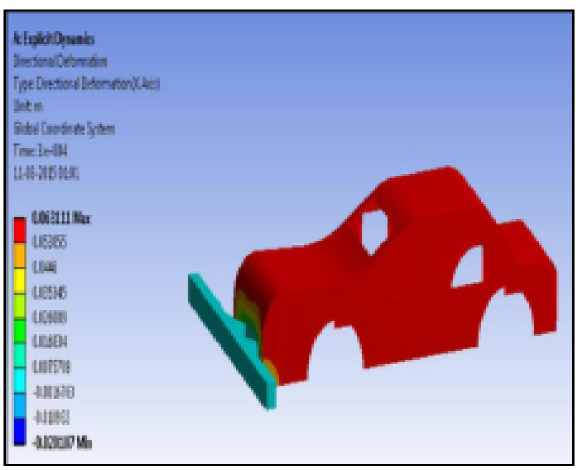


Figure 3: Deformation Plot

Table I: Result Table

S.no	Parameter	Value
1	Max. vonmises Stress	331 MPa
2	Max. deformation	63 mm

Design Iterations

- Plate Type Design

Table II: Material Table

S.No	Material Property	Value
1	E	210 Gpa
2	Poisson ratio	0.3
3	Yield Stress	180 Mpa
4	Impact Time	0.5 sec
5	Thickness	4 mm
6.	Force	2695 KN

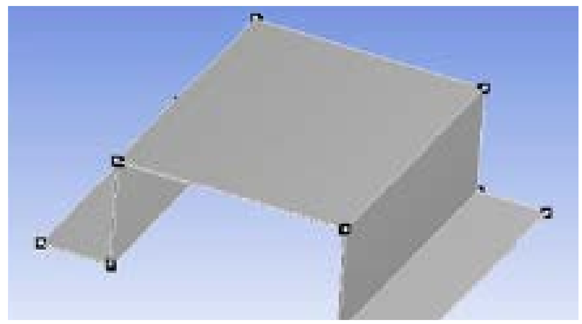


Figure 4: CAD design (3mm Plate)

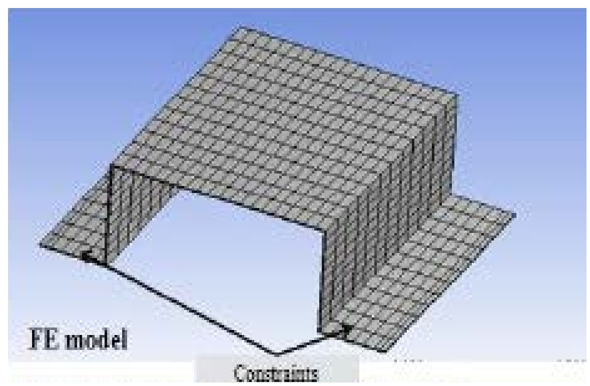


Figure 5: FE Model (Shell Meshing)

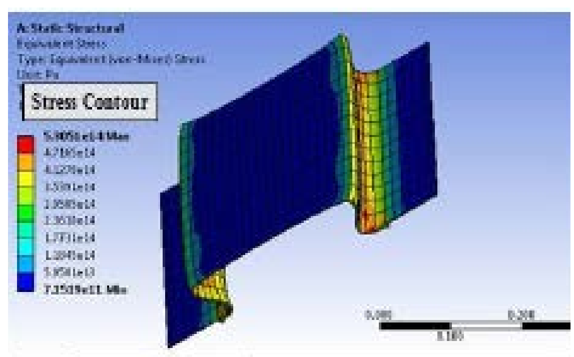


Figure 6: Stress contour

International Journal of Scientific and Research Publications, Volume 5, Issue 8, August 2015
ISSN 2250-3153

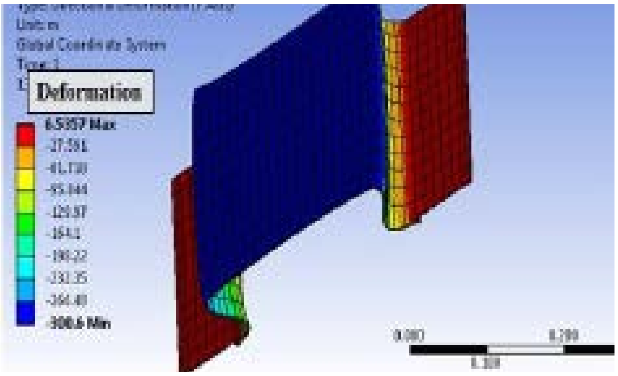


Figure 7: Deformation contour

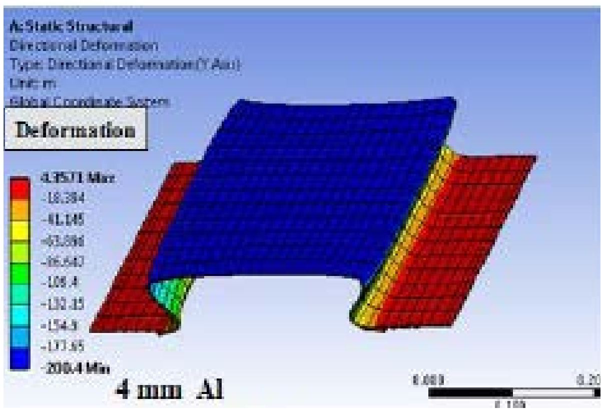


Figure 8: Deformation contour (4mm Al)

The analysis is repeated with different material but the deformation in each case is way beyond the permissible limit. Although this design was safe when the vehicle speed used to be low but nowadays with better road conditions and better technology, the speed of the vehicle has considerably increased and so does the vehicle force, thus making this type of crash box design obsolete now.

- Box Type design

Analysis Parameters

- Mass of vehicle + Passenger = 1150 Kg.
- Vehicle Speed = 90 Kmph (25m/s)
- Shell Thickness= 3.0 mm
- Force of impact= 2695 KN
- Software used = Ansys 13.0
- Material= E150

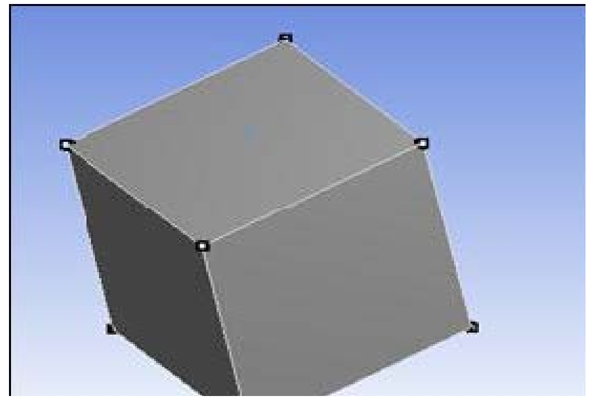


Figure 9: CAD model (Box Type Design)

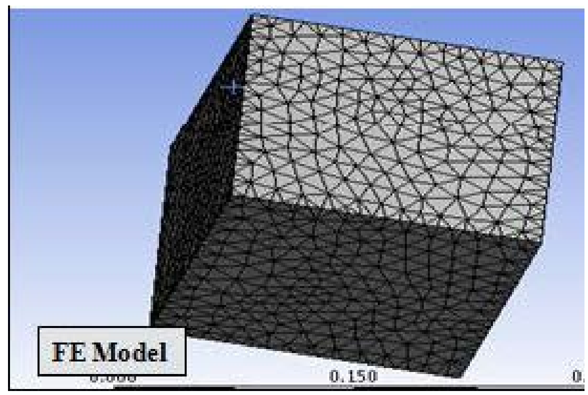


Figure 10: FE Model

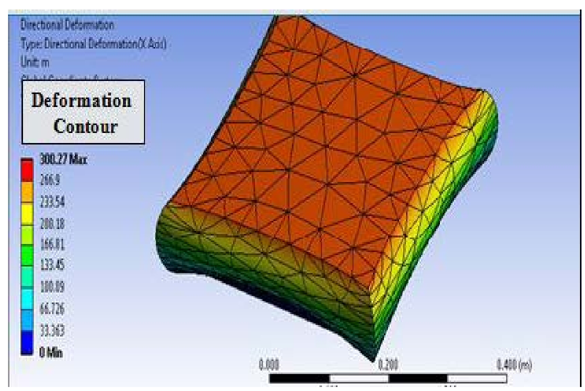


Figure 11: Deformation contour

The deformation at the standard loading condition is 300.27 mm. This design is safe but is not optimized.

- Honey Comb Structure

Honeycomb structures made either natural or man-made. They

International Journal of Scientific and Research Publications, Volume 5, Issue 8, August 2015
ISSN 2250-3153

have the geometric similarity with that of a honey comb prepared by bee. It allows optimization by using very less amount of material thus reaching minimal weight and minimal material cost

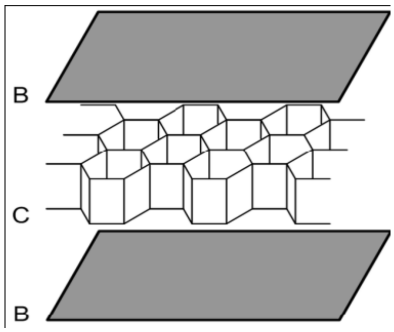


Figure 12: Honeycomb

Honey comb structure offers a compressed panel with minimal weight and excellent rigidity. The behavior of the structure is orthotropic and hence the panel behaving differently in different orientation.

Table III: Material Table

S.No	Material Property	Value
1	E	80 Gpa
2	Poisson ratio	0.334
3	Yield Stress	240Mpa
4	Impact Time	0.5 sec

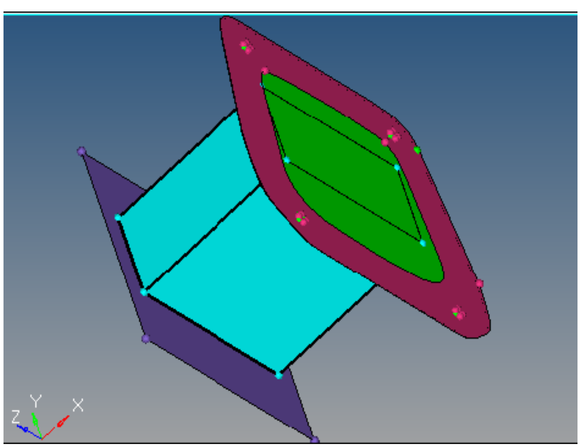


Figure 13: CAD Model

Analysis Parameters

- Mass of vehicle + Passenger = 1150 Kg
- Vehicle Speed = 90 Kmph (25m/s)
- Force of impact= 2695 KN

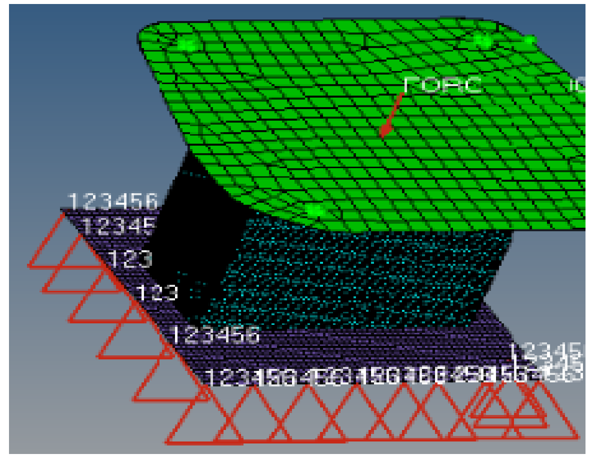


Figure 14: FE Model

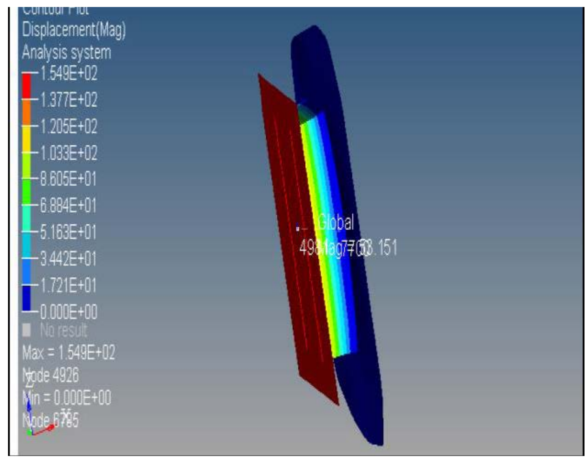


Figure 15: Displacement contour

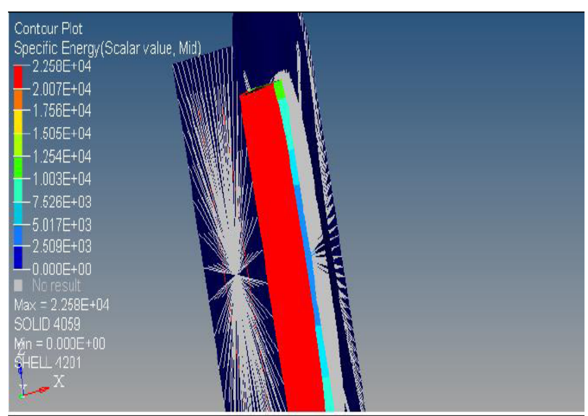


Figure 16: Strain Energy plot

Table IV: Result Table

S.No.	Parameter	Unit	Value
1	Max. deformation	mm	140
2	Max Strain Energy	KJ	22.58

The design is safe and as per regulatory standards. The system has been analyzed and found safe but also, it is very cost effective in comparison to the previous design.

V. DESIGN CO-RELATION

Numerical methods like FEA, FDM are approximate methods and assumptions are made to simply the solution. It is applicable even if physical prototype of the model is not available. How it is used for solving real life problem but its results cannot be believed blindly and must be verified by experiment or hand calculations for knowing the range of result.

Numerical Methods like FEM are based on discretization of integral form of equation. Basic theme of all numerical methods is to make calculation at only limited number of points and then interpolate the results for entire domain. Even before getting into the solution, we assume how the unknown is going to vary over entire domain. Say for example, when meshing is carried out using linear Quad. Elements, assumption are parabolic variation. This may or may not be case of real life and hence all numerical methods are based on initial hypothetical assumption.

Validation

Design is validated using Numerical Methods by doing Mathematical modelling and solving governing equations. The basic equation used to describe any transient dynamics is described as:

$$[m]\ddot{\vec{x}} + [c]\dot{\vec{x}} + [k]\vec{x} = \vec{F} \quad \text{-------------(1)}$$

Where, $[m]$ - Mass matrix
$[c]$- Damping matrix
$[k]$- Stiffness matrix
$\vec{x}$- Displacement Vector
$\vec{F}$- Force Vector

All that we have to do is just determine the evolution of the basic quantities such as displacement, velocity and acceleration with the help of boundary conditions. All other quantities can be derived from these and the most important is element stress, plastic strains, contact forces, and kinetic energies. Most software would commonly solve the dynamic equilibrium equation in an implicit way but the most popular way that should be used for highly nonlinear problem is to use explicit time integration scheme such as numerical central difference method.

Also, The Solver that we have used in our analysis are LS-Dyna which uses Numerical Integration or central difference Methods for Crash Test Analysis. Therefore, we have tried to validate the Software FEA result using Numerical Central difference method.

Central Difference Method

The Central Difference formula for velocity vector at time $t_i = i\Delta t$ is

$$\dot{\vec{x}}_i = \frac{1}{2\Delta t}(\vec{x}_{i+1} - \vec{x}_{i-1}) \quad \text{------ (2)}$$

Similarly the acceleration vector is given as

$$\ddot{\vec{x}}_i = \frac{1}{(\Delta t)^2}(\vec{x}_{i+1} - 2\vec{x}_i + \vec{x}_{i-1}) \quad \text{--- (3)}$$

Using these two formulas, the equation of motion for time t_i can be written as

$$\left([m]\frac{1}{(\Delta t)^2} + [c]\frac{1}{2\Delta t}\right)\vec{x}_{i+1} =$$

$$\vec{F}_i - \left([k] - \frac{2}{(\Delta t)^2}[m]\vec{x}_i\right) - \left([m]\frac{1}{(\Delta t)^2} - [c]\frac{1}{2\Delta t}\right)\vec{x}_{i-1}$$

Therefore, from the above equation we can calculate the solution vector $\vec{x}_{i+1}$ if we know $\vec{x}_i$ and $\vec{x}_{i-1}$. The above equation is to be used for i= 1,2,3,…….n, so for calculating $\vec{x}_1$ we will require $\vec{x}_0$ and $\vec{x}_{-1}$. Therefore we need some procedure to find $\vec{x}_{-1}$ which is equal to $\vec{x}$ at $t=-\Delta t$.

Now, equations (1), (2), and (3) are to be evaluated at $i=0$.

$$[m]\ddot{\vec{x}}_0 + [c]\dot{\vec{x}}_0 + [k]\vec{x}_0 = \vec{F}_0 = F(t=0) \quad \text{--------- (5)}$$

$$\dot{\vec{x}}_0 = \frac{1}{2\Delta t}(\vec{x}_1 - \vec{x}_{-1}) \quad \text{------------------------------- (6)}$$

$$\ddot{\vec{x}}_0 = \frac{1}{(\Delta t)^2}(\vec{x}_1 - 2\vec{x}_0 + \vec{x}_{-1}) \quad \text{----------------------- (7)}$$

From equation (5), we get the initial acceleration vector as;

$$\ddot{\vec{x}}_0 = [m]^{-1}(\vec{F}_0 - c\dot{\vec{x}}_0 - [k]\vec{x}_0) \quad \text{------------------ (8)}$$

And from equation (6), we get the displacement vector at $t=t_1$ as;

$$\vec{x}_1 = \vec{x}_{-1} + 2\Delta t\dot{\vec{x}}_0 \quad \text{----------------------- (9)}$$

On substituting equation (9) into equation (7) we get

$$\ddot{\vec{x}}_0 = \frac{2}{(\Delta t)^2}(\Delta t\dot{\vec{x}}_0 - \vec{x}_0 + \vec{x}_{-1})$$

Further, re arranging, we get,

$$\vec{x}_{-1} = \vec{x}_0 - \Delta t\overrightarrow{\dot{x}_0} + \frac{(\Delta t)^2}{2}\overrightarrow{\ddot{x}_0} \text{ ----------- (10)}$$

Thus we have found the $\vec{x}_{-1}$ needed for solving equation (4) at i=1

By dividing the time frame into no of discrete interval, we can reach to the convergence where the variation in the result with the FEA plots is within the acceptable limits.

The computational algorithm can be stated in the following steps as:

- First of all from the given initial conditions, (i.e. $\vec{x}_0$ and $\dot{\vec{x}}_0$), find out $\ddot{\vec{x}}_0$.
- Then select a time step Δt keeping in mind that $\Delta t<\Delta t_{critical}$.
- Calculate $\vec{x}_{-1}$ from equation (10).
- Calculate $\vec{x}_{i+1}$, starting with =0 , from equation (4) .
- Repeat the above step until $\vec{x}_{n+1}$ ($i=n$) is determined.

VI. CONCLUSION

The simulation of the car crash was successfully carried out and results were also obtained successfully. The various designs have been analyzed and found out the design is optimized in case of honey comb structure. Also, the methodology we used for carrying out for simulating the crash impact is validated using central difference technique. The results were satisfactorily complying thus countenancing our methodology. With the help of advanced computers and workstations, real time simulations can be achieved with a high percentage of realism. It is also concluded that most of the deformations and stresses were developed in the frontal part of the car. Therefore, it is a good practices to install the engine in the frontal part with little or no margin in the frontal part for added safety.

REFERENCES

[1] Ellouz, M., Völzke, H., Protz, C.: ―ContainerSafety Assessment for the Disposal of Non-Heat Generating Radioactive Waste in the KONRAD Repository.‖ INMM 53rd Annual Meeting, Orlando, Florida USA, July 15-19,2012

[2] LS-DYNA Explicit Finite Element Code Version ls971d, Release 4.2.1, Revision53450, Livermore Software Technology Corporation.

[3] Nitin Gokhale, 2013, Finite to infinite, Finite to infinite publication, Page no. 321

[4] Nayak, Ashish, 2004, ―Crash Simulation of a RTV‖, M-Tech Thesis, IIT Delhi.

[5] Bose, Dipan and Gupta, Rahul, 2001, ―Finite element analysis of child restraint system in car crash situations.‖ B-Tech Project, IIT Delhi.

[6] Müller, K., Quercetti, T., Melnik, N., Droste, B.: ―Impact target characterization of BAM drop test facility.‖ Packaging, Transport, Storage and Security of Radioactive Material, Volume 19 (2008), Number 4, pp. 217-221, Maney Publishing, UK

[7] Kazimi, S.M.A, 2002, Solid Mechanics, Tata MacGraw-hill, New Delhi.

[8] Norton, R.L., 2004, Machine Design, Pearson Education, Singapore.

[9] Ghosh, Amitabha and Malik, Asok Kumar, 2002, Theory of Mechanisms and Machines, East-West Press, New Delhi.

AUTHOR

First Author -Vikas Shrivastava, (Mechanical Engineering), Maulana Azad National Institute of Technology, Bhopal, vikas2907@gmail.com

Second Author –Dr. Amit Telang, Assistant Professor, (Mechanical Engineering), Maulana Azad National Institute of Technology, Bhopal, atelang7@rediffmail.com

Correspondence Author -Vikas Shrivastava, vikas2907@gmail.com, 09826469895