International Journal of Mechanical and Industrial Technology ISSN 2348-7593 (Online)
Vol. 4, Issue 2, pp: (38-50), Month: October 2016 - March 2017, Available at: www.researchpublish.com

Side Impact Analysis on Automotive Door Beam

Madasamy Panneerselvam

Wayne State University, Detroit, MI, USA
Caresoft Global Inc, Burr Ridge, IL, USA

Abstract: **The project involves dynamic analysis of an automotive door beam subjected to impact by a ram, which is moving with a particular velocity. For the analysis of the side-door impact beam, the Finite Element Method (FEM) was used since it is the most widely used computational method in automotive design applications. A finite element basic model of the beam and the ram was developed in hypermesh 12.0. The objective is to run a FE analysis for this model and collect animation results, force-displacement plots, stresses etc. It was also required to collect plots at 6 inch of ram travel. It is also necessary to keep the stresses in the beam lesser than the ultimate stress of the material being used. Phase II is to simulate the door beam in complete vehicle with different design and different material composition.**

The energy - time plots collected give us an idea about the amount of energy the beam could absorb under impact. Similarly the force displacement plots can give information about the load carrying capacity of the door beam. Two design improvements have been suggested for the door beam. These changes in the design have shown a good improvement in the load carrying capacity of the beam. For all these analyses it was observed that the stresses obtained at 6 inch ram travel were below the ultimate stress of the beam material and this prevents the beam from failing

Keywords: **Side impact; door beam; LS-DYNA; Impact analysis.**

I. INTRODUCTION

Safety aspects of an automobile are always carefully considered. The goal of any auto industry is to transfer high safety level to the passenger cars. This goal has been achieved by most of these industries by extensive use of computer aided engineering simulations using finite element methods.

The finite element code LS-DYNA is used by many car manufacturers and suppliers and has become more and more standard software tool for crash simulation. It can take care of highly nonlinear structural deformations allowing consideration of sudden dynamic changes including quasi-static situations.

With proper formulation of these crash simulations using these FEA software, including correct representation of material properties, very little differences can be observed between these simulations and the actual crashes. The simulation results are utilized during the early design stage where few, if any tests are conducted. These simulations and results can be obtained in accordance with the US standards.

II. SOFTWARE USED IN THIS PROJECT

- LS-DYNA as the solver
- Hypermesh Version 12.0 as the preprocessor
- Hyperview & Hypergraph as the post processor
- UNIX operating system

III. THE DOOR BEAM

The function of the safety beam in automotive applications is to provide a high level of safety for the passenger in the case of side impact by another vehicle. The beam should have the ability to absorb as much deformational energy as possible without failing. The door beam is generally placed on the inside of the vehicle door frames. The exact position of the member depends on the structural joints of the door and on the position of the car seat. Proper placement considerably contributes to the passenger safety. Steel is still the most widely used material for such members. However, other lighter materials, such as FRP and composites are being used in the automotive industry. Proper fiber orientation and stacking sequence of the cross-ply laminate help in absorbing more energy than the conventional steel beams.

IV. PROCEDURE

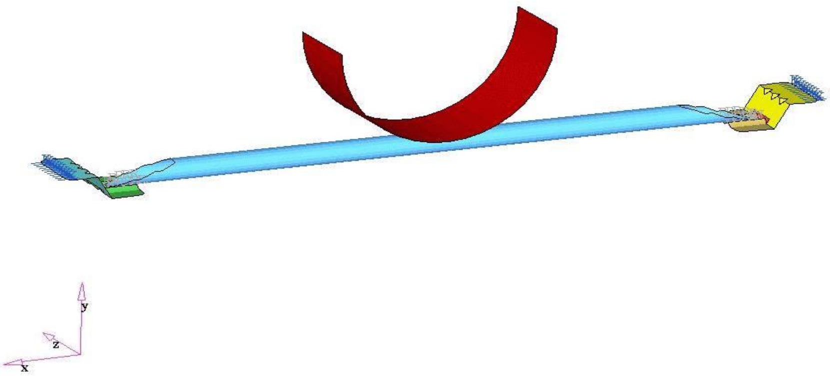


Fig 1. The ram and the beam in hypermesh format

V. ASSUMPTIONS

- The analysis conducted is a quasi-static analysis
- The material used for different parts are considered isotropic and homogeneous
- The geometry of the model and boundary conditions were already provided.

VI. MODELING INFORMATION

Model was given in NASTRAN bulk data format that was converted into LS-DYNA key format. The CELAS elements in the NASTRAN file were deleted and converted to DISCRETE spring elements in LS-DYNA format.

In this model the beam is connected to the end brackets through rigid links (missing data in the given NASTRAN file). In the real structure the beam is welded on to the brackets. These brackets are supported by springs with a stiffness of 15000lb/in at each end.

After making these changes various collectors were defined. Material collectors were created at first. Basically 3 material cards were used in this project. MAT 24 MAT20 and SDMAT1. MAT 24, which is represented as MAT_PIECEWISE_LINEAR_PLASTICITY which was used for the door beam, the brackets and the end plates. High strength steel, which has an ultimate strength of 1620 Mpa, was used for the door beam. MAT20 that is represented as MAT_RIGID in the input deck was used for defining the ram material. Since the ram is made up of a rigid material it is assigned with MAT20. SDMAT1 was used for defining spring elements that is represented as MAT_SPRING_ELASTIC. Stiffness was defined for the springs using this card.

Since the beam and ram are made up of 2 D shell elements the thickness for all the components had to be specified. Thickness of each component is given in table 1.

A load collector with the card image INT VELOCITY is created for giving the input velocity for the ram. A node set is created for the nodes of the ram for this purpose. The analysis was run for different velocities of the ram ranging from 2000 mm/ sec to 20,000 mm/ sec. In each case the displacement plots, energy plots and stress results were collected.

A) Contact:

Many engineering problems involve contact between two or more components. In these problems a force normal to the contacting surfaces acts on the two bodies when they touch each other. The general aim of contact simulations is to identify the areas on the surfaces that are in contact and to calculate the contact pressures generated. In a finite element analysis contact conditions are a special class of discontinuous constraint, allowing forces to be transmitted from one part of the model to another. The constraint is discontinuous because it is applied only when the two surfaces are in contact. When the two surfaces separate, no constraint is applied. The analysis has to be able to detect when two surfaces are in contact and apply the contact constraints accordingly. Similarly, the analysis must be able to detect when two surfaces separate and remove the contact constraints.

The two Contact Cards used in this model are:

B) Automatic Single Surface Contact:

Single surface contact is established when a surface of one body contacts itself or the surface of another body. In single surface contact, the LS-DYNA program automatically determines which surfaces within a model may come into contact. Therefore, single surface contact is the simplest type to define because no contact or target surface definitions are required. When it is defined, single surface contact allows all external surfaces within a model to come into contact. This option can be very powerful for self-contact or large deformation problems when general areas of contact are not known beforehand. Unlike implicit modeling, where over-defining contact will significantly increase computation time, using single surface contact in an explicit analysis will cause only minor increases in CPU time. Most impact and crash-dynamic applications will require single surface contact to be defined. Since automatic general (AG) contact is very robust and includes shell edge (SE) contact as well as improved beam contact, it is recommended as the first choice for self-contact and large deformation problems when the contact conditions are not easy to predict.

C) Automatic Node-to-Surface:

Node-to-surface contact is a contact type, which is, established when a contacting node penetrates a target surface. This type of contact is commonly used for general contact between two surfaces.

The flat or concave surface is the target. The convex surface is the contact surface.

The coarser mesh is the target surface. The finer mesh is the contact surface.

D) Automatic Contact:

Along with the general contact family, the automatic contact options are the most commonly used contact algorithms. The main difference between the automatic and general options is that the automatic contact algorithms automatically determine the contact surface orientation for shell elements. In automatic contact, checks are made for contact on both sides of shell elements.

***E)** Defining contacts b/w ram and beam:*

Contacts are defined between the ram and the beam. Without contacts the ram may not recognize the presence of the beam. Since the beam deforms penetration between its own surfaces can occur which is not desirable. To prevent these problems contacts have to be defined. Two types of contact surfaces are provided. One is “nodes to surface” and the other is “single surface” type of contacts. Here the ram is considered as a rigid body and hence it is made the master surface. The beam is considered as a slave surface. The second type of contact is called single surface contact used for the beam surface. This type of contact prevents the surface of the beam to overlap and penetrate onto its own surface. Both the types of contacts are of automatic type. Use of the automatic card adjusts the normal of the contact surfaces to face in opposite direction to each other even if they are not.

F) Control Cards:

The following control Cards were defined to obtain the results:

*CONTROL_TERMINATION - To define the End-Time of analysis

*CONTROL_CONTACT - To change defaults for computation of contact surfaces

*CONTROL_ENERGY - Provide controls for energy dissipation option

G) Database Option:

The following database cards were defined to obtain the output from the analysis. The output in ASCII format.

*DATABASE_NODOUT: To get output for nodes specified.

*DATABASE_ELOUT: To get output for elements.

*DATABASE_GLSTAT: To get energy output from the analysis.

*DATABASE_DEFORC: To get discrete elements force

*DATABASE_RCFORC: To get output for contact force.

*DATABASE_SLEOUT: To get sliding interface force.

*DATABASE_BINARY_D3PLOT: To get animation.

Boundary conditions are defined such that the ends are free to rotate, and the beam ends are constrained in both lateral and vertical translational D.O.F.

•LS-DYNA input deck is created and analysis is run

TABLE 1: COMPONENT DESCRIPTION

Name of the component	Gauge (mm)	Poisson's ratio	Young's modulus (MPa)	Yield strength (MPa)	Density Tonne/mm3	Mass (Tonne)
Beam	2.0	0.28	207000.0	1170	7.83e-9	0.001254
Ram	1.0	0.28	207000.0	210	7.83e-9	4.32e-4
RH-L-bracket	2.2	0.28	207000.0	210	7.83e-9	1.21e-4
RH-bracket	2.2	0.28	207000.0	210	7.83e-9	7.98e-5
RH-end plate	2.0	0.28	207000.0	210	7.83e-8	3.93e-5
LH-L-bracket	2.2	0.28	207000.0	210	7.83e-9	1.21e-4
LH-bracket	2.2	0.28	207000.0	210	7.83e-9	7.98e-5
LH-end plate	2.0	0.28	207000.0	210	7.83e-8	4e-5

TABLE 2: MATERIAL PLASTIC STRAIN TABLE FOR BEAM

Strain	Stress (MPa)
0.5e-1	1490
0.1e1	1620

TABLE 3: MATERIAL PLASTIC STRAIN TABLE FOR ALL OTHER PARTS

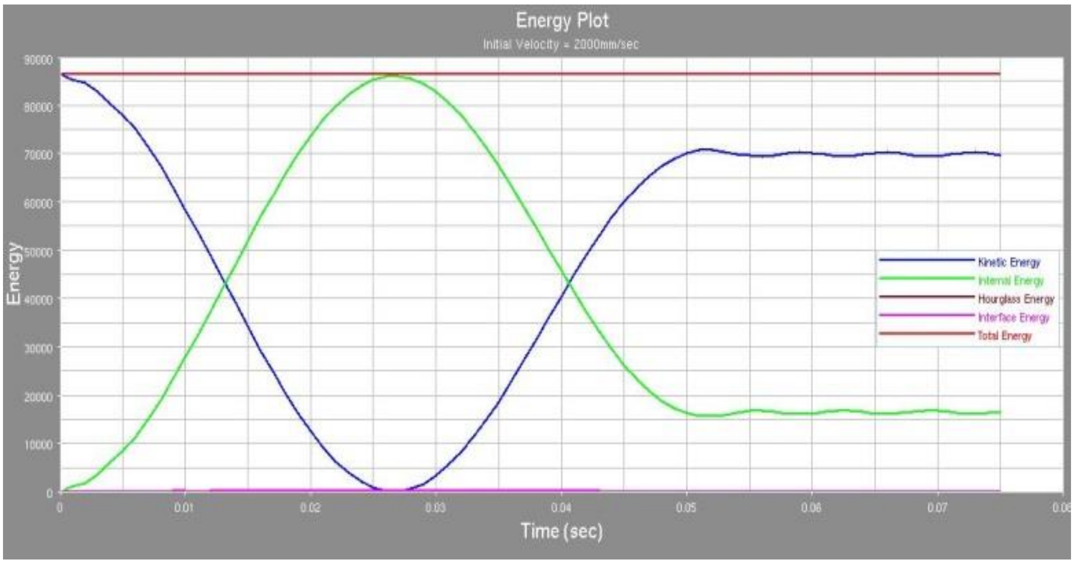
Strain	Stress (MPa)
0.0309	0.3e+3
0.0409	0.3410e+3
0.05	0.3250e+3
0.15	0.3900e+3
0.301	0.4380e+3
0.701	0.5050e+3
0.901	0.5270e+3

International Journal of Mechanical and Industrial Technology ISSN 2348-7593 (Online)
Vol. 4, Issue 2, pp: (38-50), Month: October 2016 - March 2017, Available at: www.researchpublish.com

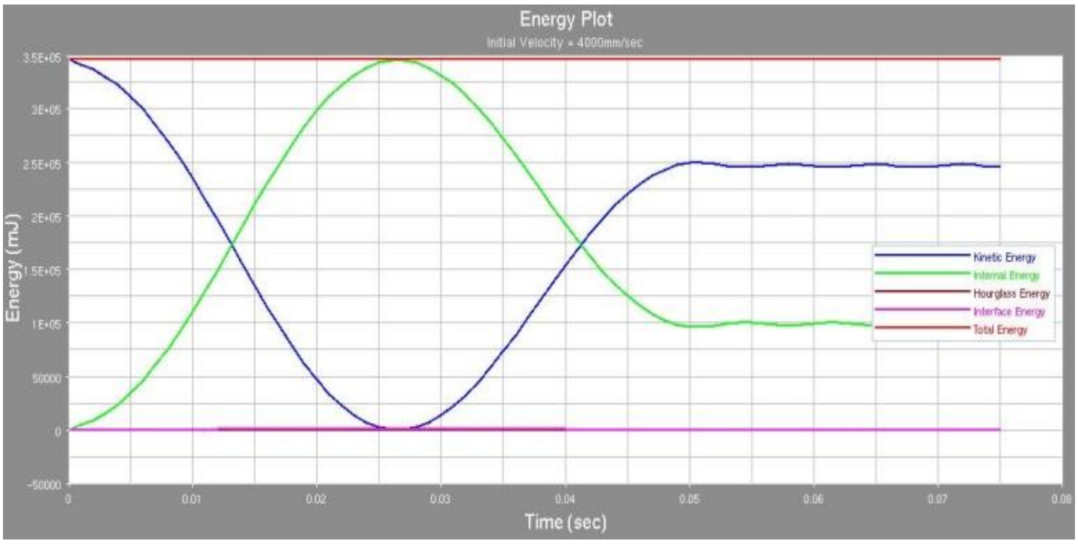
VII. RESULTS OF EXISTING DESIGN

A) Energy plots for various Initial velocities of the ram:

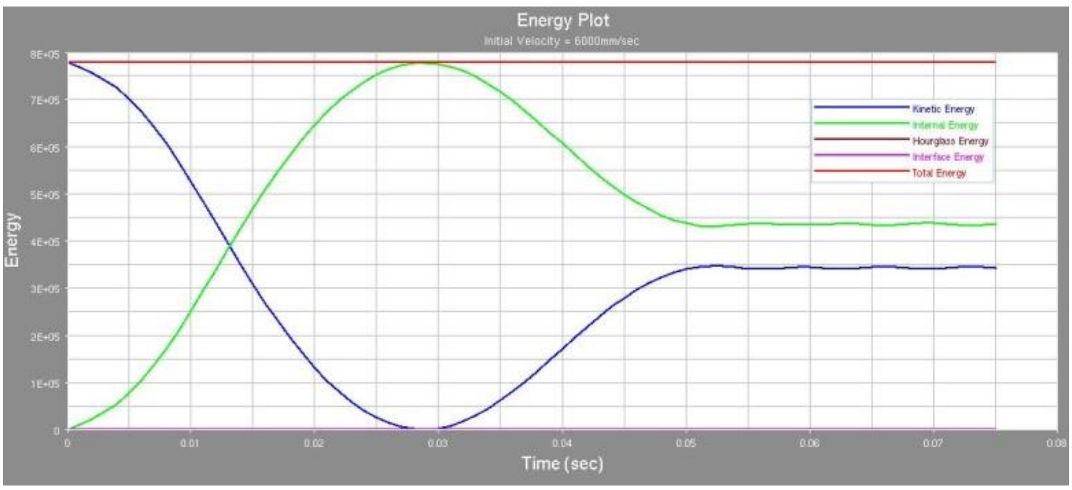
i) Energy (mJ) Vs time (s) for initial vel =2000mm/sec



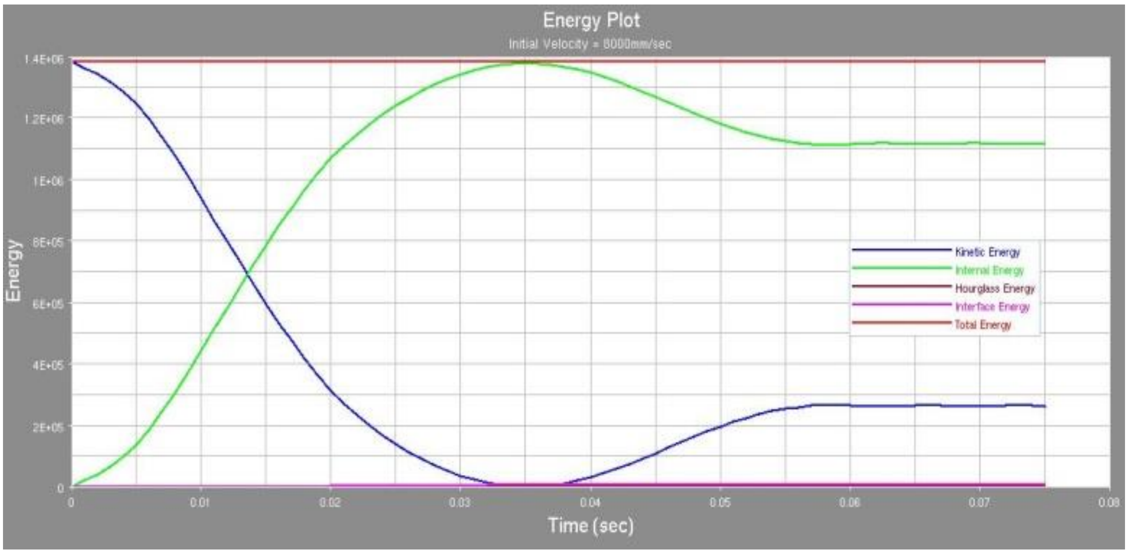
ii) Energy (mJ) Vs time (s) for initial vel =4000mm/sec



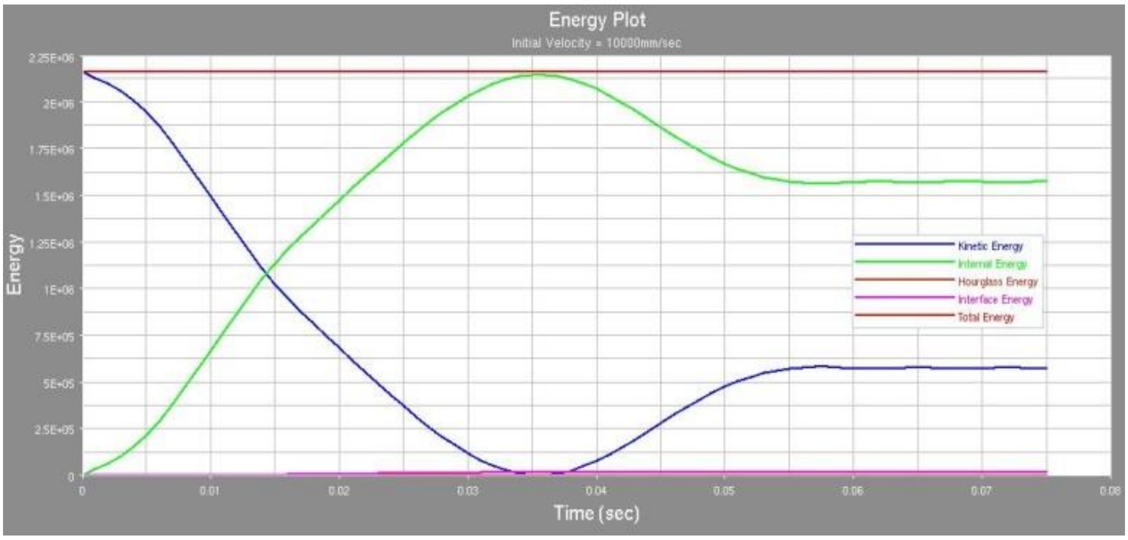
iii) Energy (mJ) Vs time (s) for initial vel =6000mm/sec



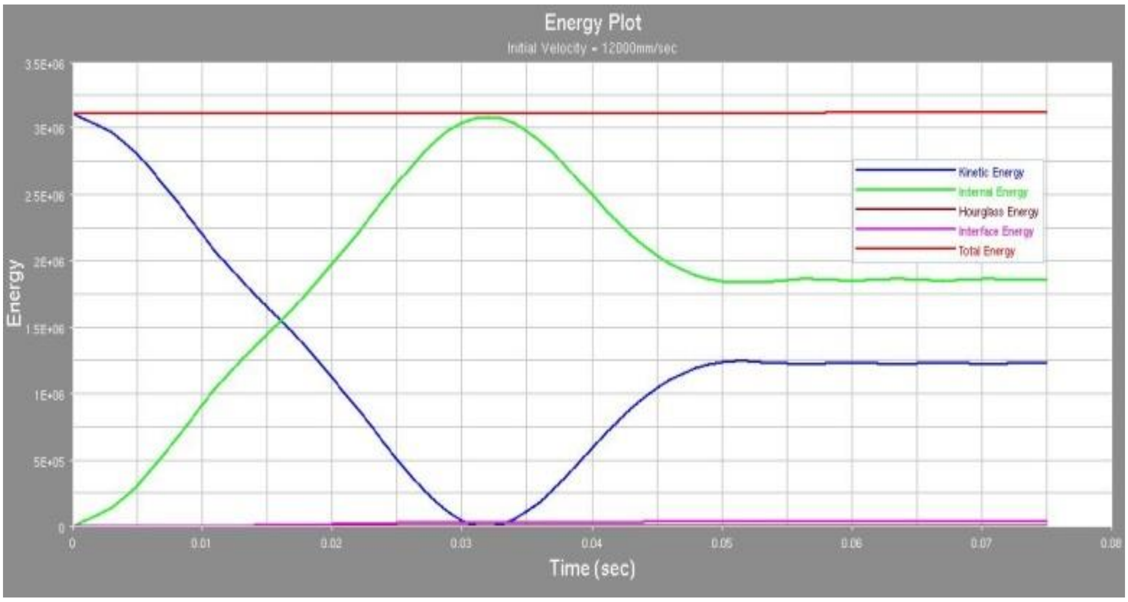
iv) Energy (mJ) Vs time (s) for initial vel =8000mm/sec



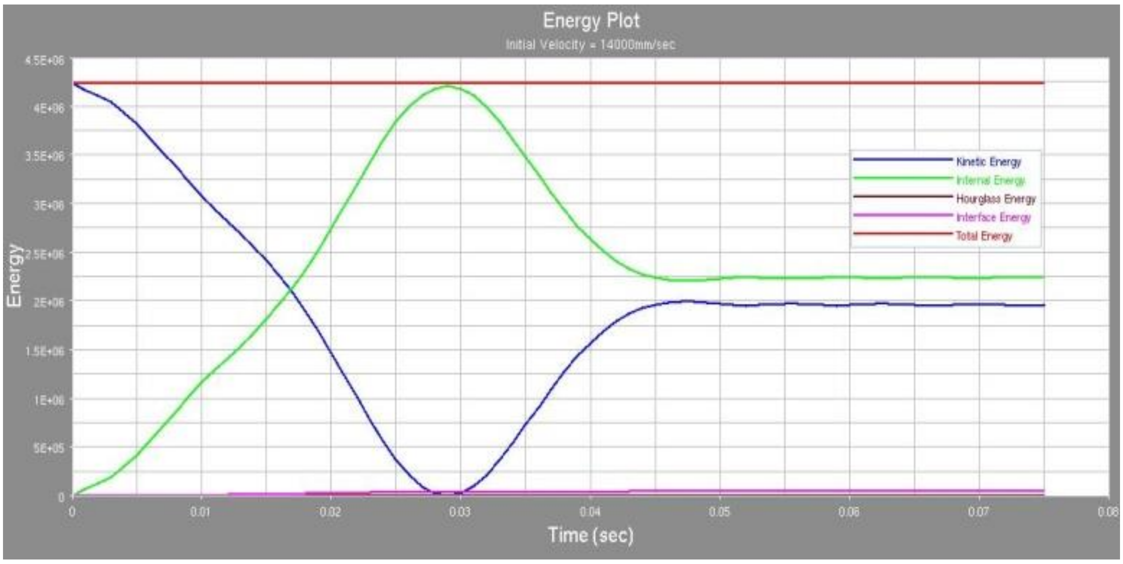
v) Energy (mJ) Vs time (s) for initial vel =10000mm/sec



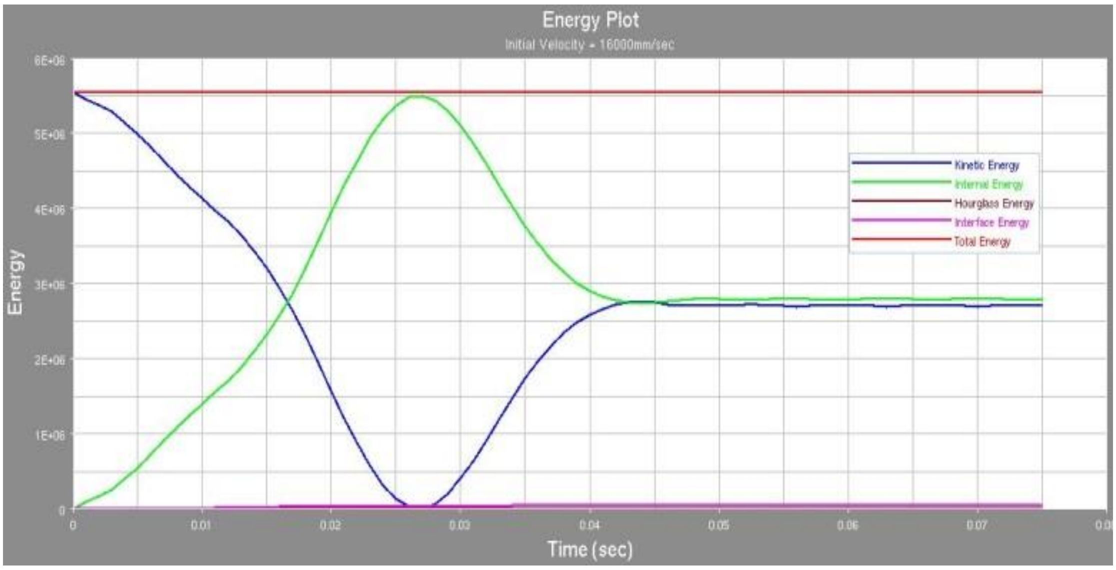
vi) Energy (mJ) Vs time (s) for initial vel =12000mm/sec



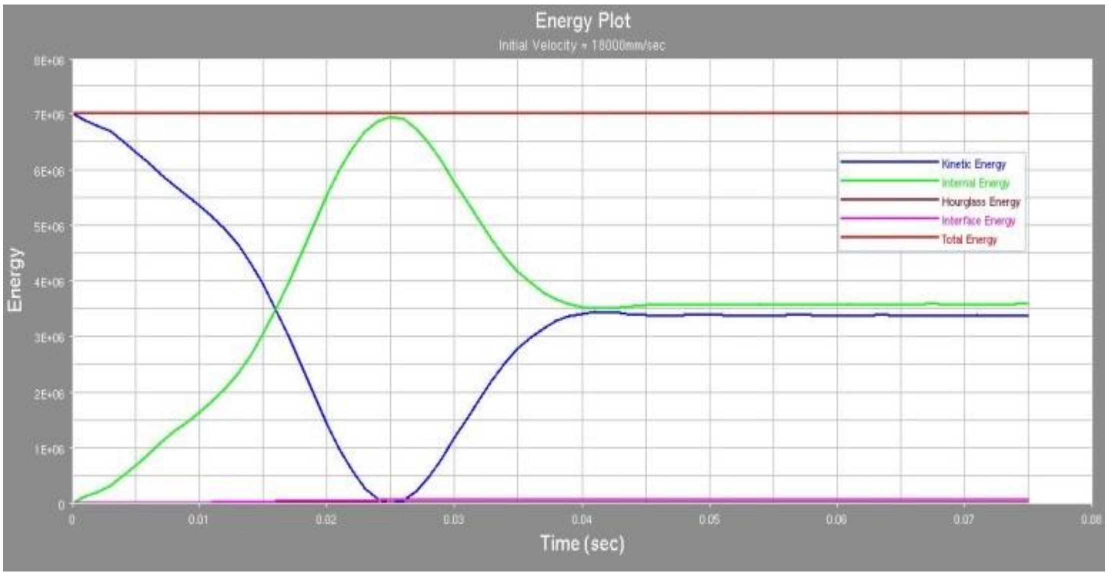
vii) Energy (mJ) Vs time (s) for initial vel =14000mm/sec



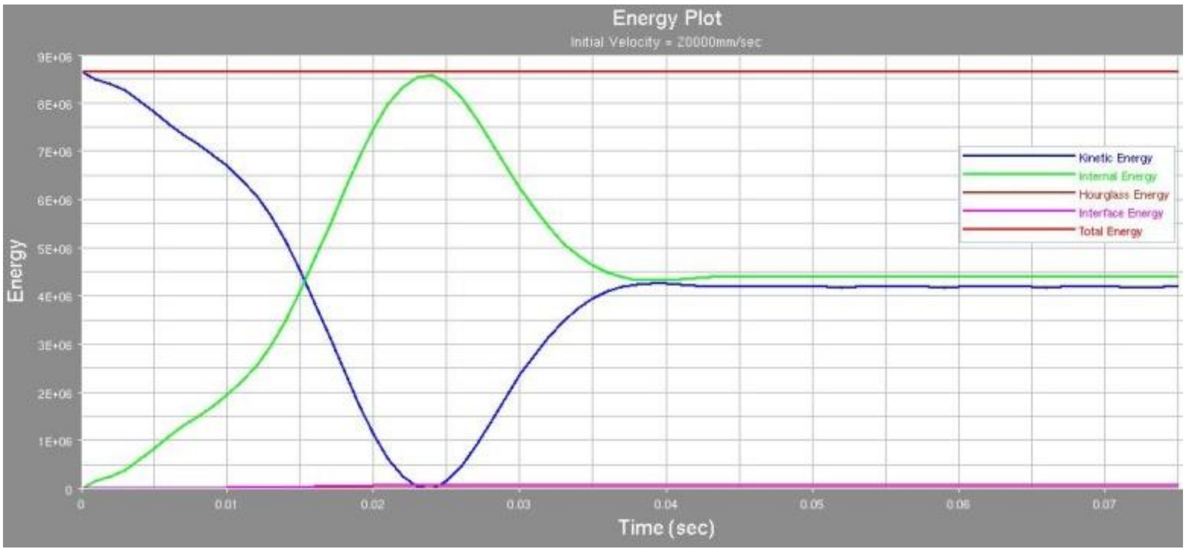
viii) Energy (mJ) Vs time (s) for initial vel =16000mm/sec



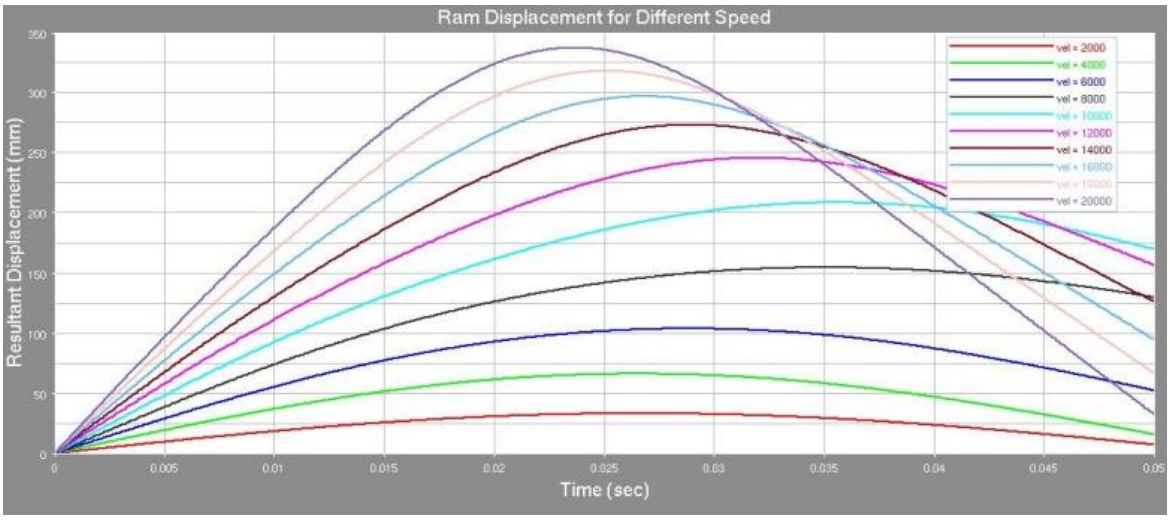
ix) Energy (mJ) Vs time (s) for initial vel =18000mm/sec



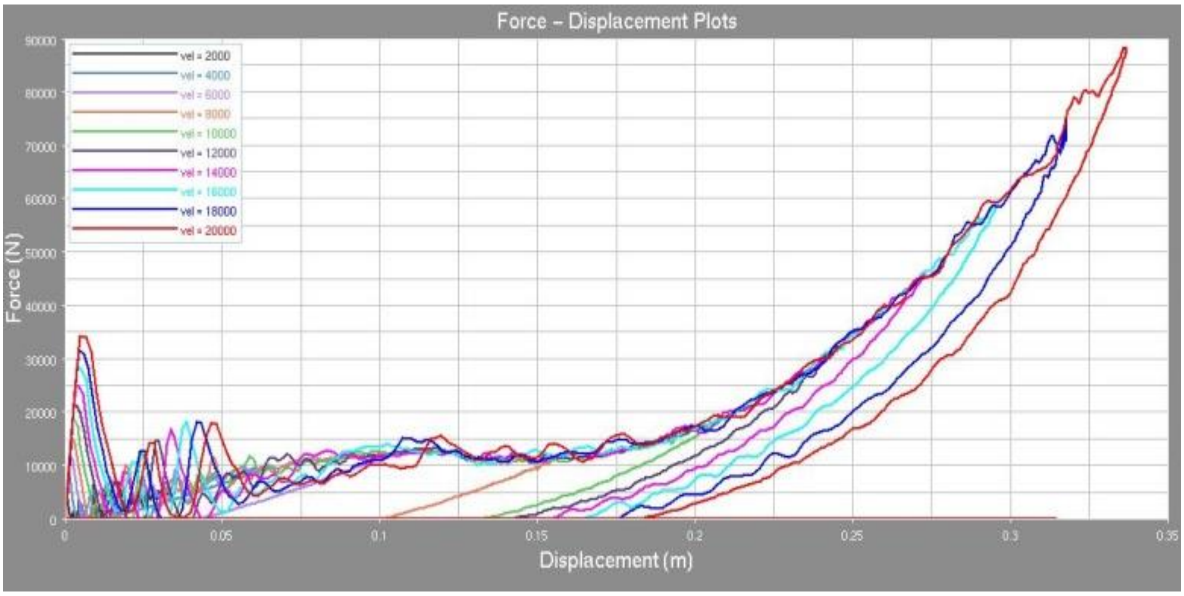
x) Energy (mJ) Vs time (s) for initial vel =20000mm/sec



B. Comparison of displacement vs time for all the velocities of RAM



C. Comparison of force displacement plots for all the velocities of RAM



International Journal of Mechanical and Industrial Technology ISSN 2348-7593 (Online)
Vol. 4, Issue 2, pp: (38-50), Month: October 2016 - March 2017, Available at: www.researchpublish.com

D. Stress contours for the base model for 6” ram travel for a ram velocity of 20,000mm/sec

Max stress obtained=1.54e3 MPa which is less than the ultimate strength for the beam material (high strength steel, UTS= 1620 MPa).

VIII. DESIGN IMPROVEMENT 1

A few design improvements were made to the door beam in order to increase its load carrying capacity for the same ram velocities. In the first design modification, the cross section of the door beam was modified as shown below:

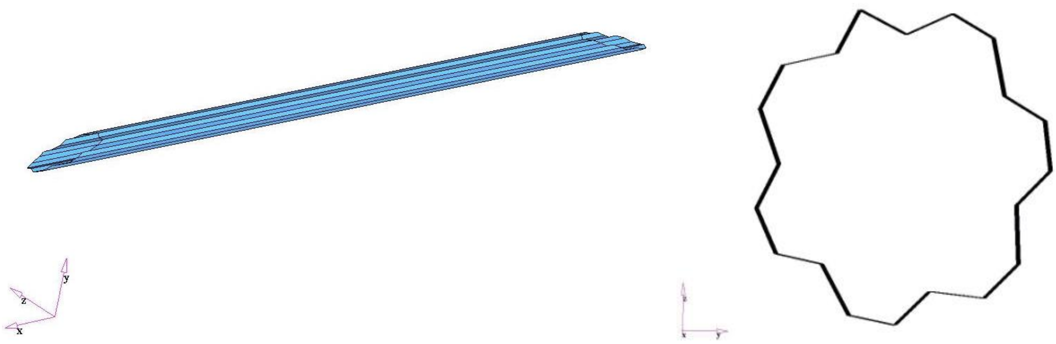
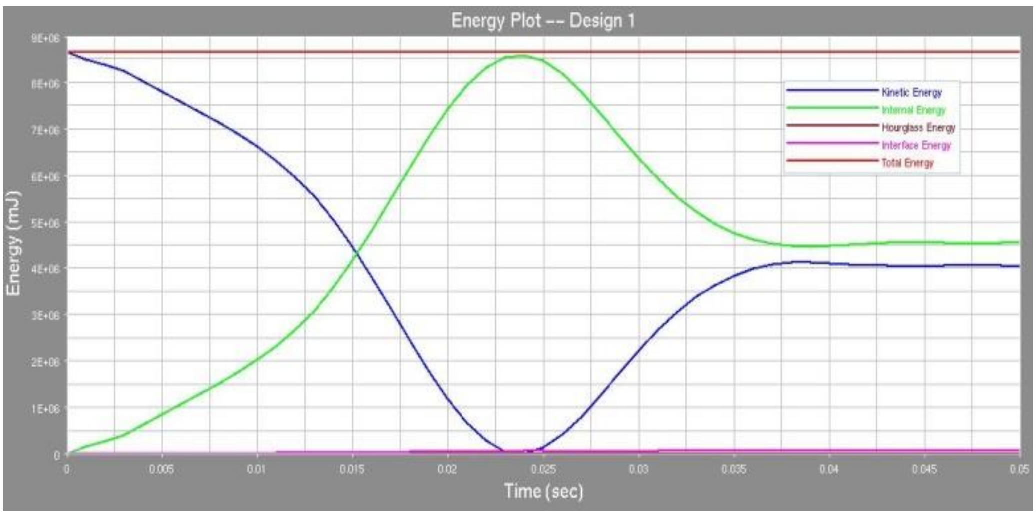


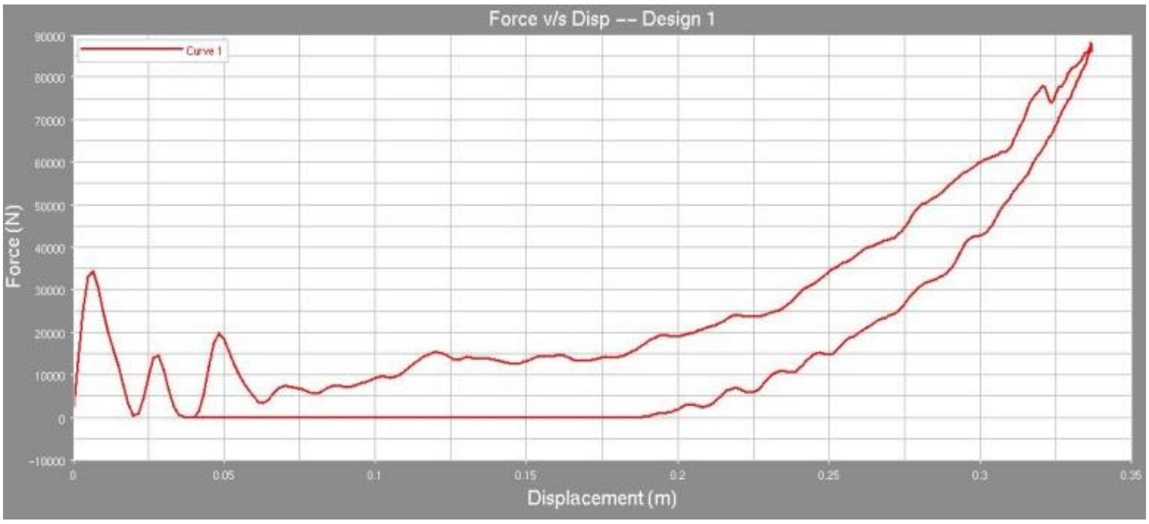
Fig 2. Modified C/S of the beam

i. Energy vs time for design modification 1



International Journal of Mechanical and Industrial Technology ISSN 2348-7593 (Online)
Vol. 4, Issue 2, pp: (38-50), Month: October 2016 - March 2017, Available at: www.researchpublish.com

ii. Force vs displacement for design modification 1



IX. DESIGN IMPROVEMENT 2

In the second design modification, solid circular rings were added on to the beam at the region where it comes in contact with the ram. 3D solid elements were used to create these rings. Again the analysis was run for this model and the results were compared.

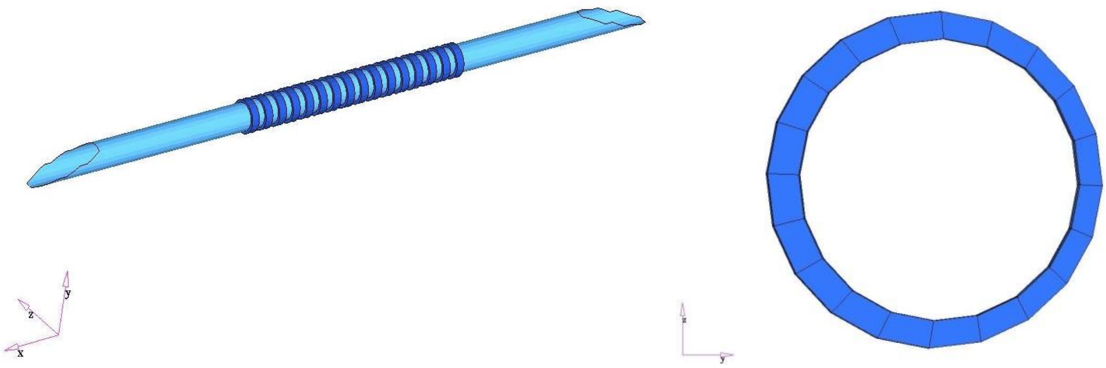
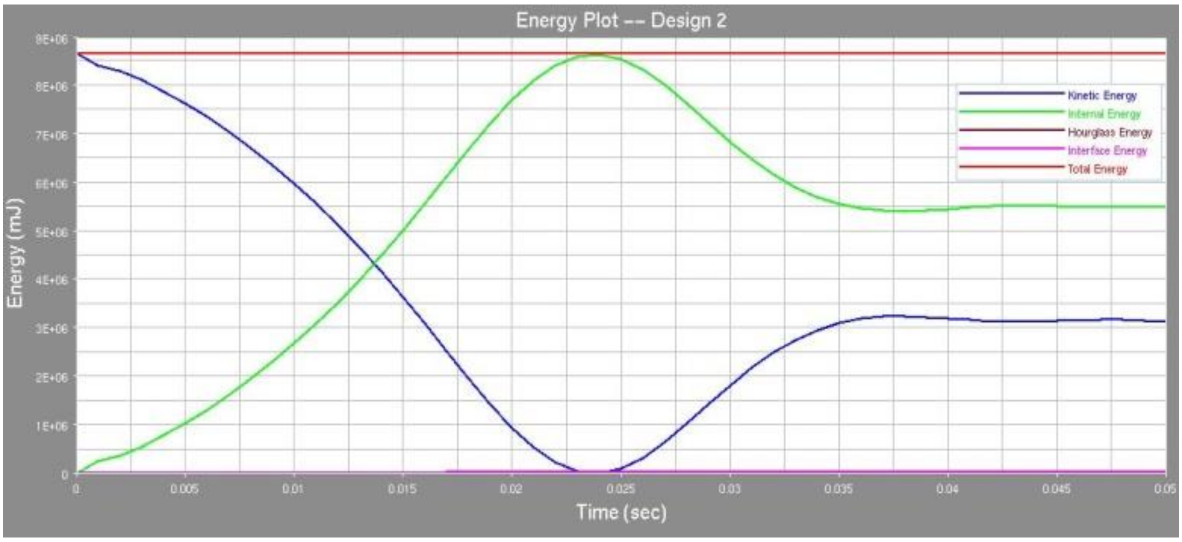


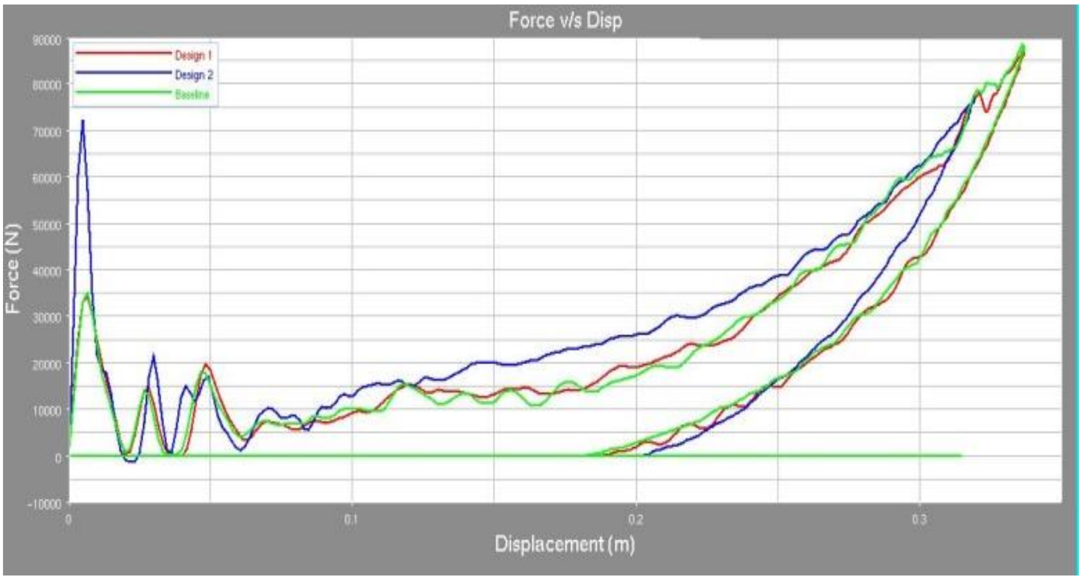
Fig 3. Modified C/S of the beam

i. Energy vs time for design modification 2



ii. Force vs Displacement for design modification 2

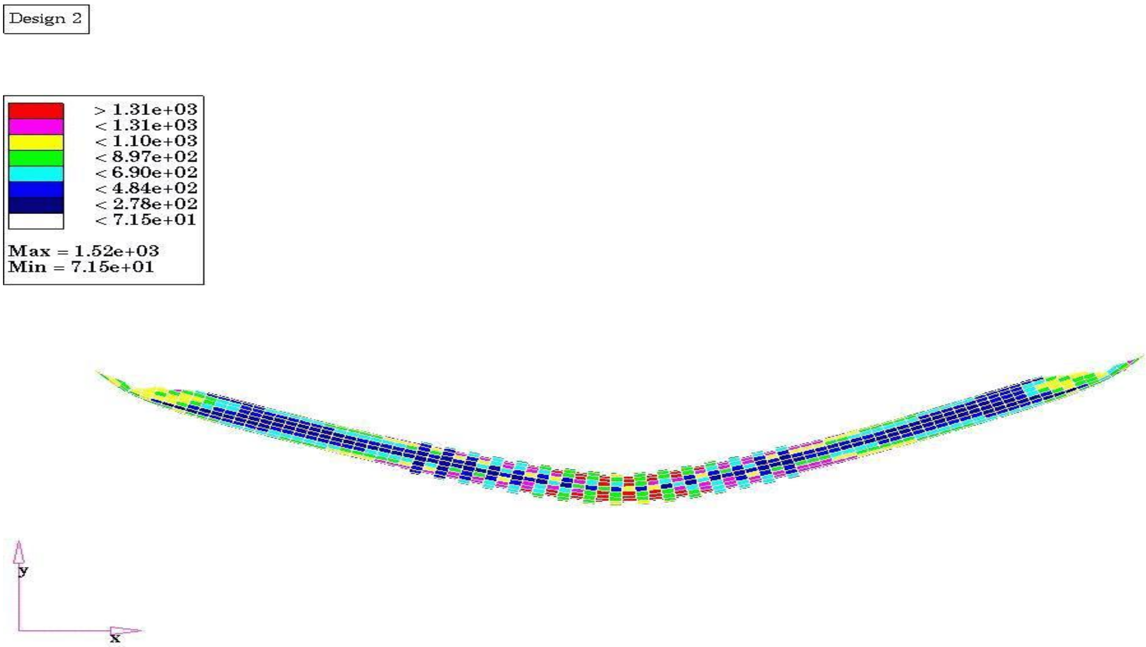
iii. Comparison of force - displacement plots for the 3 models used :



iv. Stress plot for design modification 1 for ram velocity of 20000mm/ sec and 6" ram travel:

Max von misses stress obtained =1.54 e3 MPa < UTS for High strength steel

v. Stress plot for design modification 2 for ram velocity of 20000mm/ second 6” ram travel:



Max Von misses stress oobtained =1.52 e3 MPa < UTS for High strength steel

X. RESULTS OF MODIFIED DESIGN

	Base model	**Design 1**	**Design 2**
Von Misses stresses at 6” ram travel	1550 MPa	1540 MPa	1520 MPa
Plastic strain	44 %	40%	23.7%
Load carrying capacity at 6” ram travel	11, 857 N	13, 369 N	20, 003 N
% Increase in load carrying capacity	-	12.75%	68.75 %
Energy absorbed by Door beam	4388 J	4530 J	5479 J
Percentage increase in energy	-	3.24%	24.8%
Energy absorbed at 6” ram travel	1571J	1627J	2103J
Percentage increase in energy absorption for 6” ram travel	-	3.56%	33.86%

XI. CONCLUSIONS

From the displacement plots it can observed that, 6” ram travel cannot be achieved for an initial ram velocity 2000 mm/sec. Hence the analysis was run for different velocities and it was observed that at 8000mm/sec and beyond, 6” ram travel has been achieved as mentioned in the problem. From the energy vs. time plots, it can be seen that kinetic energy decreases with time and the internal energy of the system increases at the same rate. Kinetic energy increases with increase of the initial velocity. This comparison can also be made from the graphs. Since kinetic energy and the internal energy curves follow the law of conservation of energy it can be concluded that the modeling of the system and FEA analysis has provided satisfactory results. The von misses stresses obtained at 6 “ ram travel can also be observed to be less than the ultimate tensile stress limit for the door beam material. So the beam does not fail at six inch ram travel.
From the force displacement plots, the load carrying capacity of the beam can be obtained. From these plots, the maximum energy the beam can absorb upon impact by a rigid body can also be calculated. The results obtained after making design modifications for the base model were satisfactory and showed improvements. From the results table it can be seen that the load carrying capacity of the beam has increased for both the design changes. With these modifications the beam can withstand stronger impacts and still not fail at 6 “ram travel. The Animation results give a clear picture of the ram hitting the beam and rebounding.

Scope For Further Improvements:

Use of composite materials instead of steel for the door beam can increase the strength and at the same time reduce weight, which is most desirable in a car**.** Composites have very high strength and stiffness-to-weight ratios in the fiber direction, as well as in the direction perpendicular to the fiber even though their Young’s modulus is 15-times lower than that of the steel. This means that composite members will necessarily have higher load carrying capacities than steel. The disadvantages of composites, in comparison to steel, are higher production and tooling costs. The method of manufacture is also expensive.

REFERENCES

[1] Nitin S. Gokhale, Sanjay S.Deshpande, Sanjeev V. Bedekar "Practical Finite Element Analysis"

[2] Ahmed Elmarakbi “Advanced Composite Materials for Automotive Applications", Structural Integrity and Crashworthiness.

[3] Drabek, Pavel, Milota, Jaroslav "Methods of Nonlinear Analysis", Applications to Differential Equations.

[4] Giancarlo Genta "Motor Vehicle Dynamics" Modeling and Simulation

[5] Matthew Huang "Vehicle Crash Mechanics"

[6] William F. Milliken, Douglass L.Milliken, Maurice Olley "Chassis Design" Principles and Analysis

[7] Shun-Hsyung Chang, Ivan Parinov, Vitaly Yu. Topolov, Advanced Materials: Physics, Mechanics and Applications

[8] FMVSS Standards - https://icsw.nhtsa.gov/cars/rules/import/FMVSS/

[9] NCAP - http://www.globalncap.org/