International Journal of Technical Innovation in Modern Engineering & Science (IJTIMES)
Impact Factor: 5.22 (SJIF-2017), e-ISSN: 2455-2585
Volume 5, Issue 03, March-2019

CAR BUMPER CROSS SIMULATION WITH VERIOUS MATERIALS

K. Hariprasad Reddy[1], K. Trinath kumar[2,] B.Anjaneyulu[3], S.L.V Prasad[4]

[1]*PG Scholar Dept. of Mechanical Engineering, Gates Institute of Technology, Gooty, A.P.-515401, India*
[2]*Individual Researcher of Mechanical Engineering, Anantapur, A.P. India*
[3]*M.Tech, (PhD), MIAENG and MIE*
[4]*M.Tech, (PhD)*

Abstract— Bumper is one of the main parts which are used as protection for passengers from front and rear collision. The most important parameters including material, thickness, shape and impact condition are studied for analysis of an automotive front bumper beam to improve the crash worthiness design in low velocity impact. In this project, a front bumper beam made of three materials: aluminium, plastic (glass mat thermoplastic) and steel (high-strength sheet moulding compound) is studied by impact modelling to determine the deflection, impact force, stress distribution and energy absorption behaviour. The mentioned characteristics are compared to each other to find best choice of material, shape and thickness. The results show that a modified SMC bumper beam can minimize the bumper beam deflection, impact force and stress distribution and also maximize the elastic strain energy. In addition, the effect of passengers in the impact behaviour is examined. The time history of the calculated parameters is showed in graphs for comparison. Furthermore, beside the above-mentioned benefits, some more advantages like easy manufacturing due to simple shape without ribs, economical aspects by utilizing low-cost composite material and reducing weight with respect to others can be achieved by steel material.

Keywords— Car bumper, shape and thickness, Aluminium, Plastic (glass mat thermoplastic) and Steel, CAD, Hypermesh (LS_DYNA), Deflection, Impact force, Stress distribution and Energy absorption.

I. INTRODUCTION

Car accidents are happening every day. Most drivers are convinced that they can avoid such troublesome situations. Nevertheless, 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. The cover of the car bumper is called fascia. An automobile 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. Car frontal and rear fascia is designed to prevent or reduce physical damage to the front or rear ends of passenger motor vehicles in collision condition. When the bumper is impacted by a stiff object, such kind may happen in a parking accident or in the legislative low speed impact pendulum test, then the bumper fascia alone may not be there to withstand the impact without considering the forces acting on it. Thus, there were four main strategic parameters being studied during the test. Firstly, how the type of material can affect the impact specifications and what kind of materials can be used as replacement in order to lower part weights. The effect of module of elasticity, yield strength and Poisson's ratio on impact behaviour of bumper beam was under investigation in this section. Secondly, how the bumpers beam thickness can affect the impact specifications. Thirdly, how even small changes and modifications can result in easier manufacturing processes and lessening material volume without lowering the impact strength. A bumper is a car shield made of steel, aluminium's, 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. Some bumpers use energy absorbers or brackets and others are made with a foam cushioning material. 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.

1.1 Components of bumper system

Bumper is a one of the main structures of passenger cars that protect them from front and rear collisions. It is the front-most or rearmost 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. The components of bumper system are as follows,

a) Fascia *b) Energy absorber* *c) Face bar* *d) Reinforcing beam*

1.2 Types of bumper systems

Metal face bar & Plastic fascia and reinforcing beam
Plastic fascia, reinforcing beam and mechanical energy absorbers
Plastic fascia, reinforcing beam and foam or honeycomb energy absorber
Plastic fascia, reinforcing beam, foam, and mechanical energy absorber

1.3 Crash Test

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. 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. The most common types of crash tests are listed below

Frontal impact *Side impact* *Pedestrian impact* *Pole impact*

2. LITERATURE REVIEW

Hosseinzadeh et al., (2005)[1] in their paper says that bumper beams are one of the main structures of passenger cars that protect them from front and rear collisions. In their paper, a commercial front bumper beam made of Glass Mat Thermoplastic (GMT) is studied and characterized by impact modelling using LS-DYNA ANSYS 5.7 according to the E.C.E. United Nations Agreement, Uniform Provisions concerning the Approval of Vehicles with regards to their Front and Rear Protective Devices (Bumpers, etc.), E.C.E., 1994]. Three main design factors for this structure: shape, material and impact conditions are studied, and the results are compared with conventional metals like steel and aluminium. Finally, the aforementioned factors are characterized by proposing a high strength SMC bumper instead of the current GMT. Evans D and Morgan (1999),[2] have studied that as vehicle manufacturers continue to become more aggressive with the styling of new vehicles, bumper system technologies will be required to find new solutions that fit into the reduced package spaces while continuing to meet the vehicle performance and cost requirements. It was suggested to introduce new and innovative Expanded Polypropylene (EPP) foam technologies and techniques. Sayyad F.B et al., (2013), [3]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. Bautista Garret et al., (2009),[4]they optimized the shape of bumper beam by performing the software simulation. They also studied the effect of metallic energy absorber in bumper system. Maximum stress and deformation were used as design criteria. They have complied many international standards for bumper beam design. Marzbanrad JM et al., (2009),[5]discussed the most important parameters including material, thickness, shape and impact condition are studied for design and analysis of an automotive front bumper beam to improve the crashworthiness design in low-velocity impact. The simulation of original bumper under condition impact is according to the low-speed standard of automotives stated in E.C.E. United Nations Agreement Regulationno.42, 1994. In this research, a front bumper beam made of three materials: aluminium, glass mat thermoplastic (GMT) and high-strength sheet moulding compound(SMC) is studied by impact modeling to determine the deflection, impact force, stress distribution and energy-absorption behaviour. The results show that a modified SMC bumper beam can minimize the bumper beam deflection, impact force and stress distribution and also maximize the elastic strain energy. Anderson R et al.,(2002),[6] has discussed that to increase crash performance in automotive vehicles it is necessary to use new techniques such as use of energy absorber and materials. Components linked to crash safety should transmit or absorb energy. The energy absorbing capability of a specific component is a combination of geometry and material properties. Duncan C et al.,(2004),[7]Simulated crash-testing is being increasingly by various institutes to study the outcome of a vehicular in various situations under different conditions. The advantage of simulation is that the FE models can be reused again and again and also the user has the freedom to change any of the parameters of the test and also the user can vary the material properties as well as the type of material of the parts in the vehicle.

3. METHODOLOGY

3.1 General procedure for crash analysis

Crash simulation is non linear explicit in nature. In other words, it is time dependent. The deformations can be calculated at any instant of time. The following step by step procedure is to be followed to accomplish crash analysis with better results.

1) Build Finite Element Model *2) Apply Loads and Boundary Conditions* *3) Assign Material Properties* *4) Control Parameters* *5) Run the Analysis* *6) Review and Interpret Results*

3.2 Hypermesh

Hyper Mesh is product of design and development of Altair Engineering. It is software which is based on CAE models as these CAD models were imported through this software and further the Meshing process occurs. The meshing of a model plays a very important role in analysis of an object. Through this analysis various amount of forces, stresses and displacement can be calculated. As there are numerous of software which provides quite appreciable analysis of these objects. But the basic purpose to choose this software for analyzing is that it gives quite understandable results as well as it is famous for crash analysis. In today's world crash analysis is now getting importance very much for safety and

security of the passengers as well as for the vehicles. Altair Hyper Mesh is a high-performance finite element pre-processor to prepare even the largest models, starting from import of CAD geometry to exporting an analysis run for various disciplines. In the year 1990, Hyper Mesh was released by Altair and in the year 1994, Altair receives Industry Week's "Technology of the Year" award for Opti Struct. Hyper Mesh enables engineers to receive high quality meshes with maximum accuracy in the shortest possible time. A complete set of geometry editing tools helps to efficiently prepare CAD models for the meshing process. Meshing algorithms for shell and solid elements provide full level of control, or can be used in automatic mode. Altair's Batch Meshing technology meshes hundreds of files precisely in the background to match user-defined standards. Hyper Mesh offers the biggest variety of solid meshing capabilities in the market, including domain specific methods such as SPH, NVH or CFD meshing with a focus on engineering productivity.

3.3 LS DYNA

LS-DYNA is a general-purpose finite element code for analyzing the large deformation static and dynamic response of structures including structures coupled to fluids. The main solution methodology is based on explicit time integration. An implicit solver is currently available with somewhat limited capabilities including structural analysis and heat transfer. A contact-impact algorithm allows difficult contact problems to be easily treated with heat transfer included across the contact interfaces. By a specialization of this algorithm, such interfaces can be rigidly tied to admit variable zoning without the need of mesh transition regions. Other specializations allow draw beads in metal stamping applications to be easily modeled simply by defining a line of nodes along the draw bead. Spatial discretization is achieved by the use of four node tetrahedron and eight node solid shell elements, two node beam elements, three and four node shell elements, truss elements, membrane elements, discrete elements and rigid bodies. A variety of element formulations are available for each element type. Specialized capabilities for air bags, sensors and seat belts have tailored LS-DYNA for applications in the automotive industry. It is an analysis software package for use in predicting how a prototype will respond to real-world events. It enables users to produce virtual prototypes, realistically simulating the full-motion behaviour of complex mechanical systems on their computers and quickly analyzing multiple design variations until an optimal design is achieved. Among the applications that LS-DYNA is used for are crashworthiness simulations, metal forming, biomedical applications and earthquake engineering.

3.5 Mesh quality

Element quality is a subject often talked about and never fully understood. The reason for this is complex but is related to the fact that quality is relative and the solution, by definition, is approximate. In the formulation of finite elements a local parametric coordinate system is assumed for each element type and how well the physical coordinate systems, both element and global, match the parametric dictates element quality. Below you see some graphics representing element quality and you should attempt to follow them, however, there will be a point of diminishing return if you try too hard to get every element within the acceptance criteria. Check with "local experts" regarding the appropriate values for each element type required by your element checking computer programs.

3.13 Simulation steps
3.13.1 Design

*The Toyota Camry 2012 model car fascia is designed by using Auto CAD software as per the dimensions. The holes are incorporated in designing the fascia to withstand the drag forces when the vehicle is in motion. The following figure shows the CAD model of fascia as shown in Fig 3.14 and 3.*15

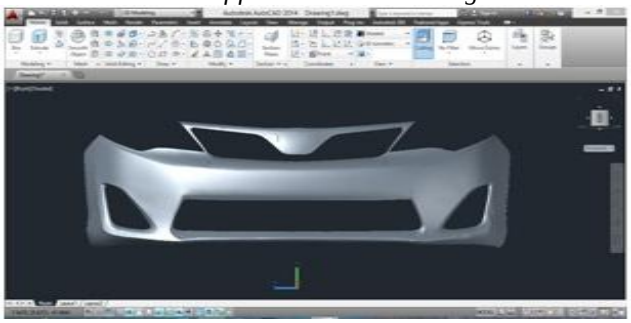


Fig 3.14 Front View of Fascia in AutoCAD

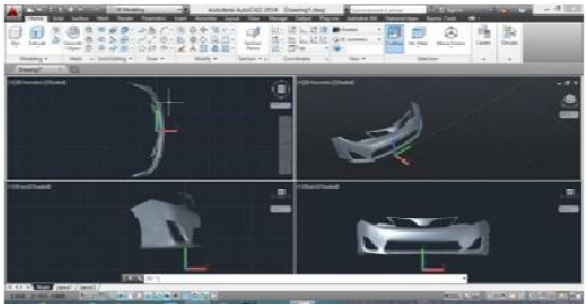


Fig 3.15 Views of Fascia in AutoCAD

3.13.2 Meshing

Toyota Camry 2012 model car fascia is taken which is designed in Auto CAD. The file then exported to Hypermesh software for meshing purpose. To import the file in hypermesh the file should be in .hm format. The imported file in hypermesh window in shown the Fig 3.16 and 3.17

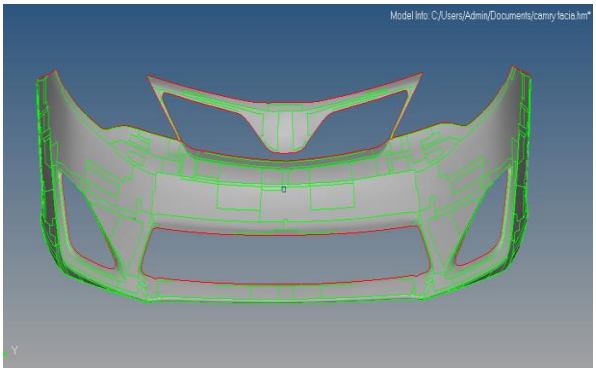


Fig 3.16 Model of Toyota Camry Fascia

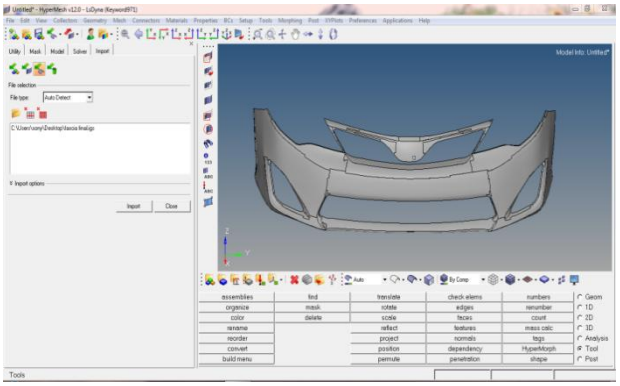


Fig 3.17 Toyota Camry Fascia Imported in Hypermesh



No of points = in finite
Dof per point = 6
Total equations = in finite



No of points = 8
Dof per point = 6
Total equations = 48

Then the imported model is meshed finely with the elements Trias and Quads (Mixed mode). Mixed mode is commonly preferred due to better mesh pattern(total tria %<5). For crash or non linear analysis symmetric mesh flow lines with all the elements satisfying required quality parameters is very important. Mix-mode instead of pure quad helps to achieve better flow lines and convergence of solution

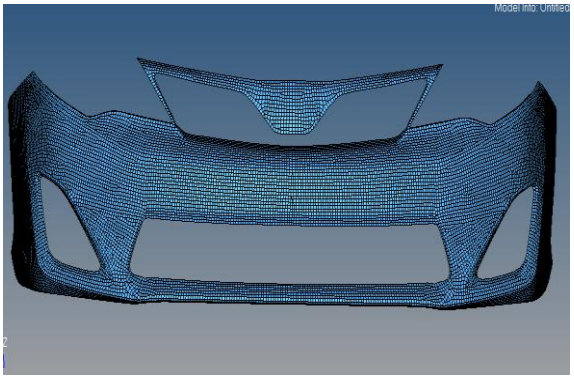


Fig 3.18 Toyota Camry Fascia Meshed

trias:

warpage	5.000	length	7.500	min angle	20.000
aspect	5.000	length	20.000	max angle	120.000
skew	60.000	jacobian	0.700	quads:	
chord dev	0.100	equia skew	0.600	min angle	45.000
cell squish	0.500	area skew	0.600	max angle	135.000
		taper	0.500		

Table 3.5 Mesh Quality Parameters

The imported fascia is not meshed directly. First, a mid surface is extracted because mathematically element thickness (specified by user) is assigned half in +Z axis (element top) and half in -Z axis (element bottom). Hence, for appropriate representation of geometry via 2-d mesh it is necessary to extract mid surface and generate nodes and elements on the mid surface. The thickness of fascia is 2.15 mm. The meshed fascia consists of 18182 nodes and 17675 elements throughout its surface. The size of element maintained throughout meshing is 4 mm.

3.13.3 Components

In this section, the components need to simulate are created and properties, boundary conditions are applied. In our work, four components are created such as Rigid Wall, Fascia, Mass Point and Rigid Connectors. The rigid wall is created by creating four nodes which represent four corners of the wall. The mass of the wall is 20 kg and area is nearly $2.603*10^6$ mm^2. After creating the wall, a mass point is created at the point where centre of gravity of car acts. It is so created because considering the entire finite element model of car and analyzing it consumes a lot of time and uneconomical. So instead of considering the entire car model, a mass point is created which is equal to the weight of the car. Then, the created mass point and fascia are connected through rigid connectors as shown in Fig. 3.19.

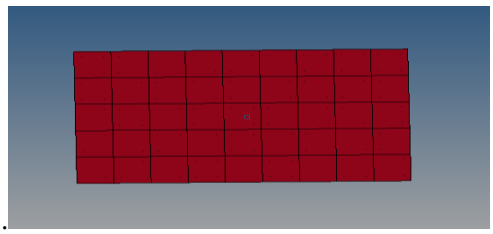


Fig 3.19 Rigid Wall

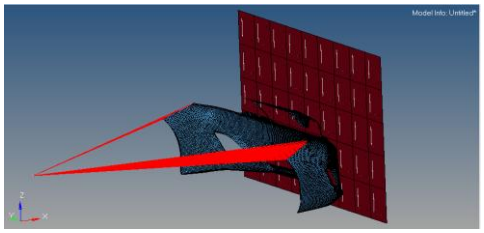


Fig 3.20 Components Created in Hypermesh

After creating the mass point that coincides with the centre of gravity of car, it is connected to the car fascia with rigid connectors. Rigid connectors are shown in red colour The rigid connectors and mass point are used instead of entire car mode as shown in the Fig 3.20.

Our work is mainly focused on the analysis of car frontal fascia. So instead of considering the entire car model, a mass point is created at the centre of gravity of car equal to that of car weight. The mass of the car is found to be 1623 kg without considering fascia. The weight of the car changes accordingly with the material used. This is discussed clearly in the next section. Considering the entire car model involves number of connections to be given to each part. So it is a risky process and time consuming. As we are not analysing the entire car model it can be eliminated by creating a mass point which is equal to the weight of car.

Co-ordinates for Mass Point:

X = -1992.50 mm Y = 63.153 mm Z = 509.404 mm

Mass point = 1623 kg, Mass of fascia with steel material = 19.136 kg

Mass of fascia with Aluminum material = 6.354 k, Mass of fascia with plastic material = 4.912 kg

Total car with Steel material fascia = 1623 + 19.136 = 1642.136 kg

Total car with Aluminum material fascia = 1623 + 6.354 = 1629.354 kg

Total car with Plastic material fascia = 1623 + 4.912 = 1627.912 kg

3.13.4 Materials

*In this section, materials are assigned to the components. If the any component is given material *MAT_20 RIGID, the software considers rigid properties by default. The values of young's modulus, Poisson ratio, mass density, yield strength doesn't affect the deformation of rigid wall during impact. However, young's modulus value must be far higher than the slave. But the constraints given to the wall plays a major role.*

Table 3.6. Material Properties of Master and Slave

Materials	***Properties***				
	Young's Modulus(GPa)	***Density (kg/mm^3)***	***Poisson Ratio***	***Yield Stress(N/mm^2)***	***Ultimate Stress(N/mm^2)***
Plastic	*1*	*2.01e-06*	*0.45*	*26*	*40*
Aluminum	*80*	*2.60e-06*	*0.33*	*35*	*90*
Steel	*210*	*7.83e-06*	*0.30*	*250*	*420*

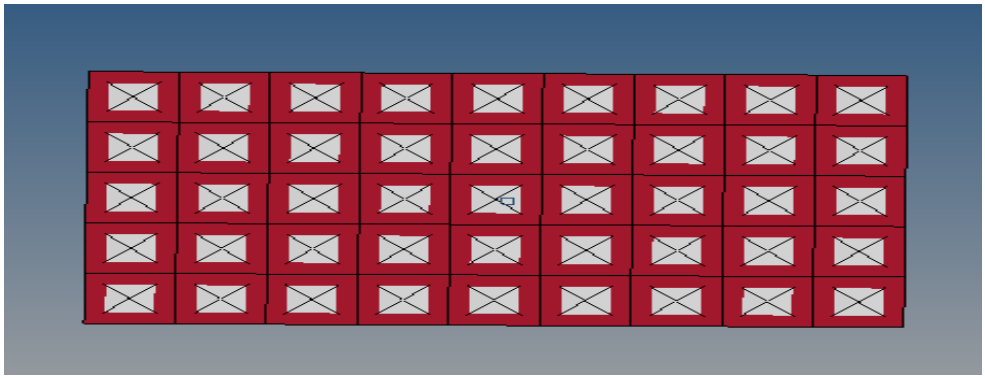


Fig 3.21 Rigid Wall with Constraints

3.13.5 Contacts

Contact occurs when two bodies come towards each other during deformation process. A force is transmitted at the interface during the contact. If there is no contact, the two components simply penetrate each other. There are two considerations in giving contacts namely master and slave. Master is the component with higher stiffness and higher Young's modulus. In our work, rigid wall is considered as master. Slave is the components which is deformable and have less stiffness and young's modulus value compared to master. Hence, fascia is considered as slave. The color of fascia and rigid wall changes to red which indicates that the contact is applied as shown in Fig. 3.22 and 3.23.

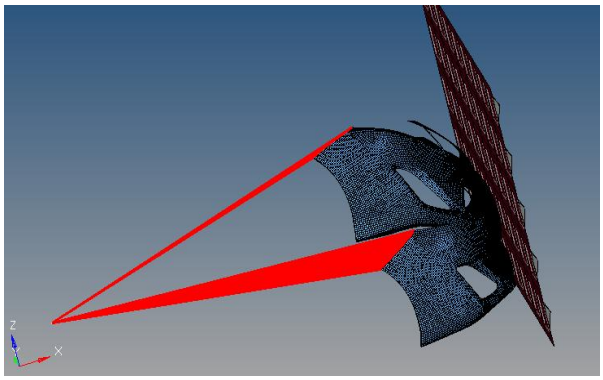


Fig 3.22 Before Contact is Applied between Fascia & Rigid Wall

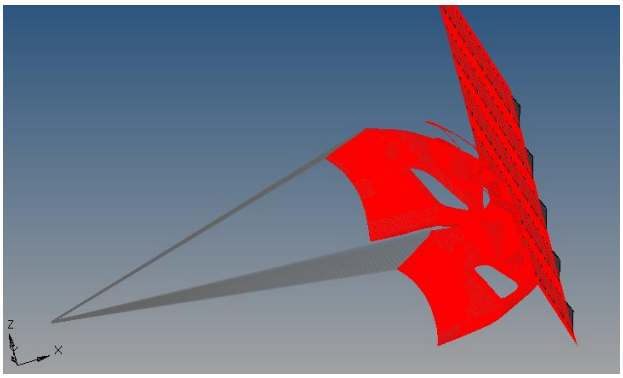


Fig 3.23 After Contact is Applied between Fascia and Rigid Wall

3.13.6 Sets

To assign the velocity for fascia, the velocity should be given to all the nodes on fascia. It is difficult to assign velocity to each and every node. So it is better to create a set for all the nodes so that the velocity for all the nodes on the fascia can be assigned at a time.

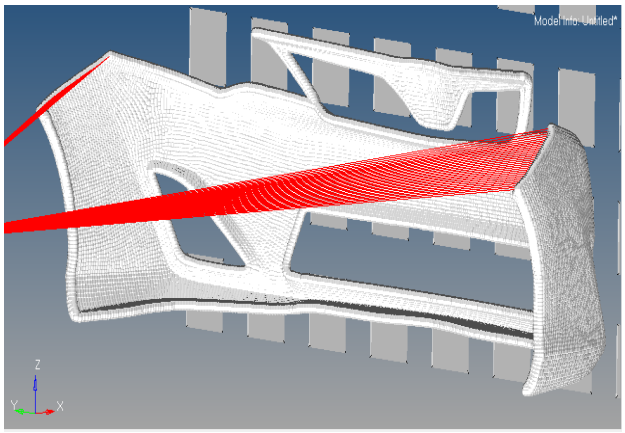
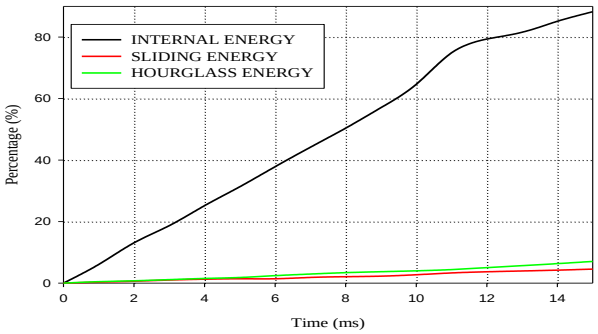


Fig 3.24 Node Set for Assigning Velocity

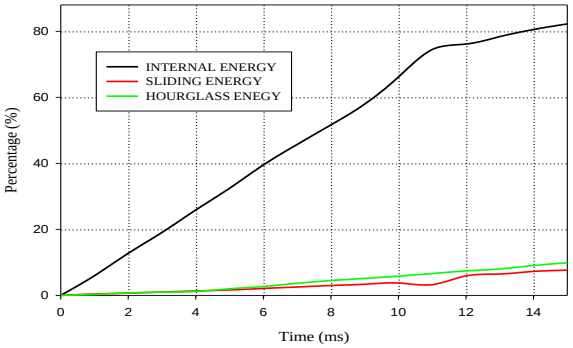
RESULTS AND DISCUSSIONS

The Toyota Camry model 2012 is considered for analyzing the energy absorption characteristics. Three different materials such as Steel, Aluminum and Plastic are considered. A velocity of 30 mm/ms i.e. 108 km/hr is assigned and a run time of 15 ms is given and the results are interpreted as follows.

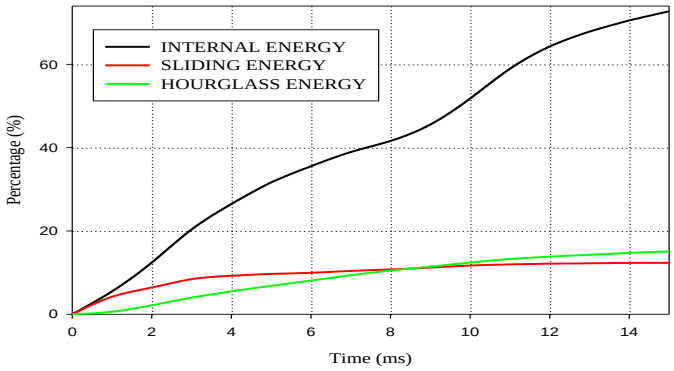
4.1 Energy balance curves



4.1 Energy Balance Curve for Steel Material



4.2 Energy Balance Curve for Aluminum Material



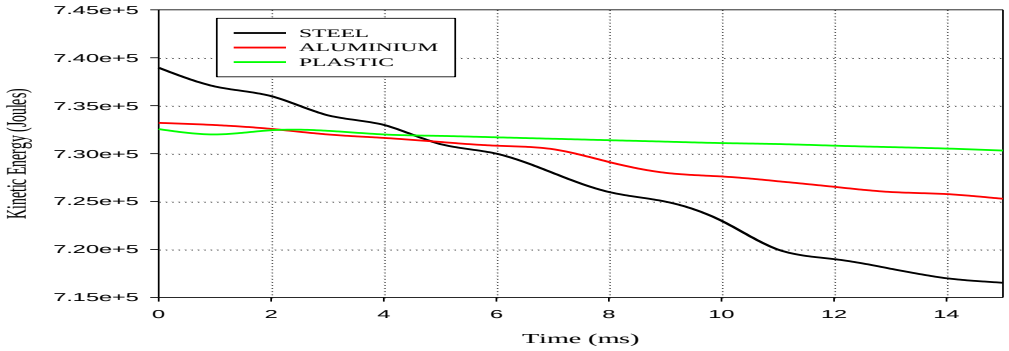
4.3 Energy Balance Curve for Plastic Material

From the Graph 4.1 it is clear that the absorption of Internal Energy (IE), Sliding Energy (SE) and Hour glass Energy (HE)for steel is 88.25, 7.05, 4.55% respectively. The summation of all the energies leads to 99.85% which indicates the energy ratio is approximately one with an error of 0.15%. At 0 m/s, the percentage of internal energy. It shows that the kinetic energy lost during impact is appeared in the form of IE, SE and HE.

From the Graph 4.2 it is clear that the absorption of Internal Energy (IE), Sliding Energy (SE) and Hour glass Energy (HE)for aluminum is 82.25, 7.63, 9.88% respectively. The summation of all the energy leads to 99.76% which indicates the energy ratio is approximately one with an error of 0.24%. It shows that the kinetic energy lost during impact is appeared in the form of IE, SE and HE.

From the Graph 4.3 the absorption of Internal Energy (IE), Sliding Energy (SE) and Hour glass Energy (HE) for plastic is 72.73, 12.35, 15.08% respectively. It is observed clearly that up to 8 m/s the rise of sliding energy is higher than Hour glass energy but after 15 m/s sliding energy is less than Hour glass Energy. Initially the plastic deforms more without absorbing the energy which makes it unstable.

4.2 Variation of kinetic energy

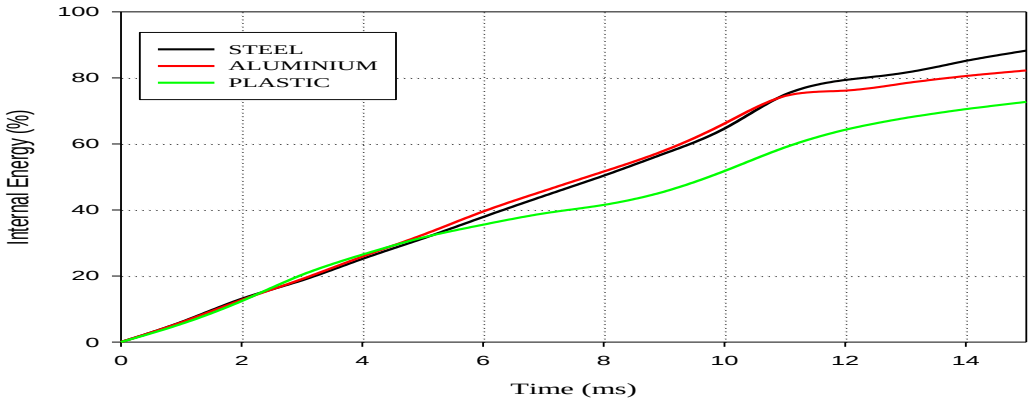


Graph 4.4 Kinetic Energy with respect to Time

Kinetic energy lost during impact by steel, aluminum and plastic is 22430, 7920 and 2240 J respectively. In terms of percentages, the kinetic energy lost from the total energy for steel, aluminum and plastic is 3.04, 1.09 and 0.31%

respectively. It is noted that, the initial kinetic energy for these materials is different due to different masses. The kinetic energy for steel falls drastically about 3.04% from the total energy.

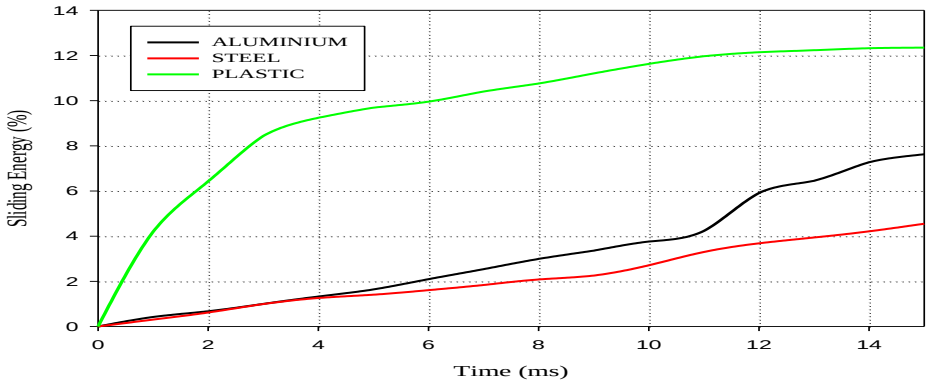
4.3 Variation of internal energy



Graph 4.5 Internal Energy with respect to Time

From Graph 4.5, Internal Energy with respect to Time, it is clear that, the percentage of internal energy absorbed by steel, aluminum and plastic is 88.25, 82.25 and 72.74% respectively. It is because steel is much stiffer than aluminum and plastic. Also, the young's modulus value of steel is higher than aluminum and plastic which enables it to absorb more energy. It means for the same strain value the stresses induced in steel are much higher than aluminum and plastic indicating steel has more energy absorption capacity comparatively.

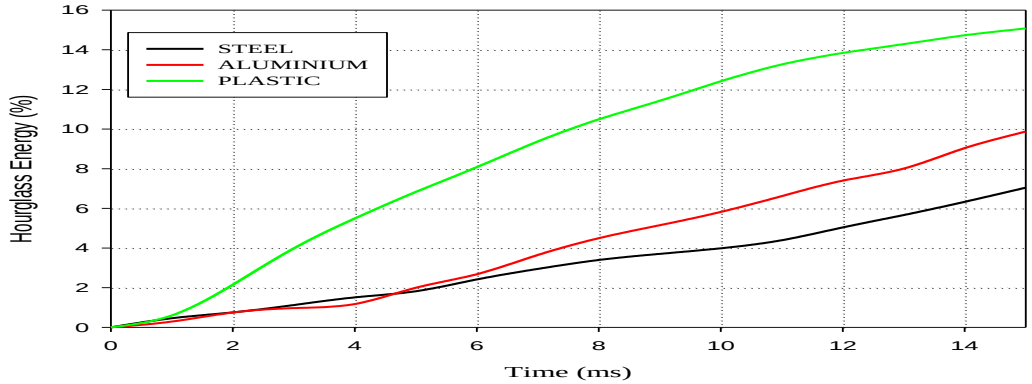
4.4 Variation of sliding energy



Graph 4.6 Sliding Energy with respect to Time

From Graph 4.6, Sliding Energy with respect to Time, it is clear that sliding energy values for steel, aluminum and plastic are 4.55, 7.63 and 12.35% respectively. The higher sliding energy value indicates the instability of material. Higher the sliding energy value, higher the deformation of material and instability [Hallowell (1996)]. So material with good absorption capacity posses less sliding energy value. As steel posses less value, one can say it has the capacity to absorb more energy with sustainable deformation.

4.5 Variation of hourglass energy



Graph 4.7 Hourglass Energy with respect to Time

From Graph 4.7, Hourglass Energy with respect to Time, it is clear that hourglass energy values for steel, aluminum and plastic is 7.05, 9.88 and 15.08% respectively. Hourglass energy is also called Zero Energy modes which is nothing but deformation of elements without prior energy absorption. Considering the above values, plastic has higher value than steel and aluminum. It means the elements in plastic deform more without absorbing energy.

CONCLUSION

Amount of energy absorbed	*Steel*	*Aluminum*	*Plastic*
Internal Energy	*88.25%*	*82.24%*	*72.73%*
Sliding energy	*4.55%*	*7.63%*	*12.35%*
Hourglass energy	*7.05%*	*9.88%*	*15.07%*

Steel absorbs more internal energy and less sliding energy & hourglass energy Compare steel with aluminum and plastic, ***Steel is the best material***

SCOPE OF FUTURE WORK

To integrate the car frontal fascia with Energy absorbers, Bumper beam and Crash box and simulate to calculate the energy absorption of the bumper system and energy transferred to the occupant's space. Modify the design of the bumper system to sustain the impact during crash. Use the composite materials to increase the energy absorption capacity of bumper system.

REFERENCES

1. *Hosseinzadeh RM, Shokrieh M, and Lessard LB, "Parametric study of automotive composite bumper beams subjectedto low-velocity impacts", Composite Structures 68 (2005) 419–427, Elsevier Ltd.*
2. *Evans D and Morgan T, "Engineering Thermoplastic Energy Absorbers for Bumpers", SAE Paper, 1999.*
3. *Sayyad F.B, "Numerical Simulation of Bumper Impact Analysis and to Improve Design for Crash Worthiness", The International Journal of Engineering and Science (IJES)2015.*
4. *BautistaGarret, HardinChad, "Best-In-Class Global Bumper Reinforcement Beam'', AISI 123-129.California Polytechnic State University San Luis Obispo2009.*
5. *Marzbanrad J M, Alijanpour M,"design and analysis of an automotive front bumper beam in low speed frontal crashes'',Thin-Walled Structures 47(2009) 902-911, Elsevier Ltd.*
6. *Andersson R, Schedin E, Magnusson C, Ocklund J, "The Applicability of Stainless Steel for Crash Absorbing Components", SAE Technical Paper, 2002*
7. *Duncan C, Corben B, "Vehicle and Safety Barrier Crash-Testing'', performing organization is Accident Research centre, Monash University, Victoria 3800.*