Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 8 Issue 07, July-2019

Simulation of Car Frontal Fascia During Crash using LS-DYNA

Md Atiqur Rahman[1]
[1.]Department of Mechanical Engineering,
Research Scholar, Birla Institute of Technology,
Mesra,

D. Praveen Babu [2]
[2.] Department of Mechanical Engineering,
SVPCET, Puttur

Abstract – **Automotive bumper beam assembly plays very important role in absorbing impact energy and protects passengers from front and rear collisions. 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. The simulation of vehicle crashes by using computer software's has become an indispensible tool for shortening automobile development time and lowering costs. This paper on the simulated crash test of car frontal fascia. The model used here was that of a Toyota Camry 2012 passenger car. The car fascia is designed in AUTOCAD version 14.0 with thickness of 2.15 mm. The designed car fascia was meshed in HYPERMESH-12 with mixed elements of size 4 mm for getting better accuracy and simulated in LS-DYNA.**

The results are interpreted by using LS-PREPOST to analyze the energy absorption characteristics during crash for different materials at a velocity of 30mm/ms which is approximately 108 km/hr for the duration of 15 ms. The project is carried out for three cases and they are different material models, constant velocity for particular selected material model and using same thickness for particular material model. With the help of LS-DYNA codes nonlinear dynamic contact analysis by using different materials can be done effectively and accurately. The results are found that steel material absorbs maximum internal energy of 88.25% followed by aluminium and plastic materials with 82.28 and 72.23% respectively. This is because steel has high Young's Modulus when compared to aluminum and plastic material and also impact force distribution is uniform in steel material.

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 [Hosseinzadeh et al. (2005)].

These numbers call for the necessity to improve the safety of automobiles during accidents. A car bumper is a front part of the car that covers the car's chassis. 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. They protect the hood, trunk, grill, fuel, exhaust and cooling system as well as safety related equipment such as parking lights, headlamps and taillights etc [Suddin et al. (2007)].

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 behavior 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, aluminums, 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. Automobile bumpers are not typically designed to be structural components that would significantly contribute to vehicle crashworthiness or occupant protection during front or rear collisions [Nitesh Joshi et al. (2016)]. It is not a safety feature intended to prevent or mitigate injury severity to occupants in the passenger cars.

Automotive bumper system plays an important role not only in absorbing impact energy but also in a styling stand point. A great deal of attention with in the automotive industry has been focused upon light weight and sufficient safety in recent years. Therefore, the bumper system equipped with thermoplastic and energy absorbing element is a new world trend in the market. The major point for the design of bumper system is summarized as a degree of absorption of impact energy in a limited clearance between back face of bumper and body parts of the vehicle. While experimental test is rather costly and time consuming, finite element analysis helps engineers to study design concept at an early design stage when prototypes are not available.

I SIMULATION STEPS

1.1 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 Figure 1.14 and 1.15.

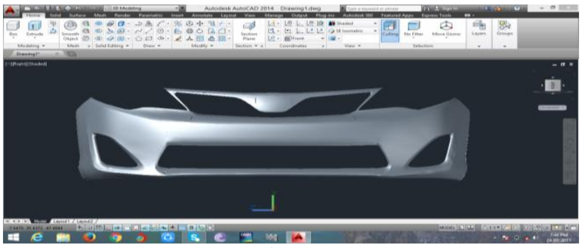


Figure 1.14 Front View of Fascia Designed in AutoCAD

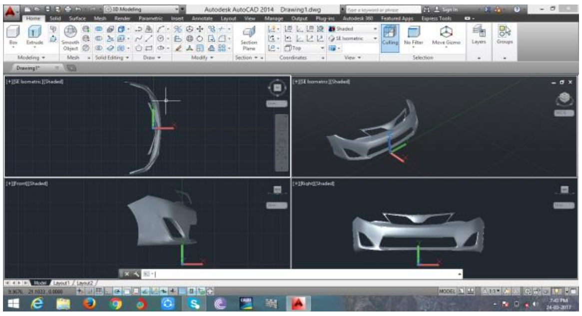


Figure 3.15 Views of Fascia Designed in AutoCAD

3.2 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 Figure 1.16 and 1.17

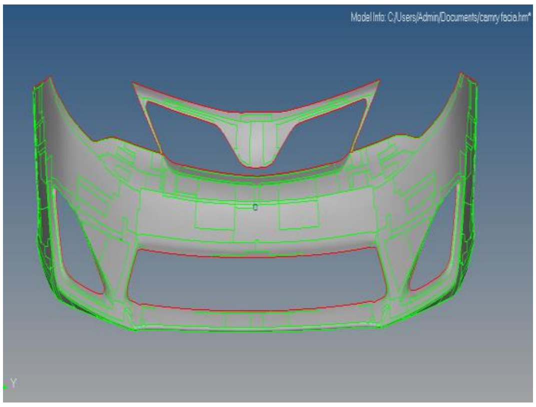


Figure 1.16 Designed Model of Toyota Camry Fascia

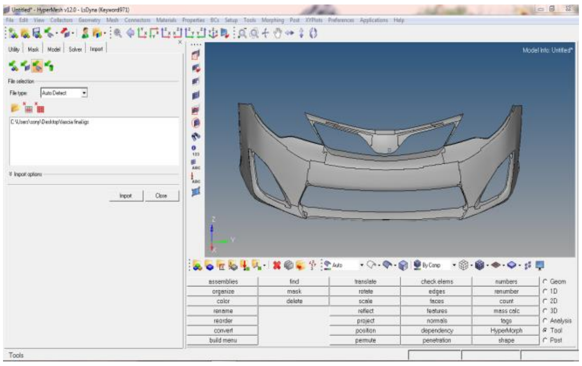


Figure 1.17 Toyota Camry Fascia Imported in Hypermesh

Basic theme of FEA is to make calculations at only limited number of points and then interpolate the results for entire domain(Surface or volume). Any continuous object has infinite degrees of freedom and its just not possible to solve the problem in this format. Finite Element Method reduces degrees of freedom from infinite to finite with the help of discretization i.e. meshing (nodes and elements)


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 as shown in the Figure 1.18.

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.

Different quality parameters like skew, aspect ratio, included angles, Jacobian, stretch etc are the measures of how far a given element deviates from ideal shape. Square means all angles 90^0 and equal sides, while equilateral triangle is all angles 60^0 and equal sides. Some of the quality checks are based on angles (like skew, included angles) while others on side ratios and area (like aspect, stretch)

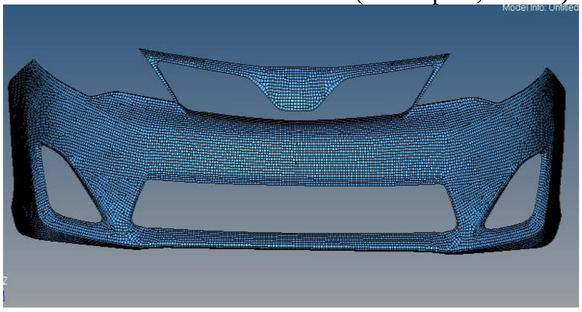
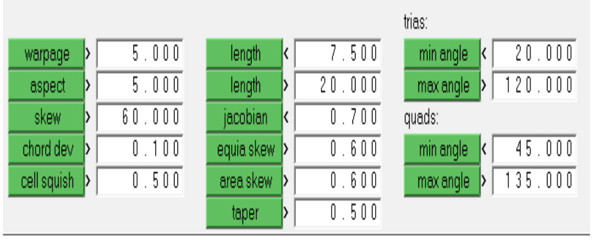


Figure 1.18 Toyota Camry Fascia Meshed in Hypermesh

Table 1.5 Mesh Quality Parameters



						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			

1.3 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. 1.19.

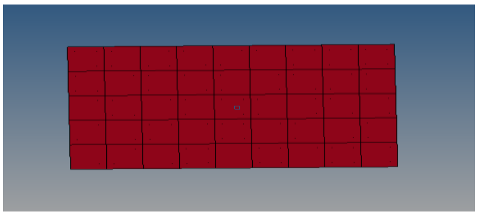


Figure 1.19 Rigid Wall

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 Aluminium material= 6.354 kg
Mass of fascia with plastic material = 4.912 kg
Total car with Steel material fascia = 1623 + 19.136
= 1642.136 kg
Total car with Aluminium material fascia= 1623 + 6.354

= 1629.354 kg
Total car with Plastic material fascia = 1623 + 4.912
= 1627.912 kg

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 in fig: The rigid connectors and mass point are used instead of entire car mode as shown in the Figure 1.20.

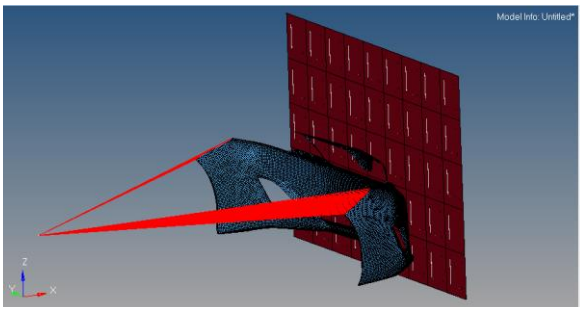


Figure 1.20 Components Created in Hypermesh

1.4 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.

Car fascia is tested with three different materials namely Steel, Aliminium and Plastic. All the properties of the materials are given under data card *MAT_PIECEWISE_LINEAR_PLASTICITY.The properties of the different materials of fascia and rigid wall are shown in the Table 1.6. The behavior of fascia during impact depends on material properties like Young's Modulus, Poisson ratio, Density and-Yield strength. Generally in Ls-Dyna, metals and plastics are given under *MAT_PIECEWISE_LINEAR_PLASTICITY with their specific properties

Table 1.6. Material Properties of Master andSlave

MATERIALS	**PROPERTIES**				
	Young's Modulus	**Density (kg/mm^3)**	**Poisson**	**Yield Stress**	**Ultimate**
Plastic	1	2.01e-06	0.45	26	40
Aluminium	80	2.60e-06	0.33	35	90
Steel	210	7.83e-06	0.30	250	420

The constraints are generally applied to the master component. During simulation, the master should be rigid without any deformation. Moreover, the master components is constrained in all the six degrees of freedom. It is not intended to translate and rotate in the prescribed Cartesian co-ordinate system. Generally in giving the constraints, there are two parameters i.e. primary and secondary constraints. primary constraints are given to restrict the translation of master in all the directions and secondary constraints are given to restrict the rotation of master in all the three directions of co-ordinate system as shown in Figure

3.21. The following data is given Hypermesh software to constraint the rigid body

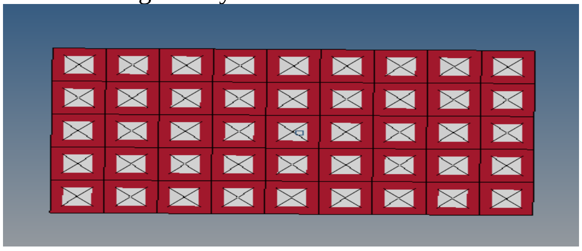


Figure 1.21 Rigid Wall with Constraints

Center of mass constraint option, CMO	= 1
First constraint parameter, CON1	= 7
Second constraint parameter, CON2	= 7

CMO= 1.0, indicates constraints applied in global directions.
CON1=7 indicates,
EQ.0: no constraints,
EQ.1: constrained x displacement,
EQ.2: constrained y displacement,
EQ.3: constrained z displacement,
EQ.4: constrained x and y displacements,
EQ.5: constrained y and z displacements,
EQ.6: constrained z and x displacements,
EQ.7: constrained x, y, and z displacements.
CON2=7 indicates
EQ.0: no constraints,
EQ.1: constrained x rotation,
EQ.2: constrained y rotation,
EQ.3: constrained z rotation,
EQ.4: constrained x and y rotations,
EQ.5: constrained y and z rotations,
EQ.6: constrained z and x rotations,
EQ.7: constrained x, y, and z rotations

1.5 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. By default, the software doesn't have any logic to detect the contact unless the user indicates. 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 component which is deformable and have less stiffness and young's modulus value compared to master. Hence, fascia is considered as slave. The colour of fascia and rigid wall changes to red which indicates that the contact is applied as shown in Fig. 1.22 and 1.23.

1.6 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. The selected nodes on the fascia are shown in Fig. 1.24.

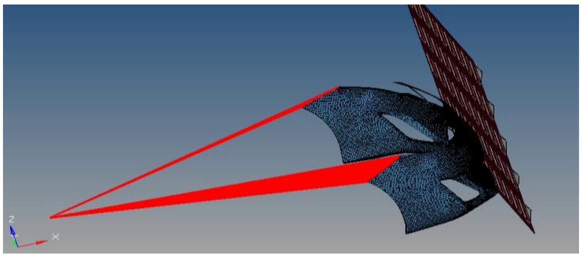


Figure 1.22 Before Contact is Applied between Fascia and Rigid Wall

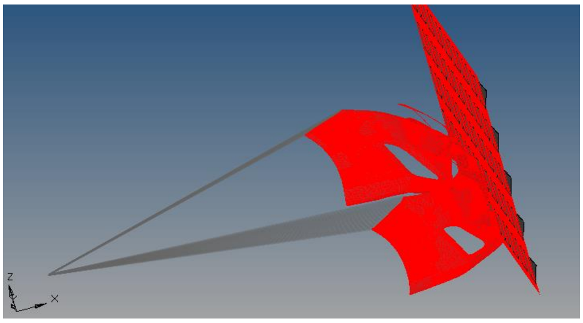


Figure 1.23 After Contact is Applied between Fascia and Rigid Wall

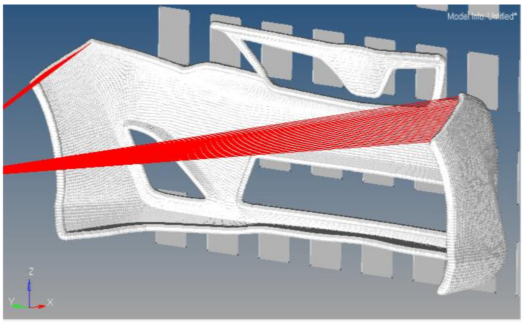


Figure 1.24 Node Set for Assigning Velocity

1.7 INITIAL-In this section, the velocity is assigned to fascia. The velocity is assigned for all the nodes that are created as a set. So that the assigned velocity is distributed for the surface nodes equally. Velocity of 30 mm/ms i.e. 108 km/hr in +X-direction is given to fascia as the vehicle in reality moves in X-direction.

1.8 CONTROL CARDS-Control cards are used to control the time step and run time of the file. These cards are necessary to control the duration of simulation time.

1. DT min
2. DT

The run time of file is 15ms. The results are plotted for the energies absorbed by the fascia in 15 ms. DT min is the interval in which the codes are generated in processing the file in ls-dyna. DT is the time step value. It is the minimum time taken by the sound wave for travelling from one end to other of the shortest element. It is also used in plotting the graph i.e. for every 0.1 interval a point on the graph is plotted. Decreasing the value below 0.1 increasing the smoothness of the curve but at the same time increases time.

1.9 DATABASE CARDS-Database cards are given to obtain the output i.e. to generate results and to interpret plots. Database cards are used to analyze the energy absorption characteristics that are discussed in next chapter namely result and discussions. Some of the database plots used are :

1. D3PLOT (database for complete output states)
2. D3DUMP (complete database for restart)

II. RESULTS AND DISCUSSION

The Toyota Camry model 2012 is considered for analyzing the energy absorption characteristics. Three different materials such as Steel, Aluminium 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.

2.1 ENERGY BALANCE CURVES-According to Law of Conservation of energy, Energy can neither be created nor be destroyed, but it can be converted from one form of energy to other form. Applying the same principle to crash analysis, the amount of kinetic energy lost during impact must be converted to other forms of energy such as internal energy, sliding energy and hour glass energy. It is also noted that, there may be negligible errors in calculating energy ratio because all the processes in this universe are irreversible and some losses are always included which deviates energy ratio slightly from one.
From the Fig. 2.1 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 ms, the percentage of internal energy. It show that the kinetic energy lost during impact is appeared in the form of IE, SE and HE.
From the Fig. 2.2 it is clear that the absorption of Internal Energy (IE), Sliding Energy (SE) and Hour glass Energy (HE) for aluminium 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 show that the kinetic energy lost during impact is appeared in the form of IE, SE and HE.

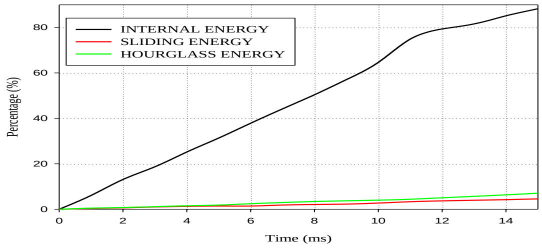


Figure 2.1 Energy Balance Curve for Steel Material

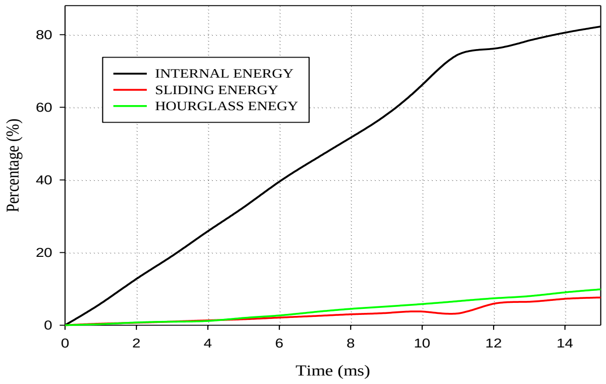


Figure 2.2 Energy Balance Curve for Aluminium Material

From the Fig. 2.3 it is clear that the absorption of Internal Energy (IE), Sliding Energy (SE) and Hour glass Energy (HE) for plastic is 72.73, 12.35, 15.08% respectively. From the graph, it is observed clearly that up to 8 ms the rise of sliding energy is higher than Hour glass energy but after 15 ms sliding energy is less than Hour glass Energy [Srikanth (2012)]. It means, initially the plastic deforms more without absorbing the energy which makes it unstable.

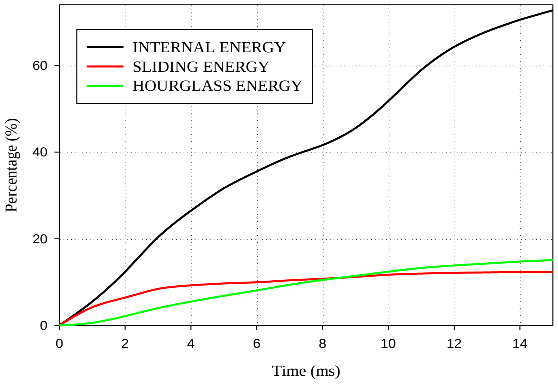


Figure 2.3 Energy Balance Curve for Plastic Material

2.2 VARIATION OF KINETIC ENERGY-From Fig. 2.4, Kinetic Energy respect to Time, it is clear that kinetic energy lost during impact by steel, aluminium and plastic is 22430, 7920 and 2240 J respectively. In terms of percentages, the kinetic energy lost from the total energy for steel, aluminium 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 because the impact force acting on the fascia made of steel is uniformly distributed over the surface such that it absorbs large amount of energy [Pecht (2005)].

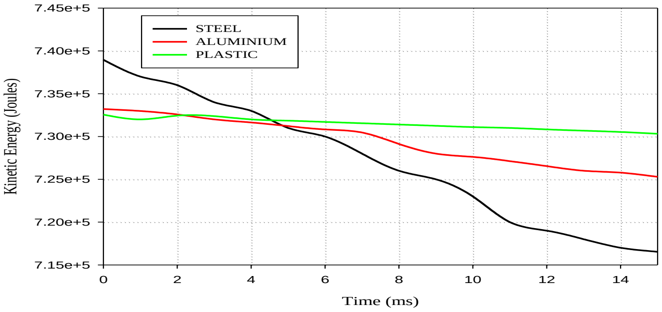


Figure 2.4 Kinetic Energy with respect to Time

2.3 VARIATION OF INTERNAL ENERGY-From Fig. 2.5, 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 [David (2013)]. 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.

2.4 VARIATION OF SLIDING ENERGY-From Fig. 2.6, Sliding Energy with respect to Time, it is clear that sliding energy values for steel, aluminium 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, a 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.

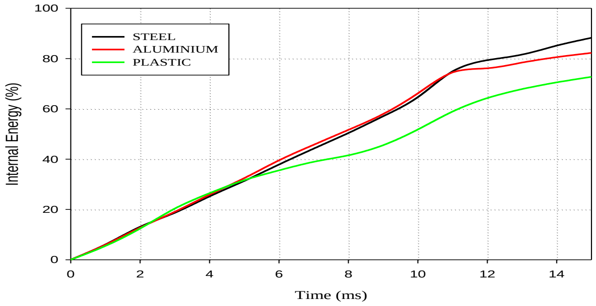


Figure 2.5 Internal Energy with respect to Time

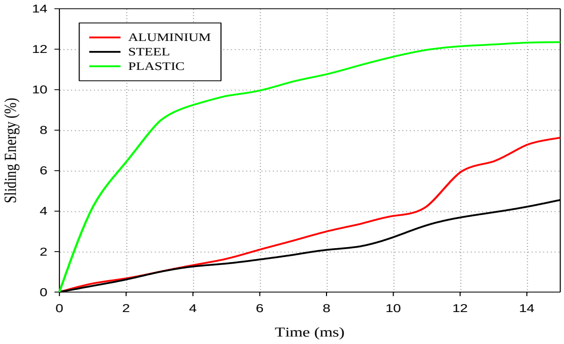


Figure 2.6 Sliding Energy with respect to Time

2.5 VARIATION OF HOURGLASS ENERGY

From Fig. 2.7, Hourglass Energy with respect to Time, it is clear that hourglass energy values for steel, aluminium and plastic is 7.05, 9.88 and 15.08% respectively. Hourglass energy is also called Zero Energy Mode which is nothing but deformation of elements without prior energy absorption. Considering the above values, plastic has higher value than steel and aluminium [Chandan (2013)]. It means the elements in plastic deforms more without absorbing energy. That's why, the plastic deforms more and absorbs less energy when compared to steel and aluminium.

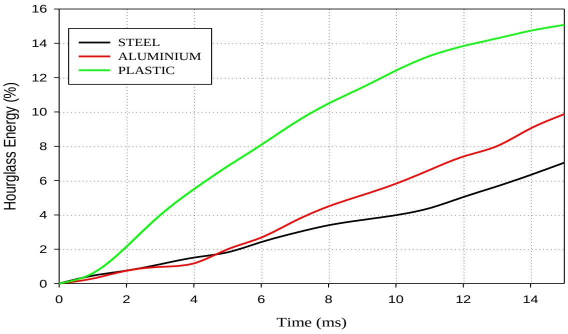


Figure 2.7 Hourglass Energy with respect to Time

III CONCLUSION

Car fascia is the first and foremost part that gets damaged and absorbs energy during a crash. If fascia is made of suitable materials, then it absorbs substantial part of energy during crash and protects the occupants. In our work, Steel, Aluminium and Plastic used as materials for fascia and crash simulation is carried out at 30 mm/ms i.e. 108 km/hr for 15 ms. From the results, the following conclusions can be drawn.

1. Among the three materials, steel material absorbs maximum internal energy of 88.25% followed by aluminium and plastic materials with 82.28 and 72.73% respectively. This is because, steel material has high Young's Modulus when compared to aluminum and plastic material and also impact force distribution is uniform in steel material.
2. The conversion of sliding energy from the available kinetic energy is more in plastic material followed by aluminium and steel materials. It shows the instability of plastic because of lowest young's modulus.
3. The conversion of hourglass energy from the available kinetic energy is more in plastic material followed by aluminium and steel materials. This is because elements in plastic material deforms more without absorbing energy due to less element stiffness when compared to aluminium and steel materials.
4. The decrease in kinetic energy during impact is more in steel material with 3% followed by aluminium and plastic materials with 1 and 0.3% respectively

IV REFERENCES

[1] Alok Singh, Savitha Singh “Study of Mechanical Properties and Absorption Behaviour of Coconut shell Power-Epoxy Composite” PSIT College of Engineering,Kanpur,2013.
[2] Naheed Saba, Paridah Md Tahir “A Review on Potentiality of Nano Filler/Natural Fiber Filled Polymer Hybrid Composite’’Universiti Putra Malaysia, Malasia,2014.

[3] R.Udhayasankar And B.Karthikeyan"A Review on Coconut Shell Reinforced Composite"Annamalai University, Tamilnadu,2015.
[4] Rajendra Kumar And Tejeet Singh" Natural Fibers Polymeric Composite With Particulate Fillers-A Review Report"Guru Nanak Dev Engineering College, Ludhiana Punjab, 2015.
[5] D.Ravindran, T.Sornakumar" Development of Hybrid Coconut Shell Powder-Wood Dust Polyester Resin Based Composite"2015.
[6] B.Sudharsan,M.Lava Kumar"The mechanical Behaviour of Eggshell And Coconut Coir Reinforced Composite" Siddarth Institute of Engineering And Technology,Puttur,2014.
[7] K.Vignesh, G.Ramasivam" Mechanical Properties of Bone Powder And Coir Fiber Reinforced Polyester Composite"Aksheyaa College of Eng, Tamilnadu,2016.
[8] Rakshit Agarwal, M.Ramachandran" Tensile properties of Reinforced Plastic Material Composite with Natural Fiber and Filler" MPSTME, Maharashtra, 2015.