International Journal of Scientific & Engineering Research, Volume 4, Issue 8, August 2013
ISSN 2229-5518

CRASH SIMULATION IN ANSYS LS-DYNA TO EXPLORE THE CRASH PERFORMANCE OF COMPOSITE AND METALLIC MATERIALS

Tony Punnoose Valayil, Dr. Jason Cherian Issac

Abstract— During an automobile crash, some parts in the front of an automobile body will have plastic deformation and absorb a lot of energy. Hence it becomes necessary to check the car structure for its crash ability so that safety is achieved together with fuel economy. A simple finite element (FE) model of a car is developed in ANSYS and it is solved for full frontal impact in ANSYS LS-DYNA explicit code. Computational simulations and various results are plotted and analyzed. There are various test configurations. We have limited our analysis to frontal impact with a rigid wall at a speed of 35 mph, corresponding to a NHTSA (National Highway Traffic Safety Administration) full frontal impact. It was noted that composite materials could be used more effectively for light-weightiness other than metallic materials without affecting the necessary impact energy absorbing capacity of the car body. Since composite materials and metallic materials absorb approximately the same energy during a car crash we conclude that composite materials can be used for light-weightiness in automobiles.

Index Terms— Plastic deformation, crash ability, computational simulation. Light-weightiness, finite element analysis, specific energy absorption.

◆

1 INTRODUCTION

CAR body light weighting and crashworthiness are two important aspects of auto design. During an automobile crash, some parts in the front of an automobile body will have plastic deformation and absorb a lot of energy. Structural members of the vehicle are designed to increase this energy absorption capacity and thus to enhance the safety and reliability of the vehicle. As the dynamic behavior of the structural member is different from static one, the crashworthiness has to be assessed by impact analysis. Hence it becomes necessary to check the car structure for its crash ability so safety is achieved together with fuel economy. There are two ways by which the safety features can be achieved:

1. Performing an actual crash test.
2. Simulating the crash in some FE code like ANSYS LS-DYNA

Though the first option is more accurate and reliable, it demands time and high cost. A more practical solution which results in a compromise between the factors of accuracy, cost and time is simulation. With appropriate initial conditions, loads and element formulations, engineers can develop a precise enough FE model to judge the crash response in an actual accident. This technique has superseded the testing using an actual model.Thus computer simulations are used to find the automobile model's crash ability. There are various test configurations. We have limited our analysis to frontal impact with a rigid wall at a speed of 35 mph, corresponding to a NHTSA (National Highway Traffic Safety Aistration) full frontal impact[1].

In the automotive industry weight reduction is important because fuel consumption is directly related to vehicular weight. Composite materials are often used to reduce the weight of structures. Composite materials can offer significant safety benefits for vehicles compared with metals in terms of high specific energy absorption (SEA) and strength compared to weight.

Before starting this project, some of the published literatures and previous researches have been reviewed to build up a solid background in the area of car simulation and finite element analysis.

Yucheng liu's [2] paper shows that the experiment results and FEA results matches very well and the validity of the computer model is then verified. Simonetta Boria's [3] study is the design of a crash-box for a Formula SAE car and the investigation, through a numerical approach, of its dynamic behavior in frontal impact conditions. The obtained results show that the impact attenuator by itself is able to absorb the total kinetic energy with dynamic buckling and plastic deformation of its structure with an average deceleration limited under a 20g value.

Bižalana [4] et al., studied the numerical simulation of crash test using Student Roadster (SR). The results illustrate the importance of indentations in energy absorbent beams for plastic deformations formed in those areas.

Un-koo Lee [5] et al., illustrated a sub frame type fuel tank and evaluated to increase the vehicle safety and the design flexibility against rear crash of automotive vehicles. The proposed design of fuel tank not only provides the design flexibility but fulfils the requirements of the revised Federal Motor Vehicle Safety Standards.

- *Tony punnoose valayil is currently working as assistant professor in Coimbatore institute of technology,coimbatore, PH-09489911023. E-mail: tonyisdesigning@mail.com*
- *Dr.Jason cherian issac is currently working as Professor in SAINTGITS college of engineering kottayam,kerala,India, PH-.09447733536 E-mail: jason.cherian@saintgits.org*

1.1 Crashworthiness

Crashworthiness is the ability of a structure to protect its occupants during an impact. This is commonly tested when investigating the safety of aircraft and vehicles. Depending on the nature of the impact and the vehicle involved, different criteria are used to determine the crashworthiness of the structure. Crashworthiness may be assessed either prospectively, using computer models (e.g., LS-DYNA, MSC-Dytran, MADYMO) or experiments, or retrospectively by analyzing crash outcomes.

2 FINITE ELEMENT ANALYSIS

Finite Element Analysis. is a robust and mature technique for approximating the physical behavior of a complex system by representing the system as a large number of simple interrelated building blocks called elements.

2.1ANSYS LS-DYNA

ANSYS LS-DYNA is an explicit dynamic program intended to solve short duration dynamic problems. ANSYS LS-DYNA technology is the result of a collaborative effort between ANSYS, Inc. and Livermore Software Technology Corp. (LSTC). Introduced in 1996, the capabilities and robustness of ANSYS LS-DYNA software have helped thousands of customers in numerous industries resolve highly intricate design issues.

2.2 Explicit dynamic analysis

ANSYS explicit dynamics engineering simulation solutions are ideal for simulating physical events that occur in a short period of time and may result in material damage or failure. These types of events are often difficult or expensive to study experimentally. Simulation provides insight and a detailed understanding of the fundamental physics taking place and gives engineers a chance to make necessary changes before their products are put into service, when mistakes in design can be costly.

3 CRASH SIMULATION USING ANSYS LS-DYNA

The crash simulation is done by performing a crash analysis using appropriate FE software. Here an FE model is created using ANSYS LS-DYNA with appropriate initial condition, loads and element formulation to judge the crash response as in an actual accident. During the course of this project different journals were studied and FE model from reference [1] was chosen to validate the crash analysis procedure.

The first step in doing this project was to create a car model. Using ANSYS LS-DYNA a car model was made as shown in figure 1

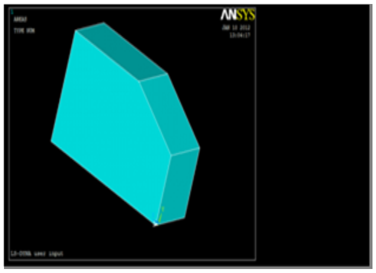

Fig 1 ANSYS LS-DYNA car model

3.1 Material properties

The crash simulation was first performed with typical structural steel properties. Second an aluminum alloy was selected to replace the steel structure. Next, E Glass Fabric composite properties were chosen to replace the aluminum structure. Finally, Carbon Fiber Fabric composite material was selected to replace E Glass Fabric composite.

3.2 Load curve

The relationship between the stress and strain that a particular material displays is known as that material's Stress-Strain curve. It is unique for each material and is found by recording the amount of deformation (strain) at distinct intervals of tensile or compressive loading (stress). These curves reveal many of the properties of a material (including data to establish the Modulus of Elasticity, E). Stress-strain curves of various materials vary widely, and different tensile tests conducted on the same material yield different results, depending upon the temperature of the specimen and the speed of the loading. It is possible, however, to distinguish some common characteristics among the stress-strain curves of various groups of materials and, on this basis, to divide materials into two broad categories; namely, the ductile materials and the brittle materials. The load curve that relates the stress with the strain is drawn using following data points [1].

The load curve that relates the stress with the strain is drawn using following data points:

Strain rate	Yield stress(Newton/square metre)
0	207e6
0.08	250e6
0.16	275e6
0.4	290e6
1.0	300e6

3.3 Finite Element Model

Finite element model of a car with bumper system for performing a crash test is simulated on the computer using ANSYS LS-DYNA. In creating the finite element model the shell element, SHELL 163, is used to mesh the bumper surface.

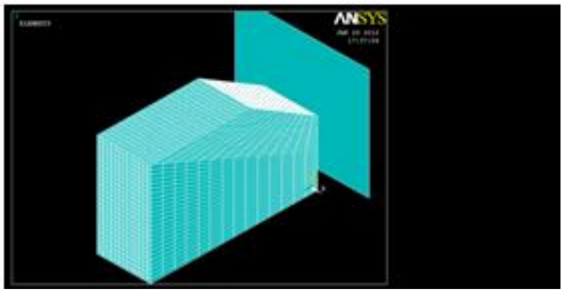

Fig 2 Finite Element Model

3.4 Element type

Elements are selected on the basis of their DOF and different options applicable to them. Since in a rigid body, all nodes have same DOFs, the rigid wall need not to be defined an element. The rigid wall is also not meshed. For the rest of the model, shell 163 was selected.SHELL163 is a 4-node element. The element has 12 degrees of freedom at each node: translations, accelerations, and velocities in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes. This element is used in explicit dynamic analyses only. The three material models chosen are compatible with it.

4. COMPOSITE AND METALLIC CRASH PERFORMANCE

Composite materials are increasingly being utilized in automotive parts. Because of their high strength and stiffness to weight ratio, typical composite parts are about 30 to 40 percent lighter than steel components. Composite and thermoplastics are being utilized as materials for body panels, hoods and bumpers of automobiles. High performance parts, such as drive shafts, transmission parts, flywheels, leaf springs and brake drums, are areas in which CFRP (Composite Fiber-Reinforced Polymer) have applications.. Utilizing ANSYS LS-DYNA, the crash performance of two composite structures was analyzed and compared to those of an aluminum alloy and steel structure.

4.1 Internal energy

The law of conservation of energy explains that energy inside a system cannot be created or destroyed, and it can be transferred from one form into another without changing the Total amount of energy. Considering mechanical systems, such as the vehicle systems, the absorbed work or internal energy of a system cannot exceed the work input. In theory, internal energy is equal to the work (E) done by external Forces on the system, which is equal to the product of the exerted force (F) and the distance (d) through which the force moves: During the impact of a vehicle, its kinetic energy is predominantly transformed into plastic deformation of the respective structures, for which the internal energy can be calculated. To simplify the concept, the energy of a head-on vehicle impact against a rigid barrier is provided. The rigid barrier is fixed during the impact so its kinetic energy is constantly zero. Due to the rigid body definition, it does not deform and absorb energy. Therefore, the energy of the vehicle is equal to the energy of whole system. This means that the kinetic energy of vehicle is absorbed by itself during impact time. The total amount of internal energy to be absorbed by the chosen vehicle models is calculated using ANSYS LS-DYNA.

4.2 Crash simulation

The crash simulation of a full frontal impact of the model at a velocity of 35 mph with a rigid wall is carried out and analyzed in detail as in [1]. The initial velocity was applied to the car which served as the impacting object

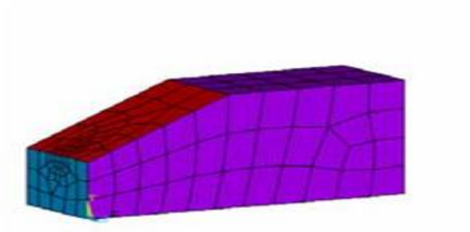

Fig 3 FE Model showing the material models Violet=HS Steel, Red=Glass, Green=bumper

The green colour portion represents the front portion of the car. The bumper of the car was first made of metallic material properties, that is steel and aluminum. Then it was replaced by composite material properties like E Glass Fabric and CF Fabric. The image captured during the crash simulation using ANSYS LS-DYNA is shown below.

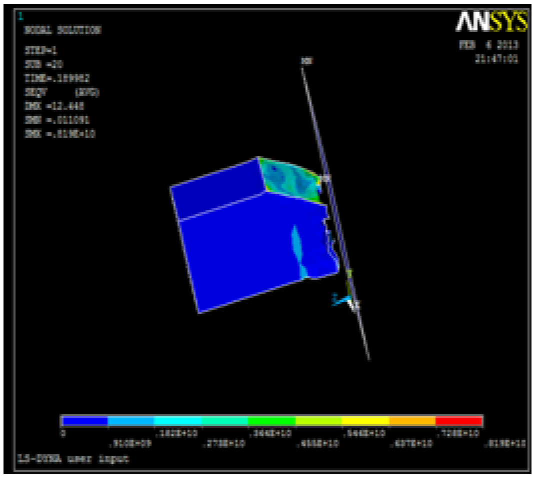

Fig 4 Image captured during crash simulation

5. RESULTS

5.1 Mass of materials

The annual world oil production, which peaked in 2000-2010, is expected to drop over the next 50 years due to diminishing oil reserves. Global growth in transportation is accelerating the demand for oil. In the automotive industry weight reduction is important because fuel consumption is directly related to vehicular weight, so if the mass of the vehicles decreases the fuel consumption also decreases. For achieving this vehicles made of light weight materials can be used.

The mass of steel, aluminum, E glass fabric, and standard CF fabric in the respective car models used in the study is given below in figure 5.

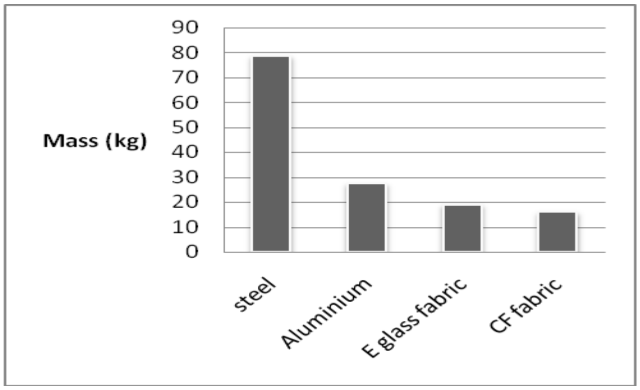

Fig 5. Chart showing mass of materials

5.2 Energy absorbed by materials

The energy absorbed by the different materials are analyzed by a graph showing internal energy along y-axis and time along x-axis

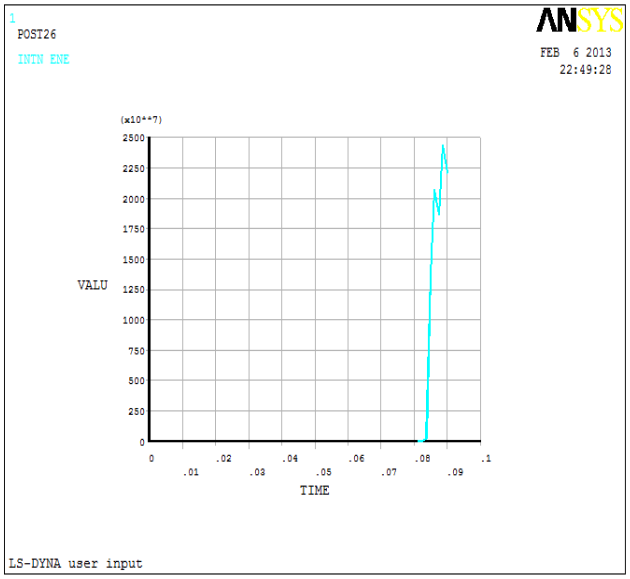

Fig 6 Energy absorbed by steel

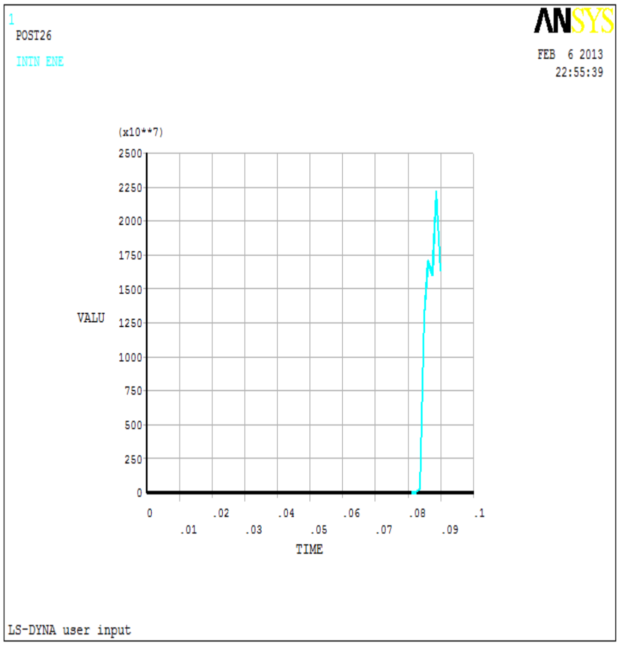

Fig 7 Energy absorbed by aluminium

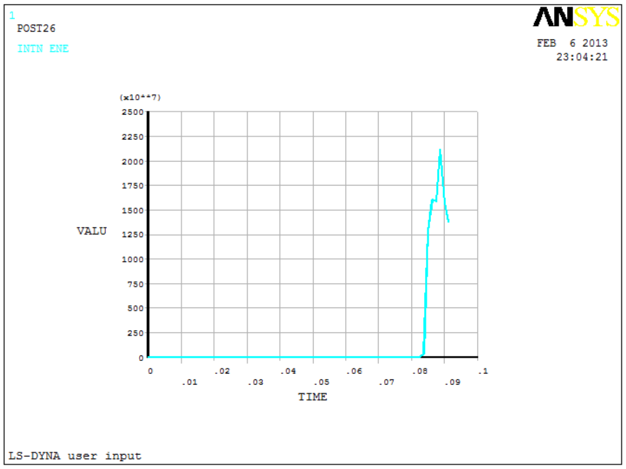

Fig 8 Energy absorbed by CF fabric

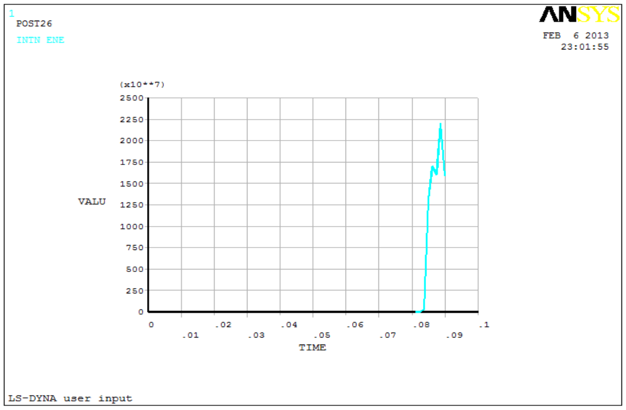

Fig 9 Energy absorbed by E glass

5.3 Total energy

The total absorbed energy is the amount of energy that the specimen absorbed during the entire impact test – from start to end. This value may be the same as energy to maximum load when the specimen abruptly fails at the maximum load point. The value is calculated from the time the load begins to rise until the first occurrence of zero loads after the maximum point. This value can be used as an indicator to a materials ductility or toughness – the higher the value the stronger the material. However, care needs to be taken to ensure that when the data collection ends, the load has fallen below the zero thresholds. If it hasn't, this value is no longer a valid number to be used. The total energy is calculated by the summation of energy before crash and after crash of the different materials. The chart showing the comparison of the total energy of the different materials is shown in figure 10.

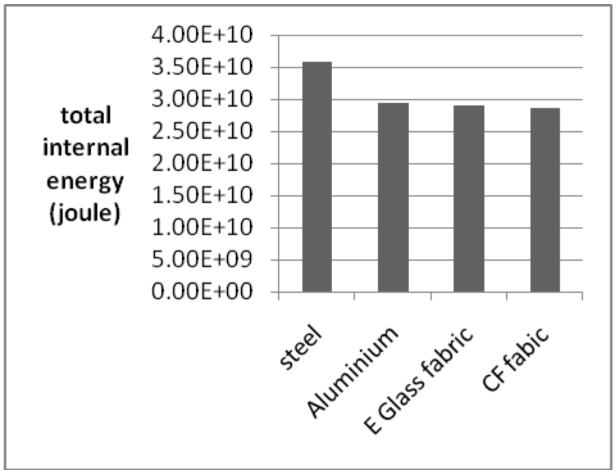

Fig 10 Total energy absorption of materials

5.4 Specific energy

Fundamentally, the key requirement of crash protection is to absorb and dissipate energy. In an impact, the crash structure must dissipate the energy of the impact whilst ensuring that the occupants of the vehicle are not subjected to excessive accelerations/forces, and that the "survivable" zone within the car remains intact (i.e. the crash structure does not ingress too far into the vehicle). This energy dissipation is achieved through plastic work done in deforming the material in the crash structure. Therefore, by comparing the capacity for energy dissipation of different materials, and considering their density, it is possible to assess which materials provide optimal energy dissipation per unit mass – "specific energy absorption". Composites have dramatically higher specific energy absorption than steel or aluminum.. The chart showing the comparison of the specific energy of the different materials is shown in figure 11.

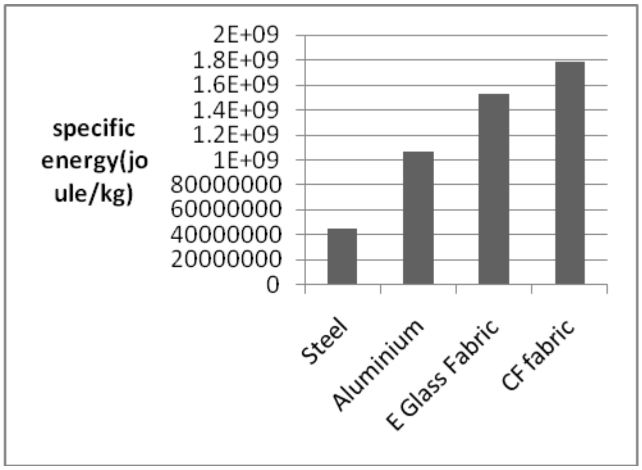

Fig 11 Specific energy absorption of materials

6. CONCLUDING REMARKS

Although the structure modeled was a fairly simple one, the ANSYS LS-DYNA crash simulations show that E Glass Fabric and CF Fabric composites for bumper of the car can absorb the same amount of energy as the steel or aluminum .A good measure of crashworthiness is the specific energy absorbed by the structure. the CF Fabric composite structure has the highest specific energy absorption followed by E Glass Fabric, aluminum and steel. Hence, for this simulation, the crashworthiness of carbon composite and E Glass composite, with a much lower weight, is superior to the other two materials.

7. REFERENCES

[1] Waseem Sarwar and Nasir Hayat, "Crash Simulation and Analysis of a Car Body Using ANSYS LS- DYNA," *Failure of engineering materials and structures,* code 09, 2007, pp. 65-72..

[2] Yucheng Liu, "Impact Experimental Analysis and Computer Simulation," Department of Mechanical Engineering, University of Louisville.

[3] Simonetta Boria, Giuseppe Forasassi, University of Pisa, " Numerical simulation of crash-test for a formula sae car," University of Pisa , Pisa, Paper Number 09-0152, pp.1-8.

[4] Bižal Ana, Klemenc Jernej, Rosa Uroš, Fajdiga Matija, "Numerical simulation of crash test for the vehicle student roadster," University of Ljubljana, Faculty of mechanical engineering, Slovenia, F2008-SC-030.

[5] Un-Koo Lee, Seong Bae and Jang Seong S. Cheon," Development of a Subframe Type Fuel Tank for Passenger Cars," *Journal of mechanical design,* Vol. 132/044501, April 2010, pp.1-7.