Car Crash Analysis Using Explicit Dynamics

Sagar Bhamre
Dept of Automobile
ITM Universe, Vadodara

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 CATIA and it is solved for full frontal impact in ANSYS LS-DYNA explicit code. Computational simulations and various results are plotted and analysed. There are various test configurations. We have limited our analysis to frontal impact with a rigid wall, pole, pyramid at a speed of 27.7 m/s, corresponding to a NHTSA (National Highway Traffic Safety Administration) full frontal impact. It was noted that as was observed, the bumper, engine and the rails absorb most of the energy before the wheel impacts the wall. Almost half of the energy of the crash is absorbed by these components after about 0.04 sec of the crash initiation. It has been observed that there is minimum deformation of the cabin and also there was minimum intrusion of the components into the cabin. Therefore, it can be assumed that the occupants in the cabin would not be caused any injury by a component intruding into the cabin in the event of the crash.*

Keywords*-* Ansys LS-DYNA, CATIA V5 R19, Crash Test, Automobile Safety.

I. INTRODUCTION

With the development of society, people have more and more stringent demands for automobile passive safety and fuel economy, which requires the improvement of automobile structure crashworthiness and light weighting degree. Car body light weighting and crashworthiness are two important aspects of auto-design. A major concern of both the industry and government is the development of vehicles that would consume less fossil fuel, thus compromising the safety of occupant resulting from the reduced weight of the automobile.

A crash test is a form of destructive testing usually performed in order to ensure safe design standards in crashworthiness and crash compatibility for various modes of transportation or related systems and components. 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, by analyzing crash outcomes. Several criteria are used to assess crashworthiness prospectively, including the deformation patterns of the vehicle structure, the acceleration experienced by the vehicle during an impact, and the probability of injury predicted by human body models. Injury probability is defined using criteria, which are mechanical parameters (e.g., force, acceleration, or deformation) that correlate with injury risk. A common injury criterion is the Head impact criterion (HIC). Crashworthiness is assessed retrospectively by analyzing injury risk in real-world crashes, often using regression or other statistical techniques to control for the myriad of confounders that are present in crashes.

II. TYPES OF CRASH TEST CONDUCTED

1. **Frontal-impact tests**: which is what most people
Initially think of when asked about a crash test. These are usually impacts upon a solid concrete wall at a specified speed, but can also be vehicle-vehicle tests. SUVs have been singled out in these tests for a while, due to the high ride-height that they often have.

2. **Moderate overlap frontal test-** In the moderate overlap frontal test, a vehicle travels at 40 mph toward a barrier with a deformable face made of aluminum honeycomb. The barrier face is just over 2 feet tall. A Hybrid III dummy representing an average-size man is positioned in the driver seat. Forty percent of the total width of the vehicle strikes the barrier on the driver side. The forces in the test are similar to those that would result from a frontal offset crash between two vehicles of the same weight, each going just under 40 mph.

3. Small overlap frontal test- In the small overlap frontal test, a vehicle travels at 40 mph toward a 5-foot-tall rigid barrier. Twenty-five percent of the total width of the vehicle strikes the barrier on the driver side. Small overlap frontal crashes primarily affect a vehicle's outer edges, which aren't well protected by the crush- zone structures. Crash forces go directly into the front wheel, suspension system and firewall.

4. **Side impact test-** Side collisions are vehicle crashes where the side of one or more vehicles is impacted. These crashes often occur at intersections, in parking lots, when two vehicles

pass on a multi-lane roadway. When a vehicle is hit on the side by another vehicle, the crumple zones of the striking vehicle will absorb some of the kinetic energy of the collision. TheCrumple zones of the struck vehicle may also absorb some of the collision's energy, particularly if the vehicle is not struck on its passenger compartment. Both vehicles are frequently turned from their original directions of travel. If the collision is severe, the struck vehicle may be spun or rolled over, potentially causing it to strike other vehicles, objects, or pedestrians. After the collision, the involved vehicles may be stuck together by the folding of their parts around each other.

III. CRASH ANALYSIS USING CAE TOOLS

CAE tools are very widely used in the automotive industry. In fact, their use has enabled the automakers to reduce product development cost and time while improving the safety, comfort, and durability of the vehicles they produce. The predictive capability of CAE tools has progressed to the point where much of the design verification is now done using computer simulations rather than physical prototype testing. CAE dependability is based upon all proper assumptions as inputs and must identify critical inputs. Even though there have been many advances in CAE, and it is widely used in the engineering field, physical testing is still used as a final confirmation for subsystems due to the fact that CAE cannot predict all variables in complex assemblies.

IV. LS DYNA INTRODUCTION

CAE tools are very widely used in the automotive industry. In fact, their use has enabled the automakers to reduce product development cost and time while improving the safety, comfort, and durability of the vehicles they produce. The predictive capability of CAE tools has progressed to the point where much of the design verification is now done using computer simulations rather than physical prototype testing. CAE dependability is based upon all proper assumptions as inputs and must identify critical inputs. Even though there have been many advances in CAE, and it is widely used in the engineering field, physical testing is still used as a final confirmation for subsystems due to the fact that CAE cannot predict all variables in complex assemblies.

LS-DYNA is a general-purpose finite element program capable of simulating complex real world problems. It is used by the automobile, aerospace, construction, military, manufacturing, and bioengineering industries. LS-DYNA is optimized for shared and distributed memory. Unix, Linux, and Windows based platforms and it is fully QA'd by LSTC. The code's origins lie in highly nonlinear, transient dynamic finite element analysis using explicit time integration.

V. LITERATURE SURVEY

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

Tony Punnoose Vayalil *et al.* illustrates 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. **[1]**

M. Raghupathi *et al.* improved the performance of the seat cross member for the lateral crash through Finite Element Techniques with the help of Beta-CAE ANSA for model build up and ESI PAM-Crash for solving and Beta-CAE Meta Post for extracting the solutions. Having a good compromise between the computational time and the accuracy the element size is chosen in the range of 4mm - 8mm. The model is built with 1.2 million elements approximately for the analysis. **[2]**

Faizan Khatri *et al.* buses form an integral part of public transport vehicle fleet in India. A large number of people use public transport buses for their daily conveyance. Crash tests used for assessing the safety and crash worthiness of automobiles can be used to assess the safety of passenger buses as well. A Finite Element model of a passenger bus was used in a crash simulation to assess the safety and crashworthiness of these buses. Evaluation of vehicle structural durability is one of the key requirements in design and development of today’s automobiles. **[3]**

Mr.C.Sadhasivam *et al.* Vibration and Crash Analysis of Car Body using ansys is carried out including dynamic, static, crash analysis and so on. The main objective of the project was to find fundamental characteristics like frequency, stress and displacement for different material and velocity influence in the car body structure. **[4]**

Tejasagar Ambati *et al.* the simulation of vehicle crashes by using computer software has become an indispensable tool for shortening automobile development time and lowering costs. It also has huge impact on the crashworthiness of an automobile. This work reports on the simulated crash test of an automobile. The objective of this work is to simulate a frontal impact crash of an automobile and validate the results. The aim is also to alter some of the materials of the components with a view to reduce the forces experienced during the crash. **[5]**

Waseem Sarwar ***et al.*** the current paper discusses the development, modification, and analysis of a finite element model of car body using H.S steel. A simple FE model 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 analysed. **[6]**

a) SUMMARY OF LITERATURE SURVEY

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.

The FE model was then used to simulate crash test. The FE software used here to carry out the simulation was LS-DYNA. One of the tests carried out was the Frontal-offset crash at 40 mph. Before the simulation could be carried out, several other pre-processing conditions have to be specified. The test results were verified using results from actual crash-test reports. Present runtimes on high-end workstations for LS-DYNA vehicle models are still measured in days, while multi-body run- times are typically less than 1h, even for the most complex models.

VI. METHODOLGIES

Flowchart shows the steps to solve via cad software.

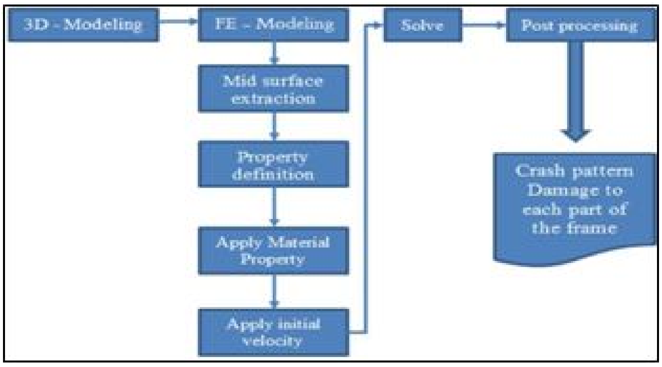


Fig 1- step by step flow chart for analysis

a) BLUEPRINT

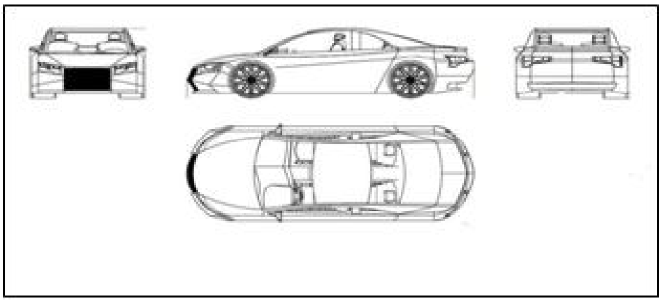


Fig 2- Blueprint of a car

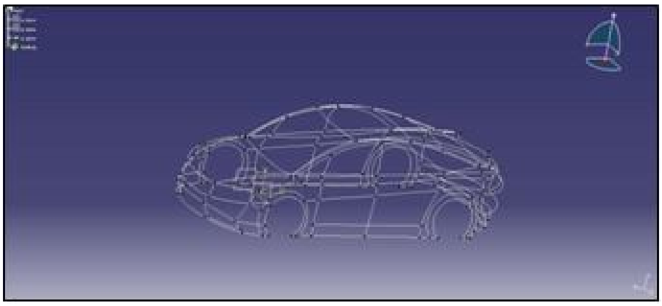


Fig 3- Wireframe model of car

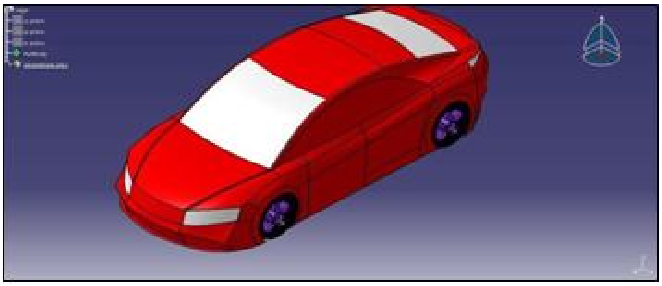


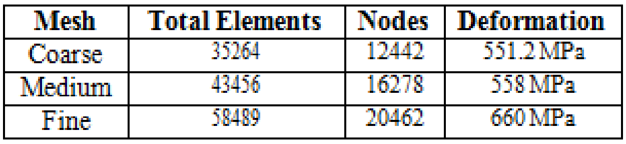
Fig 4- surface model of car

VII. DISCRETIZATION

Domain is discretized into a finite set of control volumes or cells. The discretized domain is called the “grid” or the “mesh.” The creation of the mesh is the most important step in CFD. The mesh density, or inversely, the size of the control volumes, determines the accuracy of the simulation.

The mesh creation consists of three major steps. In the first step the geometry of the model is established, a computer aided design (CAD) program is used to determine dimensions. The second step is the creation of the surface mesh, which is placed on the surfaces created in the CAD geometry modelling phase.

The final step is the interpolation of the surface mesh to the final fully three-dimensional volume mesh.



Mesh	Total Elements	Nodes	Deformation
Coarse	35264	12442	551.2 MPa
Medium	43456	16278	558 MPa
Fine	58489	20462	660 MPa

Based on the results of mesh independence test, a fine grid with approximately 50 thousand elements was selected for present study to computational time.

VIII. RESULT AND DISCUSSION

Simulating a crash-test of a vehicle model moving at a velocity of 27.7 m/s (≈100kmph) in to a rigid immovable barrier is to be carried out and analyzed. It is assumed that the brakes are not applied during the crash event. Weight of the vehicle was assumed to be 1ton (1000 kg).

Three simulations were carried out for the frontal impact. The sequence of images shown below is the image of the vehicle and after it impacts the rigid wall with the specified velocity of 27.7 m/s (≈100kmph). The parameters considered are Total Deformation, Strain energy, Normal Stress.

The results obtained will then be validated and compared with the results of the same crash analysis performed in research paper of an international journal.

i. Total Deformation

- Total Deformation of car with wall

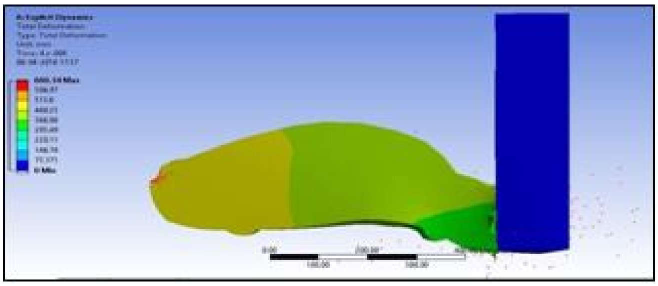


Figure 5- Total Deformation of car with wall

- Total Deformation of car with pole

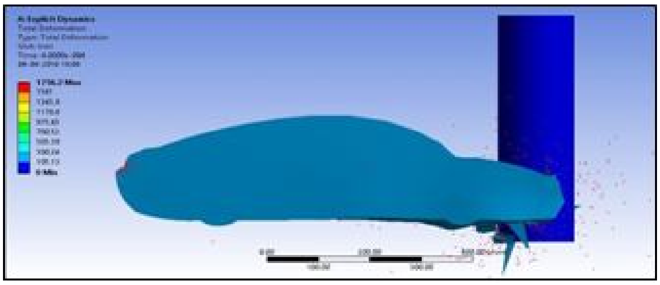


Figure 6- Total Deformation of car with pole

- Total Deformation of car with pyramid

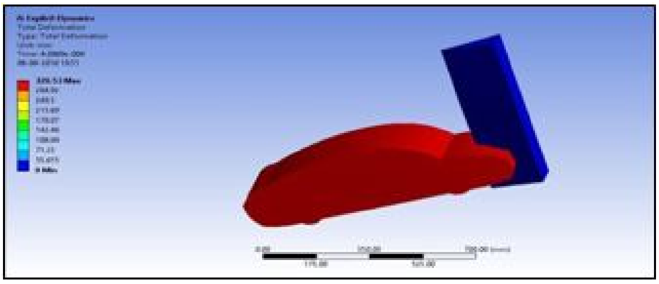


Figure 7- Total Deformation of car with pyramid

Total deformation in the structures due to various impacts is shown. Location for maximum deformation is shown as well.

ii. Normal stress

- Normal stress of car with wall

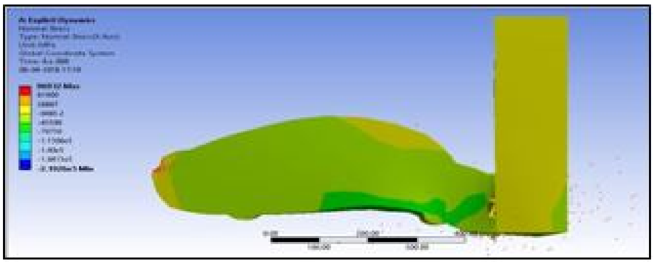


Figure 8- Normal stress of car with wall

- Normal stress of car with pole

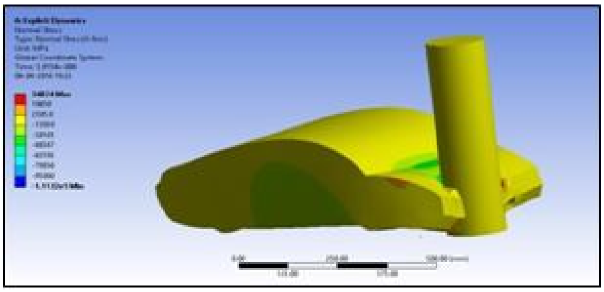


Figure 9- Normal stress of car with pole

- Normal stress of car with pyramid

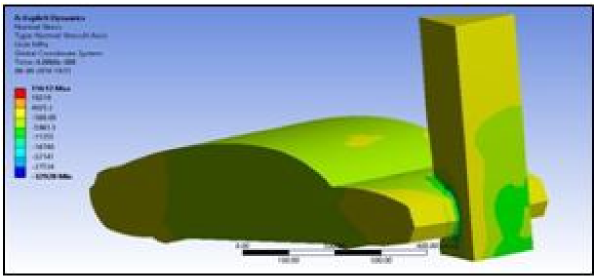


Figure 10- Normal stress of car with pyramid

iii. Strain Energy

- Strain energy of car with wall

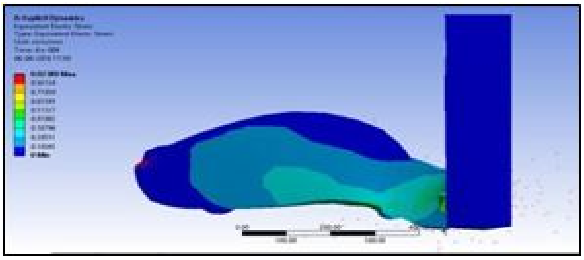


Figure 11- Strain energy of car with wall

- Strain energy of car with pyramid

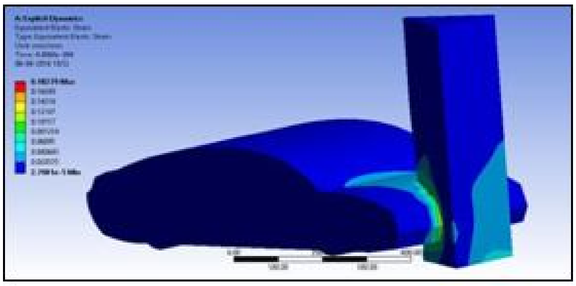


Figure 12- Strain energy of car with pyramid

- Strain energy of car with pole

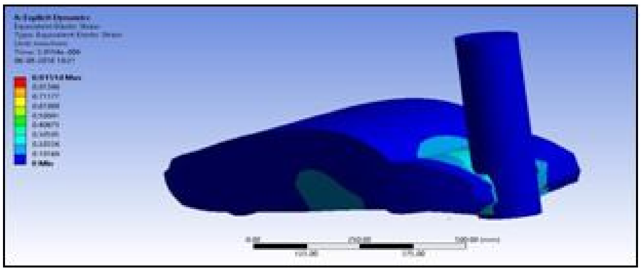


Figure 13- Strain energy of car with pole

Strain energy produced in the members due to frontal and side impact is shown. The location of maximum value is shown as well.

IX. CONCLUSION

- Values of various parameters after impact
- The overall objective of the work was to simulate a Frontal crash-test and validate the results of the simulations obtained from the crash-test. Simulation was performed using the LS-DYNA software package.
- As was observed, the bumper, engine and the rails absorb most of the energy before the wheel impacts the wall. Almost half of the energy of the crash is absorbed by these components after about 0.04sec of the crash initiation.
- It has been observed that there is minimum deformation of the cabin and also there was minimum intrusion of the components into the cabin. Therefore, it can be assumed that the occupants in the cabin would not be caused any injury by a component intruding into the cabin in the event of the crash.
- For more accurate results a more accurate model would be required but the computer resources required for the simulations would have been much higher. Therefore a compromise had to be found wherein the simulation could be performed without the result deviating too much.

PARAMETERS	TOTAL DEFORMATION (mm/mm)	STRAIN ENERGY (J)	NORMAL STRESS (MPa)
Pole	1756.2	0.91514	34874
Wall	660.34	0.92389	96932
Pyramid	320.53	0.18279	15612

Table 3- Values of various parameters after impact

X. FUTURE SCOPE

- The FE model can be used for further simulation of in the simulations of the offset frontal impact test, where one side of the front of the vehicle is impacted against a barrier or another vehicle.
- Other tests include the side impact test, where a vehicle is impacted from the side by and oncoming vehicle and oblique car-to-car impacts the two or more vehicle take part in a collision.
- Rollover simulation can also be carried out wherein the vehicle rolls on its sides due to the cause of an impact or other factors.

REFRENCES

[1] S. W. Kirkpatrick, J. W. Simons, and T. H. Antoun "Development and Validation of High Fidelity Vehicle Crash Simulation Models" 1998

[2] Yucheng Liu "Impact Experimental Analysis and Computer Simulation" 2000

[3] M. Raghupathi, M. Saravanan , G. Sivakumar "Comparison of Crash Worthiness of Two Materials Used in Bus Skeletal Frame" 2015

[4] Mr. Abdullatif K. Zaouk, Dr. Nabih E. Bedewi, Dr. Cing-Dao Kan, and Mr. Dhafer Marzougui "Development and Evaluation of a C-1500 Pick-Up Truck Model For Roadside Hardware Impact Simulation" 1994

[5] Faizan Khatri, Sanchit Jain, Prof. P. Bhaskar "Crash Test Simulation and Analysis of a Passenger Bus Frame" 2014

[6] Mr.C.Sadhasivam, Prof. R.Vijayan. "Vibration and Crash Analysis of Car Body Using ANSYS and 3D LS DYNA" 2011

[7] Tejasagar Ambati, K.V.N.S. Srikanth & P. Veeraraju. "Simulation of Vehicular Frontal Crash-Test". 2012

[8] Waseem Sarwar and Nasir Hayat. "Crash Simulation and Analysis of a Car Body Using ANSYS LS- DYNA" 2013

[9] T. Ananda Babu, D. Vijay Praveen, Dr.M.Venkateswarao. "Crash Analysis of Car Chassis Frame Using Finite Element Method" 201