Crash Simulation and Analysis of a Car Body Using ANSYS LS-DYNA

Waseem Sarwar ¹ and Nasir Hayat ²

¹ Student and ² Professor, Mechanical Engineering Department, UET Lahore

ABSTRACT 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 analyzed.

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 [1]. 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.

Today’s automobile manufacturers are increasingly using lightweight materials to reduce weight; these include plastics, composites, aluminum, magnesium and new types of high strength steels. Many of these materials have limited strength or ductility, in each case rupture is a serious possibility during the crash event. Furthermore, the joining of these materials presents another source of potential failure [3]. Both material and joining failure will have serious consequences on vehicle crashworthiness and must be predicted.

During an automobile crash, some parts in the front of the automobile body may have plastic deformation and absorb a lot of energy. Structural members of a vehicle are designed to increase this energy absorption efficiency and thus to enhance the safety and reliability of the vehicle. The crashworthiness of each member needs to be evaluated at the initial stage of vehicle design for good performance of an assembled vehicle. As the dynamic behavior of structural members is different from the static one, the crashworthiness of the vehicle structures has to be assessed by impact analysis [4].

Hence it becomes necessary to check the car structure for its crash ability so that safety is achieved together with the fuel economy. There are two ways by which this safety feature can be assessed.
a. Performing an actual crash test.
b. 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.

The model to be simulated is usually developed using data obtained from the disassembly and digitization of an actual automobile using a reverse engineering technique. This approach is necessary because the models developed by the manufacturers are proprietary, and not available either to the public or to the government [2].

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.

MATERIAL PROPERTIES.

ANSYS LS-DYNA includes over 40 material models that can be used to represent a wide range of material behavior. We, in our analysis, have used four material models. This keeps the analysis simple. The wall is considered as rigid, while three material models have been incorporated in the FE model of automobile body. All material properties are obtained from ANSYS help manual. In the analysis all material properties have been taken at room temperature (298 K).

The wall is made from Rigid Material. Using rigid bodies to define stiff parts in a finite element model can greatly reduce the computational time required to perform an explicit analysis. [5].Rigid material considered in this paper is steel.

The hyper elastic continuum rubber model defined by Blatz and KO is used in the front part of the model for investigation purposes. Normally, the bumpers are made of M.S or H.S steels. Linear Viscoelastic material model e.g. Glass, has been used as windscreen. The body is made from H.S steel which has been added using the material model Stain rate dependent plasticity. In this case, a load curve is used to describe the initial yield strength, as a function of effective strain rate.

We have based our choice on the research work of Yuxuan Li [1].

FINITE ELEMENT MODEL.

The major step in the use of explicit numerical methods in crash simulations is the proximity of the FE model to reality. Closer the model to reality, the more reliable the results would be. Therefore before the start of the analysis, the model has to be validated to ensure that it is close
to the real car. The method employed is the comparison of the acceleration responses at various locations with the actual physical data [2]. Once close approximation is achieved, the model is validated and it can be used with confidence in the crash simulation.

Table: 1 Material Properties.

<table>
<thead>
<tr>
<th>Model</th>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rigid Material Model</td>
<td>MP,ex</td>
<td>1,207e9</td>
</tr>
<tr>
<td></td>
<td>MP,nuxy</td>
<td>1,0.3</td>
</tr>
<tr>
<td></td>
<td>MP,dens</td>
<td>1,7580</td>
</tr>
<tr>
<td>Blatz-Ko Rubber Elastic Model</td>
<td>MP,dens</td>
<td>1,1150</td>
</tr>
<tr>
<td></td>
<td>MP,gxy</td>
<td>1,104e7</td>
</tr>
<tr>
<td>Viscoelastic Model</td>
<td>MP,dens</td>
<td>1,2390</td>
</tr>
<tr>
<td></td>
<td>TBDATA,46</td>
<td>27.4e9</td>
</tr>
<tr>
<td></td>
<td>TBDATA,47</td>
<td>7.0</td>
</tr>
<tr>
<td></td>
<td>TBDATA,48</td>
<td>60.5e9</td>
</tr>
<tr>
<td>Strain Rate Dependant Plasticity Model</td>
<td>MP,ex</td>
<td>1,209e9</td>
</tr>
<tr>
<td></td>
<td>MP,nuxy</td>
<td>1,0.29</td>
</tr>
<tr>
<td></td>
<td>MP,dens</td>
<td>1,7850</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Strain Rate</th>
<th>Yield Stress</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>207e6</td>
</tr>
<tr>
<td>0.08</td>
<td>250e6</td>
</tr>
<tr>
<td>0.16</td>
<td>275e6</td>
</tr>
<tr>
<td>0.4</td>
<td>290e6</td>
</tr>
<tr>
<td>1.0</td>
<td>300e6</td>
</tr>
</tbody>
</table>

Fig: 1 Graph showing the relation between yield stress and strain rate

Fig: 2 FE Model showing the material models. Violet=HS Steel, Red=Glass, Green=Rubber.

Since, no such data was available to us; there was no obvious way to validate the model. However, the deceleration responses were matched with those of the accepted FE models. Though, the numerical values were different, the desired shape of the curves was obtained.

MED UET Taxila (2007)
Owing to the limitation of time and resources, the model was kept as simple as was possible.

To ensure the correctness and effectiveness of FE model, the following efforts were made:

- Mapped meshing was used, since the goal is to simulate the frontal impact of the car, the meshing of front car body, where the stresses were maximum, was refined.
- As the number of integration points directly impact the CPU time, reduced integration is employed. Despite being robust for large deformations and saving extensive amounts of computer time, the one-point integration causes hourglass deformations. The general principles are to use a uniform mesh and to avoid concentrated loads on a single point and the hourglass energy should not exceed 10% of the internal energy to avoid hourglass deformations. Though our model was more or less uniformly meshed, they did appear. In order to avoid this, fully integrated shell formulation was used that completely eliminated the hourglass modes as shown in the graph.

![Graph showing reduced integration vs fully integrated shell](image)

(a) Reduced Integration  (b) fully integrated shell

Fig: 3 Where red one =hourglass energy, green one =internal energy.

- Automatic general single surface contact algorithm is adopted aiming at simplification as in this case Ls-DYNA program automatically determines which surfaces within a model may come into contact. Moreover it is robust.
- In order to transmit effects of HS steel onto the glass, it is coupled from the three sides.

**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.

The choice was made due to the following options available with shell 163:

MED UET Taxila (2007)
a. Since our model was basically a shell with minor thickness, it was reasonable to choose shell 163 for analysis.

b. Hourglass control. Fully integrated shell formulation was available in this element. The final FE model had 4 material models and 371 elements.

**NUMERICAL RESULTS.**

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. The initial velocity was applied to the car which served as the impacting object. At t=0.12s, the collision occurs. The initial distance between the front of the car and the rigid wall was kept at 4 meters. Total simulation time was 0.25s.

![Fig: 4 Showing the initial position](image)

**Stresses:** The figure 5 gives the von misses stresses at the point of impact and at the end of simulation. Maximum stresses are attained at the front at t=0.25s.

![Oblique view. t=0.12s](image) ![Left view .t=0.25s](image)

**Fig: 5**

**Gross Motions:** Gross motion reveals the overall dynamic response of the model and is akin to rigid body motion.
Absorbed energy: There are two main concerns in the crashworthiness study of vehicles; the absorbed energy, and the peak decelerations at the driver’s cabin. The basic idea is to design vehicle structures which minimize the amount of injury-causing crash energy that reaches the occupants. Generally, this is accomplished by developing structural zones that absorb crash energy outside the passenger compartment – these are called "crush zones" and they collapse in a prescribed way at specified loads, thereby providing the appropriate energy absorption and deceleration of the passenger compartment. In the case of the crush zones, the energy is absorbed by the folding and bending deformation of the metal structure. To the strongest extent, automotive engineers attempt to maximize the transmission of crash energy through the structure axially (from front to back) so that the structure folds like an accordion as it absorbs the crash energy. During the very short time period (in milliseconds) of impact, the kinetic energy must be absorbed primarily by the vehicle’s crush zone. The curved shape of the below graph between the t=0.12-0.20s is due to the breakage of the glass as is clear from the gross motions.

Fig: 6

Fig: 7 Variation of internal energy with respect to time.
Acceleration Responses: It must also be noted that decelerations are inversely related to the stiffness of the structure. It is neither cost-effective, nor safe, for a vehicle to be designed as rigid or heavy as to survive any collision imaginable without damage. The human body can only withstand deceleration to a certain limit, beyond which severe internal injury or death occurs. A crashworthy vehicle must be designed to deform according to a deceleration-time response, or crash pulse. Ideally, engineers try to design the deformation of the structure to achieve a uniform deceleration, for example 20-25 G’s when measured in a fixed barrier, frontal crash at 30 mph (where G is the pull of gravity).

A small, light car will always experience a higher deceleration level in a crash test as compared to its heavier opponent, as Deceleration = crush load / car mass.

In conclusion, the frontal structural stiffness of a low mass vehicle must be at least equal or slightly higher than the stiffness of its heavier counterpart. As, the yield stress of H.S steel (220MPa) is greater than the yield stress of M.S steel (160MPa), it can be employed in lighter vehicles, replacing M.S.

Fig: 8 Acceleration responses. The first fig shows the locations where the acceleration responses are potted.

Fig: 9 First Principal Strains at t=0.13s
CONCLUDING REMARKS.

The H.S steel simplified FE model was investigated using ANSYS LS-DYNA. Since, the main aim of the project was to develop expertise in the field of crash analysis; the analysis was kept simple using assumptions. It was noted during the course of the project that H.S steel could be used effectively for lightweight ness without affecting the necessary impact energy absorbing capacity of the car body.

REFERENCES