Design and Crash Analysis of a Passenger Car using ANSYS Workbench

C. Sai Kiran[1], J. Sruthi[2] and S. Chandrahas Balaji[3]
[1]Asst. Professor, CVR College of Engineering/Mechanical Engg. Department, Hyderabad, India.
Email: csaikiran001@gmail.com
[2]Asst. Professor, CVR College of Engineering/Mechanical Engg. Department, Hyderabad, India.
Email: sruthij02@gmail.com
[3]Asst. Professor, Nova College of Engg. & Technology/Mechanical Engg. Department, Hyderabad, India.
Email: chandrahas.b2011@gmail.com

Abstract: **In this paper, an automobile (car body structure) was designed and analyzed which can be used for travelling of the passengers and goods. The car body structure is made of aluminum and the crash analysis is performed on the car body structure to know how the car body structure deforms in a crash accident. Crash analysis is performed on car body designs for ensuring safety of passengers in accidents. A detail car body deformation analysis has been performed in ANSYS based on 3D LS-DYNA. A frontal impact crash of a car is performed. A Finite Element model of a car structure was used in a crash simulation to assess the safety and to know crashworthiness of the car.**

Index Terms**: Crash analysis, Crashworthiness, ANSYS, LS-DYNA.**

I. Introduction

A car is a wheeled, self-powered motor vehicle and a product of the automotive industry which is mainly used as transportation. Cars are constructed primarily for transportation of people rather than goods, which generally have seating for one to eight people. Cars are mainly designed to run on roads, which typically have four wheels with tyres. Automobiles can be classified by size or weight. Size classification of an automobile is based on wheelbase and weight classification of an automobile is based on curb weight.

Andrew Hickey et al. [1] has performed a quasi-static simulation, to simulate the car crash by using finite element method (FEM). Safety of passengers is one of the most important design considerations in the automobile community. Therefore, a crash test is a crucial step to validate the car design. Experimental crash tests results in higher cost, and acquired data might not be correct. Therefore, a numerical modelling and the simulations are used for studying a car crash than to perform experimental testing. Hence, a powerful numerical tool, FEM plays a crucial role in crash test simulations.

C. Sadhasivam et al. [2] has performed a detail car body mode analysis and stress analysis based on 3D LS-DYNA in ANSYS. Modal analysis has been performed to know the natural frequencies and mode shapes of a car body structure. Vibration and crash analysis of the car body is performed, which includes dynamic, static and crash analysis. Most of the automobile manufacturers generally prefer lightweight materials to reduce weight and these include composites, aluminium, magnesium or new types of high strength steels. These materials have a limited strength or ductility, in case of rupture which is a most common seen phenomenon during a crash accident. Material joining failure is also one of the consequences on the vehicle crashworthiness. In a car crash, front-part of the automobile structure absorbs a lot of impact and undergoes plastic deformation. Most of the vehicles are designed to increase the absorption efficiency, to enhance the safety of passengers and reliability of the vehicle. Crashworthiness of different parts of a vehicle needs to be evaluated at the initial stage of the vehicle design only. The dynamic behaviour of a structural member is always different from the static behaviour, therefore crashworthiness of the vehicle can be known by impact analysis. Hence, it is necessary to check the crash ability of car structure for both safety and fuel economy. The two ways of ensuring safety are by performing a crash test of a car or by simulating the crash analysis of the modelled car structure in analysis software.

Byeong Sam Kim et al. [3] has performed a crash analysis of sub frame and upper body for neighborhood electric vehicle (NEV). NEV's front platform assembly behaviour was simulated in LS-DYNA and results were observed when it is subjected to a frontal car crash. The safety of passengers at low cost reduction has been researched. When a vehicle crashes, the passengers inside the vehicle must be free from injury and the vehicle must be able to withstand impact loads. In crash accidents, capability of the vehicle structure to absorb the energy can be defined as crashworthiness. The vehicle structure should be designed to withstand higher speed and the passengers should not experience a net deceleration.

Lin et al. [4] had performed the computer simulation of a car crash analysis. They have analysed two crash situations: a higher speed car crashing into a wall and a high velocity car crashing into a static car. The objective of the research was to know the sources which can harm the driver and the passengers when car accidents occur and to create a model of a bumper for knowing its potentiality to withstand impact loads on it. The Simulations on bumper are performed to assure that the bumper design meets the safety requirements.

Tejasagar et al. [5] has studied different car crash simulations by using computer softwares because to reduce automobile developing time and to reduce the cost of manufacturing. A frontal impact crash analysis of a car was researched, which made an impact on car crashworthiness. Crash test is one of the destructive tests performed on the car for ensuring safer design in crashworthiness and to know the crash compatibility of automobiles. The vehicle manufacturers perform different crash tests to ensure safety of the cars under various conditions such as various types of crashes, from different sides, different angles and with different objects, including other vehicles.

Praveen et al. [6] has performed a car crash analysis in non-linear transient dynamics. In the crash test, frontal collision and sideways collision analysis is performed to know deformations of the car. Crashworthiness of the car simulations is performed in Finite Element Analysis (FEA). The chassis frame takes the loads of a heavy vehicle, its function is to carry the loads on the vehicle safely for each operating condition. The frame of chassis should be able to support different chassis components and vehicle structure. Chassis frame should withstand both static loads and dynamic loads without any distortion or deflection in the vehicle. The frontal collision and side collision conditions are tested on the generated model, the total deformations and stresses developed are determined.

Saeed Barbat et al. [7] has modelled a car to evaluate the effects of design variables on dummy responses for front-to-side vehicle crash analysis. The striking or hitting vehicle was selected to be a SUV while the struck vehicle was a small size passenger car which consists of four seats. A deterministic approach that allowed analytical prediction equations for dummy responses was generated. A baseline front crash vehicle to the side crash was modelled in FE and correlated to a physical front crash to the side crash.

Ravinder Reddy et al. [8] has performed a crash analysis on the passenger car frontal bumper beam by using Hypermesh and RADIOSS software. A bumper is used for passenger's protection from both front and rear collisions. Bumpers are used for absorbing impact energy. Design of a bumper is based on the degree of absorbing impact loads. The analysis is performed on the bumper beam model with different materials and different designs of the bumper. The analysis was done against a fixed wall with different materials of bumper. Behaviour of the bumper in crash simulations is observed and modifications are done in the design of bumper. The materials of bumper used for crash test analysis are carbon fibre composite, E-Glass epoxy and aluminium 6063-T6.

Xinping Song et al. [9] has performed a car crash analysis in FEM. The front collision of crash test was simulated in the LS-DYNA software and car model is designed in Hypermesh software. The longitudinal front beam is used for absorbing the kinetic energy during crash test of the vehicle. The longitudinal front beam can efficiently absorb the kinetic energy and avoid high acceleration of the car. The acceleration time history curves and the deformations of the car during the crash test were analysed. From the results, anti-impact capacity of the car is observed. These simulation results show that optimization of the car structure can improve the results in the car crashworthiness, but further improvements are necessary in the car structure for the safety of passengers. Selecting the correct material of the car in the vehicle collision experiment, will influence the results.

A. Types of Car Body Structures

The car body structures vary from country to country. The different car names are given to different car models based on the technology used, utility, customization and design. General car body styles used in India are given below:

1. Convertible
2. Hatchback
3. Wagon
4. Sedan
5. Jeep
6. MUV/SUV
7. Van
8. Coupe

A hatchback model of the car is modelled in CAD software and crash analysis is performed on the car model in ANSYS software.

B. Material Used For Car and Wall

Aluminium material is widely used for car body structures because it is corrosion resistant, ductile, has a high electrical conductivity and is a soft material compared to other available engineering materials. It is one of the lightest engineering metals available in nature, which has a superior strength to weight ratio when compared with steel. Therefore, aluminium is suited as the material for car body, and concrete is taken as the wall material. The properties of aluminium and concrete material are shown in Table 1.

TABLE I.
PROPERTIES OF ALUMINIUM

S. No	Properties	Aluminium	Concrete
1.	Density (g/cm^3)	2.70	2.35
2.	Young's Modulus (GPa)	70	33
3.	Poisson ratio	0.35	0.2
4.	Shear modulus (GPa)	26	16
5.	Bulk modulus (GPa)	76	18
6.	Thermal Conductivity (w/mK)	205	0.8

II. MODELLING AND ANALYSIS

A. Assembled Model

A car body outer part is modelled by using different commands in SOLIDWORKS software. After creating the car model, another car model is placed exactly opposite to the first car structure at a distance of 10 m as shown in Fig. 1.

Figure 1. Assembling of the car structures.

The outer part of a car body and the wall are modelled in SOLIDWORKS software and are placed exactly opposite to each other at a distance of 10 m as shown in Fig. 2.

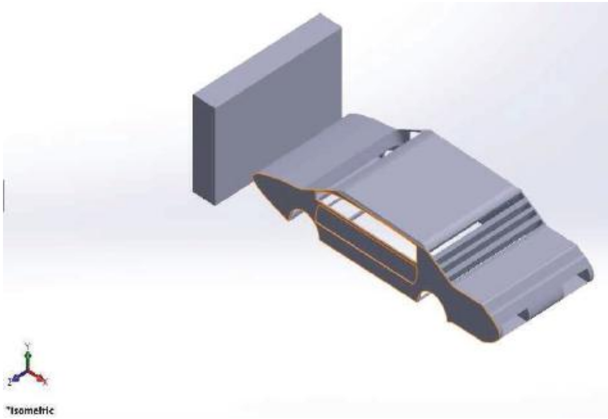

Figure 2. Assembling of the car and wall.

B. Element Type

The element type used is SOLID 186. SOLID 186 is a higher order 3-D with 20-node having three degrees of freedom per node, solid element that exhibits quadratic displacement behaviour. This element supports large deflection, plasticity, large strain capabilities, hyper elasticity, stress stiffening, and creep.

C. Meshing

After assigning the element type of the car structure, the solid model is converted into the IGES format and imported into ANSYS Workbench. Meshing is an important process of an analysis and it should be performed on the car structure. Meshing is the process of dividing the created model in number of divisions or elements which consists of nodes. By applying meshing process, we can determine the efficiency and effectiveness of any analysis. An automated mesh generation is as shown in Fig. 3.

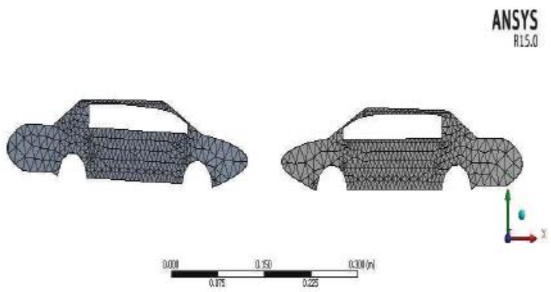

Figure 3. Applying mesh on the car structures.

Meshing is applied by using automatic mesh. Under mesh sizing, mesh was set to fine mesh to achieve accurate and precise results. Rather than using a fine mesh all over the components, coarse mesh was used on large area and fine mesh was used at the area of stress concentration as shown in Fig. 4.

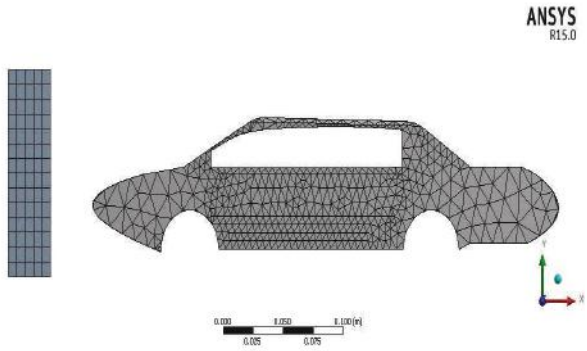

Figure 4. Applying mesh on the car and wall.

D. Applying Loads

In the analysis setting, a velocity of 1000 m/s is assigned to the car structure in X-direction and the end time is 0.001 seconds as shown in Fig.5

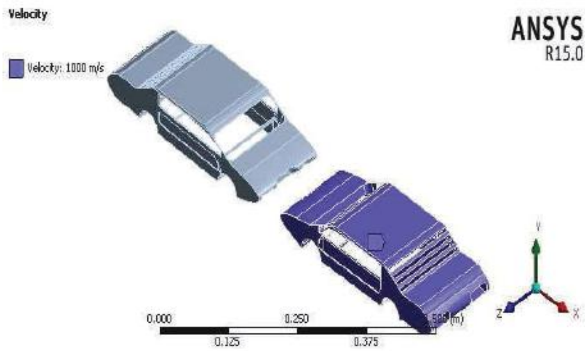

Figure 5. Assigning velocity of 1000 m/s to the car structure.

In the analysis setting, a velocity of 1000 m/s is assigned to another car structure in X-direction such that car to car collision would take place and the end time is 0.001 seconds as shown in Fig.6.

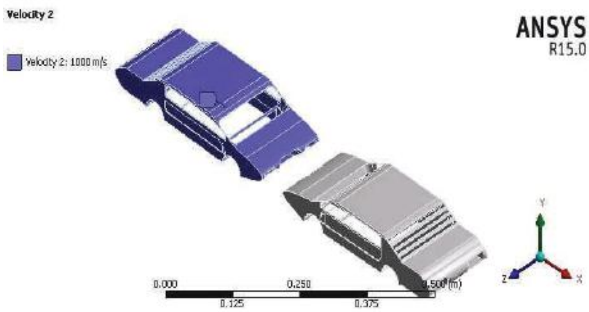

Figure 6. Assigning velocity of 1000 m/s to another car structure.

In the analysis setting, fixed support is applied to the concrete wall, so that the wall is constrained in all degrees of freedom and it would withstand different types of loads as shown in Fig.7.

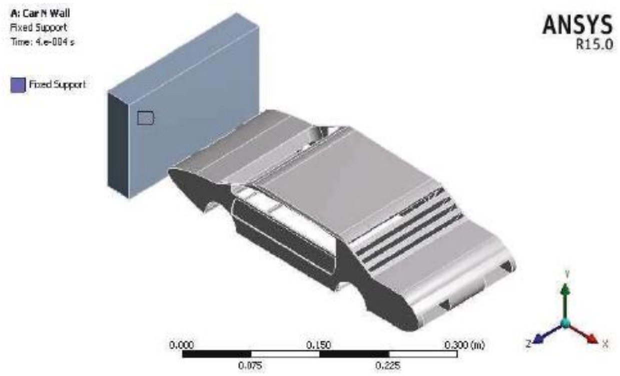

Figure 7. Fixing the wall.

In the analysis setting, a velocity of 1500 m/s is assigned to the car structure in the X-direction so that the car would hit the wall as shown in Fig.8.

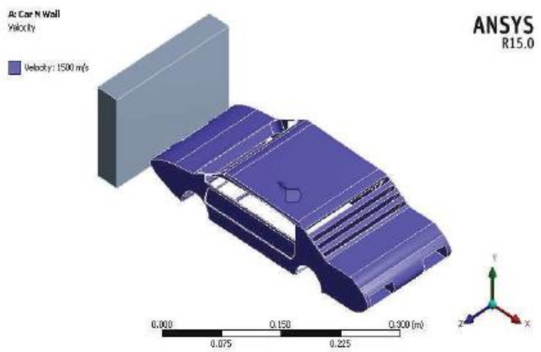

Figure 8. Assigning velocity of 1500m/s to the car structure.

III. Results And Disscussions

After assigning velocity to car structures and fixing the wall, crash analysis is performed in ANSYS workbench and the following results were observed in crash analysis.

A. Total Deformation of Car to Car

After performing crash analysis of one car hitting another car with a velocity of 1000 m/s, a maximum total deformation of 0.09 m and minimum total deformation of 0.03 m is observed from the Fig.9.

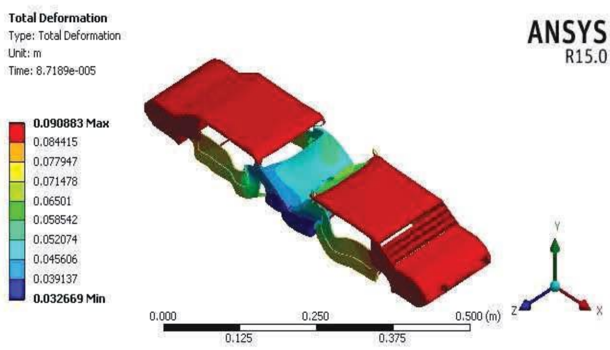

Figure 9. Total deformation of the car to car.

B. Total Deformation of a Car to Wall

After performing crash analysis of the car hitting the wall with a velocity of 1500 m/s, maximum total deformation of 0.33 m and minimum total deformation of 0.03 m is observed in the car from the Fig.10.

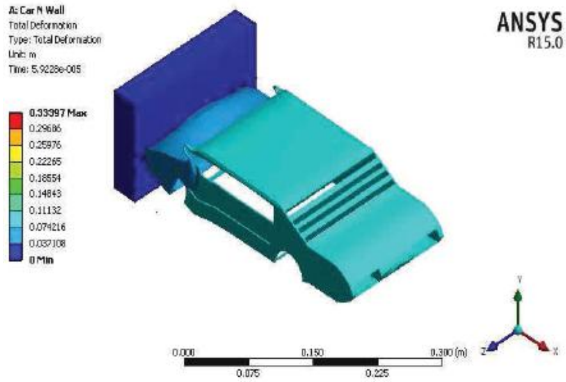

Figure 10. Total Deformation of the car hitting the wall.

IV. Conclusions

In this paper, a car structure is modelled in SOLIDWORKS, a crash analysis of one car to another car with the initial velocity of 1000 m/s and car to wall with the initial velocity of 1500 m/s is performed in ANSYS software. The total deformation of the car structure is analyzed. From the results, it is observed that the maximum deformations in car structures crashing a car and wall are within the limits. Therefore, the modelled car structure is safe for the passengers to travel in view of car accidents.

In the future scope, different car body structures can be modelled and different car body materials can be taken by assigning different velocities for crash analysis.

References

[1] Andrew Hickey, and Shaoping Xiao, "Finite element modelling and simulation of car crash," *International Journal of Modern Studies in Mechanical Engineering*, ISSN 2454-9711, Volume 3, Issue 1, 2017, pp. 1-5.

[2] C. Sadhasivam, and S. Jayalakshmi, "Simulation of car body by development of static and dynamic analysis," *International Journal of Automobile Engineering Research and Development*, ISSN (P): 2277-4785; ISSN (E): 2278–9413 Vol. 4, Issue 3, June 2014, pp. 1-6.

[3] Byeong Sam Kim, Kyoungwoo Park, and Youn-Min Song, "Finite element frontal crash analysis of vehicle's platform with upper and sub frame body," *International Association for Automation and Robotics in Construction*, June 2011, pp. 1326-1329.

[4] Lin C. S, Chou K.D. and Yu C.C, "Numerical simulation of vehicle crashes," *Applied Mechanics and Mat*erials, 590, December 2014, pp. 135.

[5] Tejasagar ambati, K.V.N.S. Srikanth and P. Veeraraju, "Simulation of vehicular frontal crash-test," *International Journal of Applied Research in Mechanical Engineering*, ISSN: 2231 –5950, Volume-2, Issue-1, 2012.

[6] T. Ananda Babu, D. Vijay Praveen and Dr. M. Venkateswarao, "Crash analysis of car chassis frame using finite element method," *International Journal of Engineering Research & Technology (IJERT)*, ISSN: 2278-0181 Vol. 1 Issue 8, October - 2012.
[7] Saeed Barbat, Xiaowei Li and Priya Prasad, "Vehicle-to-vehicle front-to-side crash analysis using a CAE based methodology," in *Passive Safety Research and Advanced Engineering*, Ford Motor Company, United States, Paper Number 07-0347.
[8] P. Ravinder Reddy, and Thota Harish, "Design and crash analysis of passenger car frontal bumper beam using Hypermesh and Radioss," *International Journal of Emerging Research in Management & Technology*, ISSN: 2278-9359, Volume-4, Issue-12.
[9] Chunke Liu, Xinping Song, and Jiao Wang, "Simulation analysis of car front collision based on ls-dyna and hyper works," *Journal of Transportation Technologies*, Vol. 4, Issue 3, June 2014, pp. 337-342.