International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395 -0056
Volume: 03 Issue: 05 | May-2016 www.irjet.net p-ISSN: 2395-0072

Design Evaluation of Fuel Tank & Chassis Frame for Rear Impact of Toyota Yaris

Ashish T. Patil[1], N.K. Kelageri[2]

[1]Dept. of Mechanical Engineering, KLE Dr M.S. Sheshgiri College of Engg. & Tech. Belagavi, Karnataka, India
[2]Professor, Dept. of Mechanical Engineering, KLE Dr M.S. Sheshgiri College of Engg. & Tech. Belagavi, Karnataka, India

---***---

Abstract – *Vehicle crash is a highly non-linear transient dynamics phenomenon. . Crashworthiness simulation is one typical area of application of Finite-Element Analysis (FEA). This paper deals with behavior of the Toyota Yaris car model in the event of the rear crash and its effect on the fuel tank. The crash simulation is done by using CAE tools like Hypermesh v11.0, LS-DYNA and LS-PrePost. The standards followed were FMVSS 301R and AIS-101. The crash analysis simulation and results can be used to assess both the crashworthiness of the current model and to investigate ways to improve the design in order increase the crashworthiness of the Fuel Tank. The modified design model showed better results as compared to the baseline model. The validation of the crash simulation is done by comparing the 'g' values of the baseline and modified model with the experimental value.*

Key Words: **Rear Crash, Hypermesh v11.0, LS-DYNA, LS-PrePost, Fuel Tank, FMVSS 301R, AIS-101**

1. INTRODUCTION

Automobiles get impacted in different types of crashes in the real world today which may injure or kill the occupants. Crash can be classified as frontal, side, rear and roll over. In United States side and rear impacts are the second and third most common type of vehicle impact, respectively after frontal impacts. The National Highway Traffic Safety Administration (NHTSA) introduced the Federal Motor Vehicle Safety Standard (FMVSS) No. 301 "Fuel System Integrity" to reduce the deaths and injuries resulting from post-crash vehicle fires. The rear impact crash situation is as shown in the Fig 1.1 below

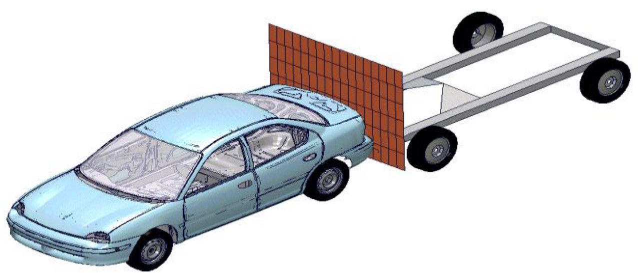

Fig -1.1 : Rear Impact Crash Situation.

Automobiles tend to have inertia effects due to rear crash which causes sloshing or damage to the fuel tank resulting in fire hazards. Comparison between Steel and Plastic material for the fuel tank is done to check the performance attributes and competitive analysis. Stiffness and energy absorption are important criterion.

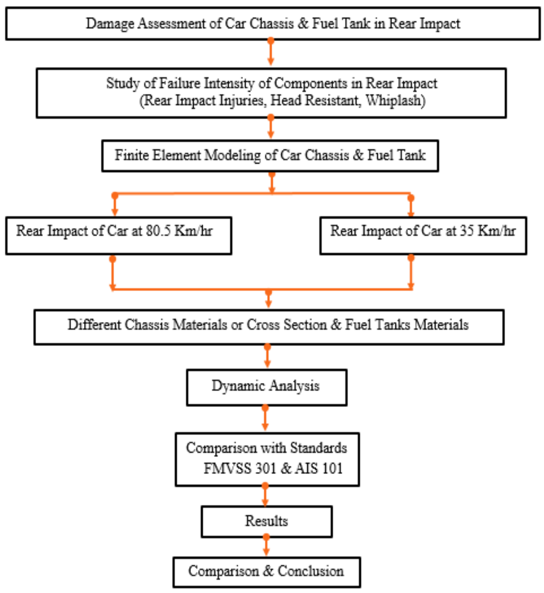

Fig-1.2: Methodology of the crash analysis.

Fig 1.2 shows the methodology used in the rear crash analysis.

2. CAD MODEL OF THE CAR

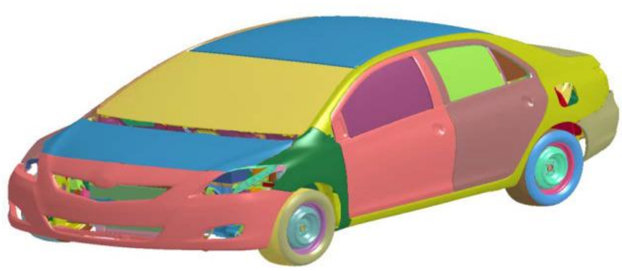

Fig-2.1: CAD model of Toyota Yaris.

Fig 2.1 shows the CAD model of the Toyota Yaris. There are total 917 components of the car which are assembled together. The dimensions of the car model are same as that of the actual car. This model is saved in .IGES format and is imported to Hypermesh v11.0 for meshing and adding material cards to the FE model. The meshing includes combination of linear, Quad2, Tri3, Tetra4, Hexa8 and Penta6 elements. In this analysis the performance of the fuel tank is only evaluated during the rear crash analysis. The main intension is to prevent the fuel tank from damage and fire hazards.

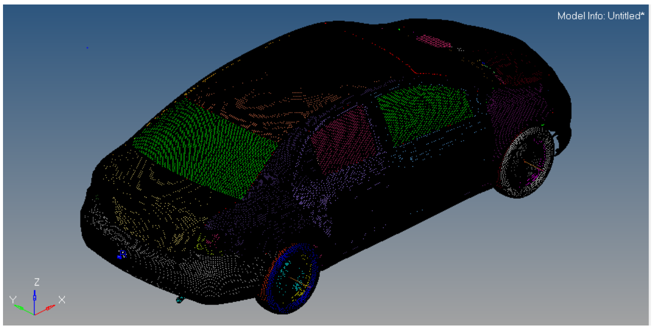

Fig 2.2: Meshed model of the car.

The meshed model the car is shown in Fig 2.2. A standard mesh sensitivity analysis is carried out in order to ensure that the results obtained are effectively insensitive to the size of the elements used. After meshing all the components with respective elements and sizes, quality criteria of the elements is checked and maintained. LS-DYNA deck is prepared in the Hypermesh by assigning the materials and properties and by giving proper control cards and contacts.

3. MATERIAL PROPERTIES

Material properties need to be selected from standard material handbook. Most of the Automobile parts are generally made of steel. Properties were assigned to all components as shown in the Table 3.1

Table -3.1: Material Properties

	Baseline Model	Modified Model
MATERIAL PROPERTIES	HIGH STRENGTH LOW ALLOY STEEL	HIGH DENSITY POLYETHYLENE
Composition	Al- 0.015%, C- 0.05%, Fe- 97.5%, Mn- 0.88%, Nb- 0.09%, P- 0.025%, S- 0.025%, Si- 0.32%, Ti- 0.12%	Polymer- 100%
Density	7800 Kg/m^3	952 Kg/m^3
Price*	35 INR/Kg	76.9 INR/Kg
Young's Modulus	2 x 10^5 N/mm^2	1.07 x 10^3 N/mm^2
Poisson's Ratio	0.286	0.41
Elongation (Strain %)	0.8 %	13 %
Maximum Service Temperature	502 °C	129 °C
Minimum Service Temperature	- 63 °C	- 82 °C

Materials are assigned using *MAT cards available in LS Dyna. The main cards used are MATL1, MATL9, MATL20 and MATL24. Table 3.1 shows the material properties for the baseline model and modified model respectively.

4. SIMULATION AND RESULTS

Model deck is prepared in Hypermesh and solved in LS-DYNA. The simulated results are viewed in LS-PrePost. The crash simulation model setup for the rear crash is as shown in the Fig 4.1

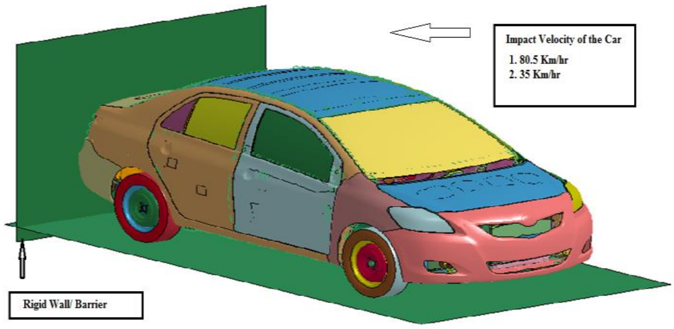

Fig 4.1: Rear crash simulation model setup.

The car model is placed on the surface and has been given velocity in 'x' direction and is made to impact on the rigid wall which is fixed. The crash simulation is done for the following four cases as shown in Table 4.1

Table–4.1: Different simulation cases.

Case I		Case II		Case III		Case IV	
Impact Velocity (Km/hr)	Material Used For The Fuel Tank	Impact Velocity (Km/hr)	Material Used For The Fuel Tank	Impact Velocity (Km/hr)	Material Used For The Fuel Tank	Impact Velocity (Km/hr)	Material Used For The Fuel Tank
80.5	High Strength Low Alloy Steel	35	High Strength Low Alloy Steel	80.5	High Density Polyethylene	35	High Density Polyethylene

4.1 Simulation Result For Case I and Case II

The crash simulation results for Case I and Case II are evaluated and we find that there is a severe damage to the fuel tank at the speed of 80.5 Km/hr i.e. for the case I and there is not much damage to the car as well as the fuel tank when the car is impacted to the rigid wall at the velocity of 35 Km/hr i.e. for the case II. Fig 4.2 shows the amount of damage caused to the car and the fuel tank. This damage is severe and it will make the fuel tank to burst and cause gasoline spillage which may result in fire hazard. Whereas there is very little damage to the fuel tank for the speed of 35 Km/hr and the fuel tank remains safe after the rear crash.

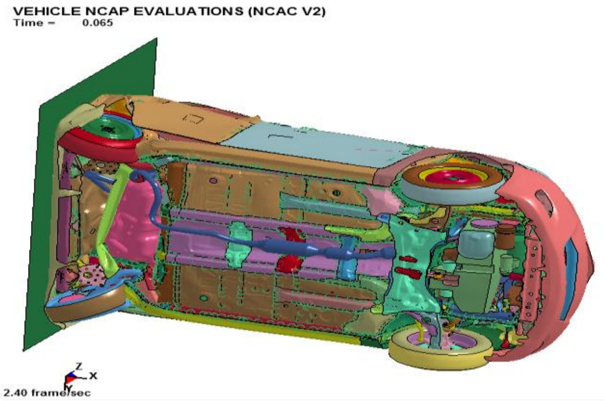

Fig- 4.2: Result of simulation for case I.

The simulation result for the case II i.e. for 35 Km/hr is as shown in Fig 4.3 below.

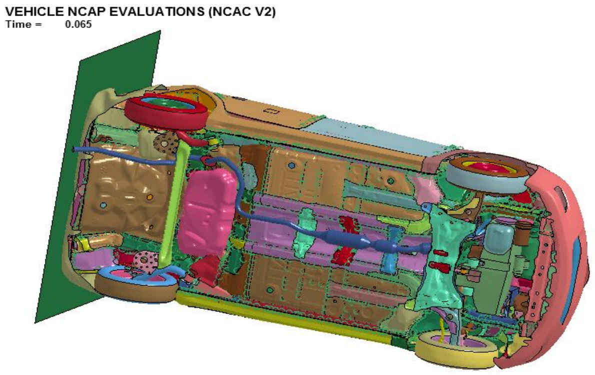

Fig- 4.3: Result of simulation for the case II.

Since the damage to the fuel tank is severe in case I we need to make some modifications in the design of the car.

4.1.1 Effective Plastic Strain For Case I

The effective plastic strain on the fuel tank is evaluated after the simulation for the case I is done which is shown in Fig 4.4 below.

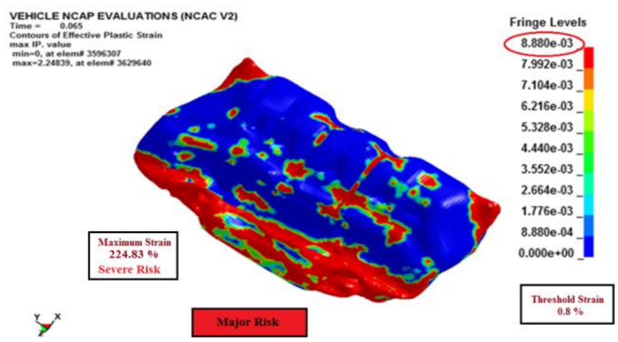

Fig- 4.4: Effective plastic strain on the fuel tank.

The maximum strain percentage of the fuel tank with High Strength Low Alloy Steel after the simulation is 224.83 % which is very severe and surely cause the fuel tank to burst.

5. DESIGN MODIFICATIONS

1. Fuel tank material is changed to High Density Polyethylene.

2. The thickness of the cross member is increased. Cross member of the rear part of the car chassis is as shown in Fig 5.1 below.

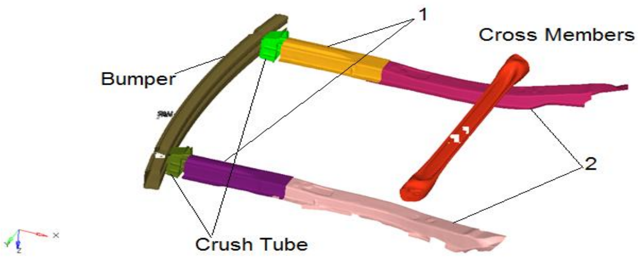

Fig-5.1: Rear chassis cross members.

The thickness of the cross members (1) is changed from 1.5 mm to 2.0 mm and also this cross member is coated with Carbon core. The thickness if the cross members (2) is increased from 1.9 mm to 2.4 mm.

3. The depth of crush extrusion is increased by 0.5 mm.

4. The profile of the rear Bumper is changed to Double Hat Profile and also increasing the thickness to 2.5 mm from 1.5 mm as shown in Fig 5.2 below.

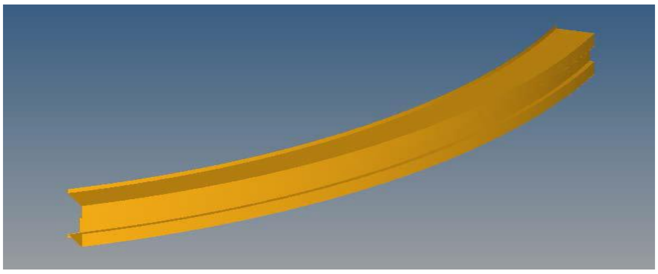

Fig-5.2: Double hat profile of the bumper.

Now after doing these design modifications i.e. for case III the crash simulation is done again. The result of the simulation for case III showed improved results by preventing the fuel tank from damage as shown in Fig 5.3.

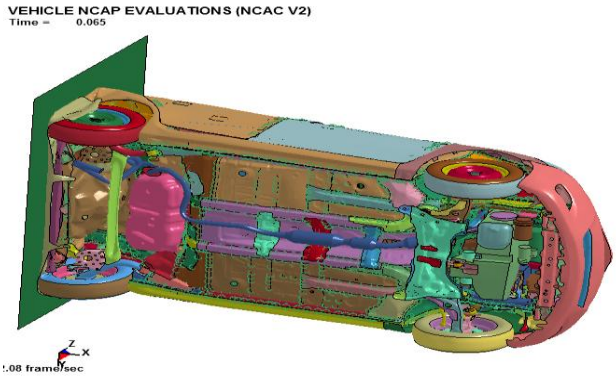

Fig-5.3: Simulation result for case III.

Crashworthiness of the fuel tank is increased because of the design modifications which transfers minimum amount of impact force to the fuel tank by absorbing the maximum energy during the rear crash. This prevents the fuel tank from bursting and fire hazards.

5.1 Effective Plastic Strain For Case III

The effective plastic strain in the fuel tank after the simulation for case III is done is shown in the Fig 5.4 below.

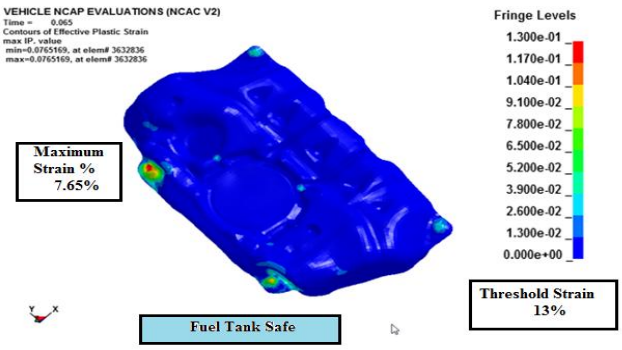

Fig-5.4: Effective plastic strain in the fuel tank for case III.

The permissible strain percentage for the HDPE material is 13%. When this material is used for the fuel tank during the design modification the maximum plastic strain percentage was 7.65% which confirmed that the fuel tank was safe in the event of the rear crash.

6. COUNTER PLOTS

The counter plots were obtained after the simulation is done for all the cases. The counter plots gives us the clear idea of how the design modification has been successful in reducing the damage to the fuel tank.

6.1 Force Plots in Rear Chassis Frame

Force impact on the rear chassis cross member for both case I and case III is shown in the Chart 6.1 below.

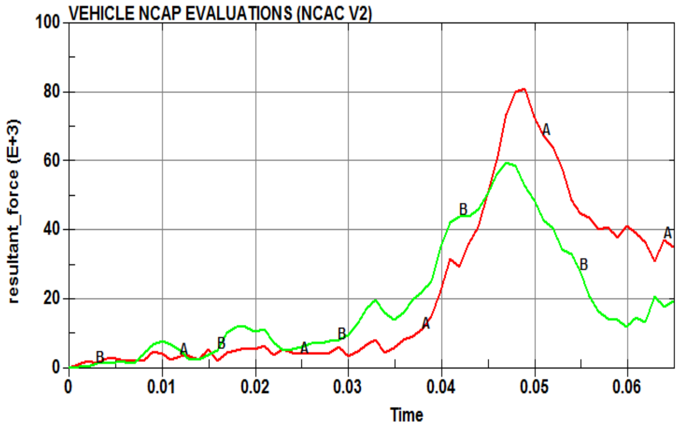

Chart- 6.1: Force v/s Time plot for Case I & Case III.

As seen from the graph the red line (A) indicates the impact force on the cross members in case I and the green line (B) indicates for the case III. The maximum impact force in case I is 80 KN and in case III is 60 KN respectively. A decrease of 25% in the impact force on the rear cross members is found after the modification.

6.2 Force Plots for Connections in the Fuel Tank

The effect of impact force was also recorded at the bolting locations of the fuel tank. The bolting locations are as shown below in Fig 6.1.

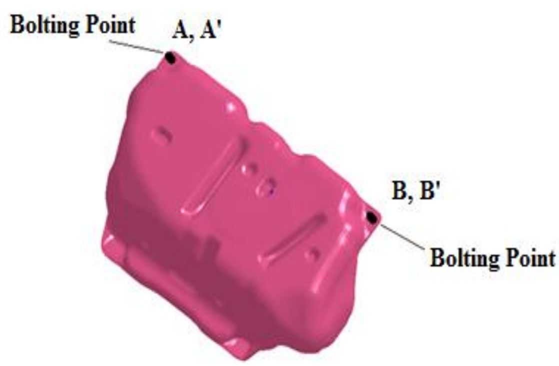

Fig-6.1: Bolting location in the fuel tank.

The result of simulation for the impact force on the bolt locations is as shown in Chart 6.2 below. A and B represents bolt locations in case I whereas A' and B' represent bolt locations in case III simulation. The difference in the impact force can be easily analysed from the graph of Force v/s Time.

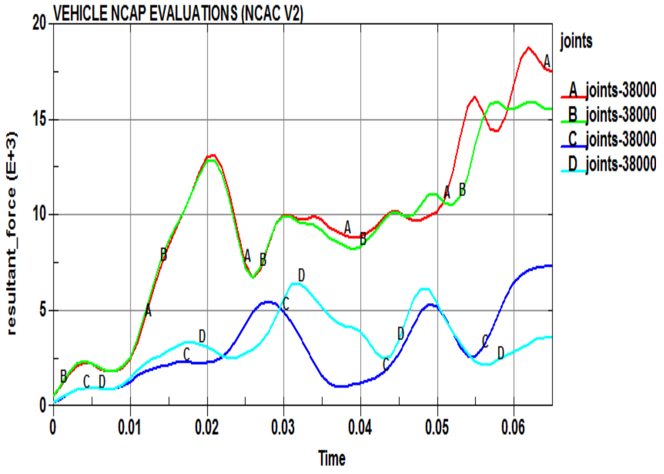

Chart- 6.2: Force v/s Time plot at Bolt connections for Case I & Case III.

The value of Force at point A is 18.75 KN and at point B it is 16 KN as seen from the graph. Similarly the magnitude of Force at point A' is 7.5 KN and at point B' is 6.25 KN after the modification. It is found that there is a reduction of 60% in the impact force at bolt locations A & A' and 60.93% at the bolt locations B & B'.

6.3 Plot for Internal Energy in the Fuel Tank

The internal energy recorded in the fuel tank for the baseline and modified model is shown in the Chart 6.3 below.

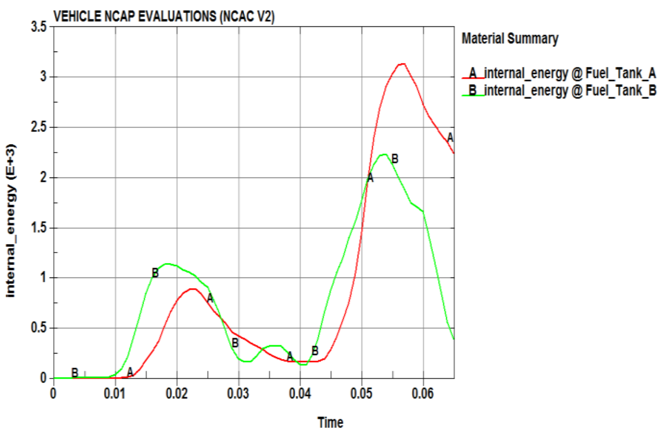

Chart-6.3: Internal Energy v/s Time graph for the Case I and Case III.

The maximum internal energy recorded during case I is 3.2 KJ and that in case III is 2.25 KJ respectively. A decrease of 29.68% is observed in the internal energy of the fuel tank.

6.4 Energy Balance Plots

The global energy plots were obtained from the simulation for the baseline and the modified model. Energy balance is seen throughout the simulation. The kinetic energy decreases and the internal energy increases as the car strikes the rigid wall. The Total energy remains constant for case I and case III. The global energy plot for the baseline model is as shown in Chart 6.4 below.

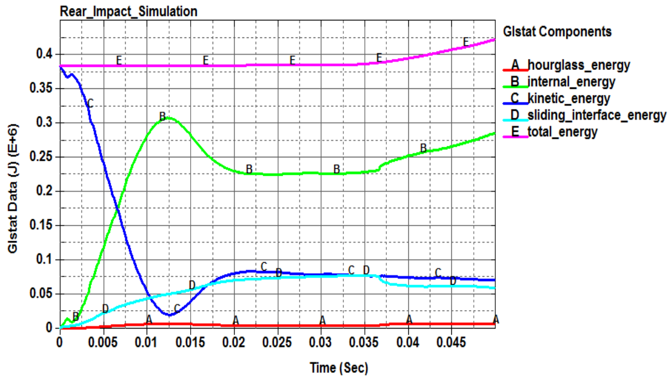

Chart-6.4: Simulation Energy Balance Analysis for Case I.

The global energy plot for the modified model is as shown in the Chart 6.5 below.

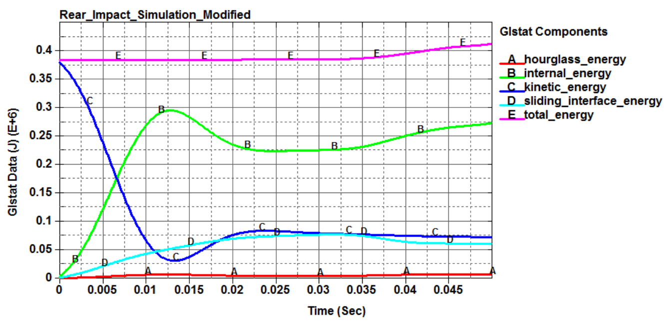

Chart-6.5: Simulation Energy Balance Analysis for Case III.

A total energy of 380 KJ is observerd in both the cases which shows that the energy balance is similar in the baseline and the modifed car model.

7. VALIDATION

The validation is done by comparing the acceleration v/s time graph for the baseline model and modfied model to that obtained after experimental testing of the actual Toyota Yaris car. In actual rear crash testing an Accelerometer is place near the fuel tank before the crash and during the crash the magnitude of acceleration is recorded near the fuel tank. Acceleration value is always expressed as 'g' values which gives the measure of the severity of the impact or damage. The accelerometer card used for simulation in LS-DYNA is *ELEMENT_SEATBELT_ACCELEROMETER keyword card which is specified near the fuel tank which gives same results as that of an actual accelerometer. A Tri-axial Accelerometer is used in automobiles during crash testing. The acceleration output is as shown in the Chart 7.1 below.

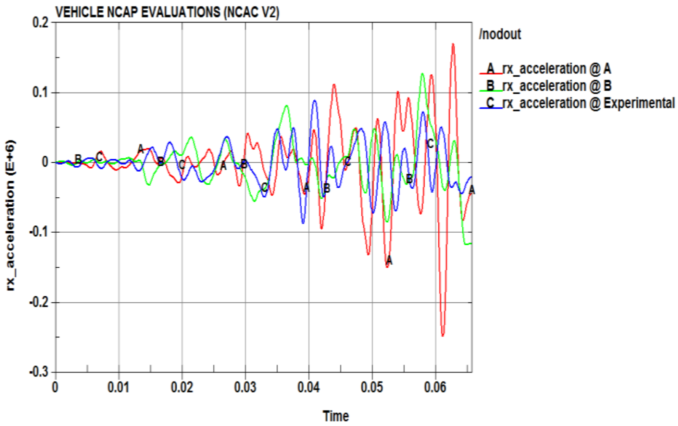

Chart-7.1: Comparison of Acceleration (mm/s^2) v/s Time (seconds) for Case I, Case III and the Experimental Crash Test for the time frame of 0.065 sec.

The red line (A) indicates the acceleration at the fuel tank for the case I with maximum acceleration of 250g. The green line (B) represents the acceleration at the fuel tank for the case III with maximum acceleration of 120g and the blue line shows the behavior of acceleration during the experimental rear crash test has maximum value of acceleration of 95g. Since the acceleration is reduced from 250g to 120g at the fuel tank the design modification is successful and the acceleration value of 120g in case III is near to the average experimental value of 95g. The Toyota Yaris car model passes the rear crash test.

8. CONCLUSION

The crashworthiness of the fuel tank has been increased in the event of the rear crash of the Toyota Yaris by suitable design modifications. The fuel tank is prevented from the damage for the speed of 80.5 Km/hr which is one of the severe case in the rear crash. The fuel tank with High Density Polyethylene material will also get deformed to some extent during the rear crash but it will not get burst and cause fuel spillage because the deformation of the fuel tank is within the permissible limits i.e. the strain percentage of High Density Polyethylene is 13% and the maximum effective plastic strain percentage is 7.65% which is safe. The main aim of the analysis is to save the occupants as well as the car from fire hazard by increasing the fuel tanks crashworthiness.

The impact force on the rear chassis is reduced by 25% after the design modification. A reduction of the impact force at the bolt locations by 60% at A & A' and by 60.93% at bolt locations B & B' is found. Internal energy absorbed by the fuel tank is reduced by 29.68%. Acceleration near the fuel tank is reduced from 250g to 120g which indicated that the fuel tank remains safe after the rear crash. Also there is energy balance throughout the simulation. So finally it is concluded that the car model of Toyota Yaris passes the rear crash test after the proper design modifications.

REFERENCES

[1] Dhafer Marzougui et al, July 2012, "Extended Validation of the Finite Element Model for the 2010 Toyota Yaris Passenger Sedan" , The George Washington University.

[2] D. Brown et al, " Development & Validation of a Finite Element Model for a Mid-Sized Passenger Sedan" , Centre for Collision Saftety and Analysis, George Mason University.

[3] Paul T. Eichbrecht, 2013, "Crash Test Protocol Development for Fuel System Integrity Assessment", Department of Transportation, NHTSA-98-3588-72

[4] Department of Transportation, National Highway Traffic Safety Administration, "Laboratory Test Procedure for FMVSS 301R, Fuel System Integrity – Rear Impact," TP-301R-02, January 17, 2007

[5] Department of Road Transport & Highways, Government of India, "Requirement for the Protection of Fuel System in the Event of Rear Impact of a Motor Vehicle ", Automotive Industry Standard (AIS-101).