International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 07 Issue: 10 | Oct 2020 www.irjet.net p-ISSN: 2395-0072

Crash Analysis of Truck Cabin Structure at a speed of 60KMPH

Rajat Bhagwat Eklare[1], Abhijeet Shah[2], Shamal Santosh Gowale[3]

[1]Department of Mechanical Engineering, Rajarambapu Institute of Technology, Shivaji University, Sakharale, Sangli, 415414, India
[2]Associate Professor, Mechanical Engineering Dept., Rajarambapu Institute of Technology, Sakhrale
[3]Department of Mechanical Engineering, Rajarambapu Institute of Technology, Shivaji University, Sakharale, Sangli, 415414, India

---

Abstract - *As the research and development of new design technologies in automobile sector, vehicle or a automobile safety has taken a drastic change in these days. Due to this development in automobile sector, the industries are giving their continuous effort to improve the automobile safety and to reduce count of deaths in an accident. In the present work one such effort is shown by carrying out crash analysis of a truck structure at a speed of 60kmph using ANSYS Explicit dynamic approach. A rigid wall was used as an obstacle and aluminium alloy has been chosen as a body material for truck cabin structure model. By carrying out the analysis at speed different then 60kmph, it was observed that for velocity less than 60kmph I.e. for low velocity most of the impact forces were absorbed by front part of truck cabin structure model with small deformation. But at velocity greater than 60kmph I.e. at high velocity impact results in permanent deformation of the truck structure. The extent of plastic deformation of the truck structure increases with increase in speed, with most of the impact load taken by the truck cabin structure, due to which most of the fatal injuries takes place. Also from different graphs it can be clearly seen that internal energy increases and kinetic energy decrease drastically during the course if impact.*

Key Words: **Crashworthiness, Impact**

1. INTRODUCTION

It is being important to analyse the "Crashworthiness of an automobile structure". Crashworthiness is defined as ability of vehicle structure to sustain impact loading and to prevent the fatal injuries at the time of accident or it may also be defined as capability of vehicle structure to absorb energy during accident. There may be two type of impacts during accidents front impact or head-on collision and second is side impact. The design of vehicle should be such that its occupants should not experience net deceleration during impact at high speed. The analysis of Crashworthiness can be done using computer tools or my experimental methods.

1.1 Crash Test

Crash test is a form of destructive testing usually performed to ensure safe design standards in terms of crashworthiness and crash compatibility for vehicle with related components. The procedure conducted by destructive testing for determining the safety performance of the vehicle is called as Crash test. The most common type of crash test are front impact test, front offset crash test, side impact test and roll over test etc.

1.2 Basic Definitions

Stress : The intensity of internally distributed forces that tends to resist change in shape of the body during impact is called as stress

Strain : Change in length per unit original length in linear direction of the body during impact loading is known as strain.

Elastic Range : The greatest stress up to which the material exhibits the characteristics of regaining its original shape and dimensions on removal of impact load is known as elastic range of the material.

Hooke's law : This states that when a material is loaded with in it's elastic limit, the stress is directly proportional to strain.

Deformation : The transformation of a structure from reference shape to current shape during impact is known as deformation. Deformation may be of different types directional deformation, total deformation, plastic deformation and elastic deformation.

2. METHODOLOGY

In order to analyze crashworthiness of a truck structure model ANSYS Explicit dynamic tool and LS-Dyna is used. This complex simulation methodology involves :-

i. Creation of Geometry (Truck structure model and wall as obstacle) using CATIA V5 modelling tool.

ii. Meshing the truck structure model.

iii. Setting up the boundary conditions.

iv. Solving the dynamic response of truck structure model with respect to time at speed of 60kmph.

v. Examine the results obtained using ANSYS and LS-Dyna post processing tool.

3. COMPUTATIONAL ANALYSIS OF TRUCK STRUCTURE CRASH

3.1 Geometry Creation

As the main objective of the present work was to simulate the truck structure crash, to know the material behaviour for a speed of 60kmph and at different intervals of time, simulation of wheels was neglected so that the complexity of simulation may be reduced. Scaled down commercially available truck structure model was chosen for analysis having Length = 5.125m, Width = 1.5m and Height =1.970m. The wall was modeled with a dimension having Length =2m, Width =2m and Thickness =0.5m. The initial distance between the truck structure model and wall was made to be 0.3m i.e. at t = o sec.

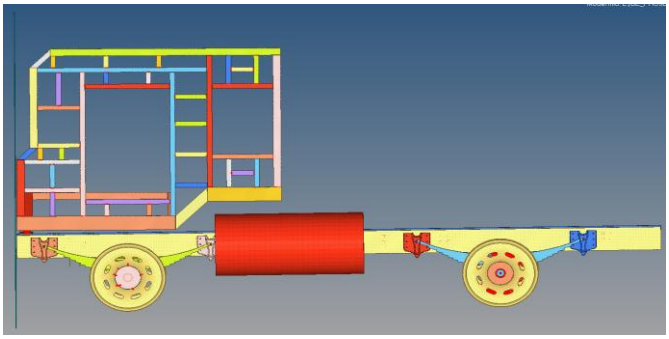

Chart -1: Model of Truck Structure (Side View)

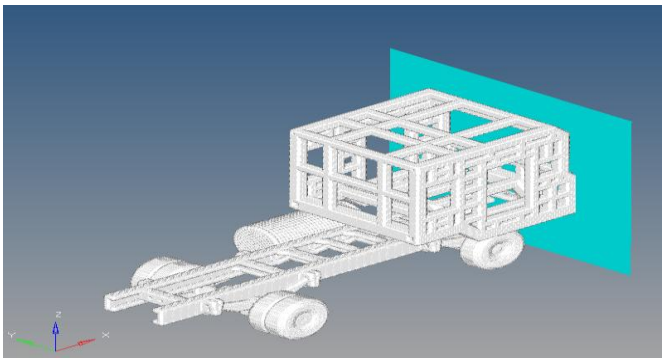

Chart -2: Model of Truck Structure (Isometric View)

3.2 Meshing

Meshing process was carried out by using meshing tool ANSYS workbench. Auto mesh was generated, with element as tetragonal in shape for truck structure model and elements of wall being hexagonal in shape, with high smoothing and slow transition. The total number of elements and node were found to be as 6606 elements and 11060 nodes.

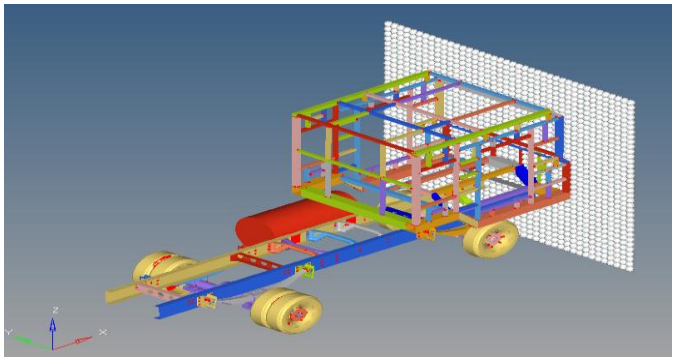

Chart -3: Meshed model of Truck Structure

4. RESULT AND DISSCUSSION

Simulation for scaled down truck structure model was carried out by using ANSYS Explicit Dynamic Approach and LS-Dyna. The initial distance I.e. at t = 0, between the wall and the truck structure was kept to 0.3m. After the simulation for crash test of truck structure model at speed of 60kmph following results were obtained.

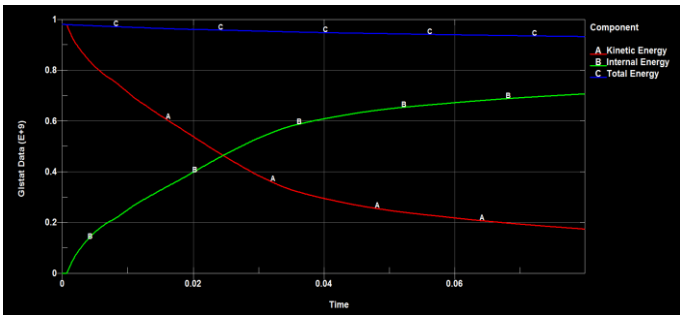

Chart -4: Energy Curve

Energy curve shows the variation of kinetic and internal energy with respect to collision time. Fig. 4 shows the energy curve we can clearly see that after the impact the kinetic energy of truck structure model decreases suddenly (because after impact the velocity of the vehicle decreases suddenly) and internal energy of vehicle increases suddenly after the impact.

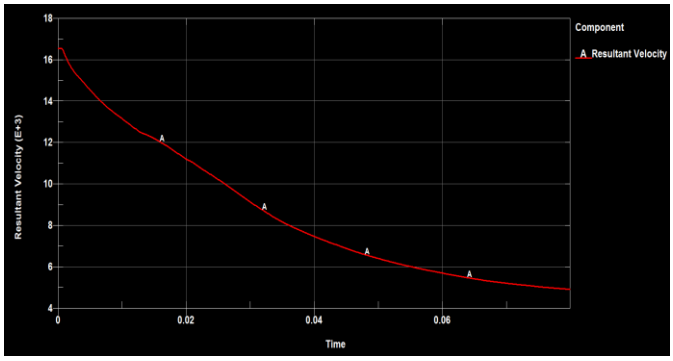

Chart -5: Velocity Curve

From the stress contour of the crash test, it can be concluded that during impact maximum load is absorbed by cabin part of the truck structure due to which fatal injuries occurs, results in loss of life of occupant. So due to this reason crashworthiness of the vehicle should be analyzed.

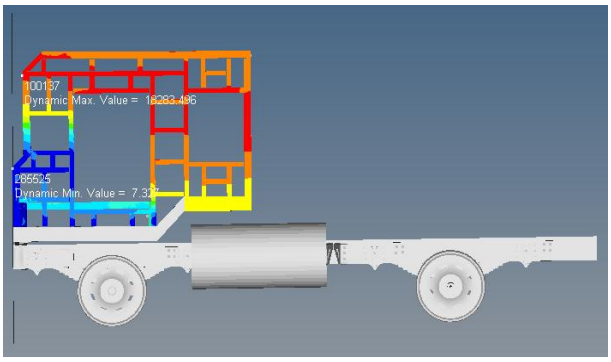

Chart -5: Stress Contour

5. CONCLUSION

Crashworthiness of the scaled down model of truck structure was tested using the computational approach using ANSYS Explicit dynamic tool and LS-Dyna in this work. It can be clearly seen that extent of plastic deformation of the truck structure model increases with increase in speed of the truck, with the front part of the truck absorbs the major part of the impact energy. Wind shields of the cabin structure was the major part to undergo plastic deformation. Also from the graphs it can be clearly seen that internal energy increases and kinetic energy decreases drastically during the course of impact. After the impact the truck structure body rebounds back and regains its kinetic energy, while the internal energy goes on decreasing.

ACKNOWLEDGEMENT

The previous version of this paper was presented by P. Veeraraju at the 40st Australasian Transport Research Forum (ATRF 2018). Thanks for all the members for their direct and indirect support in the survey.

REFERENCES

[1] T. Ambati, Srinath, P. Veeraraju "Simulation of vehicular frontal crash test" International journal of applied research in Mechanical Engineering vol-2 (2012).

[2] Byeong S.K, K. Park, Youn-Min Song "Finite element frontal crash analysis of new vehicles platform with upper and sub frame body" journal of dept. of automotive engineering hoseo university (2013).

[3] Erika Fechova, Jozef Kmec, Alena Vagaska Drazan Kozak "Material properties and safety of car at crash test",journal of international conference on manufacturing engineering and materials, science direct (2016).

[4] Saeed Barbat, Xiaowei Li, Priya Prasad "Vehicle to vehicle front to side crash analysis using a CAE based methodology" Journal of passive safety Research and Advanced Engineering (2004).

[5] Touraj Gholami, Jürgen Lescheticky, Ralf Pabmann "Crashworthiness simulation of automobile with ABAQUS/Explicit", Journal of Abaqus user conference BMW Group(2003).

[6] Wlodzimierz Bedkowski "Assessment of the fatigue life of machine components under service load –A review of selected problems" Journal of theoretical and applied mechanics,(2014).

[7] Jadav Chetan, Panchal Khushbu, Moulvi Nauman "The fatigue analysis of a vehicle suspension system –A review article" International journal of advanced computer research (2012).

[8] Hariharan Sankarasubramanian Sudipto Mukherjee, "Optimization of vehicle front for safety of pedestrians" Indian institute of technology,Delhi, India, 2014.

[9] C. Sadhasivam & S. Jayalakshmi, "Simulation of car body by development of static and dynamic analysis" Department of mechanical engineering, pet engineering college, Tirunelveli, Tamilnadu,India 2011.

[10] Mandeep Kumar Vandanapu1, Ratnakar Pandu, "Design optimization of passenger car front bumper" international journal of science and research (IJSR) 2015.