Crashworthiness Analysis of Finite Element Truck Chassis Model Using LS-DYNA®

Yucheng Liu
Department of Mechanical Engineering
University of Louisiana
Lafayette, LA 70504, USA

Abstract

This paper presents a detailed multi-purpose finite element model of a light duty truck chassis and evaluates this model in computational simulations of full frontal, offset frontal, and corner impacts. The simulation results are analyzed which correctly describe the characteristics and performance of a truck chassis during under impact scenarios. Through the validation and computational simulations, the presented model is proved to be computationally stable, reliable, repeatable, and useful for vehicle crashworthiness analysis. LS-DYNA is used for finite element modeling and crashworthiness analysis

1. Introduction

Crashworthiness is the ability of a vehicle to withstand a collision and to prevent the occupants from injuries in the event of a vehicular accident. It is one of the most important criteria used in designing and evaluating a vehicle or a vehicle component. In modern automotive industry, vehicle manufacturers vastly make use of computer modeling and simulation to test crashworthiness and safety features in new designs. By building a finite element model of a vehicle/component and run simulation on the computer, they can save lots of time, effort, and cost that would otherwise be required to build a unique prototype and test it. Important crashworthiness characteristics and safety features of their designs can be estimated and reflected from the computer analysis results.

The paper presents a finite element model of a light duty truck chassis (figure 1) and uses it for the crashworthiness analysis. This paper briefly addresses the details of this truck chassis model and carefully discusses crash responses of the model during full frontal, offset frontal, and corner impacts. Suggestion for further improvement in developing finite element truck chassis models is also included. Dynamic explicit code LS-DYNA is used for running the crashworthiness analysis

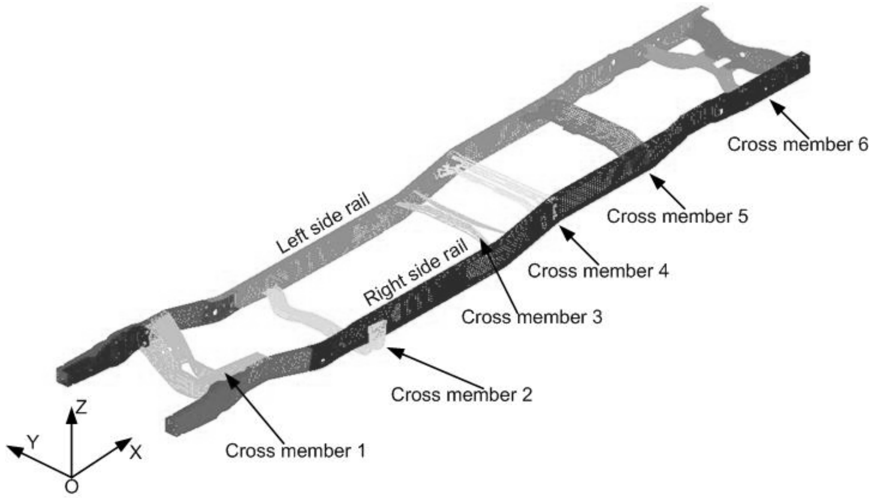


Figure 1. The finite element model of light duty truck chassis

2. Background

Finite element models of vehicles and vehicle components have been increasingly applied in preliminary design analysis, vehicle crashworthiness evaluation, and component design. Due to its importance in vehicle design and crash simulation, several investigators did study on design and analysis of vehicle chassis models. Cosme et al. [1] performed specific case studies in the design and analysis of heavy-duty frames. In their work, a detailed heavy truck model was analyzed and all the changes to the truck chassis in the event of impact were simulated and studied thoroughly. Karaoglu and Kuralay [2] performed stress analysis of a truck chassis by using finite element method. In their research, an improved truck chassis model with reduced weight and stresses was achieved by increasing its side rail thickness locally.

All the modeling and simulation involved in this paper were performed using LS-DYNA [3]. LS-DYNA is an explicit code very capable of solving high-speed impact problems that requires small time steps, which is commonly used by researchers in vehicle modeling, analysis, and crashworthiness evaluation [4].

3. Model Description

3.1 Finite element chassis model

Figure 1 displays the finite element model of truck chassis, from which it can be found that the entire chassis model is composed of two side rails and six cross members. The chassis model has a total length of 6.2 meters and a width of 0.95 meter. The presented finite element model is meshed using full integration shell element: 4-node Belytschko-Tsay shell element with five integration points through the thickness. Overall 55529 shell elements and 56741 nodes exist in this model. The truck chassis is made of mild steel with material properties listed in table 1. In LS-DYNA, this material is modeled with material type 13, the isotropic elastic plastic material model with failure mode.

Table 1. Isotropic elastic-plastic material model

Material Properties	
Young's modulus	2.07E5Pa
Density	7830kg/m^3
Yield Stress	250MPa
Ultimate Stress	448MPa
Hardening modulus	630MPa
Poisson's ratio	0.3

Parts are connected using constrained nodes. Two types of nodal constraint, nodal rigid body constraint and spot weld, are used. The impact contacts between the different components of the chassis are simulated using automatic single surface contact algorithm, which is also provided in LS-DYNA. This finite element chassis model will be used for computational simulations of different impact tests.

3.2 Rigid wall

The rigid wall is defined by a rigid wall card (*RIGIDWALL_PLANAR). Rigid shell elements are used to model the wall, but which is only for visualization purposes. For the full and offset frontal impacts with a rigid wall, sliding interface is used to model the contact between the chassis and the wall. For the corner impact, the geometric contact entity option in LS-DYNA is employed, which provides high accuracy and computational efficiency for metal forming problems.

4. Crash Simulations

In this project, the truck chassis model is used for three types of crash simulation: full frontal impact, offset frontal impact, and corner impact. During these simulations, the chassis model impacts the rigid wall at an initial speed 15m/s (≈ 33.8mph) and the simulation time is set as 0.1sec (100ms) for the frontal impacts and 0.24 sec (240ms) for the corner impact. After computer analyses, accelerations are taken at the center of gravity of this model to estimate the responses of truck chassis during different impact cases. Energy balance analyses are also performed in order to investigate the crash energy dissipation among different chassis components during crashes. The performances of truck chassis involved in impacts will then be discussed. The overall weight of this chassis is 236 kg and its center of gravity location is measured at X = 3638.74 mm, Y = -2.54 mm, and Z = 1674.23 mm, which are in a global coordinate system. The weight of each chassis component is listed in table 2. It should be noted that each of these components is composed of several parts.

Table 2. Weight of chassis components

Component	Weight (kg)
Left side rail	89.2
Right side rail	89.2
Cross member 1	23.7
Cross member 2	8.2
Cross member 3	2.8
Cross member 4	4.2
Cross member 5	8.7
Cross member 6	10

4.1 Full frontal impact

In the full frontal impact simulations, the chassis model impacts the stationary rigid wall at 15m/s (≈ 33.8mph). Fig. 2 shows the crashed configurations of the model at 32, 64, and 100ms. From that figure, it can be seen that during the low-velocity full frontal impact, the front part of side rails seriously buckled and no obvious plastic deformation was observed in the remaining side rails. As for the cross members, cross member 1 experienced a small bending due to the transferred impact force. Cross members 2, 5, and 6 almost kept their original shapes through the crash simulation. Only cross members 3 and 4 suffered large deformation because of their weak structures. Fig. 2 verifies that during the simulation, most crash energy was absorbed by the front part of chassis model (high impact zone) and only a few impact forces were transferred to the rear part of this model.

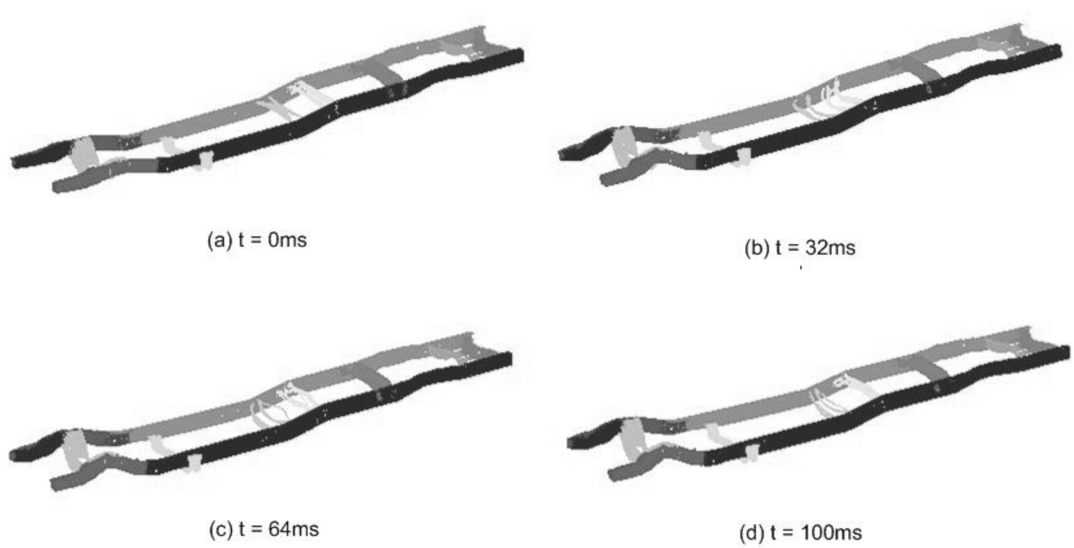
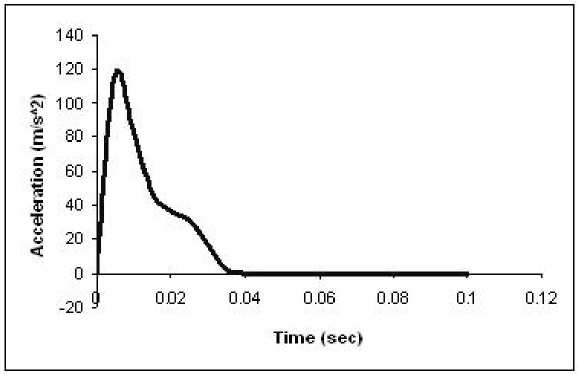
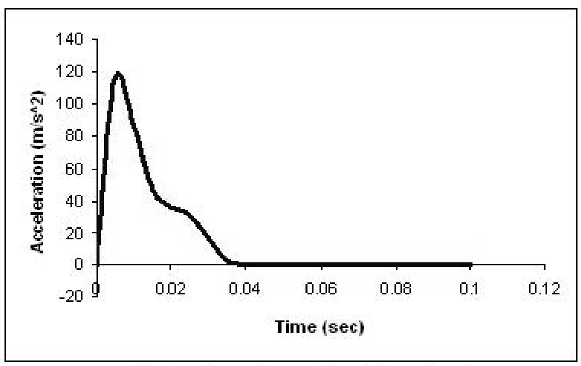


Figure 2. The gross motion of the model in a full frontal impact simulation

The acceleration history was measured at center of gravity of this model. Fig. 3 displays and compares its X-acceleration and resultant acceleration. From that figure, it is observed that during the full frontal impact, the model suffered an immerse impulse instantly after the impact and then this impulse dramatically dropped down and faded away at about 40 ms. After that time, the rear part of this model still moved forward because of the inertia but there was no distinct impact force transferred through the model. Also, by comparing the measured X-acceleration and resultant acceleration, it can be found that in the full frontal impact, the impact force and acceleration were mainly generated along the impact direction, X-direction. The model's responses along other directions were inconspicuous and can be neglected in numerical analysis.



(a) Center of gravity - X



(b) Center of gravity - Resultant

Figure 3. Acceleration at center of gravity for the full frontal impact

It is important to analyze the energy absorption by the different components in the chassis. This can be obtained in the simulation by using the *database_matsum data entry in LS-DYNA. Table 1 lists the percent of crash energy dissipated through different components for the full frontal impact simulation.

The internal energy absorbed by different components is displayed in table 1. From the table it can be found that about 90% of energy was absorbed by left and right side rails. It verifies that the side rails, which are basically longitudinal thin-walled beams, have excellent energy absorption capability and are used in the chassis model as major energy-absorbing components. Meanwhile, as mentioned before, another portion of energy was absorbed by the cross member 1 along with the bending deformation. From table 1 it can be calculated that these three components absorbed 97.27% of total internal energy and only less than 3% of the energy was dissipated into the remaining components. The energy absorption analysis results fully coincide with the observations come from Fig. 2.

Table 3. Internal energy for the full frontal impact

Component	Internal Energy (kJ)	Percentage
Whole chassis	26.34	100%
Left side rail	11.87	45.06%
Right side rail	11.42	43.36%
Cross member 1	2.33	8.85%
Cross member 2	0.07	0.27%
Cross member 3	0.16	0.98%
Cross member 4	0.11	0.42%
Cross member 5	0.14	0.53%
Cross member 6	0.14	0.53%

4.2 Offset frontal impact

In the offset frontal impact, 50% of the chassis frontal width impacts the wall at 15m/s (≈ 33.8mph), as illustrated by Fig. 4. From that figure, it can be seen that in this simulation, the right part of model impacted the rigid wall and the left part continued to move forward. The fact that the right part "stagnated" at the rigid wall and the other part kept on moving caused the model seriously twisted (Fig. 4(d)). Also, the front part of right side rail significantly buckled during this impact and most crash energy should be absorbed by this side rail. The deformation of the cross members are comparatively small except for the cross member 3 and 4, which is similar to the full frontal impact case.

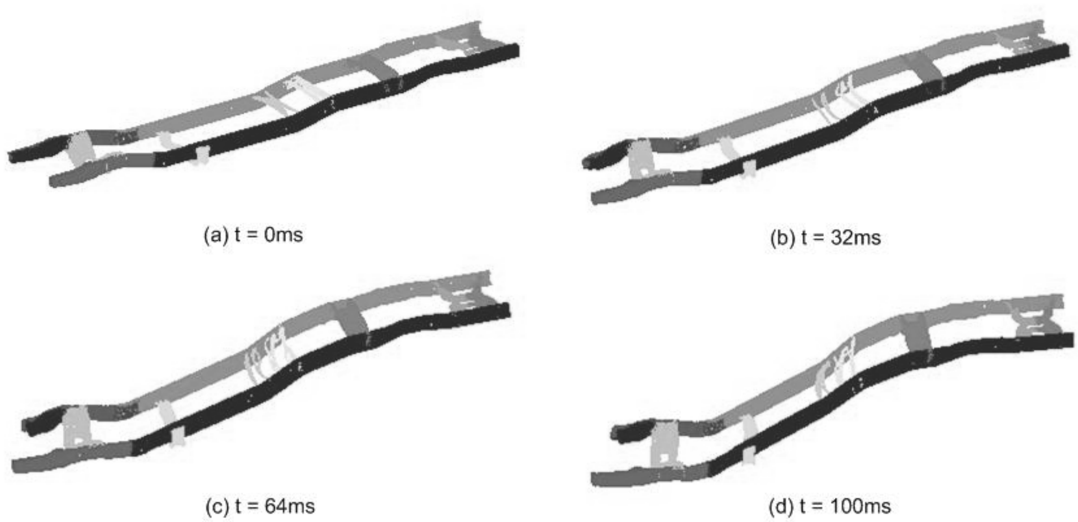


Figure 4. The gross motion of the model in an offset frontal impact simulation

Fig. 5 compares the resultant acceleration and accelerations along X, Y, and Z direction, which were measured at the model's center of gravity. In this simulation, different from the full frontal impact, detectable accelerations were generated on Y and Z directions. Even they are much smaller than the X-acceleration, they did contribute to the overall response of the model during the simulation. During this simulation, besides the impact force along X direction, the twisting of the chassis model also caused internal forces along other directions, which generated the accelerations. Another observation is that after dropping down from the peak values, the X-acceleration and resultant acceleration did not approach to zero as shown in Fig. 3, but oscillated around a certain level. This is because that in the offset frontal impact, the right part of the model continued to move forward after the left part impacted the wall, and then it can be considered that the overall chassis model kept on impacting the rigid wall during the entire simulation.

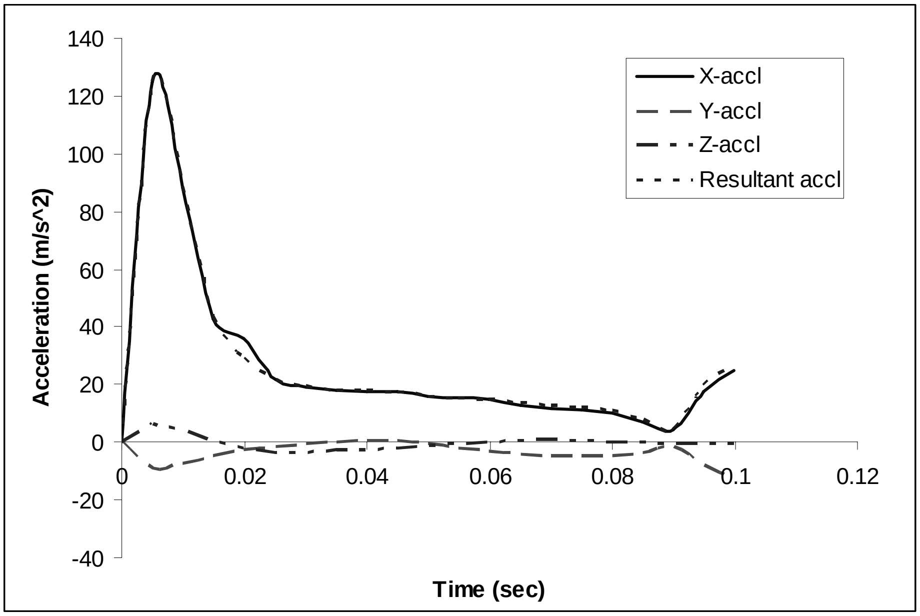


Figure 5. Acceleration at center of gravity for the offset frontal impact

The internal energy absorbed during the simulation is also analyzed. From table 2, it is found that the total internal energy is 24.36 kJ. The difference of 2.19 kJ still remained in the chassis model in the form of kinetic energy. As concluded from Fig. 4, in the offset impact case, most of the internal energy was absorbed by the right side rail, which seriously buckled at its high impact zone. The left side rail, as a major energy absorber, still absorbed 14.20% of the impact energy. Cross member 1 and 2 each absorbed a certain amount of energy, which is because both cross members were very close to the impact zone due to the moving forward of the left part of model. As displayed in table 2, most of the internal energy was absorbed in the high impact zone: 97.16% of the total energy was absorbed by the left and right side rails and the cross member 1 and 2.

Table 4. Internal energy for the offset frontal impact

Component	Internal Energy (kJ)	Percentage
Whole chassis	24.36	100%
Left side rail	3.46	14.20%
Right side rail	17.56	72.09%
Cross member 1	1.37	5.62%
Cross member 2	1.28	5.25%
Cross member 3	0.08	0.33%
Cross member 4	0.04	0.16%
Cross member 5	0.5	2.05%
Cross member 6	0.07	0.3%

4.3 Corner impact

In this simulation, the chassis model impacts frontally on the rigid wall at a 30º angle, as illustrated in Fig. 6. The initial velocity components for the chassis model are 13m/s (≈ 29.3mph) parallel to the wall and 7.5m/s (≈ 16.9mph) normal to it. The gross motion displayed in Fig. 6 agrees well with the motion observed from the earlier stage of corner impact simulation and test of a truck model [1]. From that figure, it can be found that during the 30º corner impact, most of the plastic deformation appeared on the front part of left side rail and the cross member 1 is the only cross member that directly impacted the wall. If the chassis model is assembled with suspension systems which have wheels and higher controllability, it will show the same motion of an entire vehicle assembly during the corner impact simulation.

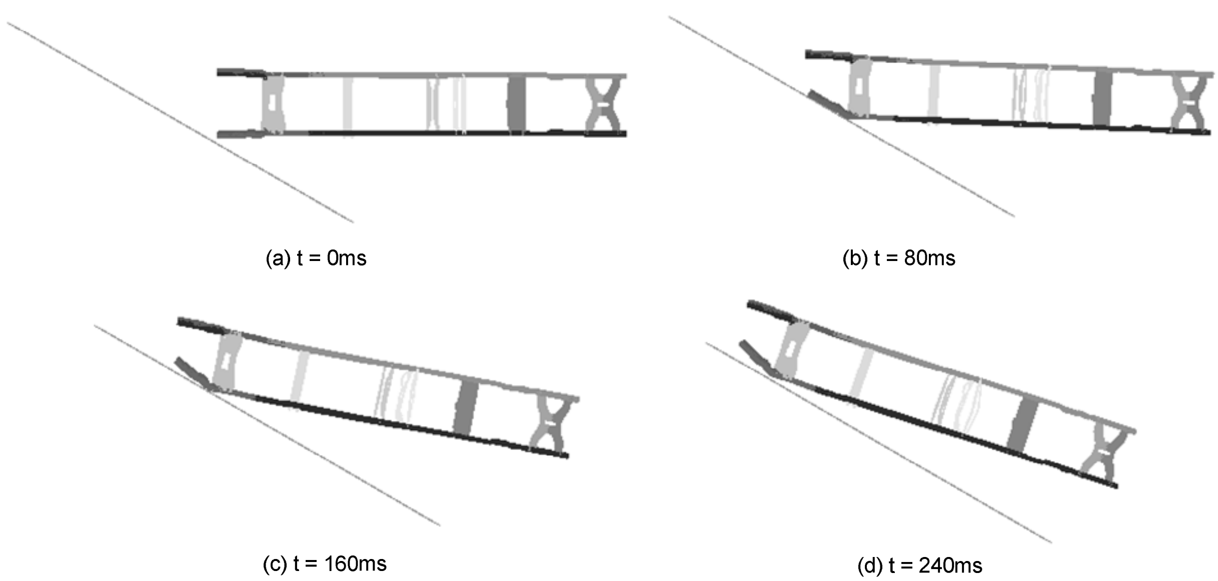


Figure 6. The gross motion of the model in a corner impact simulation

Acceleration response of this chassis model is displayed in Fig.7, from which it can be found that in the corner impact, the overall impulse is much lower than the frontal impacts. The model suffered accelerations along X and Y directions while the Z-acceleration is too small and can be neglected. After 150ms, the accelerations approached to zero because at that time, the model already adjusted its motion direction and began to "slide" along the wall.

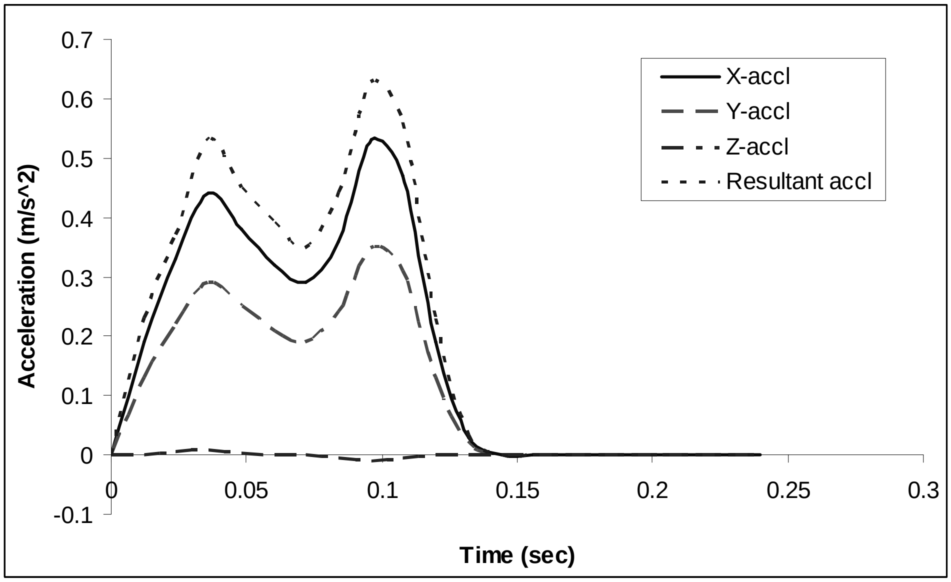


Figure 7. Acceleration at center of gravity for the corner impact

Table 3 shows the percent of total internal energy mitigated through the different components. Only 10.42 kJ kinetic energy was transferred to the internal energy and stored in the model. About 96% energy was absorbed by the left side rail and cross member 1, which highly impacted the wall during the simulation. The energy absorption and distribution seems to be consistent with the gross motion illustrated in Fig. 6.

Table 5. Internal energy for the corner impact

Component	Internal Energy (kJ)	Percentage
Whole chassis	10.42	100%
Left side rail	7.03	67.44%
Right side rail	0.21	2.05%
Cross member 1	3.01	28.89%
Cross member 2	0.06	0.57%
Cross member 3	0.01	0.09%
Cross member 4	0.00	0.00%
Cross member 5	0.10	0.96%
Cross member 6	0.00	0.00%

5. Conclusions

This paper presents a detailed truck chassis model and uses this model for crash simulations. The simulation results can correctly represent the model's responses during real impact tests. Through this project, the chassis model's performances during full frontal, offset, and corner impacts are investigated. Characteristics of different impact cases are revealed and compared through acceleration and internal energy analyses. From the paper, it is concluded that LS-DYNA is a very powerful finite element software package that can be efficiently used for computer modeling and crashworthiness analysis.

References

1. Cosme, C., Ghasemi, A., Gandevia, J., "Application of computer aided engineering in the design of heavy-duty truck frames", *SAE Transactions*, 1999-01-3760, 1999
2. Karaoglu, C., Kuralay, N.S., "Stress analysis of a truck chassis with riveted joints", *Finite Elements in Analysis and Design*, 38, 2002, 1115-1130
3. Livermore Software Technology Corporation, "LS-DYNA® Keyword User's Manual", Livermore, CA, 1999
4. Mei, L, Thole, C.A., "Data analysis for parallel car-crash simulation results and model optimization", *Simulation Modelling Practice and Theory*, 16, 2008, 329-337