Mishra et al. 2016. *Int. J. Vehicle Structures & Systems*, 8(4), 250-254
ISSN: 0975-3060 (Print), 0975-3540 (Online)
doi: 10.4273/ijvss.8.4.12

International Journal of Vehicle Structures & Systems
Available online at www.maftree.org/eja

Crashworthiness of a Truck Cabin using Finite Element Simulation

Neeraj Kumar Mishra[a] and Sharad K. Pradhan[b]

Dept. of Mech. Engg., National Institute of Technical Teachers Training & Research Institute, Bhopal, India
[a]Corresponding Author, Email: neeraj9391@gmail.com
[b]Email: spradhan@nitttrbpl.ac.in

ABSTRACT:

A crash test is a form of destructive testing in order to ensure safe design in case of an impact or collision. These impact tests are carried out to understand a components structural behaviour and its response under different collision conditions. But it is not always economical to destroy a specimen to ensure its safety, especially in case of small scale production. Moreover actual crash testing in different conditions requires high resources and time. Crash simulation is very popular now a days because it is a virtual representation of a destructive crash test of a vehicle using a computer simulation in order to examine the level of safety of the vehicle and its occupants in different conditions like collision velocity, collision direction, colliding object, material and number of component of colliding object etc and hence saves the time and cost. Data obtained from a crash simulation indicate the capability of the vehicle body to protect the vehicle occupants during a collision against injury. In this study a FE crash analysis of commercial truck cabin in two different cases (Bare cabin and loaded cabin) using Finite Element Approach (FEA) is focused. Crash analysis of truck cab is performed using FEA in order to predict the design parameters for minimization harm to the occupants of the vehicle. A bare truck cabin and a rigid wall were used to simulate crash conditions with different crashing velocity and impact angles. CATIA V5R20 CAD software is used for the modeling of the selected cabin components followed by FE meshing through Hypermesh and then analysis is done using LS-DYNA by setting the boundary conditions, material properties etc. appropriately.

KEYWORDS:

Crashworthiness; Finite element analysis; LS-Dyna; Hypermesh; Truck cabin; Crash Simulation

CITATION:

N.K. Mishra and S.K. Pradhan. 2016. Crashworthiness of a truck cabin using finite element simulation, *Int. J. Vehicle Structures & Systems*, 8(4), 250-254. doi:10.4273/ijvss.8.4.12.

1. Introduction

A crash test is a form of destructive testing in order to ensure safe design in case of an impact or collision. These impact tests are carried out to understand a components structural behaviour and its response under collision condition. But it is not always economical to destroy a specimen to ensure its safety, especially in case of small scale production. This gave rise to the concept of crash simulation which is a non-destructive method to witness the effects of a crash situation. This is performed with the help of well-defined computerized software, which make the simulation process quick and inexpensive [1]. Crash simulation is very popular now a days because it is a virtual representation of a destructive crash test of a vehicle using a computer simulation in order to examine the level of safety of the vehicle and its occupants and hence saves the time and cost. Data obtained from a crash simulation indicate the capability of the vehicle body to protect the vehicle occupants during a collision against injury.

This simulation technology have greatly increased the protection, dependability and producing potency in today's vehicles. Numerical methods are now extensively applied in engineering due to the advances in computing. of all the numerical methods, the finite element method is the most popular and convenient approach, because it is easy to implement for all kinds of boundary and loading conditions and it can be used for the analysis of large complex structures. Experimental measurement is considered as a powerful and accurate data acquisition approach and is generally used to approve most of the mathematical models. The only difficulty with experimental data is its high cost and validity for a particular situation only. In this study a FE crash analysis of commercial truck cabin in two different cases i.e. bare and loaded using Finite Element Approach (FEA) is focused and comparison is made to encounter the major difference.

Crash analysis of truck cab is performed using FEA in order to predict the design parameters for minimizing harm to the occupants of the vehicle. In both the cases truck cabin and a rigid wall were used to simulate crash condition with different crashing velocity and impact angles. CATIA V5R20 CAD software is used for the modeling of the selected cabin components followed by FE meshing through Hypermesh and then analysis is done using LS-DYNA by setting the boundary conditions, material properties etc. appropriately. Krishnaswami et al [2] presented the results of a study to examine the feasibility and benefits of improving truck occupant injury outcomes through the use of appropriate protection systems. Samavedam et al [3] presented the test results and finite element correlations of a full-scale dynamic collision between a locomotive and a highway truck loaded with two heavy steel coils. Parnell et al [4] shown that, despite the relatively long time span of the event, it is possible to apply the techniques of vehicle crashworthiness simulations to heavy truck rollover.

Mirzaamiri et al [5] investigated the behaviour of Iran Khodro (IKCO) 2624 truck subjected to a complex crash test according to regulation ECE-R29. Philip et al [6] presented simulations that are performed to verify various safety aspects to ensure crashworthiness of the truck cabin. Wang et al [7] built a finite element model of a high-top cabin of a heavy truck with a manikin on the driver seat with commercial code Hypermesh. Ambati et al [8] presented the simulated crash test of an automobile. The objective of this work was to simulate a frontal impact crash of an automobile and validate the results. Guosheng et al [9] took the type of 6900 passenger car as an object reference, referred protection of the occupants. Consolazio et al [10] described the process by which numeric simulation was recently used to carry out conceptual development of a new temporary concrete work zone barrier. Raine et al [11] described the second part of the Swedish cab safety test, where the cylinder hits the A-pillar on the driver side.

Raich et al [12] presented the application of the numerical method using example of new of ACTROS mega space cab. Bonin et al [13] presented the characteristics of the HGV FE model, developed and validated using LS-DYNA FE code. Krusper et al [14] made a comparison of structures in heavy goods vehicles and passenger cars along with in-depth accident investigations using specially made 3D geometric models of the vehicles in position just before a collision. Lonn et al [15] used a meta-model based Monte Carlo method to evaluate the robustness of a vehicle structure. Chen et al [16] built a reliable finite element model of a light truck. Ajith et al [17] aimed at analysing an existing, non-compliant commercial vehicle cab, redesigning it to ensure its compliance against regulations using numerical methods.

2. Finite element simulation

Finite element method (FEM) is a numerical technique for finding approximate solutions to boundary value problems for partial differential equations. It uses subdivision of a whole problem domain into simpler parts, called finite elements, and variational methods from the calculus of variations to solve the problem by minimizing an associated error function. FEM encompasses methods for connecting many simple element equations over many small sub domains, named finite elements, to approximate a more complex equation over a larger domain. Crash simulations [21] are used by automakers during Computer-Aided Engineering (CAE) analysis to ascertain crashworthiness in Computer-Aided Design (CAD) process of modelling a new vehicle. During a crash simulation, the kinetic energy, or energy of motion, that a vehicle possess before the impact is transformed into deformation energy, mostly by plastic deformation of the body material, at the end of the impact. Data obtained from a crash simulation indicate the capability of the vehicle body or guard rail structure to protect the vehicle occupants during a collision against injury. To model real crash tests, crash simulations include virtual models of crash test dummies and passive safety devices. Guide rail tests evaluate vehicle deceleration and rollover potential, as well as penetration of the barrier by vehicles.

Crash and occupant safety analysis must be able to handle large deformations, sophisticated material models (for steel and aluminium, rubbers, foams, plastics, and composites), complex contact conditions among multiple components, and short-duration impact dynamics. The simulating environment must be capable of simulating different types of crash events: frontal impact, side impact, rear impact, and rollover. Crashworthiness simulation is less expensive and yields more information than experimental techniques. LS-DYNA is used worldwide by leading automobile manufacturers because of its extensive capabilities for handling crashworthiness and occupant safety simulations.

3. Truck cabin CAD models

A truck cabin comprises of many components for the analysis purpose but here only few components are chosen based on the involvement of these components during an accident and computational time involved in considering all the components. The components of a loaded cabin selected for the crash analysis are truck cabin, wind shield, front doors (left & right), mudguard (left & right), window front (left & right), chassis, floor, bumper, foot rest (left & right), bolts, radiator, engine and dashboard. Solid models of bare and loaded truck cabins are prepared using CATIA software as depicted in Fig. 1. Loaded truck cabin refers to cabin with all typical components and bare cabin refers to cabin only without any other component.

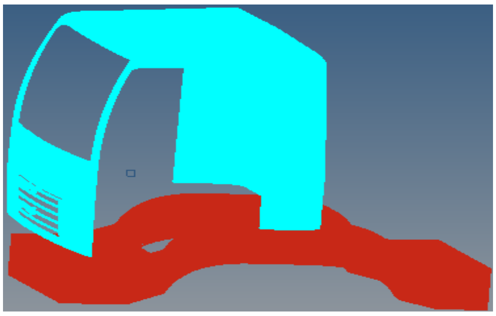

Fig. 1(a): Solid model of selected bare truck cabin

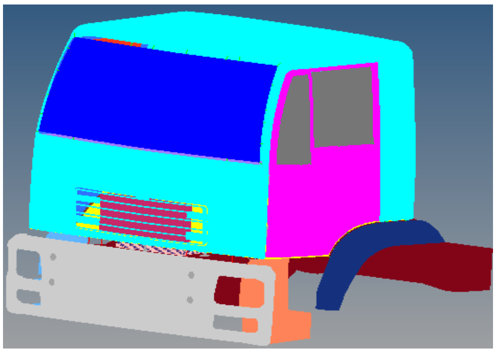

Fig. 1(b): Solid model of selected loaded truck cabin

4. Finite element crash simulation of bare and loaded truck cabins

After development of the solid models using CATIA software, Hypermesh software is used to create the FE models of both the cabins. Shell and solid elements are used to descritize the solid models. There are 542364 shell elements and 104219 solid elements in the FE model. Material properties for each part are chosen such that it should be in line with the material used for the real truck cabin. LSDYNA version 971 with 2 processors is used to create simulated environment. Most of the run of this work are analyzed for 0.2 seconds and the results are observed. In this study following twelve different cases of the truck cabin-rigid wall impact under two different categories are analyzed to capture the dynamic behaviour of truck cabin during various patterns of accident which are shown in Table 1. Out of all above twelve cases, detailed discussion on results obtained for following two cases are presented as a sample in this paper through snapshots taken at 100 milliseconds intervals.

- Case-I: Rigid wall stationary, bare truck-cabin impacting at velocity of 56km/h at 90°.
- Case-II: Rigid wall stationary, loaded truck-cabin impacting at velocity of 56km/h at 90°.

Table 1: Crash simulation cases

Cases	Impact velocity (km/hr)	Impact angle (°)
Loaded truck cabin	40	90, 30
	50	90, 30
	56	90, 30
Bare truck cabin	40	90, 30
	50	90, 30
	56	90, 30

5. Results and discussion

5.1. Case-I: Rigid wall stationary, bare truck-cabin impacting at velocity of 56 km/h at 90°

Fig. 2 to 4 depicts the behaviour of bare truck cabin impacting a rigid wall at 90° with 56km/h (Case-I) for 200 milliseconds. The screen shots taken at 0, 100 and 200 milliseconds clearly depict the dimensional changes happening in the bare cabin frame due to this impact. Fig. 5 shows the graph between the behaviour of different energies with the time. As it is depicted in the graph that before impacting the kinetic energy remains constant for some time at a value of 440 kJ and after the impact, it suddenly goes down due to approximately zero velocity. Due to deformation/ deflection in the frame of the bare truck cabin this KE is being absorbed by the frame and is convert into internal energy, thereby, increasing the internal energy of the vehicle to almost same level. As shown in the graph, after 0.15 sec both the energies become constant. It can be seen that within 0.15 sec complete impact energy has been absorbed by the cabin structure. Fig. 6 shows the graph between displacements of the vehicle from its original position with the time. In total the cabin frame is deformed by 220 mm with in 0.15 sec. Although the energy is completely absorbed by the bare cabin but the magnitude of deformation is too high to be too dangerous for the driver and occupants.

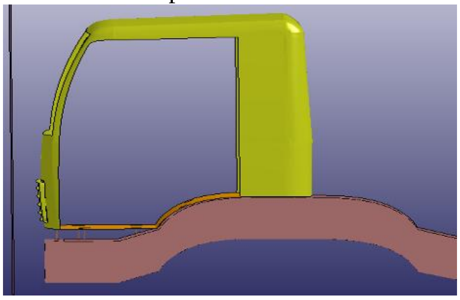

Fig. 2: Deformation of bare cabin for Case-I at 0 milliseconds

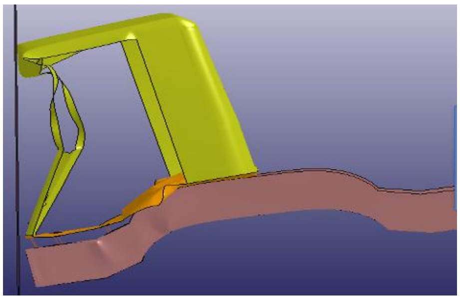

Fig. 3: Deformation of bare cabin for Case-I at 100 milliseconds

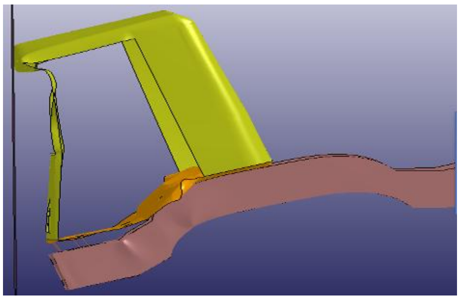

Fig. 4: Deformation of bare cabin for Case-I at 200 milliseconds

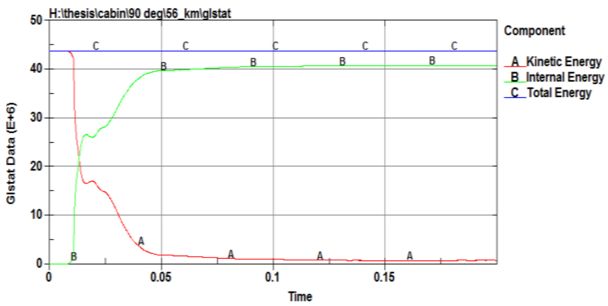

Fig. 5: Case-I: Energy vs. Time (sec)

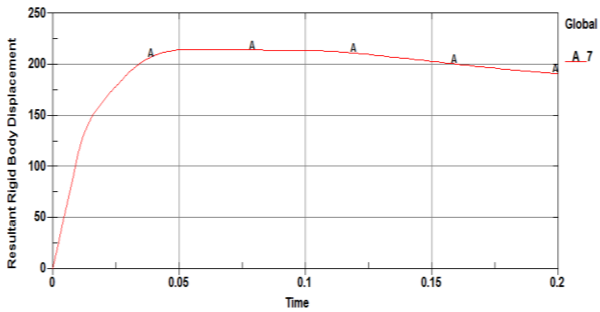

Fig. 6: Case-I: Displacement (mm) vs. Time (sec)

Fig. 7 shows the graph between the velocity vs. time. It can be seen that the velocity remain constant for some time before impacting the rigid wall and then it suddenly went down due to impact. After 0.05 sec it almost remains constant at very low value of 1 mm/s which is the velocity of cabin due to plastic deformation. Fig. 8 shows the graph between the negative acceleration of the vehicle with time. As we know that the acceleration is nothing but differentiation of velocity, Initially, when velocity is constant (for 0.01 sec duration) the acceleration is zero but as the velocity decreases from 11mm/s to 1mm/s due to impact with in 0.04 sec, the retardation goes very high up to 1.45 mm/s^2. After some fluctuations again it becomes zero for duration of 1.15 sec due to constant velocity.

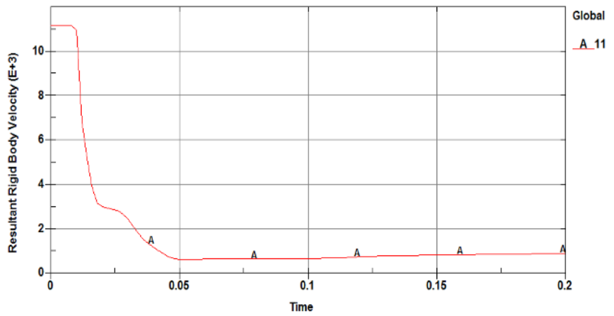

Fig. 7: Case-I: Velocity (mm\s) vs. Time (sec)

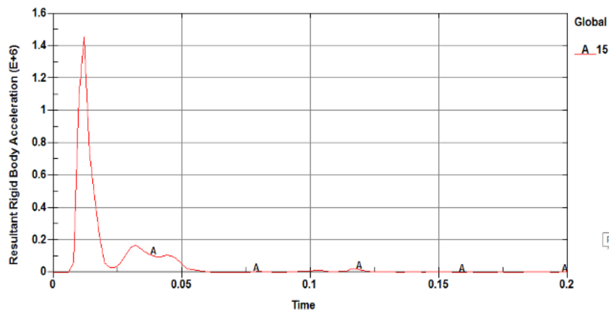

Fig. 8: Case-I: Retardation (mm\s^2) vs. Time (sec)

5.2. Case-II: Rigid wall stationary, loaded truck-cabin impacting at velocity of 56 km/h at 90°

Figs. 9 to 11 show the loaded truck cabin impacting a rigid wall at 90° with velocity of 56km/h (Case-II) for different time steps up to 200 milliseconds. Different parts like radiator, front doors, wind shield, bumper, door glass, mudguard and front cabin are being deformed during this simulated crash situation. Fig. 12 shows the graph between the behaviour of different energies with the time. As it is depicted in the graph that before impacting the kinetic energy remains constant for some time at a value of 806 kJ and after the impact, it suddenly goes down due to approximately zero velocity. Fig. 13 shows the graph between displacement of the vehicle from its original position with the time. In total the cabin frame is deformed by 240 mm with in 0.15 sec. Although the energy is completely absorbed by the loaded cabin but the magnitude of deformation is too high to be too dangerous for the driver and occupants. Fig. 14 shows the graph between the velocity vs. time. It can be seen that the velocity remain constant for some time before impacting the rigid wall and then it suddenly went down due to impact. After 0.15 sec it almost remains constant at very low value of less than 1 mm/s which is the velocity of cabin due to plastic deformation. Fig. 15 shows the graph between the negative acceleration of the vehicle with time. As the velocity decreases from 12mm/s to 0.8mm/s due to impact with in 0.04 sec, the retardation goes very high up to 1 mm/s^2. After some fluctuations again it becomes zero for duration of 0.06 sec due to constant velocity.

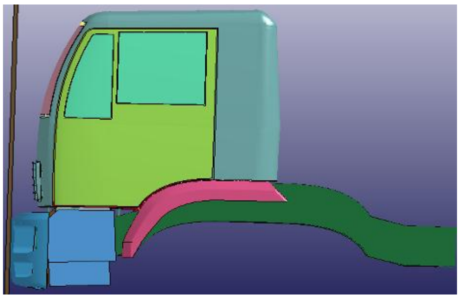

Fig. 9: Deformation of loaded cabin for Case-II at 0 milliseconds

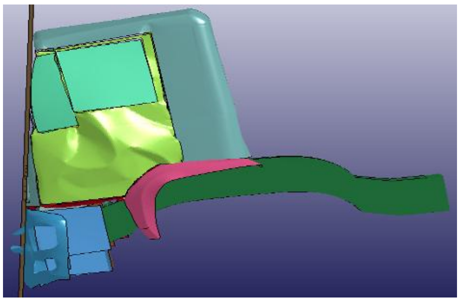

Fig. 10: Deformation of loaded cabin for Case-II at 100 milliseconds

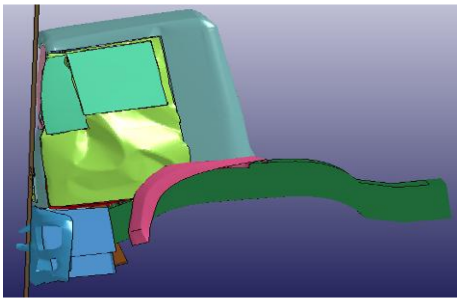

Fig. 11: Deformation of loaded cabin for Case-II at 200 milliseconds

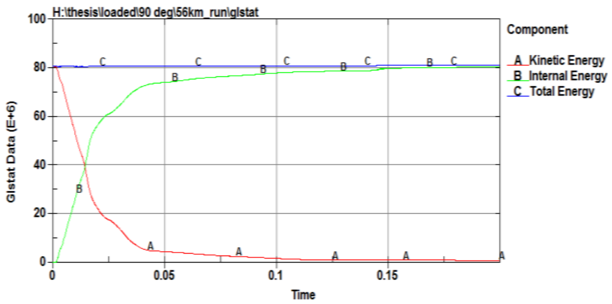

Fig. 12: Case-II: Energy vs. Time (sec)

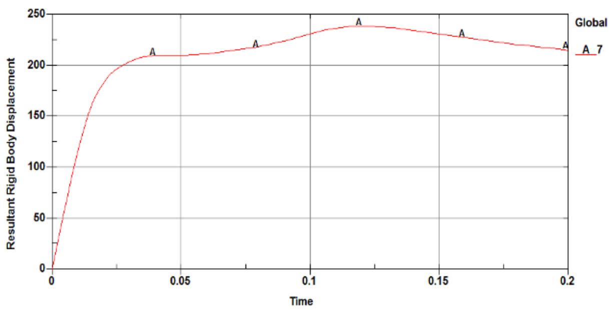

Fig. 13: Case-II: Displacement (mm) vs. Time (sec)

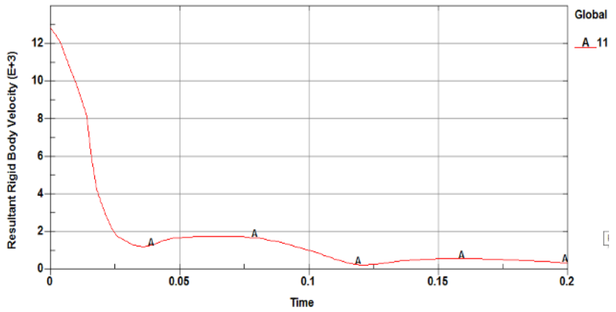

Fig. 14: Case-II: Velocity (mm\s) vs. Time (Sec)

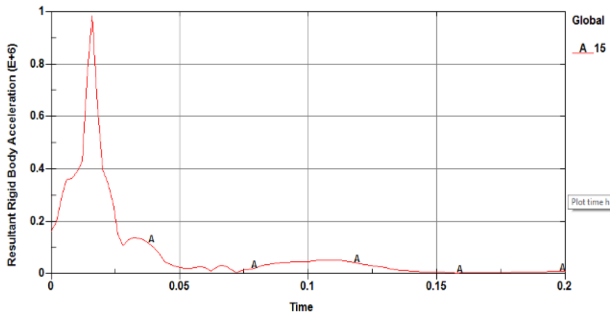

Fig. 15: Case-II: Retardation (mm\s^2) vs. time (sec)

6. Conclusions

In this research work, bare and loaded truck cabin frontal impact on rigid wall situations according to the various crash test used in testing are simulated using FE approach. CATIA V5 R20 CAD software is used for modelling the selected cabin components followed by FE meshing through Hypermesh and then analysing all crash situation cases of a truck cabin with rigid wall using LS-DYNA. Appropriate elements, boundary conditions and material properties are selected for the crash simulation. The crash simulation is run for 200 milliseconds and results are observed and discussed. At the time of crash, kinetic energy gets converted into internal energy. It is felt that FE approach is a viable way to evaluate the crashworthiness of automobile. The energy, displacement, velocity and acceleration graphs obtained are reasonably good and showing good agreement with the mathematical model.

REFERENCES:

[1] N.K. Mishra and S.K. Pradhan. 2014. Crashworthiness and formability assessment of complex system using FE crash simulation, *Proc. Int. Conf. on Advances in Design and Manufacturing*, NIT Trichy, India.

[2] V. Krishnaswami and D. Blower. 2003. *Feasibility of Heavy Truck Occupant Protection Measures*, Transportation Research Institute, The University of Michigan.

[3] G. Samavedam and K. Kasturi. 2011. *Full-Scale Locomotive Dynamic Crash Testing and Correlations: Locomotive Consist Colliding With Steel Coil Truck at Grade Crossing*, U.S. Dept. of Transportation.

[4] T.K. Parnell, C.V. White and S.E. Day. 1999. Finite element simulation of 180° rollover for heavy truck vehicles, *Proc. ASCE Engg. Mechanics Conf.*, Baltimore, USA.

[5] R. Mirzaamiri, M. Esfahanian and S. Ziaei-Rad. 2012. Crash test simulation and structure improvement of IKCO 2624 truck according to ECE-R29 regulation, *Int. J. Automotive Engg.*, 2(3), 180-192.

[6] J. Philip and S. Mungale. 2013. Correlation of test results for ECE-R29 load cases with CAE simulation, *Proc. Altair Technology Conf.*, India.

[7] W. Wang, X.L. Xie, F.L. Shen and X.F. Wang. 2011. Virtual testing for frontal impact of high-top cab of heavy truck, *Applied Mechanics and Materials*, 148-149, 1081-1084. http://dx.doi.org/10.4028/www.scientific.net/AMM.148-149.1081.

[8] T. Ambati, K.V.N.S. Srikanth and P. Veeraraju. 2012. Simulation of vehicular frontal crash-test, *Int. J. Applied Research in Mech. Engg.*, 2(1), 24-29.

[9] Z. Guosheng, L. Qiang, W. Yang and L. Zicheng. 2014. Occupant risk evaluation based on frontal collision of bus, *Proc. Int. Conf. on Logistics Engg. Management and Computer Science*, Shenyang, China.

[10] G.R. Consolazio, J.H. Chung and K.R. Gurley. 2002. Impact simulation and full scale crash testing of a low profile concrete work zone barrier, *Computers and Structures*, 81, 1359-1374. http://dx.doi.org/10.1016/S0045-7949(03)00058-0.

[11] J. Raine and H. Engstrom. 2007. *Finite Element Based Parametric Studies of a Truck Cab Subjected to the Swedish Pendulum Test*, Dept. of Management and Engg., Linköping, Sweden.

[12] H. Raich, D. Chrysler and A.G. Stugart. 2003. Safety analysis of the new ACTROS megaspace cabin according to ECE-R29/02, *Proc. 4th European LSDYNA Conf.*, Ulm, Germany.

[13] G. Bonin, G. Cantisani and G. Loprencipe. 2004. Development of a HGV FEM for road safety analysis, *Proc. 8th Int. Symposium on Heavy Vehicle Weights and Dimensions*, Muldersdrift, South Africa.

[14] A. Krusper and R. Thomson. 2009. *Crash Compatibility between Heavy Goods Vehicles and Passenger Cars: Structural Interaction Analysis and In-depth Accident Analysis*, Chalmers Publication Library, 180-190.

[15] D. Lonn, M. Oman, L. Nilsson and K. Simonsson. 2009. Finite element based robustness study of a truck cab subjected to impact loading, *Int. J. Crashworthiness*, 14(2). http://dx.doi.org/10.1080/13588260802412992.

[16] D.Y. Chen, L.M. Wang, C.Z. Wang, L.K. Yuan, T.Y. Zhang and Z.Z. Zhang. 2015. Finite element based improvement of a light truck design to optimize crashworthiness, *Int. J. Automotive Technology*, 16(1), 39-49. http://dx.doi.org/10.1007/s12239-015-0004-7.

[17] K.P. Ajith, J. Madan and C.R. Kumar. 2009. Analysis and design of a truck cabin structure for static and impact loads, *Book of Abstracts*, 15.