DEVELOPMENT OF VALIDATED FINITE ELEMENT MODEL OF AN ARTICULATED TRUCK SUITABLE TO SIMULATE COLLISIONS AGAINST ROAD SAFETY BARRIERS

AUTHORS:

M. Pernetti, Department of Civil Engineering
Second University of Naples

S. Scalera, Department of Transportation Engineering "Luigi Tocchetti"
University of Naples "Federico II"

G. Cibllis, Department of Transportation Engineering "Luigi Tocchetti"
University of Naples "Federico II"

CORRESPONDENCE:

M. Pernetti
Second University of Naples
Real Casa dell'Annunziata – Aversa (CE)
Telephone: +39 0817683775
Email: pernetti@unina.it

ABSTRACT

Crashworthiness is one of the most important aspect which is taken into account in road design. The effectiveness of Finite Element Method (FEM) to solve major design problems and as a tool to perform parametric studies, has been plainly demonstrated in literature. Of course this is possible only when available models of vehicles and devices are calibrated in a wide range of impact conditions. This research, was intended to develop a well defined multipurpose finite element model of an articulated truck. The model has been set up taking into account two real test impacts, the first against a concrete wall and the second against a steel bridge safety barrier. The fundamental steps of the modelling process will be described along with any requirements needed to reproduce the two full scale tests. The results obtained demonstrate that the modelling processes of vehicle and safety devices were accurate and that, in particular, the articulated truck FE model is suitable for a wide range of impact conditions. As a conclusion, the validated model is reliable to foresee the impact behaviour without needing expensive crash tests.

KEYWORDS:

Articulated Truck, Nonlinear Finite Element Analysis, Road Vehicle Crashworthiness, Road Safety Steel Barrier, Concrete Wall

INTRODUCTION

Crashworthiness is one of the most important aspect which is taken into account in road design. In Italy, run off the road is the fourth type of accident causing about the 19 %of mortalities and the 9.0% of injuries, with a ratio between fatality and frequency (about 6.0 %) twice as much as the overall accidents (about 3.0 %) [1]. Due to the aforementioned reason, the development of new road restraint systems, with high containment level and great level of safety for the vehicles occupants, is a fundamental aim ..

Road safety barriers in Europe have to fulfil the European standard EN 1317[2],[3], which defines a set of crash tests for each safety barriers containment levels. Full scale tests of vehicle collision against road safety barriers have a huge importance to assess the outcomes of real accidents and, more in general, to identify barriers and vehicles' features which influence crashworthiness in a meaningful manner. On the other hand, this kind of tests is really expensive and many parameters are hard to control and measure. Due to the aforementioned reasons, numerical analysis of vehicles collisions against safety barriers has become a convenient methodology that supports and integrates the previous one, especially considering the continuous technological hardware/software progress. [4], [5], [6]. Besides, the chance of controlling and evaluating each factor which influence full scale crash tests, makes such a methodology an important tool to perform parametric studies to assess the influence of different factors on crashworthiness [7], [8]. Of course this is possible only when available models are validated in a wide range of impact conditions.

In this paper a well defined multipurpose finite element model of an articulated truck will be presented. The model has been set up taking into account two full scale impact tests, the first against a concrete wall and the second against a steel bridge safety barrier. The two different situations have been chosen in order to consider the complete range of impact conditions. Indeed, in case of a vehicle colliding with a concrete wall, the kinetic energy of moving truck negligibly transfers to the barrier, while largely turning into the internal energy of the truck; conversely, with the steel barrier the kinetic energy mostly changes into internal energy of the barrier itself.

In the following, the fundamental steps of the modelling process are described along with the requirements needed to reproduce the two full scale tests - suspensions, tires, steering system, frame side members, fifth wheel, cab, connection between cross and longitudinal members, etc. Afterwards, the comparison between full-scale and simulated test outputs is presented. The results obtained simulating the two impacts, demonstrate that the modelling processes of vehicle and safety devices were accurate and that, in particular, the articulated truck FE model is suitable for a wide range of impact conditions. As a conclusion, the validated model is reliable to foresee the impact behaviour without needing expensive crash tests. Similar investigations regarding a rigid truck are provided in another contribution by the same authors [9].

FINITE ELEMENT MODEL OF THE ARTICULATED TRUCK

The articulated truck model has been developed starting from CAD 2-D draws of SCANIA-like vehicles. The 3-D model, the FE modelling stage and the mechanical characterization have been entirely performed in our laboratories.

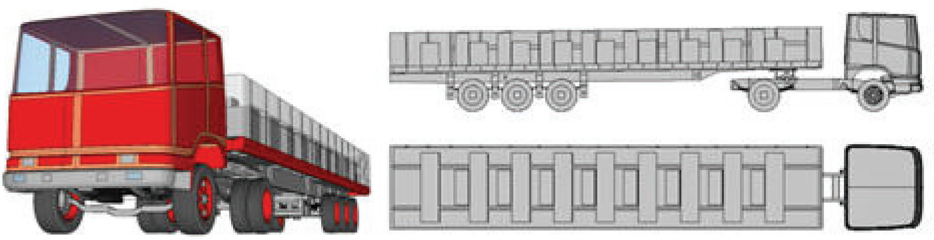


figure 1: FE model of the articulated truck

The global model is composed of 246000 and 227000 degree of freedom. The elements used for the regions experiencing large deformations are "fully integrated Hughes-Liu" shells with 5 integration points through-the-thickness. For a shell element with 5 integration points through-the-thickness, fully integrated Hughes-Liu formulation requires 35367 mathematical operations compared to the 725 ones for Belytschko-Lin-Tsay formulation. This choice, despite an increase of simulation time, drastically reduces the deformations in the zero energy modes. For the other portions of the model, the Belytschko-Lin-Tsay formulation has been chosen [10]. The modelling process has been properly carried out to limit in every portion of the vehicle the hourglass energy below the 5% of the deformation energy. The steel material of the truck was characterized by using the elastic piecewise linear plasticity material model of Ls-Dyna with a specific curve stress/strain and a yield strength amounting to 550 MPa, except for the leaf springs, for which a yield strength equal to 918 MPa was adopted. Failure criteria based on maximum plastic strain were considered. The modelling process of the vehicle was very accurate [11]; particular attention was paid to all the regions and joints that significantly affect the behaviour during the collision of the truck against a safety device. Indeed the vehicle model includes:

- Suitable joints allowing the wheel rolling and steering;
- The ballast, composed by frangible concrete wall
- The stabilizer bars, modelled by Belytschko-Schwer resultant beam, connecting axis and chassis

- The fifth wheel – whose modelled representation exactly matches the real one – is composed of two parts, the upper and lower ones. The upper one is connected to the lower via a revolute joint which axle is transversal to the chassis of the tractor. The lower one is connected to tractor through a properly shaped plate, in order to guarantee an even pressure distribution on the support chassis surface. Owing to ensure the relative rolling between the upper part of the fifth wheel and the trailer, a revolute joint is defined, which is always orthogonal to the trailer platform.
- The suspensions were modelled by reproducing all their components, i.e., leaf springs, spring eyes, centre clamps, alignment clips and plain end mountings. During the model development, the overall stiffness of each suspension was compared with the theoretical value [12]. The good qualitative agreement with the cinematic behaviour of the full-scale vehicle was verified by means of runs on humps and on irregular pavements.
- The crossmembers and the frame side members of the platform, being exposed to large deformations, were modelled performing a stress state convergence analysis in order to select the most convenient mesh [13].
- The model of the wheels includes both tires and rims: in particular, to describe the rubber material, the #27 one embedded in Ls-Dyna was adopted [14]. Moreover, in order to properly describe the friction between wheels and pavement, a pre-processing simulation only accounting for the gravity force on the loaded weights vehicle was performed, which allowed achieving the actual wheels configuration and their tensional stress state (*INTERFACE_SPINGBACK_LSDYNA).

Some of the parts described above are showed in detail in Figure 2

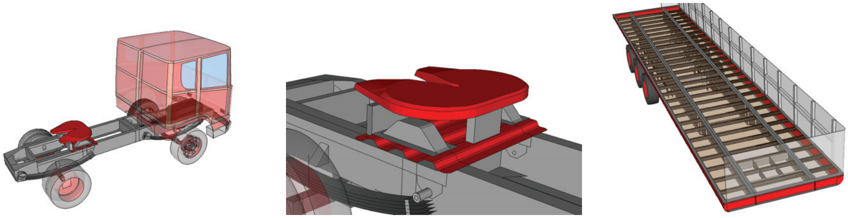


figure 2: some details of the articulated truck FE model

VEHICLE MODEL VALIDATION

The potentiality of Finite Element Method both from design of new safety devices point of view and from parametric analysis of collisions one, was clearly inferred from the proposed reference literature. However, also a refined FE model of vehicle like the one described above, needs to be validated in a wide range of impact condition, through an extensive comparison between full scale and simulated outputs. Due to the aforementioned reasons, two impacts have been chosen, the former against a very strong concrete wall and the latter against a road safety steel barrier. These collisions represent two situations extremely different considering transformation of vehicle's kinetic energy. Indeed, concerning the impact against concrete wall, a large part of kinetic energy changes in vehicle's internal energy causing a collapse in a large portion of the vehicle. Differently, in the case of impact against a steel barrier, vehicle's kinetic energy is transformed in device's internal energy, but the impact against posts stresses tires, axles and suspension in a different manner. Besides, the roll angle is grater then the one registered during the collision against the wall, because the average high above ground of the global action is less then the previous one, causing a larger upsetting moment and a significant stress on the fifth wheel and the suspensions.

Full scale results in both cases are available. As far as the impact against the concrete wall is concerned, the results are included in the report "Measurement of Heavy Vehicle Impact Forces and Inertia Properties" by Beason and Hirsch, 1989 [15]. The data compared have been the overall interaction force averaged on 50ms window. With reference to the collision against the steel barrier, a full-scale test according to EN 1317-1/2 standard for the test TB81 has been considered [2]. In this case, data comparison between the full-scale test and the FE simulation regards residual displacement of each post at different height above the ground, maximum dynamic displacement of the device and vehicle's kinetic.

To perform the simulations of the two collision, the Ls-Dyna 970 code version MPP on 8 bi-processors has been employed. A preliminary optimization to share the simulation run on the 16 processors was performed. Such an operation allowed to reach an optimum value for the Grind Time (the averaged elapsed time for computing one element at a single step) and the speed up (the ratio between the elapsed time for a simulation executed on one processor and that for a simulation on n-processor) [16], [17].

IMPACT AGAINST CONCRETE WALL

BARRIER DESCRIPTION

The concrete wall is a very rigid barrier, which could be used, equipped with a flexible protection on the side exposed to the car collisions, in situations where areas beside roads have to be particularly protected against vehicle penetration i.e., bridges crossing high speed railways, areas used to store very dangerous substances, etc

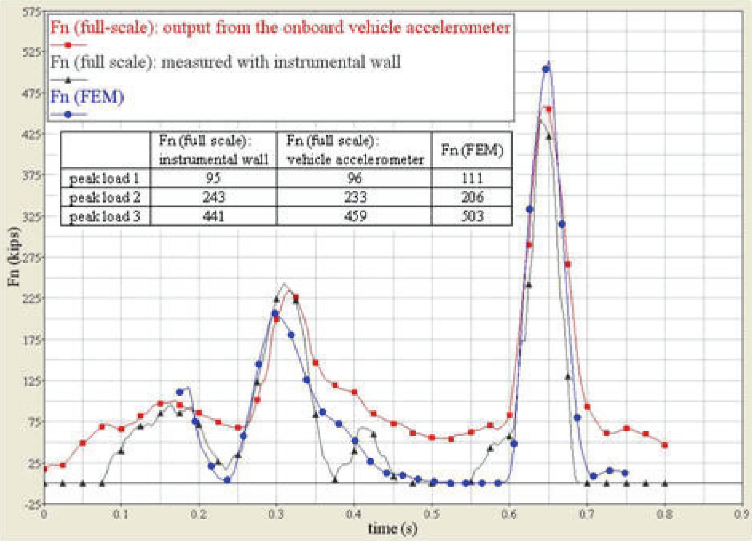
The FE model represents a reinforced concrete wall having an elastic modulus of 28500 N/mm^2 and a density equal to 2.5e-9 ton/mm^3. Its geometrical characteristics are: length 27m, height 3m and thickness equal to 0.3m. The wall has been modelled by shell elements 75 mm x 75 mm. Such values were defined through an optimization process in which the convergence of the element stress state was accounted for. The material has been assumed as indefinitely elastic and resistant. The wall is fully constrained at the lower edge.

TEST CONDITION

As said above, the employed reference to full-scale test is "Measurement of Heavy Vehicle Impact Forces and Inertia Properties" by Beason and Hirsch, 1989. In particular, the #7046-4 test is referred to, where an articulated truck collides the described wall at the following impact conditions: weight 36490kg , velocity 88.2km/h and impact angle 16°. In the same report the geometric and inertial features of the vehicle chosen for the test are addressed. Even if excellent agreement between the real values and those measured in the FE model is largely obtained, a substantial difference has not been removed: the real tractor has rear tandem tires unlike FE model. This choice has been reached to avoid the introduction of too many variables in the evaluation of the FE model outputs, considering that the real tractor used in the test against the steel barrier has a unique rear axle.

COMPARISON BETWEEN FULL-SCALE AND SIMULATED TESTS

During the full-scale test, shortly after impact, the entire front of the tractor came loose from the frame and began to shift to the left. This also occurs during the FE impact simulation after about 0.05 sec. In the real crash, after the collision of the front part of the cab, the tractor began to redirect, and at approximately 0.225sec after impact beginning, the rear tandems of the tractor hit the instrumental wall. The top of the tank-trailer made contact with the instrumental wall at approximately 0.225 sec. As the tractor moved parallel with the wall, the rear of the tank-trailer slapped at 0.618 sec and the rear wheels of the trailer hit at about 0.638 sec. The vehicle continued to move parallel with the wall until it lost contact at about 0.854 sec. The full-scale impact dynamics are described with excellent agreement by the FE model, as depicted in the diagram in the figure3, which plainly shows the time ranges where the cab, the front part and the rear side of the trailer collide, respectively. In the same diagram, the magnitudes of the 0.050 sec windows average force (expressed in kips) acting on the instrumented wall are observable. In particular a numerical comparison of the three peaks between full-scale and simulated data, occurring in the phases described above, is included in the table in the same figure. As can be observed, Fn has been measured by a twofold strategy during the full-scale test, namely, through accelerometers placed on the vehicle and by means of load cells in the instrumented wall, respectively. Accordingly to the report, the latter is believed to be more reliable; as a matter of fact, the FE counterpart better fits with the latter.



	Fn (full scale): instrumental wall	Fn (full scale): vehicle accelerometer	Fn (FEM)
peak load 1	95	96	111
peak load 2	243	233	206
peak load 3	441	459	503

figure 3: interaction between concrete wall and articulated truck – comparison between full-scale and simulated output

The error between the three peak mentioned above, recorded on the instrumental wall and during the simulation respectively, is less then 10%. Such a difference is surely caused by the presence of the rear tandem tires in the real tractor and by the consequently different mass distribution. However, the model can be considered reliable with a high rate of confidence.

IMPACT AGAINST A STEEL BARRIER

BARRIER DESCRIPTION

The model represents a three rail steel bridge barrier, with a containment energy level of 724kJ (H4b - see EN 1317-1/2). This device consists of:

- HE posts .
- Upper 2-wave and 3-wave rails whose length is 4820mm
- Upper and 3-wave rail spacers, made with 2 symmetrical steel parts
- Raised concrete beam whose high is 125mm.

figure 4: pull-out test: comparison between full scale and FE output

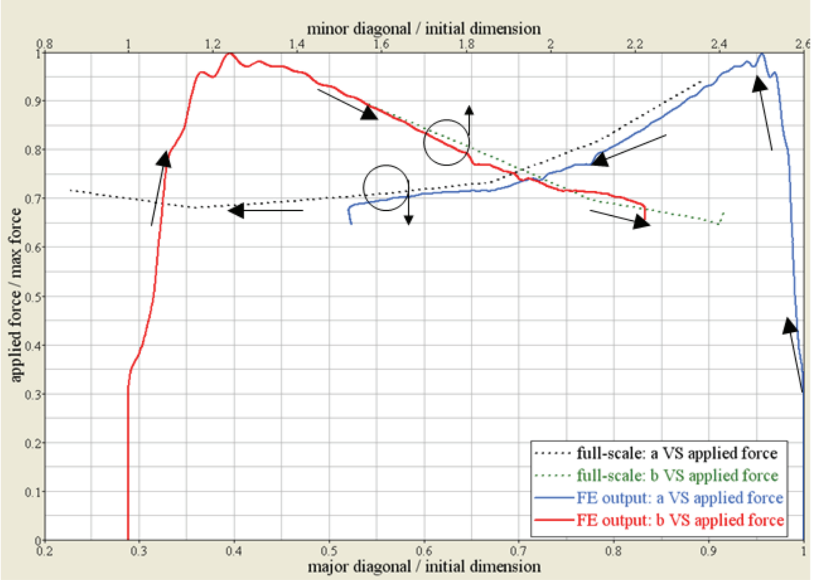


figure 5: test on the spacer: comparison between full-scale and FE output

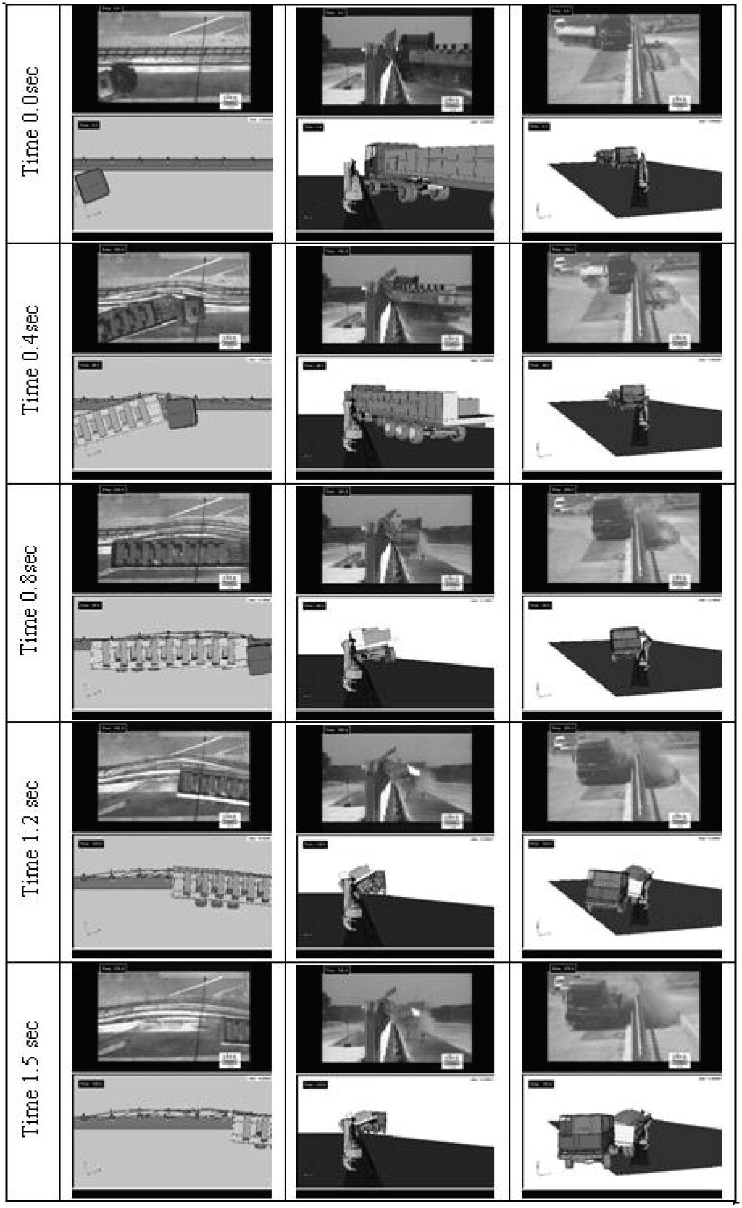


figure 6: comparison between full scale and simulated kinetic

The steel material behaviour was reproduced by the Material #24 in Ls-Dyna with the definition of stress-strain curve for the plastic yield.

To achieve accuracy in the FE simulation of the collision between the articulated truck and the considered steel barrier, preliminary tests on the post and the spacer are essential [18]. The outputs of these tests are proposed in figure 4 and 5 respectively. For the former, two measures are taken into account, increasing the pull out load, that are the displacement and the height above ground of the post's top. The reference systems are different for the two measures and specified by the arrows. In figure 5 the outputs of a crushing test on the spacer are proposed. Also in this case, increasing the load, the behaviour of two characteristic dimensions are pointed out. Obviously, the arrows included in the diagram underline that during the test, the major diagonal (dimension a in the figure) become shorter and the minor one (dimension b in the figure) grow longer. Each dimension in figure 4 and 5 is normalized with respect to a measure of reference. For both tests, the comparison between full-scale and simulated test outputs is shown and, as can be observed, both the model of the post and the spacer are able to describe the real behaviour of the correspondent structures with a high rate of confidence.

TEST CONDITION

As mentioned above, the EN 1317 -1/2 establish the impact conditions and the vehicle mass within a well-determined range for all containment levels of the barriers. In particular, it establishes for the containment level H4b a vehicle mass of 38000kg, an initial velocity equal to 65km/h and an impact angle of 20°. Such standard also defines the kind of vehicle to be employed for the test and its geometrical and inertial features, which were accounted for during the characterization stage of the FE model. The dynamic coefficient of friction of the wheels was set equal to 0.5.

COMPARISON BETWEEN FULL-SCALE AND SIMULATED TESTS

To compare the real and simulated tests, table 1 and 2 and Figure 6 are presented. In the first table permanent deflections of the barrier in four different points for each post are provided. In the second table, the summary of the outputs are included. In Figure 6 the evaluation of simulated and real test kinetic is proposed, comparing sequential pictures of the collision.

All of the results show an excellent agreement.

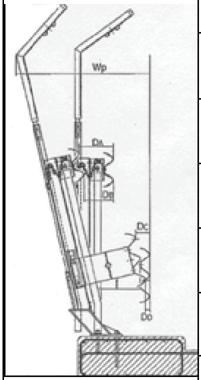
			Permanent deformations(m)				
	**Post nr**		**DA**	**DB**	**DC**	**DD**	**Wp**
	13	**Full-scale data**	0.66	0.66	0.45	0.45	1.64
		Simulated data	0.61	0.60	0.50	0.47	--
	14	**Full-scale data**	0.91	0.87	0.72	0.64	2,04
		Simulated data	0.85	0.75	0.74	0.64	--
	15	**Full-scale data**	0.89	0.85	0.84	0.71	2,14
		Simulated data	0.82	0.79	0.74	0.64	--
	16	**Full-scale data**	0.70	0.68	0.67	0.57	1,90
		Simulated data	0.83	0.74	0.67	0.54	--
	17	**Full-scale data**	0.42	0.39	0.33	0.35	1,50
		Simulated data	0.65	0.53	0.45	0.36	--

table 1: impact against steel barrier – permanent deflection of the device

	Full-scale data	**Simulated test data**	**Percentage error**
Maximal permanent deflection	0.91m	0.90m	1.1%
Maximal dynamic deflection	1.1m	0.98m	11%
Contact length	13.8m	17.1m	24%
Exit angle of the vehicle	7.5°	5.5°	26%

table 2: impact against steel barrier – summary of the output

CONCLUSION

The rigid truck FE model developed was aimed to be (i) a support to design novel devices and (ii) a tool to perform parametric studies to assess the influence of different factors. Due to this, each part of the truck has been modelled with particular attention. The model has been validated through an extensive comparison with two full scale impact tests, the first against a concrete wall and the second against a road safety steel bridge barrier. The excellent agreement attained when simulating the abovementioned impacts, characterized by noticeably different nature, demonstrates that the model of the HGV is suitable for a wide range of impact conditions. As a conclusion, the validated model is reliable to be used to predict the impact behaviour without needing expensive crash tests.

REFERENCES

[1] Italian National Institute of Statistic, ISTAT.

[2] EN 1317-1, "Road restraint system – Part1: Terminology and general criteria for test methods",1998.

[3] EN 1317-2, "Road restraint system – Part2: Performances classes, impact test acceptance criteria and tests methods for safety barriers", 1998.

[4] Reid, J. D., "LS-DYNA Simulation Influence on Roadside Hardware" Transportation Research Record

[5] Belytschko, T., "On computational methods for crashworthiness", Computers and Structure, 1992

[6] Ray, M. H, "Use of Finite Element Analysis in Roadside Hardware Design", Transportation Research Circular No.453, 1996

[7] Giavotto, V., Pernetti, M., "Comparison of Collision of Rigid Truck and Articulated Truck Against Road Safety Barriers," Transportation Research Record, 2006

[8] Montella, A., Pernetti, M., "Heavy Goods Vehicle Collision with Road Safety Barriers: Combined Influences of Position of Centre of Mass and Tire-Pavement Friction" Transportation Research Record,

[9] Pernetti, M., Scalera S., "development of validated Finite element model of a rigid truck suitable to simulate collisions against road safety barriers", 6th European Ls-Dyna Users Conference, 2007

[10] Hallquist, J.O, LS-DYNA. Keyword Theory's Manual. Version 971, Livermore Software Technology Corporation, Livermore, 2007

[11] Hallquist, J. O., LS-DYNA. Keyword User's Manual. Version 971, Livermore Software Technology Corporation, Livermore, 2007.

[12] SAE Spring Committee "*Spring Design Manual"*. Society of Automotive engineers, 1996.

[13] Karaoglu, C., Kuralay, N.S., "Stress analysis of a truck chassis with riveted joints", Finite Element in Analysis and Design, 2002.

[14] Pelc, J., "Towards realistic simulation of deformations and stresses in pneumatic tyres", Applied Mathematical Modelling, 2007.

[15] Beason, W. L., Hlrsch, T. J.,"Measurement of Heavy Vehicle Impact Forces and Inertia Properties", Texas Transportation Institute, Final Report 1989.

[16] Lin, Y. Y., "A correlation between MPP LS-DYNA performance and Various interconnection networks – a quantitative approach for determining the communication and computation cost", 4th LS-DYNA users conference, 2003.

[17] Paik H..P., Moon J.J., Kim S.J., Lee M., "Parallel performance of large scale impact simulations on Linux cluster super computer" Computers & Structures, 2006

[18] Borovinsek, M., Vesenjak, M., Ulbin, M., Ren, Z., "Simulation of crash test for high containment level of road safety barriers", Engineering Failure Analysis, 2007.