DEVELOPMENT OF VALIDATED FINITE ELEMENT MODEL OF A RIGID 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"

CORRESPONDENCE:

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

ABSTRACT

The effectiveness of FEM (Finite Element Method) to improve the crashworthiness, both of the vehicle and of the road safety hardware, has been plainly demonstrated in literature. As well known, such a methodology can be successfully employed (i) as a support to design novel devices and (ii) as a tool to perform parametric studies to assess the influence of different factors. Of course this is possible only when available models are calibrated in a wide range of impact conditions.
In this work, a well detailed finite element model of a rigid truck is presented. 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 barrier. The excellent agreement attained when simulating the abovementioned impacts, characterized by noticeably different nature, demonstrates that the modelling processes of the vehicle and devices were accurate and that, in particular, the FE model of the Heavy Good Vehicle is suitable for a wide range of impact conditions.
As a conclusion, the validated model is reliable to predict the impact behaviour without needing expensive crash tests.

KEYWORDS:

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

INTRODUCTION

Run off the road is one of the most frequent road accident. The consequences often are very severe. In Italy it represents the third type of accident and it is the second as far as the fatality rate is concerned. Safety systems, like road side barriers, crash cushions etc, are often used to reduce the injuries related to the vehicle run off the road [1].

Full scale experiments of vehicle collision against road safety barriers have a huge importance to assess impact accidents outcome 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 hardly controllable and measurable. Due to the aforementioned reasons, numerical analysis of vehicles collisions against safety barriers has become a fundamental methodology that supports and integrates the previous one, especially considering the continuous technological hardware/software progress [2], [3], [4]. Of course this is possible only when numerical analyses are carried out with vehicle and barrier models that are validated in a wide range of impact conditions.

In this paper a well detailed model for a rigid truck is proposed. All of the vehicle parts have been modelled to reproduce full scale crash tests. Indeed a lot of attention was paid to suspensions, tires, steering system, frame side members, cab, connection between cross and longitudinal members, etc [5]. 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 barrier. For the former the impact forces have been considered and for the latter the barrier displacements and the vehicle kinetic have been taken into account.

In the following, after providing the model details, it will be shown how well the finite element analysis of the two collision events describes the corresponding full scale crash test. These results demonstrate that the modelling process of vehicle and devices was accurate and that, in particular, the rigid truck FE model is suitable for a wide range of impact conditions. As a conclusion, reliable means have been obtained for predicting the actual impact behaviour without the need of extensive full-scale crash test programs, which are very time/money demanding.

FINITE ELEMENT MODEL OF THE RIGID TRUCK

The rigid 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.

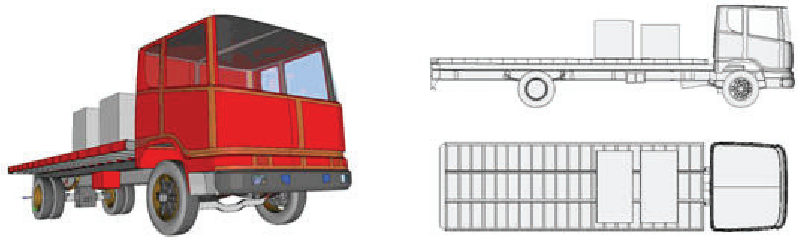


Figure 1: FE model of the truck

The global model is composed of 110000 and 105000 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 [6]. 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 [7]; 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.
- Two different types of cab (the American one for the impact against the concrete wall and the European one for the collision against the steel barrier). The cab employed in the reference full-scale test was adopted since in the HGV with European cab the engine and the front wheel are placed backward with respect to that with American cab. This noticeably affects the inertial features of the overall vehicle.
- The ballast, composed by frangible concrete wall.
- The stabilizer bars, modelled by Belytschko-Schwer resultant beam, connecting axis and chassis.
- 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 [8]. 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 [9].
- 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 [10]. 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

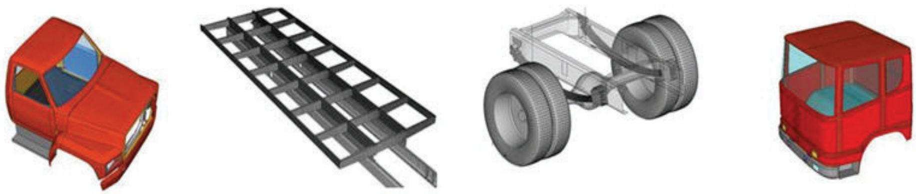


Figure 2: HGV's particular

VEHICLE MODEL VALIDATION

A well detailed Finite Element Model is a valid tool to analyze real collisions only if it is validated in a wide range of impacts. In order to provide this quality to the FE model showed in the previous paragraphs two simulations have been carried out, the former against a very strong concrete wall and the latter against a road safety steel barrier. 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 [11]. 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 TB42 has been considered [12]. 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. 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. For such a kind of impact, considerable deformations in many and wide parts of the vehicle are observed. On the contrary, during the collision against a steel barrier, the larger part of the vehicle kinetic energy is transformed in internal energy of the barrier. 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) [13], [14].

IMPACT AGAINST CONCRETE WALL

BARRIER DESCRIPTION

The wall 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 #7046-10 test is referred to, where a rigid truck collides the wall above described at the following impact conditions: weight 8187kg , velocity 83km/h and impact angle 16.8°.
In the same report the geometric and inertial features of the vehicle chosen for the test are addressed. Excellent agreement is obtained between the real values and those measured in the FE model.

FULL-SCALE AND SIMULATED TESTS DESCRIPTION

During the full-scale test, shortly after the impact, the front axle came loose from the vehicle and the front of the unit truck began to shift to the left. This also occurs during the FE impact simulation after about 0.06 sec. In the real crash, the vehicle began to redirect at 0.083sec, and at approximately 0.088sec, the front of the van-box made contact with the instrumented wall. At 0.265sec the rear of the vehicle slapped the wall and began to travel parallel to it. The vehicle continued to move parallel with the wall until it lost contact at about 0.685sec. The full-scale impact dynamics are described with excellent agreement by the FE model, as clarified by the diagram in the figure3, which plainly shows the time ranges within which the cab, the front part and the back side of the trailer collide, respectively. The only noteworthy discrepancy arises beyond 0.4 sec, when the FE rigid truck model looses contact, whilst the full-scale vehicle keeps scraping along the wall for 0.28 sec after the impact.

DATA COMPARISON

The magnitudes of the 0.050 sec windows average force (expressed in kips) acting on the instrumented wall is presented in Figure 3. 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. Both curves have three peaks. The first peak load is associated with the initial impact of the truck. In the case of full-scale test it reaches the value of 61 kips against the value of 52 kips attained with the simulation. The second peak load is associated with the impact of the front-end of the truck's trailer and it has the value of 51 kips in the case of full-scale-test and 38 in the simulated test. The third peak load is 90 kips for the real test and 107 kips for the simulated test and it is associated with the broadside impact of the truck box.

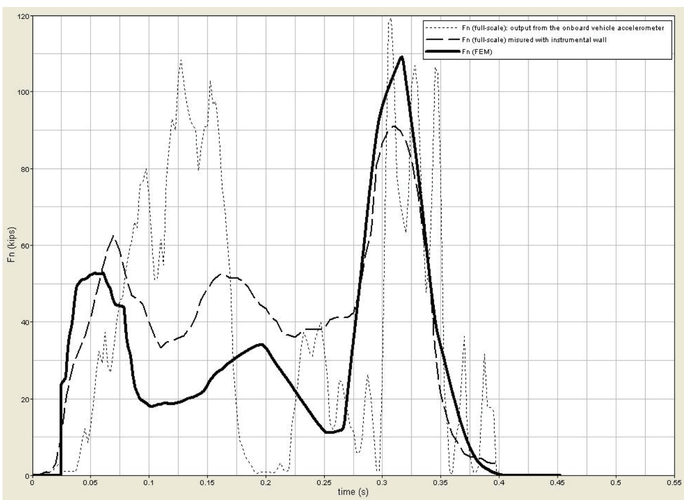


Figure 3: interaction between concrete wall and rigid truck – comparison between full-scale and simulated output

It can be concluded that, since the percentage error is lower than 20% in the overall phase where the iteration peak is maximum (i.e., 0.28-0.32 sec), the model can be considered reliable.

IMPACT AGAINST A STEEL BARRIER

BARRIER DESCRIPTION

The model represents an H1-type barrier (see EN 1317-1/2) , with a containment energy level of 127kJ (H1). This device consists of:

- C120x80x4 posts, 1.70m long, embedded into the ground and spaced every 2.00m
- W beam 3mm thick (length: 4.32m, top height: 0.75m)
- European spacer
- Rear plate (65x5mm, length:4.14m) connected to the back side of posts

Two different steel materials were used for the barrier (S275 JR and S235 JR) which behaviour was reproduced by the Material #24 in Ls-Dyna with the definition of stress-strain curve for the plastic yield. The force-displacement relationship of the posts embedded in the soil has been obtained by real pull out tests and simulated by using spring elements with elasto-viscoplatic characteristic [15].

TEST CONDITION

As mentioned above, the specifications for this kind of simulated test are EN 1317 -1/2, which 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 H1:

- Weight: 10000 kg (weight of the real vehicle 10070 kg)
- Velocity: 71 km/h (velocity of the real vehicle 71.9 km/h)
- Impact angle 15° (impact angle of the real vehicle 14.8°)

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 full-scale crash test was performed on soil. Due to this, the dynamic coefficient of friction of the wheels was set equal to 0.3.

FULL-SCALE AND SIMULATED TESTS DESCRIPTION

The guided vehicle struck the device at the 0.62 m before the end of 2 waves rail number 6. The vehicle ran along the barrier until 0.23 m after the end of w beam rail number 11 and left the device with an angle of 7.1°.

The impact created a bow 28.0 m long with a permanent deflection of 1 m (1.06 m in full scale test with a percentage error equal to 5.5%) at post number 19. The maximum dynamic deflection was 1.16 m (not measured in full scale test). The vehicle did not pass over the device, did not cross the direction line and did not rolled over within test area, both in the real and simulated tests.

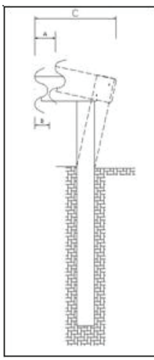
DATA COMPARISON

To compare the real and simulated tests, table 1 and 2 and Figure 4 are presented. In the first table, the summary of the outputs are included. In the second, permanent deflections of the barrier in three different points for each post are provided. In Figure 4 the comparison between roll angle and yaw rate of the vehicle in real and simulated tests is shown.

All of the results show an excellent agreement.

	test full-scale	Simulated test	Percentage error
Contact length	20.9 m	21 m	0.5%
Maximal permanent deflection	1.06 m	1.04 m	1.9%
Maximal dynamic deflection	*	1.16 m	--
Extreme lateral position of the vehicle	1.63 m	1.59 m	2.4%
Level of working width	W5	W5	W5

table 1: impact against steel barrier – summary of the output



Post number	Permanent deflection					
	A		B		C	
	Real test	Simulated test	Real test	Simulated test	Real test	Simulated test
11	0	0.01	0	0	0.47	0.47
12	0.04	0.04	0.03	0.03	--	0.48
13	0.15	0.12	0.13	0.1	--	0.53
14	0.35	0.31	0.30	0.25	0.80	0.75
15	0.62	0.65	0.56	0.60	1.03	1.04
16	0.79	0.85	0.77	0.82	1.19	1.23
17	0.95	0.96	0.85	0.89	1.25	1.29
18	1.04	0.98	0.92	0.80	1.20	1.19
19	1.06	0.85	0.95	0.76	1.23	1.02
20	0.85	0.35	0.76	0.24	1.20	0.79
21	0.49	0.15	0.50	0.10	1.02	0.49
22	0.14	0.04	0.18	0.03	0.73	0.47
23	0.01	0	0.08	0	--	--
24	-0.03	--	0	--	--	--

table 2: impact against steel barrier – permanent deflection of the device

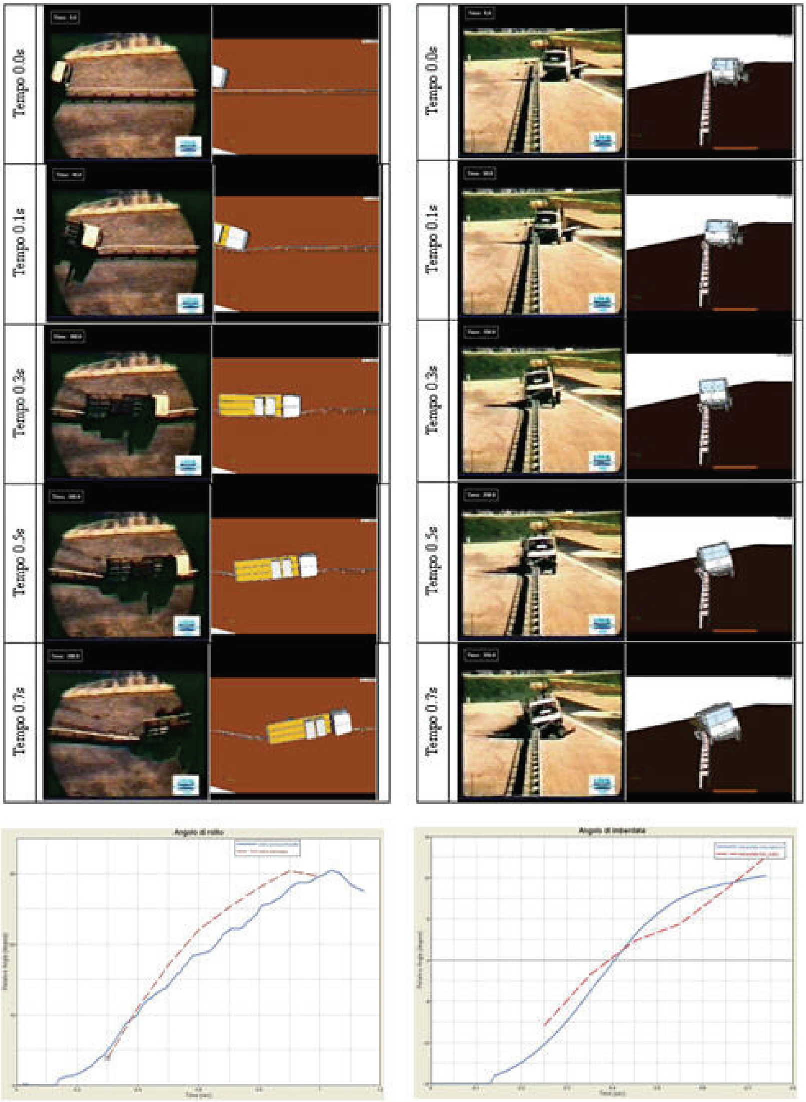


Figure 4: roll angle and yaw rate of the vehicle in real and simulated tests

CONCLUSION

The rigid truck FE model developed was aimed to be (i) as support to design novel devices and (ii) as 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 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 predict the impact behaviour without needing expensive crash tests.

REFERENCES

[1] ISTAT data 1995-2002.
[2] Reid, J. D., "LS-DYNA Simulation Influence on Roadside Hardware" Transportation Research Record
[3] Belytschko, T., "On computational methods for crashworthiness", Computers and Structure, 1992
[4] Ray, M. H, "Use of Finite Element Analysis in Roadside Hardware Design", Transportation Research Circular No.453, 1996
[5] Oldani, E. Castelletti, L., Anghileri, M., Mongiardini, M., "Impact Analysis of a 16t Truck against Road Safety System", 5th European LS-DYNA Users Conference, 2006
[6] Hallquist, J.O, LS-DYNA. Keyword Theory's Manual. Version 971, Livermore Software Technology Corporation, Livermore, 2007
[7] Hallquist, J. O., LS-DYNA. Keyword User's Manual. Version 971, Livermore Software Technology Corporation, Livermore, 2007.
[8] SAE Spring Committee "*Spring Design Manual".* Society of Automotive engineers, 1996.
[9] Karaoglu, C., Kuralay, N.S., "Stress analysis of a truck chassis with riveted joints", Finite Element in Analysis and Design, 2002.
[10] Pelc, J., "Towards realistic simulation of deformations and stresses in pneumatic tyres", Applied Mathematical Modelling, 2007.
[11] Beason, W. L., Hlrsch, T. J.,"Measurement of Heavy Vehicle Impact Forces and Inertia Properties", Texas Transportation Institute, Final Report 1989.
[12] CEN, (1998a); EN 1317-1:1998 Road restraint system – Part1: Teminology and general criteria for test methods.
[13] 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.
[14] 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
[15] Borovinsek, M., Vesenjak, M., Ulbin, M., Ren, Z., "Simulation of crash test for high containment level of road safety barriers", Engineering Failure Analysis, 2007.