

Vehicle Fatigue Load Prediction based on Finite Element TIRE/ROAD Interaction implemented in an Integrated Implicit- Explicit Approach

E. Duni, G. Toniato

FIAT Group Automobili

R. Saponaro, P. Smeriglio

FIAT Research Centre

Abstract: *This work describes a numerical methodology based on the Finite Element approach able to simulate the dynamic maneuver of the full vehicle running on fatigue reference roads. The basic idea of present work stays in combining a moderately complex and general tire model with traditional full-vehicle methods, including both implicit and explicit finite element techniques, in order to predict – within the early design phases when no prototypes are available - the loads transmitted to the vehicle running on the real fatigue reference roads. Some issues related to application of tire finite element model to a long simulation time in an explicit solution have been discussed. The real fatigue load is digitalized and implemented as a rigid body in the explicit code. The methodology has been successfully applied to Fiat light commercial vehicle, New Fiorino, chosen as test case for this work.*

Keywords: *Connectors, Constitutive Model, Coupled Analysis, Dynamics, Elasticity, Experimental Verification, Fatigue Life, Hyperfoam, Hyperelasticity, Impact, Inertia Relief, Interface Friction, Low-Cycle Fatigue, Mechanisms, Minimum-Weight Structures, Multi-Body Dynamics, Output Database, Plasticity, Postprocessing, Powertrain, Probabilistic Design, Residual Stress, Response Spectra, Rubber, Rubber Bushing, Safety, Scripting, Shell Structures, Shrink Fit, Soil-Structure Interaction, Springback, Substructures, Suspension, Tires, Vibration, Viscoelasticity, Visualization.*

1. Introduction

The current trend in car cycle development is strongly oriented in reducing the Time To Market after the concept phase, together with cost saving and structural performances improvement. In this process, building prototypes and performing experimental tests represents an heavy and expensive duty. Consequently, after the prototype is built, one of the main goals is to rely on good

confidence results concerning structural performances, in order to reduce further expensive and long-lasting trials.

The current car durability cycle adopted by FGA during the virtual development stage requires as starting point some information, in terms of acceleration and displacements of wheel centers. Normally, these parameters are missing in the early design stage of a new vehicle; therefore some assumptions must be made based on known values coming from road and bench tests on previous vehicles. The goal of the method described here is to avoid the uncertainty coming from such data on previous vehicles, transforming the virtual durability cycle into a tool able to make much more accurate and reliable the whole virtual design, at component and system level.

This requires the development of new procedures, which is possible today thanks to the combination of faster computers, knowledge on new materials and joining techniques and availability of robust computational methods. In this new environment all the past methods need to be re-evaluated to consider more general, straightforward approaches.

A critical point in modeling non-linear transient vehicle dynamics is to select the best strategy in order to combine the simulation of quasi-static and dynamic phenomena. The ideal solution is to perform quasi static analysis (equilibrium conditions prior to the dynamic simulation) using an implicit finite element method and then to use these results as the initial condition for the dynamic analysis, the latter being most efficiently conducted, in terms of tire-road contact interaction, with an explicit finite element method.

Unfortunately, the advantage that explicit solution offers in managing tire-road contact interaction is negatively balanced by the analysis time commonly required in durability evaluations. If in the crash events, where explicit codes are already widely used, the order of simulation time is milliseconds (80 ms to 120 ms), in the durability field single events have to be managed going from 60 sec to 120 sec. In this context the problem of solution accuracy due to the large number of increments and the cpu-time required represent the two main issues influencing the success of such approach.

As the minimum time increment of the explicit solver is governed by tire, the model used to simulate the response of this component represent a significant challenge, being the key factor in such kind of approach to simulate at the best level the tire/road interaction. With the finite element approach is possible to have the contact evaluation between discretized physical bodies instead of using simplified models based on a limited number of degrees of freedom; this is the main new aspect of this methodology.

2. Vehicle Virtual Durability Method: FIAT State of the Art

In a multidisciplinary vision of vehicle structural performances the virtual durability sizing represents one of the main issues to be taken into account in the development phase of a new car. The vehicle virtual durability methodology currently applied at FGA during development of new cars is summarized in the flow-chart reported in the Figure1.

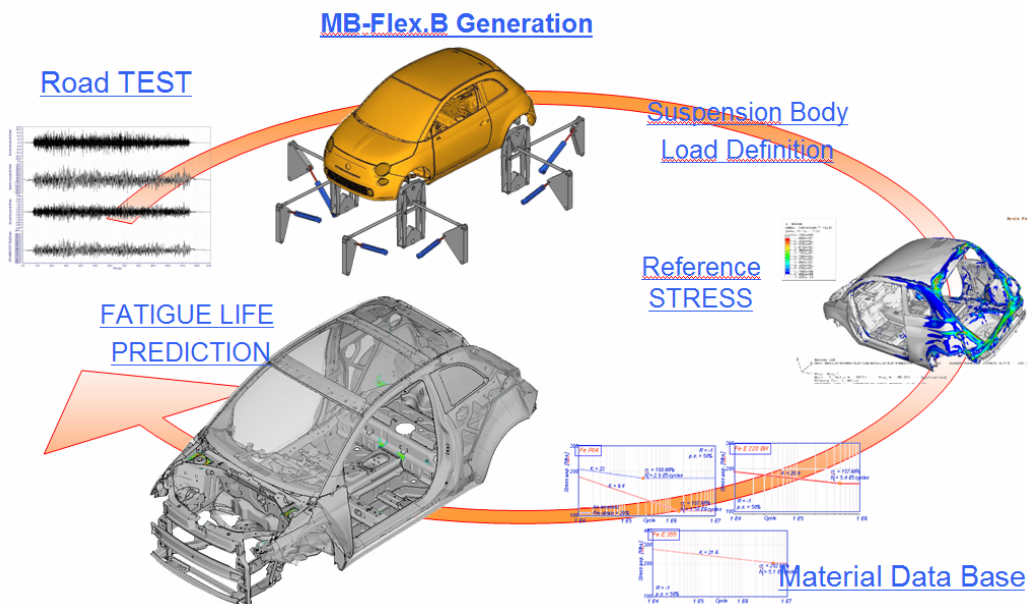


Figure 1. FIAT vehicle Durability flow-chart

In this complex procedure is important to focus on the following important issues:

- some experimental data, in terms of loads and accelerations at wheel centres, coming from previous vehicles running on real roads, similar to the new one in terms of dimensions and suspension architectures, are used as input information for the subsequent virtual calculations;
- a vehicle virtual hybrid model (MB-Flexible Body and Finite Element) is used in order to simulate the behavior of a full car on the bench test. A non linear implicit transient dynamic analysis is carried out to evaluate time history of loads acting between suspensions and body (in each degree of freedom of all attachment points) for a testing time interval of 60 sec up to 120 sec, depending on vehicle segment;
- with the assumption of linear behaviour, based on the approach used for the finite element simulation of the trimmed body (inertia relief or transient modal dynamic) the reference stresses for each unit load or modal stresses are defined;
- the combination, inside a fatigue code, of the previous information with material fatigue properties, allows the fatigue life prediction of the car in terms of cycles/kilometers with respect to targets.

This durability cycle currently applied shows two limitations:

1. it is not entirely carried out in the virtual environment, as some experimental data from previous vehicles must be available;
2. due to the previous limitation, it is not so accurate because each car, even similar to the previous, have proper particular features that make it unique and in any case different from others from the system point of view.

The need to overcome such limitations and to increase the predictive capability of fatigue loads pushed the development of new methodology reported in this paper. The main goal of this methodology based on Finite Element tire/road interaction is to simulate the dynamic maneuver of the full vehicle running on fatigue reference roads. In order to make feasible such methodology, the following issues have been faced and solved:

- the choice of the best strategy in order to combine the simulation of quasi-static and dynamic phenomena;
- the development of a reliable tire finite element model of moderate complexity which is able to run, together with a complete vehicle model, in a virtual proving ground environment;
- the availability in virtual terms of fatigue reference roads and its implementation in a context of finite element discretization.

3. The Choice of the Numerical Approach

The equations of equilibrium governing the non linear dynamic response of a system of finite elements is:

$$M \ddot{U} + C \dot{U} + K U = R \quad (1)$$

where: M , C , K are respectively, the mass, damping and stiffness matrices;
 R , is the external load vector;

U , $\dot{U}$, $\ddot{U}$ are the displacement, velocity and acceleration vectors of the finite element assemblage at time “t”.

The procedures used for the solution of general systems of differential equations can be divided in: direct integration and mode superposition. In the direct integration of the Equation 1 a numerical step-by-step procedure is used. The application of this method is based on two ideas:

- trying to satisfy the Equation 1 only at discrete time intervals “ Δt ”, instead of any time “t”;
- assume the variation of the displacements, velocities and accelerations within each time interval “ Δt ”. Obviously, the choice criteria on these assumption, determines the accuracy, stability and cost of the solution procedure.

Namely, in the solution of such problems, the choice stands between the use of explicit or implicit time integration method.

Regarding the implicit method is important to underline that this method uses the equilibrium condition at time $t+\Delta t$ in order to find the displacement field solution at the same time. The effectiveness of these integration schemes, which are unconditionally stable derives from the fact that to obtain accuracy in the integration, the time step “ Δt ” can be selected without any requirement and in many cases ‘ Δt ’ can be orders of magnitudes larger than the value used in explicit method. A big advantage of step-by-step solution scheme based on this integration method is that it can be used for solving both static and dynamic problems. Regarding contact interaction solution of the tire rolling on the road, based on the implicit integration schema, it is important to underline that this is a very complex solution and the non linear equation solving process is expensive and if the problem is very non linear (the case of tire/road contact definition) it may be difficult to obtain a solution.

On the other side, in the explicit method, known also as central difference method, the displacement equilibrium solution at time “ $t+\Delta t$ ” is based on using the conditions of equilibrium at time “ t ”. Such integration schema does not requires a factorization of the stiffness matrix in the step by step solution which can be carried out on the element level and relatively little high speed storage is required. A second very important consideration in the use of the central difference scheme is that the integration method requires that the time step “ Δt ” have to be smaller than a critical value, Δt_{cr} , which can be calculated from the mass and stiffness properties of the complete element model. An approximation to the stability limit is written as the smallest transit time of a dilatational wave across any of the elements in the mesh:

$$\Delta t_{cr} \approx \frac{L_{min}}{c_d} \quad (2)$$

where L_{min} is the smallest element dimension in the mesh and c_d is the dilatational wave speed in terms of effective Lamé’s constants, λ and $G=2\mu$.

Due to this time step requirement the explicit method is considered to be conditionally stable. If a time step larger than Δt_{cr} the integration is unstable. It means that any error resulting from the numerical integration or round off precision in the computer can affect a lot the analysis results in most cases. Since the total cost of the analysis is approximately inversely proportional to the magnitude of time step, it results that if the time step can be “ m ” times as large, the cost would be reduced by a factor of “ m ”. As it can be seen from Equation 2, the Δt_{cr} is directly proportional to minimum element length and it is very important to find a compromise between total analysis cost reduction and stress accuracy requirements. A big advantage related to commercial numerical codes based on this method is the facility to define and manage the contact problems. It is particularly indicated for numerical simulation of short dynamic phenomena and commonly used in automotive industry for crash analysis where the order of simulation time is milliseconds (80 ms to 120 ms) depending on typology of analysis: front or side crash impact. The negative aspect related to such method is that it is not possible to be applied in the simulation of static phenomena.

The best strategy in combining the simulation of quasi-static and dynamic phenomena is another big issue of this methodology. With reference to the previous comments about numerical integration methods, it can be concluded that:

- it is advantageous to use different operators to integrate the response for different phenomena involved in the transient dynamic maneuver of the car rolling on fatigue reference road. That means that the ideal solution is to perform quasi static analysis (equilibrium conditions prior to the dynamic simulation) using an implicit finite element method and then to use these results as the initial condition for the dynamic analysis, the latter being most efficiently conducted, in terms of tire-road contact interaction, with an explicit finite element method. Abaqus code is used since it offers both implicit and explicit methods, and has the ability to transfer results back and forth in an integrated modality.

- the advantage that explicit solution offers in managing tire-road contact interaction is contrasted by the total simulation time commonly required in durability evaluations. According with Fiat targets, single durability events have to be managed going from 60 sec to 120 sec. In this context the problem of solution accuracy due to the large number of increments and the total cpu time required represent the two main issues influencing the success of such approach. To achieve this result, double precision solver is used, and the stability of the code is tested beyond 2 millions explicit increments. In this long simulation the wheel models have been subjected to a big number of revolutions, which is, as known, very critical for explicit integration. Some further aspects related to these models are: the 2nd order accuracy for element integration, together with a small beta damping giving stabilizing effect and fully integrated elements (added in the Abaqus/Explicit latest releases).

4. The Finite element Tire Model

As the minimum time increment of the explicit solver is governed by tire, the model used to simulate the response of the tire represents a significant challenge, being the main crucial issue in such kind of approach to simulate at the best level the tire/road interaction. With the finite element approach is possible to have the contact evaluation between highly discretized physical bodies instead of using simplified models based on a limited number of degrees of freedom; this is the main new aspect of this methodology.

Historically, the tire has been modeled with either an analytical approach (complex spring-mass-damper system) or with an empirical one, integrated with a multibody technique for the overall system. But all these approaches showed limitations, the primary problem being a continuous need for calibration based on experimental data on real components.

In previous papers ([4], [5]) of the authors, the construction of a reliable tire finite element model of moderate complexity which provides good accuracy and true predictive capability was described; this model runs integrated with a complete vehicle model in a virtual proving ground environment. It was validated over several obstacles using quasi-static, static and dynamic analysis. In the Figure 2 the results of such static and dynamic validation are reported.

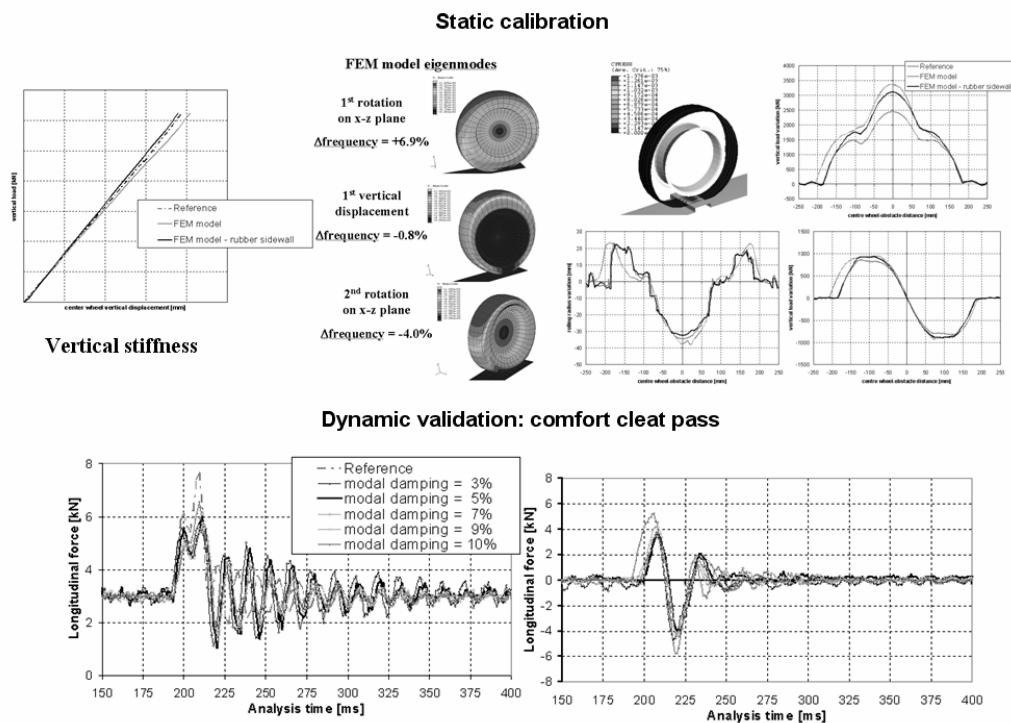


Figure 2. Finite element tire model validation method

Before to the present work, the application of such finite element tire model was tested in two different applications:

- the misuse events and specifically the pot-hole (Figure 3) and comfort obstacles were taken. Both these misuse events are characterized by a time simulation that goes from 300 ms to 600 ms (three to five order greater than common crash events). Double precision solution, mesh size, discretization level and problems related to damping were investigated;

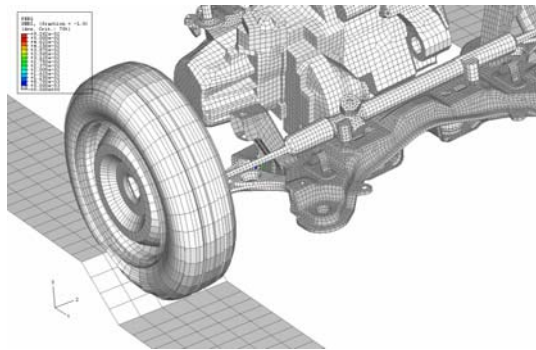


Figure 3. Finite element tire model – pothole simulation

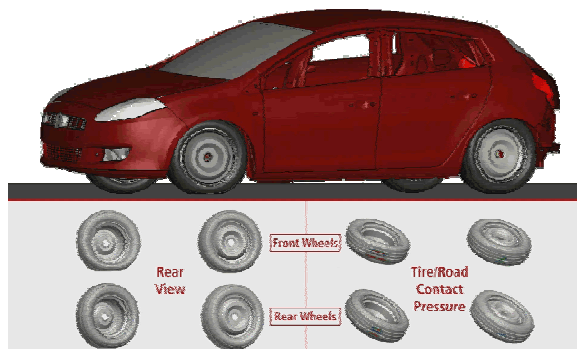


Figure 4. Finite element tire model – hand brake simulation

- an extension of such misuse events were further investigated to simulate the steering with hand brake engaged maneuver; after improvements related to hourglassing control and integration accuracy in tire models, the time simulation was successfully extended to 4 seconds (Figure 4).

In the new context of durability field application the following questions come up: - is this tire model also sufficiently described and accurate for explicit simulation when total simulation time is order of magnitude higher than misuse applications? If yes, - are there particular numerical expedients to be applied in order to make feasible this ambitious goal?

To answer these questions, a new 175/70R14 tire (one of the dimensions in New Fiorino equipment) is generated. Some long time rolling simulations over a flat road with a velocity of 30 km/h beyond a typical durability time of 60 seconds were carried out first. All the modeling parameters were re-investigated (mesh size, element integration order, damping coefficient and integration accuracy order) and, in the end, a partial revolution of modeling technique based on complete integration elements (available in explicit since the latest versions), together with the previous stabilization model issues, was necessary to achieve a sufficient stable time simulated (Figure 5).

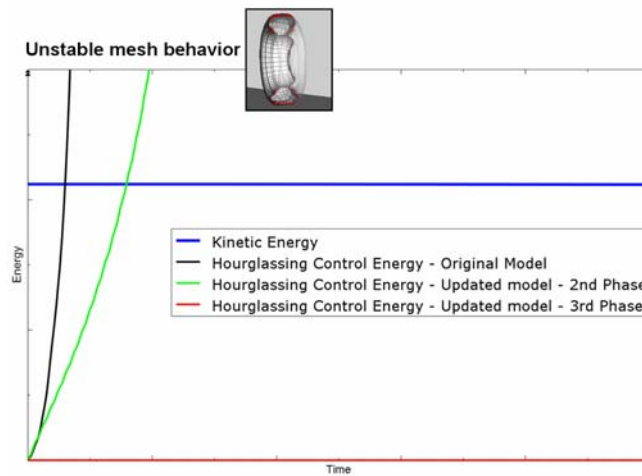


Figure 5. Hourglassing energy check during a rolling simulation after tire model improvements for fatigue typical time simulation.

As a conclusion of such work, the following main issues can be summarized as important in order to extend the application of previous developed tire model to durability field:

- fully integrated elements (lately introduced in Abaqus/Explicit and importable from an Abaqus/Implicit solution) avoid the hourglassing phenomena which is the main obstacle to explicit simulation time extension;
- the second order accuracy integration is necessary in order to control the accuracy for the application where the tire is subjected to a big number of revolution;
- the double precision integration schema is required in order to assure the accuracy of the solution for a number of increments much more greater than 2 millions.

5. The Fatigue Road Digitalisation

All car manufacturers have their own facilities for proving ground tests; these are related to some internal targets, based on vehicle segment and destination. With respect to body durability sizing, FIAT has two main reference tracks:

- “Balocco”, 400m long, mainly used for passenger car testing;
- “Mandria”, 900m long, for commercial vehicle testing.

Obviously these tracks are different in terms of roughness and are associated with different testing conditions and targets. As explained in the Paragraph 2 the possibility of a numerical description of fatigue reference tracks surface represents the starting essential information for the new methodology described in this paper.

The LMS company was chosen to make such kind of measurement: the Miniature Projection Technique is applied. This process assure a better accuracy with respect to the “pure” photogrammetric technique. As an output from the measurement campaign, a grid of vertical coordinates with a 5x5mm resolution in horizontal spacing is obtained. This is the basic information for a virtual graphic representation of track surface. In Figure 6 a graphic output for a small portion of surface track is compared to a photography of the same patch.

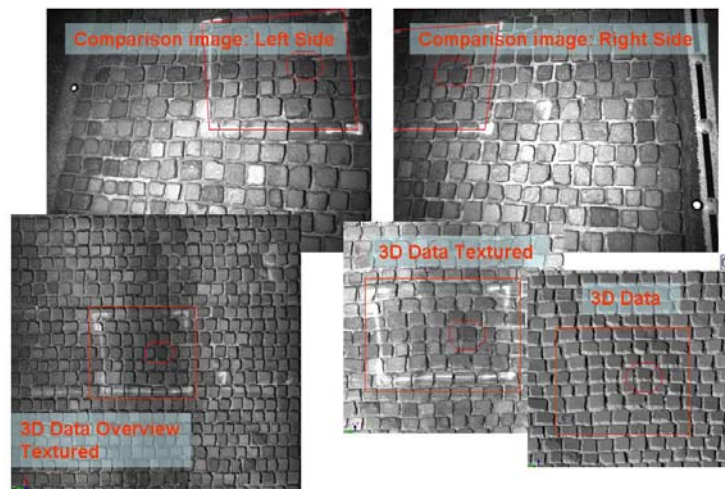


Figure 6. Track surface digitalization compared with the photography.

The road information is implemented in Abaqus/Explicit code as rigid elements associated to a rigid body based on a Python script which translates the grid information in Abaqus node information. A parametric element generation (using the *ELGEN capability available in Abaqus code), fill all this nodes network with Abaqus rigid elements of chosen dimensions (Figure 7). Using this method is possible to select and generate only the elements of the strips involved in

contact with tires, eliminating all the rest of track width and corresponding nodes. For the goal of present work, two strips, each one of 600mm width, located symmetrically with respect to car middle plane are enough to monitor tire/road interaction.

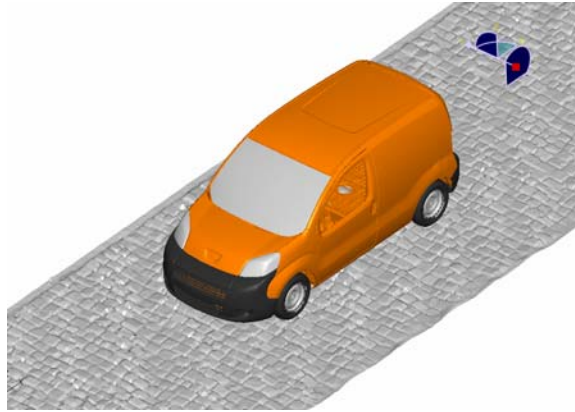


Figure 7. FE model of the track: full version.

6. Vehicle Fatigue Load Prediction

Until now, the critical issues related to the transient dynamic analysis of the vehicle rolling on reference fatigue tracks have been faced. Let's summarize what are to be considered the enabling key-points:

- the choice of the numerical approach able to combine static and dynamic phenomena (based on the integration of Abaqus implicit and explicit solvers);
- the development of a tire finite element model of moderate complexity with true predictive capability and able to run for long time simulations;
- the implementation of the real road profile as a rigid body within the explicit code.

The Fiat Light commercial vehicle, New Fiorino, has been chosen as test case for the present work in order to verify the complete system level results. The full vehicle finite element model used in this analysis was assembled using a mixed approach: only the parts of vehicle whose compliance is important in the dynamic maneuver (tires, anti-roll bar, twistbeam axle) are considered as deformable using a finite element representation. The other parts of the vehicle are modeled using a method based on multibody approach (rigid bodies joined by connector elements with appropriate equivalent stiffness and damping properties). A large use of Abaqus connector elements is done for all other suspension components. In the Figure 8 the vehicle model is shown together with rigid track representation; in the Figure 9 a detail of tire model is reported.

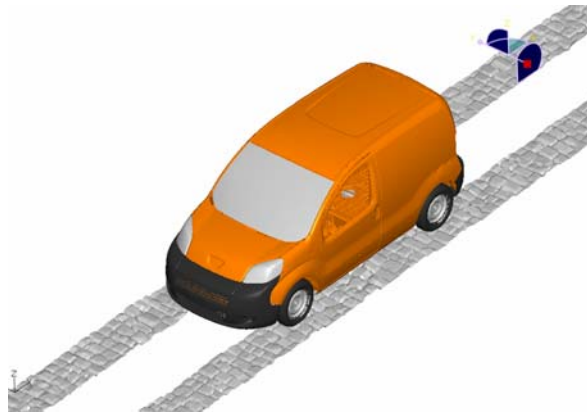


Figure 8. FE model of the track: two strips version.

Concerning the full vehicle transient dynamic procedure, in the Abaqus/Standard code the following phases are simulated using a step by step approach: the tire seating and inflation, performed for the four wheels of the model; the gravity load enforcement and consequent suspensions pre-loading; the contact enforcement between the wheels and the ground; the equilibrium between vehicle and the ground under the gravity field previously applied.

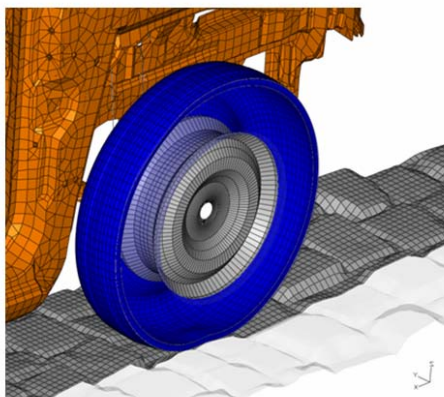


Figure 9. Tire FE model rolling on the track.

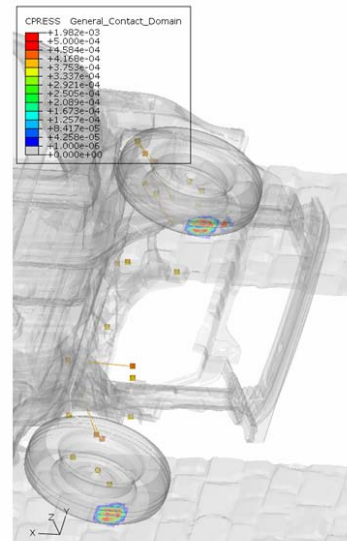


Figure 10. Tire-Road contact pressures

The equilibrium condition previously found is used as starting point for the successive transient dynamic analysis, which consists in reproducing experimental conditions. The road test is divided in two parts: the first one, corresponding to the first 200 m, is performed at a mean velocity of 30-35 km/h; the second one, corresponding to the other 200 m, much more irregular in terms of asperity content, is performed at a mean velocity of 20-25 km/h. In the attempt to reduce the finite element problem size, based on this different running conditions, also the explicit analysis is divided in two separated analyses. The general contact algorithm is applied to manage the contact between tires and ground; this choice - combined with a multiple explicit step approach where reduced contact ground patches are activated or deactivated in contact enforcement depending on vehicle position - reduced drastically the cpu-cost of dynamic simulation. In Figure 10 a contour of tire-road contact pressures is reported. For each one of the two performed analyses the problem size can be summarized as:

- total number of nodes:	1200000
- total number of elements:	1200000
- total number of degrees of freedoms:	3000000

With a time increment of approx. 6 micro-seconds the total number of increments range from 4,500,000 to 6,500,000 for each one of the two explicit analyses.

Commonly, during the experimental road tests the following dynamic parameters are recorded:

- the vertical acceleration of the four wheels, representative of the dynamic response of unsprung masses;
- the vertical acceleration of four body points located in front and rear strut mounts. These parameters are the most important in terms of structural body behavior, because they are representative of vertical loads applied to the body by the suspensions;
- the relative displacements between wheel centers and corresponding body points.

The full car behavior evaluation is carried out by comparing all these dynamic parameters with experimental data. Due to many control variables (running velocity, driver control, exact position of the tire/road contact line and so on) that are difficult to be maintained constant during the single running test execution and more, to be repeated exactly from one test to the other, all durability cycle has to be treated not as deterministic one. In Figure 11 statistically comparison in terms of level crossing of vertical acceleration is reported for two of measurement points (respectively front-right strut mount (a) and wheel center (b)). In Figure 12 the comparison is extended to the power spectral density of the same signals, while in Figure 13 the level crossing of relative displacements of rear-left suspension is reported. In addition, the history time of rear-left tire-wheel vertical load, spring and bumpstop are reported in the Figure 14.

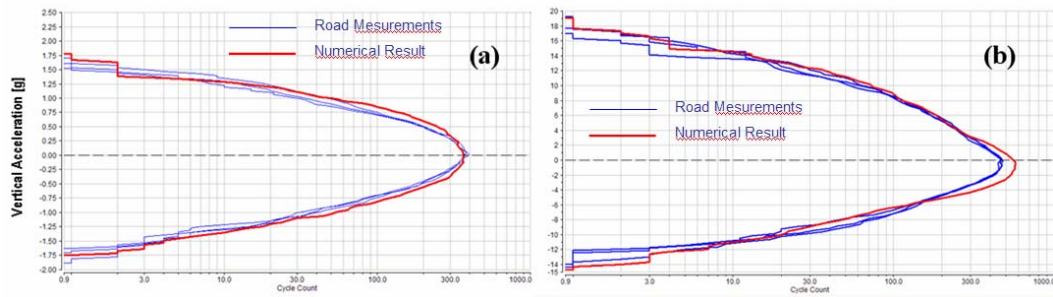


Figure 11. Level Crossing FRONT-RIGHT vertical acceleration: Body (a) Wheel (b).

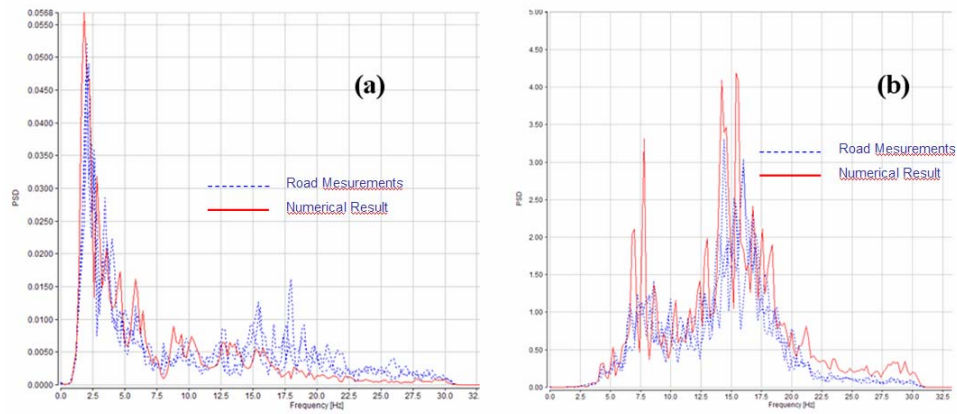


Figure 12. PSD FRONT-RIGHT vertical acceleration: Body (a) Wheel (b).

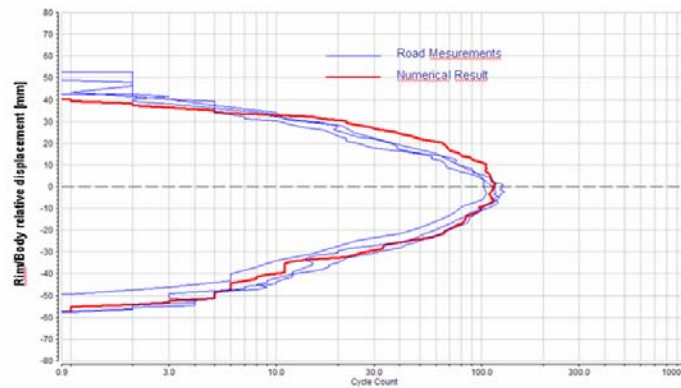


Figure 13. Level Crossing: REAR-LEFT relative displacements.

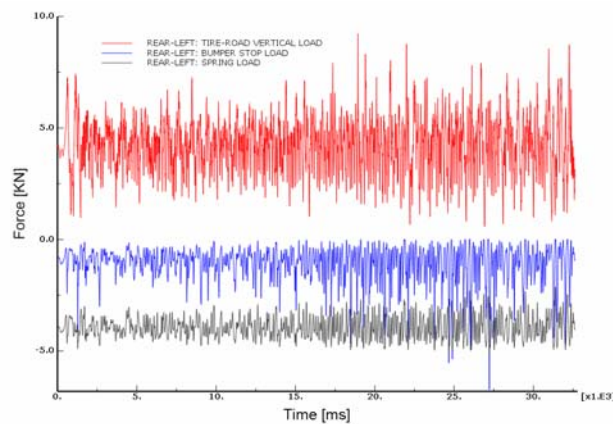


Figure 14. Time History of some representative loads.

From an overview of all these results, the following considerations can be outlined:

- in terms of *level crossing of body vertical acceleration* there is a very good agreement between numerical and experimental data both in terms of total number of events (abscissa of the diagram) and acceleration amplitude;
- the same conclusions can be extended to *wheel vertical acceleration*;
- the *power spectral density* comparison of vertical acceleration, shows a good correlation in terms of energy level and frequency content, in measurement points, either for unsprung masses and vehicle body (Figure 12);
- also in terms of relative displacements, there is a good overlap between experimental and numerical data.

7. Conclusions

An integrated CAE methodology based on finite element tire/road interaction, using Abaqus code, in a fully integrated implicit/explicit environment able to simulate the transient dynamic response of the full car running on representative fatigue loads, has been presented. Assuming as a case study the Fiat New Fiorino light commercial vehicle, the methodology has been successfully applied to predict accurately the time history of loads acting in the attachments points of the suspensions with the car body.

During the methodology development many crucial issues have been faced and solved. First of all the digitalization of reference test road has been carried out and implemented as rigid body in finite element code.

Another important issue is to select the best strategy in order to combine the simulation of quasi-static and dynamic phenomena; the ideal solution is to perform quasi static analysis (equilibrium conditions prior to the dynamic simulation) using an implicit finite element method and then to use these results as initial condition of the dynamic transient analysis, performed using an explicit finite element method for a better management of complex contact interaction problem between tires and ground. Abaqus code is used since it offers both implicit and explicit methods and the possibility to transfer results between them. It is to be mentioned that the advantage offered by explicit solution in managing tire–road contact interaction, is contrasted by the difficulty in keeping numerical stability of the model for long analysis time commonly required in durability evaluations. The problem is overcome by using full integrated elements, double precision and large parallel solution.

As the tire model has the main influence on explicit solver time increment, particular attention has been given to this issue. In particular the previous model has been improved to obtain a reliable tire model in the context of durability applications characterized by long time simulations.

8. References

1. Bakker, E., Pacejka, H.B. and Lidner, L., “A new tire model with an application in vehicle dynamics studies”, SAE paper 890087.,1989
2. Bakker, E., Nyborg, L. and Pacejka, H.B., “Tyre modelling for use in vehicle dynamics studies”, SAE paper 870421, 1987.
3. Bathe, K. J, “Finite Element Procedures in Engineering Analysis”, Prentice-Hall, Inc., Englewood Cliffs, New Jersey 07632, 1982.
4. Duni E., Saponaro R., Monfrino G, Caudano M., Urbinati F., Spinelli M, Antonino P., “Numerical Simulation of Full Vehicle Dynamics behaviour based on the interaction between Abaqus /Standard and Explicit Codes,” Abaqus Users' Conference, june Munich, 2003.
5. Duni E., Saponaro R., Monfrino G, Caudano M., Urbinati F., Spinelli M, Pizzuto A., “An integrated approach for numerical simulation of full vehicle behaviour during transient dynamic manoeuvres on arbitrary obstacles and road surfaces,” IMECE2003 - 41374, ASME Proceedings, november , WASHINGTON 2003.
6. Dixon, J. C. ,“Tires, suspensions and handling”, Cambridge University Press, England,1991.
7. Genta, G., “Meccanica dell’Autoveicolo”, Levrotto & Bella, Italy, 1993.
8. Morelli, A., “Progetto dell’Autoveicolo”, CELID, Italy, 1999.
9. Pacejka, H.B. and Bakker, E.,“The magic formula tyre model”, 1st International Colloquium on Tyre Models for Vehicle Dynamics Analysis, Delft, The Netherlands, October 21-22, 1991, Vehicle System Dynamics, **21** supplement, pp. 1-18, 1993.
10. Reimpell, J., and Stoll, H., “Automotive Chassis – Engineering Principles”, Arnold, England, 1996.