NUMERICAL SIMULATION OF FULL VEHICLE DYNAMIC BEHAVIOUR BASED ON THE INTERACTION BETWEEN ABAQUS/STANDARD AND EXPLICIT CODES

E. Duni, G. Monfrino, R. Saponaro, M. Caudano and F. Urbinati
FIAT Research Center, Orbassano (TO), Italy
S. Marco, P. Antonino
FIAT Auto Spa, Torino, Italy

Abstract: This work describes a numerical methodology based on the finite element method used for the transient dynamic simulation of the full vehicle rolling on different kind of obstacles. Some issues related to the tire finite element model development and its validation, by numerical-experimental comparison, have been discussed. The strategy to combine the static simulations such as the tire inflating, the vehicle weight application and suspension pre loading, with transient dynamic analysis of the car rolling over the obstacle has been chosen. The methodology, based on integration of Abaqus Implicit and Explicit codes, has been successfully applied for the dynamic simulation of Fiat Punto car passing over comfort and pothole obstacle.

1. Introduction

Numerical simulations are moving more and more from evaluation of the behaviour of the single component to the full system calculation. In this new scenario, analysts try to extend the virtual simulation to the problems considered in the past, for procedural and size limits, numerically impossible to be evaluated and faced only by experimental approach. Some of these problems are related to the non-linear dynamic transient behaviour of full vehicle FEM models passing over the obstacles. One critical point in facing such problems is related to the choice of the best way to combine the simulation of quasi-static and dynamic phenomena. The ideal numerical solution of this problem is to perform the quasi-static analysis (equilibrium prior to the dynamic simulation) using an implicit code and use the implicit results as initial condition of the dynamic analysis performed with an explicit code. As Abaqus code has both implicit and explicit solvers, with the possibility to transfer the information between them by the *IMPORT capability, applied in both directions, from implicit to explicit and vice-versa, it has been chosen, in the present study, as reference code for non-linear transient analysis of the car passing over the obstacle.

The most critical issue in the transient dynamic simulation based on the finite element method is related to the development of a reliable tire finite element model. Actually numerical tire models, 2D and 3D, with reduced d.o.f., integrated in a multibody approach, are commonly used by car
manufacturers for the analysis of vibrations. The major limitation of these models is that they need a continuous calibration based on data coming from the experimental tests carried out on the real component. The principal purpose of tire finite element model developed in the present work is to describe the dynamic tire response in general conditions. A finite element model for a standard 165/65R14 passenger car tire with limited d.o.f. was developed, based on information in terms of weight, stiffness, eigenmodes, usually available in the early development stage of a new car. A calibration of the model in terms of material proprieties and component thickness has been carried out for first; then, two kinds of validations have been performed for a tire rolling with constrained axle height over a rectangular obstacle with the dimensions 100x25 mm: quasi-static and dynamic validation. In the numerical/experimental comparison of the results in terms of vertical and longitudinal load, the spin acceleration and the rolling radius variations a good agreement can be observed. As case study in the present work a FIAT car of the “B” segment has been considered. The tire models, previously developed and validated, are combined with suspensions and car model. An implicit analysis has been carried out for first, in order to find the equilibrium condition of the car subjected to gravity load in presence of the road. Then, an Abaqus/Explicit analysis has been done as a restart from the implicit one.

In order to overcome the difficulties related to the problem dimensions, two different full vehicle models have been carried out in this phase:

- the semi-flexible model created by importing from implicit analysis as deformable elements, with their stress and strain information coming from the static equilibrium condition, only the components (such as tires, subframes, strut-houses) whose behaviour is particularly interesting in these kind of manoeuvres, while all the other components are imported as rigid bodies;

- the complete model where all components are imported from previous implicit analysis as deformable elements.

The explicit analysis has been carried out for two different obstacle shapes: rectangular (100x25 mm) and pothole obstacle. Two different velocity conditions are respectively applied: 50 km/h and 40 km/h. The dynamic vehicle response in terms of vertical and longitudinal forces, displacements at wheel centre and strut house has been carried out. For the rectangular obstacle, a comparison between the finite element and multibody results is reported. The stress and strain contour distributions give a complete view on real structural behaviour of critical components.

### 2- The Choice of the Numerical Approach

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

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

where: \( M, C, K \) are respectively, the mass, damping and stiffness matrices;

\( \mathbf{R} \), is the external load vector;
\( \dot{U}, \dot{\dot{U}}, \ddot{U} \) 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 term “direct” means that prior to the numerical integration no transformation of the equations into a different form is carried out. The application of this method is based on two ideas:

- trying to satisfy the Equation 1 only at discrete time intervals “\( \Delta t \)”, instead of any time “t”;
- assume the variation of the displacements, velocities and accelerations within each time interval “\( \Delta t \)”. 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.

2.1. Explicit Method

In this 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 require a factorisation 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 “\( \Delta t \)” is smaller than a critical value, \( \Delta t_{cr} \), which can be calculated from the mass and stiffness properties of the complete element assemblage. More specifically, in order to obtain a valid solution (for the case with no damping):

\[
\Delta t \leq \Delta t_{cr} = \frac{2}{\omega_{max}}
\]

where \( \omega_{max} \) is the highest frequency of the finite element assemblage with “n” degrees of freedom. 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:

\[
\Delta t_{cr} = \frac{L_{min}}{c_d}
\]

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, \( \lambda \) and \( G=2\mu \).

All integration schemes that require the use of a time step \( \Delta t \) smaller than a critical time step \( \Delta t_{cr} \), such as the central difference method, are considered to be conditionally stable. If is used a time step larger than \( \Delta t_{cr} \) the integration is unstable. It means that any errors resulting from the numerical integration or round off in the computer grow and makes the response calculations
worthless 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 3, the $\Delta_{cr}$ 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. It is particularly indicated for numerical simulation of short dynamic phenomena and commonly used in automotive industry for crash analysis.

A big advantage related to commercial numerical codes based on this method is the facility to define and manage the contact problems. In the Abaqus/Explicit code for example, two different contact algorithms are available: the kinematic and the penalty. In the case of kinematic enforcement of contact conditions, in each increment of the analysis the solver first advances the kinematic state of the model into a predicted configuration without considering the contact conditions. Then the solver determines which slave nodes in the predicted configuration penetrate the master surfaces. The depth of each slave node’s penetration, the mass associated with it, and the time increment are used to calculate the resisting force required to oppose the penetration.

The penalty contact algorithm, instead, results in less stringent enforcement of contact constraints than the kinematic contact algorithm. In this case, the algorithm searches for slave node penetrations in the current configuration. Contact forces that are a function of the penetration distance are applied to the slave nodes to oppose the penetration, while equal and opposite forces act on the nodes of master faces being penetrated. The “spring” stiffness that relates the contact force to the penetration distance is chosen automatically for the hard penalty contact, such that the effect on the time increment is minimal yet the allowed penetration is not significant in most analysis.

A negative issue related to such method is that it is not possible to be applied in the simulation of static phenomena.

### 2.2. Implicit Method

This method uses the equilibrium conditions at time $t+\Delta t$ in order to find the displacement field solution at the same time. Using standard finite difference expressions to approximate the acceleration and velocity components in terms of displacement components, the solution for displacement field at time $t+\Delta t$, $^{t+\Delta t}U$, can be obtained from the following equilibrium equation:

$$
M \, {^{t+\Delta t} \ddot{U}} + C \, {^{t+\Delta t} \dot{U}} + K \, {^{t+\Delta t} U} = {^{t+\Delta t} R}.
$$

(4)

The effectiveness of these integration schemes, which are unconditionally stable derives from the fact that to obtain accuracy in the integration, the time step “$\Delta t$” can be selected without any requirement (such as defined by Equation 3) and in many cases “$\Delta t$” can be orders of magnitudes larger than the $\Delta_{cr}$ defined by Equation 3.

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, whereas the central difference method (explicit) solution could not be used if mass and damping effect are neglected.
Regarding contact definition in the Abaqus/Standard solver, based on the implicit integration schema, a pure master-slave contact algorithm is used: nodes on one surface (the slave) cannot penetrate the segments that make up the other surface (the master). Two different formulations for the contact can be used: small-sliding and finite-sliding. When using the small-sliding formulation, the relationship between the slave nodes and the master surface at the beginning of the simulation is established and maintained throughout the analysis. The finite sliding contact formulation requires to determine constantly which part of the master surface is in contact with each slave node. This is a very complex calculation, especially if both the contact bodies are deformable. At each slave node in the contact a constraint is applied. The nonlinear equation solving process, in the solvers based on this integration method, is expensive and if the equations are very nonlinear, it may be difficult to obtain a solution.

Trying to find out the right method to perform the transient dynamic analysis, referring to the previous comments about numerical integration methods, it may be advantageous to use different operators to integrate the response for different phenomena involved in the transient mission. The use of a combination of operators for the integration of static and dynamic responses raises the questions of which operators are to be chosen and how to couple them in order to take advantage from each one. As Abaqus code has both implicit and explicit solvers, with the possibility to transfer information between them by the *IMPORT capability, applied in both directions, it has been chosen as reference code for non-linear transient analysis of the car passing over the obstacle.

3 - Tire modelling

Facing numerical dynamic problems regarding a vehicle passing over an obstacle using a finite elements approach which includes the road profile, the most critical issue is related to the construction of an appropriate tire finite element model, able to transmit correctly transient dynamic loads to the hub. Tire manufacturers commonly use, in the development stage of their products, detailed FEM models that have an excessive computational cost in a full vehicle environment. It is therefore necessary to develop a general purpose FEM model with a restricted number of d.o.f. (in any case, much greater than a multibody model) having as a principal goal to describe the dynamic tire response in general conditions, and not to be used as a tire designing tool.

3.1. Tire Multibody Model

One of the simplest multibody tire models is the 2D rigid ring model developed by Pirelli (Figure 1). It is used for the harshness analysis and consists of four components. The tire tread-band is modelled as an infinitely rigid ring and the tire sidewalls with pressurised air are modelled as springs and dampers. The contact model consists of a vertical residual stiffness, describing the large deformations in the contact patch, and a slip model. The fourth component is the rim which is modelled as a perfectly rigid body. Some model parameters (stiffnesses, dampings, inertias) are directly measured. The others are identified using measures of tire, natural frequencies and the modal dampings. The rigid ring approach is able to represent only those motions of the tire, where the shape of the tread-band/ring remains circular i.e. the flexible belt modes can be neglected.
This model has a single point tire-road interface. For relatively large wavelengths the geometry of the road surface can serve directly as input to the model while for short wavelengths an effective road description that takes into account the enveloping properties of the tire must be used. The basic assumption is that the contact patch mainly reacts quasi-statically. Consequently, the effective excitation of the tire can be assessed from the quasi-static enveloping properties of the tire. The effective road surface corresponding to an obstacle is obtained from experiments with constant axle height. In any case, this model needs, for different obstacle shapes and tires, both quasi-static and dynamic validation.

Other models (like the Swift Tire by TNO) allow the evaluation of both harshness and handling vehicle performances, combining a 3D rigid ring approach (lateral deformation of the sidewall included), with the quasi-static enveloping properties of the tire and a description of combined slip forces through synthetic formulas (the so called Magic Formulas).

In other approaches the rigid ring becomes a flexible ring and the combined slip forces are obtained from a contact patch brush model.

3.2. Tire Finite Element Model

The development of the general purpose tire finite elements model with limited number of d.o.f. is based on 165/65 dimensions tire for two reasons:

- is mounted on the FIAT Punto car, chosen as test case for the present study;
- is a standard passenger car tire and a lot of information, both experimental and multibody simulation, is available.

The tire model, with section reported in Figure 2, consists of two parts: the plies model and the rubber model. The plies are modelled using membrane elements arranged on the section perimeter according with their characteristics; in particular the material stiffness given to these elements is obtained, using the classical lamination theory, from the composed stiffness of the plies placed one upon another. The rubber parts of the tread and seat parts are modelled using solid continuum elements made of isotropic material.

As it can be seen in Figure 2, two versions of the tire model were developed, in order to study the differences in the tire behaviour due to a solid model of the sidewall rubber. Furthermore a third model was developed starting from the second version of Figure 2 to evaluate a circumferential mesh refinement influence in the near-obstacle contact patch.

The tire finite element models displayed in Figure 2 are:

Model 1, only membrane elements with equivalent material and thickness properties on the side;

Model 2, the same as model 1 with the addition of solid elements for modelling the rubber part on the side;

Model 3, different from model 2 for the presence of a finer circumferential mesh in the near obstacle contact zone.
3.2.1. Static and Modal Validation

This process includes the calibration of the rubber behaviour and of the membrane elements thickness, based on the following general data, usually available in the early development stage of a new car: weight, vertical and lateral stiffness and first in plane eigenmodes.

In order to perform the static validation of the model, Abaqus/Standard code has been used and a step-by-step procedure has been carried out. The numerical procedure can be summarised in four points:

**step 1** tire seating on the rim, performed moving the nodes interfaced to the rim to the mounted position; the shape of the rest of the tire follows this movement.

**step 2** inflating, performed applying a distributed load to the internal surface of the plies; it is possible to add the air behaviour simulation using an appropriate modelling strategy.

**step 3** loading, composed of an initial enforcement of contact interaction at ground and afterward the effective loading of the wheel at test conditions.

**step 4** the loaded configuration of the wheel, previously obtained, is the condition used to perform the modal analysis; to reproduce the test conditions the rim is completely fixed at ground except for the d.o.f. of rotation around the wheel axis.

A comparison between experimental and numerical results carried out for the tire model 1 and 2 as previously described, in terms of radial stiffness and first three in plan eigenmodes, is reported in the Figure 3. Analysing these results, the following considerations can be drawn:

- independently from sidewall modelling strategy, there is a good agreement between experimental and numerical values in terms of tire radial stiffness;

- the numerical values of the tire three first in plane eigenvalues overlap quite well with respective experimental one; indeed, the first numerical eigenvalue is 6.9 % higher than experimental one, the 2nd numerical eigenvalue is 0.8 % lower than the experimental while the 3rd numerical value is 4% lower than the respective experimental value.

3.2.2. Quasi-Static Validation

The test facility has a long flat steel road surface with the tire rolling with the wheel centre fixed in longitudinal, vertical and lateral directions allowing the experiments to be done with constrained axle height. Different obstacle shapes with different heights can be mounted on the road surface. The effective road surface velocity is very low in order not excite tire dynamics and to give the air spring system time to settle. For the tire under consideration in the present work, a rectangular obstacle with 25x100mm dimensions has been examined.

From the numerical point of view, a static analysis has been carried out starting from the last increment of step 3 analysis that represent the equilibrium conditions of the tire in the presence of a 312 kg constant vertical load. The longitudinal displacement of road profile reference node is imposed as constraints while all the other d.o.f. are constrained to the ground.
The validation is carried out in terms of vertical and longitudinal load and rolling radius variations with respect to the longitudinal distance between wheel and obstacle centre (the zero distance means that the wheel centre coincides with the obstacle centre). A comparison of numerical and experimental results is reported in Figure 4, for model 1 and 2 described before. From an overview of all these results, the following considerations can be drawn:

- in terms of rolling radius and longitudinal load variations, there is a good agreement between numerical and experimental results, independently from the finite element used;
- there are some differences between numerical and measured results in terms of peak vertical load variation;
- adding the solid model of rubber on the side, the agreement between numerical and experimental results in vertical load increases.

3.2.3. Dynamic Validation

A specific test stand can be used. It is mounted on top of a 2.5 m diameter drum and is designed to measure tire dynamic behaviour in the frequency range 0-100 Hz. The vertical height of the wheel axle can be adjusted to load the tire while during the experiments the wheel axle’s vertical, longitudinal and lateral motions are constrained. The reaction forces of the tire, both longitudinal and vertical, are measured.

The numeric simulation of tire rolling dynamic manoeuvre has been carried out based on Abaqus/Explicit code. In this case, a flat road profile provided with a rectangular obstacle of 25x100 dimensions is used instead of the bench test drum. The initial conditions for this explicit analysis have been imported (using Import capability of Abaqus code) from the last increment of static loading (step 3) previously carried out in Abaqus/Standard (implicit solver). On the reference node of road profile, modelled as a rigid body, a boundary condition of type velocity in the longitudinal direction has been imposed while all the other d.o.f. are constrained to the ground. The simulations were carried out at the velocity 40 km/h with the axle height corresponding to initial vertical load of 312kg.

Different numerical simulations have been carried out in order to investigate the influence of following parameters on dynamic behaviour of the tire rolling on the obstacle: the finite element dimensions, material damping and contact properties.

Regarding the damping, a general damping effect provided by Rayleigh damping has been introduced. In this case, the fraction of critical damping, $\xi_i$, for a particular frequency of vibration, $\omega_i$, can be defined by the following expression:

$$\xi_i = \frac{\alpha_R}{\omega_i} + \frac{\beta_R M_i}{2}$$

where $\alpha_R$ factor introduces a mass proportional damping force and $\beta_R$ factor introduces damping proportional to the elastic material stiffness. Generally, the mass proportional damping is used to
damp out the low frequency response and stiffness proportional damping is used to damp out the high modes.

For the chosen vertical load condition and longitudinal velocity, the vertical force ($\Delta F_z$), the longitudinal force ($\Delta F_x$) and the spin acceleration ($\Delta \Omega$) variations are plotted versus time in the Figure 5. In the same figures the multibody simulation results are reported too. In the Figure 6 the contact pressure distribution between tire and road before, during and after the obstacle are reported.

From a global overview of these results, the following considerations can be outlined:

- as shown in Figure 7, for damping based only on mass proportional damping, with $\alpha_R$ factor calibrated for $f^1$ in plane frequency and defined for different fraction of critical damping ranging from 2% up to 7%, no significant change in dynamic tire response is verified. For $\xi > 9\%$ instead, the tire response, especially in terms of longitudinal force amplitude is different with respect to the responses corresponding to smaller values of critical damping.

- adding the stiffness proportional damping, with $\beta_R$ factor calibrated for the 3rd frequency $f = 87$ Hz and $\xi = 5\%$, no significant change in the tire dynamic response is verified even if the total c.p.u. time is increased dramatically.

- independently from the finite element model used, there is a good agreement between multibody and the finite element results in terms of spin acceleration ($\Delta \Omega$) and longitudinal force ($\Delta F_x$) variations.

- regarding the vertical force variation ($\Delta F_z$) comparison:

  - there is the same shape of vertical force variation in multibody and finite element simulation; indeed, the two peak shape of vertical force variation of the tire overcoming the obstacle, is present in both curves reported in the Figure 5;

  - the vertical force peak value obtained by the finite element simulation, in the hypothesis of simplified finite element model, results to be 14% lower than the one obtained by multibody simulation. This value decreases in 8% by introducing the solid elements for sidewall rubber modelling.

  - in the hypothesis of model 3 (rubber solid elements on the side and fine mesh in the contact zone) the response obtained by the finite element simulation is almost equal to the multibody simulation (see Figure 5).

4. Transient dynamic analysis of the vehicle passing over the obstacle

Until now, the two most important aspects related to the transient dynamic analysis of the vehicle in the presence of road profile have been faced: developing of a tire finite element model of
reduced d.o.f. and the choice of the approach to apply in order to combine static and dynamic phenomena, based on the integration of ABAQUS implicit and explicit solvers.

Regarding the full vehicle transient dynamic procedure, in the ABAQUS/Standard code the following phases are simulated using a step by step approach:

\textit{step 1} the tire seating and inflation, performed for the four wheels of the model as described in the single tire model;

\textit{step 2} the gravity load enforcement and consequent suspensions pre-loading; loads applied to the vehicle corresponds to the standard loading configuration prescribed for the dynamic missions simulated (including passengers and baggage weight);

\textit{step 3} the contact enforcement between the wheels and the ground; performed constraining to ground the wheel centres while the road reference nodes raises in vertical direction;

\textit{step 4} the equilibrium between vehicle and the ground under the gravity field applied on the previous step, obtained removing all boundary conditions applied on vehicle parts.

During the methodology development phase of the explicit analysis, in order to overcome the difficulties related to the problem dimensions, two different vehicle models have been carried out:

- the semi-flexible model (Figure 8) created by importing from step 4 of implicit analysis, as deformable elements, with their stress and strain information corresponding to the condition of static equilibrium, the tires, the subframes, the strut-houses whose behaviour is of particular interest in these kind of manoeuvres, while all the other components are imported as rigid bodies;

- the complete model (Figure 9) where all components are imported from previous implicit analysis as deformable elements.

\subsection*{4.1. The Comfort Obstacle}

The comfort obstacle has a rectangular shape with 100x25 mm dimensions as reported in Figure 10. The experimental test is carried out with the vehicle running with velocity of 50 km/h. The Fiat PUNTO car has been considered as test case in the present study.

From the numerical point of view, during explicit analysis the following initial conditions have been imposed to the model:

- the initial velocity $V=13.9\text{mm/ms}$, in “x” car direction applied to both left and right side of the road;

- an angular velocity field $\omega=0.048\text{rad/ms}$ around the respective centre is imposed for every wheel;

The following load and boundary conditions are applied:
- gravity load applied to all the vehicle model, in order to guarantee the continuity with the static equilibrium;

- the nodes of the road are constrained on all their d.o.f. (except the “x” direction);

- a boundary of type velocity has been applied to the road profile reference nodes in the car “x” direction.

Due to the large problem dimensions the complete model analysis is limited to 60 ms necessary for the front wheel to overcome the obstacle, while for the semi-flexible model analysis a total of 400 ms dynamic analysis has been carried out.

The full car behaviour evaluation is carried out in terms of wheel centre and strut mount vertical and longitudinal load variation plotted versus the longitudinal distance between wheel and the obstacle centre (the zero distance means that the wheel centre coincides with the obstacle centre). A comparison of the finite element results, obtained from both semi-flexible and complete models, with the multibody one is reported in the Figure 11. In the Figure 12 the mises stress distribution are reported for strut house and subframe components in the correspondence of the obstacle centre. From an overview of all these results, the following considerations can be outlined:

- in terms of wheel centre longitudinal force, the finite element simulation results correspond rather well with multibody one. Leaving the obstacle, a higher than multibody amplitude vibration can be observed in the finite element simulation, probably due to different ways to introduce damping effects.

- in terms of wheel centre vertical force, while the two peak shape force distribution in the first phase of obstacle is verified in both analysis (multibody and finite element), the maximum peak value obtained from multibody simulation results to be 15% higher than one obtained from finite element analysis. It is important to underline that there are some small differences in static load distribution among the multibody and the finite element model as confirmed from different values of static loads before the obstacle.

- the mises stress contour (Figure 12) corresponding to the most critical instance of whole dynamic manoeuvre shows that the maximum mises stress value is smaller than material yield stress (a high strength material is used with $\sigma_y=320$Mpa and $\sigma_R=360$Mpa). Consequently, no plastic deformation involves the strut mount material during this dynamic manoeuvres.

4.2. The Pothole Obstacle

The pothole obstacle, which has a length of 2700mm and a depth/height of 114mm, as reported in the Figure 13, is commonly used from car manufacturers for the structural evaluation of suspension components, subframe and strut house attachments. The experimental test is carried out with a velocity of 40km/h and only the left side wheels pass over the obstacle, while the right side wheels rolls on a flat road. The same finite element model representative of Fiat Punto car has been used as test case for this dynamic manoeuvre. The tire finite element model is the same as one previously used with the exception of the fine mesh size, which is extended in order to cover the ramp. The contact between tire and rim is also introduced.
The boundary and load conditions imposed to the explicit dynamic analysis are the following:

- the initial velocity $V=11.11\, \text{mm/ms}$, in “x” car direction applied to both left and right side of the road;

- an angular velocity field $\omega=0.036\, \text{rad/ms}$ around the respective centre is imposed for every wheel;

The following load and boundary conditions are applied:

- the gravity load applied to all the vehicle model, in order to guarantee the continuity with the static equilibrium;

- the nodes of the road are constrained on all their d.o.f. (except the “x” direction);

- a boundary of type velocity has been applied to the road profile reference nodes in the car “x” direction.

In the Figure 14 a plot of the car rolling over the pothole obstacle is reported with a deformed shape of the tire in correspondence of the “ramp”. The dynamic behaviour of the car rolling over this obstacle is evaluated in terms of:

- the vertical displacement of the left wheel centre with respect to the longitudinal distance between wheel and obstacle ramp (Figure 15);

- longitudinal and vertical load variation, in correspondence of left wheel centre and strut mount, with respect to the longitudinal distance between wheel and obstacle ramp (Figure 15);

- equivalent mises stress and plastic equivalent deformation contours (Figure 16).

Analysing all the results reported in these figures, the following considerations can be pointed out:

- the peak value of both longitudinal and vertical left wheel centre load is quite the same and equal to 24000N; in terms of vertical load that means an amplification of the load due to the dynamic manoeuvre which is 6 times the static load applied to the single front wheel

- due to the wheel acceleration and energy absorbed by shock absorber, the peak value of strut mount vertical load is 10500N dramatically reduced with respect to the load passing through wheel centre.

- the mises stress contour shows that the stress level present in the strut mount is greater than material yield stress and consequently, plastic deformations are present in the component (Figure 16).
5. CONCLUSIONS

An integrated CAE methodology, based on integration of Abaqus Implicit and Explicit codes, able to evaluate the transient dynamic response of the full vehicle rolling on the road in the presence of obstacle, has been presented. Assuming as a case study the Fiat Punto car, the methodology has been successfully applied allowing to predict accurately the loads entering in the structure and the consequent damage caused to it.

The most critical point of the methodology regards the development of the tire finite element model, of limited degrees of freedom, which doesn’t require a continuous calibration with experimental data like multibody tire models. A sensitivity analysis of mesh size, modelling technique adopted and damping factor on the tire response rolling on the obstacle has been carried out. The tire validation process is composed of a static, quasi-static and dynamic evaluation where the finite element results, expressed in terms of vertical and longitudinal load and rolling radius variations, have been compared with experimental and multibody data.

Another important issue encountered regards the best way to perform the quasi static (tire inflating, vehicle weight application, suspension pre loading) simulations needed in the overall analysis to find out the equilibrium between vehicle and ground prior to the transient dynamic simulation. The strategy chosen in the present work, applied successfully for dynamic simulation of the Fiat Punto car, completed with tires (previously validated) which rolls on two different obstacles, is based on performing the static simulations in Abaqus/Implicit code (a step by step approach has been implemented) and importing the final stress/strain information into Abaqus/Explicit code where the dynamic simulation takes place. Even if there are actually in Abaqus code some limitations regarding the element types that can be imported from Implicit to Explicit code and vice-versa, the present work has demonstrated that such kind of approach can be successfully applied in automotive industry for virtual evaluation of the components particularly stressed during obstacles dynamic manoeuvres.
6. References


7. Acknowledgements

The authors would like to thank FIAT-Auto Company, that financially supported this work.
Figure 1. 2D multibody tyre model

Figure 2. FEM tyre model without (left) and with (right) solid elements for sidewall.

Figure 3. FEM tyre model results: vertical static loading; first three in-plane(x-z) eigenmodes
Figure 4. FEM tyre model results: quasi-static validation

Figure 5. FEM tyre model results: dynamic response

Figure 6. FEM tyre model: contact pressure distribution
Figure 7. Tyre internal damping sensitivity analysis

Figure 8. Semi-flexible vehicle model

Figure 9. Complete vehicle model

Figure 10. Comfort obstacle analysis
Figure 11. Comfort obstacle FEM and multibody comparison: wheel centre (left) and strut mount (right) loads

Figure 12. Stress distribution on strut mount and subframe
Figure 13. Pothole analysis

Figure 14. FEM vehicle model passing over obstacle

Figure 15. Pothole analysis results: wheel centre vertical displacement, vertical and longitudinal load

Figure 16. Pothole analysis results: mises stress and plastic equivalent strain on strut mount