IDIADA's Virtual Proving Ground (VPG) for durability analysis

Jordi Arbiol, Xavier Armengol, Jose Antonio Muñoz, Inés Lama, and Enric Aramburu

Applus IDIADA

Abstract: *It is usual that the loads applied on structural elements are in fluctuating situations. Sometimes the component can break under its ultimate or its yield strength. This breakage situation is due to the fluctuating loads that have been applied over long periods. For this purpose this breakage type is called "fatigue breaking". Fatigue breaking is a consequence of crack growth. When no prototypes are available, it is necessary to calculate the life prediction as soon as possible. Simulation through the VPG is increasingly being used for life prediction. The present work shows a methodology that uses a finite element model (FEM) that runs on a proving ground. In this case, no MBS (multibody software) model is used to obtain the loads on the structure. Because the whole simulation of the finite element model needs a lot of time, a co-simulation is carried out. An implicit FE solver is used for problems in which the response is moderately nonlinear. This implicit method is computationally expensive. On the other hand, an explicit FE solver is ideal for modelling highly nonlinear problems such as changes in contact surfaces. This explicit method is relatively inexpensive. The ideal situation is to use the correct solver where it will be needed. Components with moderate nonlinear behaviour (Body, Suspension) will be solved using the implicit method. Explicit method will be used to model the impacts of the tyre against the Virtual Proving Ground. Fundamental parts of this methodology are the digitalization of IDIADA's proving ground, its implementation as a rigid body and co-simulation.*

Keywords: *VPG (Virtual Proving Ground), Fatigue, Durability, Connectors, Co-simulation.*

1. Introduction

The Virtual Proving Ground becomes the most accurate methodology to estimate suspension component durability and is, also, useful to know the component loads on the finite element model.

Several processes are used for fatigue life prediction. Some of them come from tests and others come from virtual inputs. Some of these methods are:

- Equivalent quasi-static
- Unitary responses
- VPG using MBS
- VPG using FEM

1.1. Equivalent quasi-static

This methodology is a simple one, but it is effective in the early stages of the vehicle's development phase. It permits several loops with less CPU time consuming. The base of this process consists of applying quasi-static inputs.
These inputs are very easy ones. It is necessary to define different fatigue cycles. Each of them comes from two different states:

- turn left & turn right
- braking & accelerating
- vertical oscillations

Once they are defined, a fatigue postprocess is performed. The quasi-static method does not have to take into account the dynamic effects. For this reason it is required to perform a previous modal analysis to assure that the frequency of the components will not be excited by the inputs. Figure 1 shows the procedure to be followed.

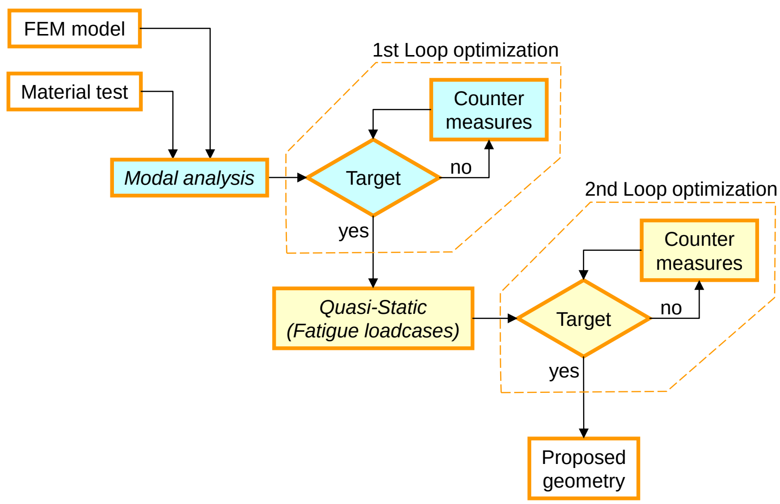

Figure 1. Procedure in quasi-static load cases

1.2. Unitary responses

If there is the information of the forces acting over time on a component, it is possible to obtain the time history of stresses by a linear superposition of the base unit response.
The base of unitary response is obtained applying unitary forces (1N) and moments (1Nmm) on each direction separately. Figure 2 shows the inputs for the unitary responses on a connection point for x, y and z directions.

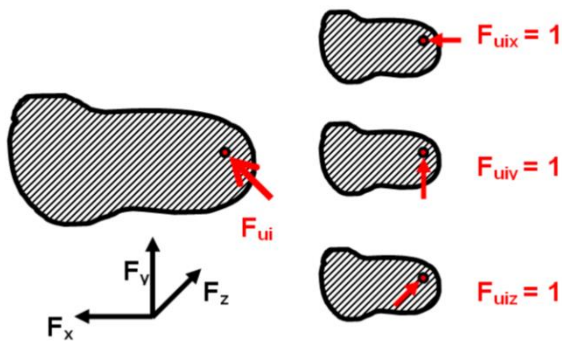

Figure 2. Unitary loads on a connection point

To know stress levels on the component, it is necessary to know the stresses for a unitary load. By multiplying these values by the force of the test at each time step, a stress state can be obtained. A linear superposition of the unitary responses is performed:

$$[\sigma]_t = \sum_{i=1}^{r} \left\{ \sum_{j=1}^{6} \left([\sigma_{ij}^{u}].[F_{ij}]_t \right) \right\}$$

where,

σ_{ij}^{u} is the stress unitary response on the *i* hardpoint due to the unitary force that acts on the *j* direction

$[F_{ij}]_t$ is the *j* component force of the force acting on the *i* hardpoint at instant *t*

r is the total number of hardpoints

1.3. VPG using MBS

This method has less limitation than the previous one, because all of the modes of the components are taken into account.
Running an MBS vehicle model on a digitalized track, it is possible to obtain the contribution of each mode at each time step on the flexile bodies and a modal superposition can be performed:

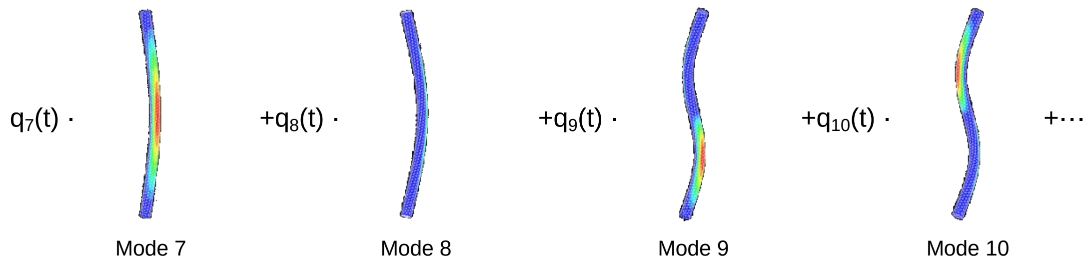

Figure 3. Superposition of modes

The stresses come from the simultaneous application of all of the modes, at time step *t*, and it is calculated (supposing elastic behaviour) as follows:

$$[\sigma]_t = \sum_{i=1}^{r} \left([\sigma_i^u] . q_{i_t} \right)$$

where,

σ_i^u is the stress unitary response for the *i* mode

q_{i_t} is the modal contribution of mode *i* at instant *t*

r is the total number of modes considered in the base

1.4. VPG using FEM

For simulations of the complete vehicle, the length of mesh elements may have an important variation range. Currently, the tendency is to develop models with a very large number of elements with a short length.

For the explicit method, the time step will be limited for the small element length of mesh and we would need a large number of increments to calculate response of the structure for the desired time interval. For the implicit method, a big time increment would not be able to accurately capture the response in mesh model. The use of a single implicit or explicit method with a uniform time step will be computationally inefficient.

Because explicit and implicit methods have advantages and disadvantages, a co-simulation process between them has been used in this work.

2. Co-simulation

The best solution for this problem is to use a methodology that divides the model in two different parts: the first will be solved with implicit methodology and the second with the explicit method. Decomposition is given in the way that each region contains elements with similar requirements of stability and precision. Both parts are solved independently and individual solutions are coupled with each other to ensure continuity of global solution through the interface between the parts (figure 1). Decomposition is performed to divide the study area into subdomains and to process in parallel. A subdomain will be calculated with implicit methodology and another with an explicit one.

This concept allows the use of different time steps for each region of the mesh, preserving the accuracy and stability from individual parts. The advantage of this system is the use of big time steps for implicit dynamic analysis. It is desirable that the total number of interface nodes be small.

The implicit scheme is very inefficient when there are several contact areas that change from open to closed or vice versa. For this reason, tyre and road contacts are simulated using the explicit scheme. The vehicle body and suspension are solved using the implicit dynamic method and they are modelled as substructures. Wheel centres are the co-simulation interface nodes.

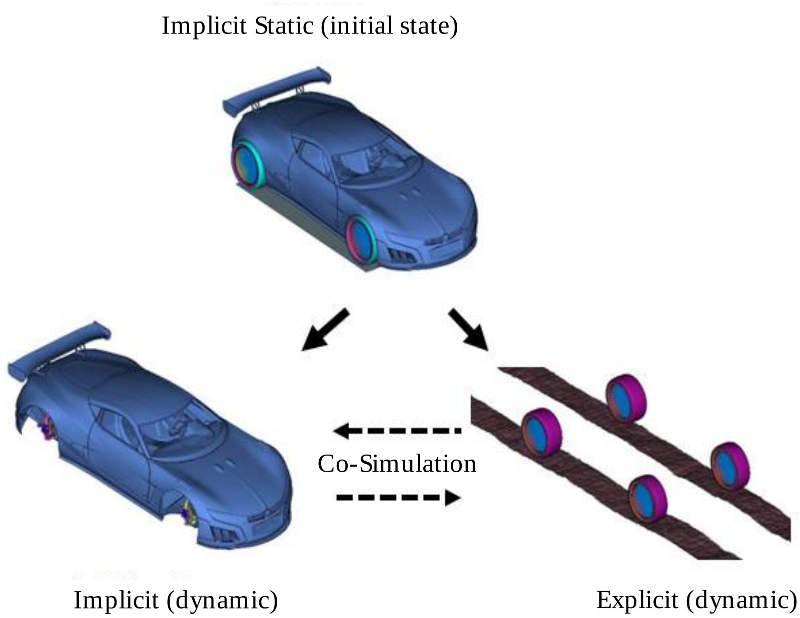

Figure 4. Implicit & Explicit subdomains

3. Description of the model

The main fields of study for fatigue prediction analysis are:

- Tyre modelling
- Modelling of body
- Modelling of suspension and components
- Assembling the implicit model
- Extraction of calculation inputs

3.1. Tyre modelling

As the minimum time increment of the explicit solver is governed by tyre, the model used to simulate the response of the tyre represents a significant challenge, and the main issue in such an approach is to simulate at the best level the tyre/road interaction.

Two versions of the tyre model were developed, as the front and rear tyres are different.

The front tyre is a 245/35R20. Its maximum load is 690 kg with a pressure of 0.29MPa.

The rear tyre is a 295/30R20. Its maximum load is 825 kg with a pressure of 0.29MPa.

The different components of the tyre are shown in figure 5.

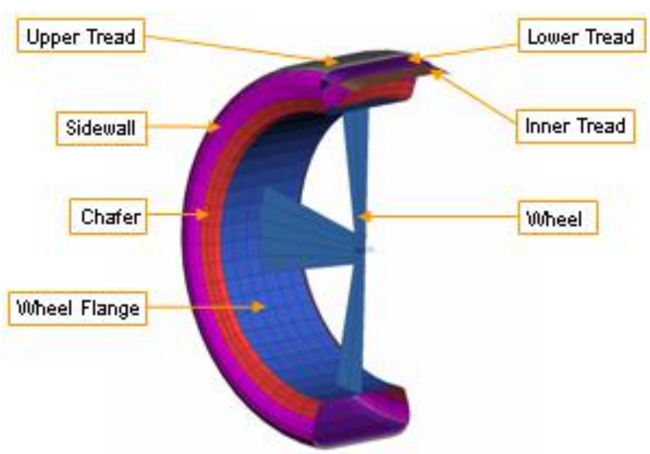

Figure 5. Part description for tyre model

The front tyre and the rear tyre have different geometries, but both models have the same materials and properties. They have about 7200 elements. A radial stiffness of 400N/mm is considered.

The upper tread and lower tread are modelled using solid elements, the chafer is modelled using shell elements, and they are made of hyperelastic material. The inner tread, sidewall, wheel and wheel flange are modelled using shell elements, and they are made of elastic material.

The solid elements used in the model are of the type C3D8R and C3D6.

- C3D8R: 8-node linear brick, reduced integration with hourglass control.
- C3D6: 6-node linear triangular prism, reduced integration with hourglass control. The shell elements used in the model are of the type S4R.
- S4R: 4-node general-purpose shell, reduced integration with hourglass control, finite membrane strains.

The enhanced hourglass control formulation was used for solid and shell. This formulation provides improved coarse mesh accuracy with slightly higher computational cost. The enhanced hourglass control approach represents a refinement of the pure stiffness method in which the stiffness coefficients are based on the enhanced assumed strain method.

Reinforcements are included on membrane elements inside the rubber material as embedded. Figure 6 shows a reinforce definition.

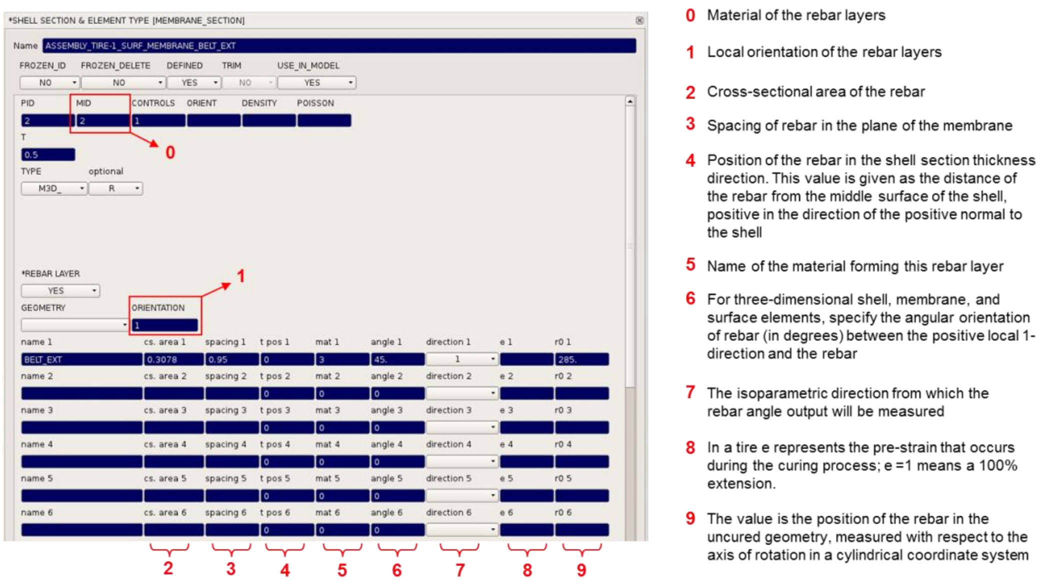

Figure 6. Plies of tyre

An initial static analysis for tyres, inflating, gravity load and vehicle preloads will be simulated. With the `*IMPORT` capability, we can import the new tyre position for nodes and elements in the dynamic explicit model.

3.2. Modelling of body

To connect all of the suspension component models to the chassis, some retained nodes are defined.

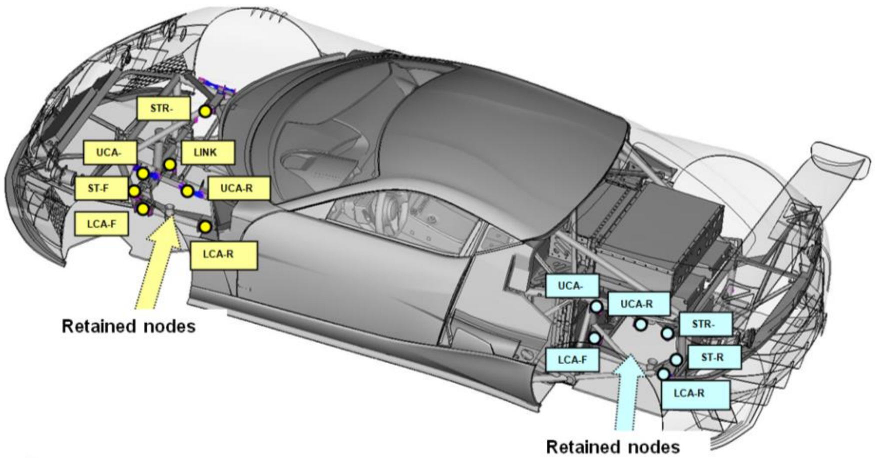

Figure 7. Body substructure and retained nodes

3.3. Modelling of suspension components

It is important to optimize the size of shell elements: in areas which are outside of fatigue study the nominal length of elements can be increased.
There are a number of good reasons to use substructures:

- Computational advantages: Efficiency is improved when the same substructure is used multiple times. The stiffness calculation and substructure reduction are done only once; however, the substructure itself can be used many times, resulting in significant savings in computational effort. Substructuring can isolate possible changes outside substructures to save time during reanalysis. During the design process large portions of the structure will often remain unchanged; these portions can be isolated in a substructure to save the computational effort involved in forming the stiffness of that part of the structure.
- Organizational advantages: Substructure libraries allow analysts to share substructures. In large design projects large groups of engineers must often conduct analyses using the same structures. Substructure libraries provide a clean and simple way of sharing structural information. Figures 8 and 9 show front and rear suspension models.

The front and rear suspension components defined as substructures are:

- upper control arm
- lower control arm
- support damper bracket
- spindle

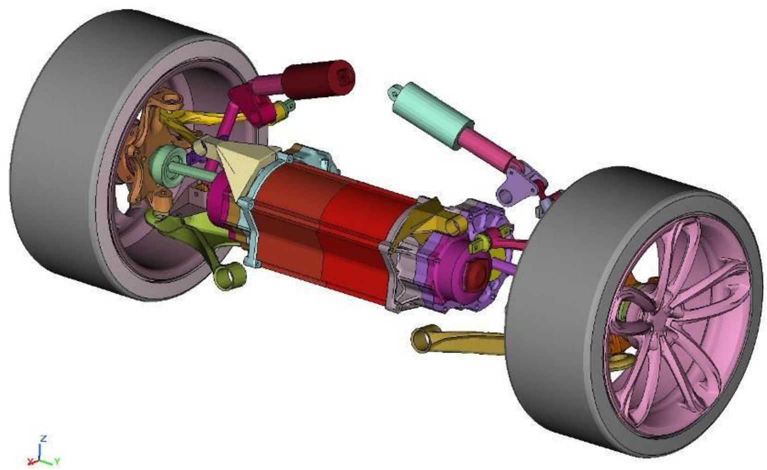

Figure 8. Front suspension description

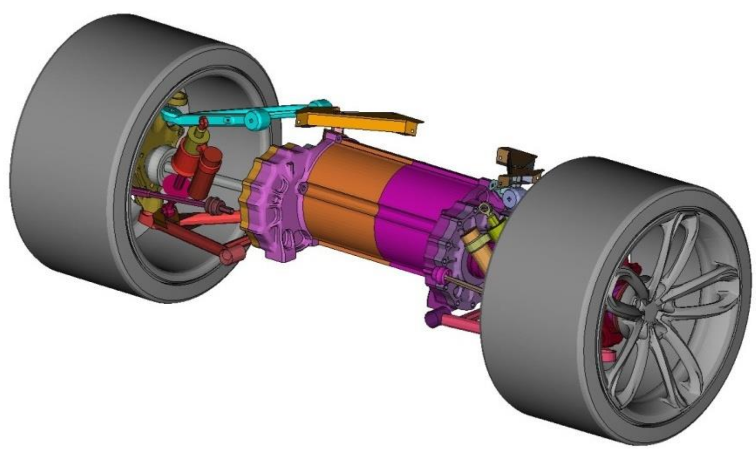

Figure 9. Rear suspension description

3.4. Assembling the implicit model

The Abaqus Standard model of the co-simulation analysis consists of the front suspension components and body; both are modelled using substructure techniques and will be assembled by connector elements.

The connections between parts are modelled using connector elements (figure 10). Join/Revolute connectors are used to model the hinges between the upper and lower control arms with chassis. Struts are modelled using an axial connection specifying nonlinear elasticity and damping. Link connectors are used to join steering and spindle. Coil springs have initial preload as given by their respective spring load deflection curves.

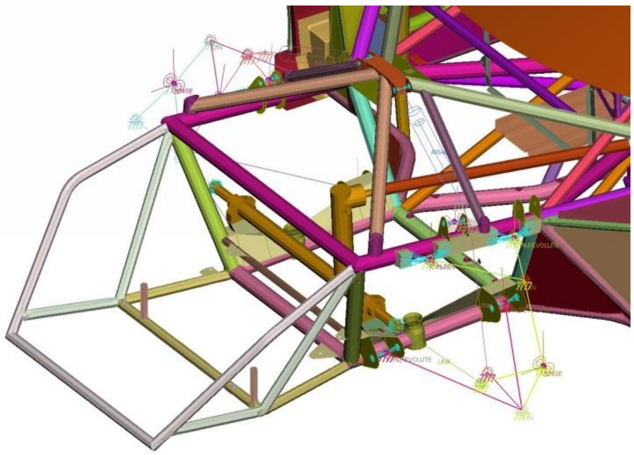

Figure 10. Front suspension to body assy

3.5. Extraction of calculation inputs

A virtual model (VPG) from Pave fatigue road (figure 11), scanned with digital technologies was developed. As a result of this technique, a high resolution surface model of this road has been obtained. This will allow us to have a realistic 3D model for automate fatigue life prediction on vehicle components.

a) Track **b) Digitalized track**

Figure 11. Pavé fatigue track

Modelling of the road surface is important. The simulation time depends on contact definition between road surface and a car. In order to greatly reduce the number of elements on our road model, only the part of the track that is in contact with tyres has been taken into account during the simulation (figure 12).

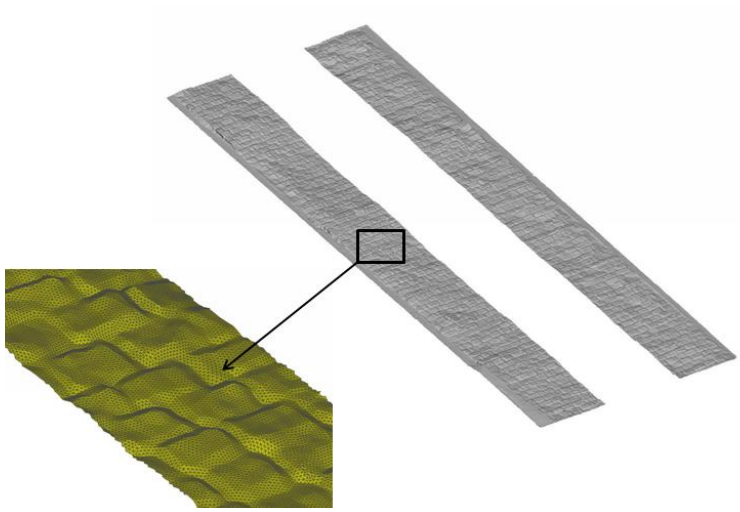

Figure 12. FEM for Pave Fatigue track

Rigid elements R3D3 (3-node, triangular facet) and R3D4 (4-node, bilinear quadrilateral) have been used to mesh road model (figure 12). Previous STL elements from 3D sampling road will be remeshed in order to decrease the density of mesh elements.

4. Analysis procedure

Several steps have been performed for the simulation model:

- Tyre inflation
- Footprint analysis
- Assembly the Implicit/Explicit domains
- Applying velocity

A previous static analysis for tyre inflating and preloading will be done, in order to know initial position for the preloaded complete vehicle model. Co-simulation regions across which data will be exchanged during the co-simulation analysis are identified on each model at the location of the wheel centre as shown in figure 13.

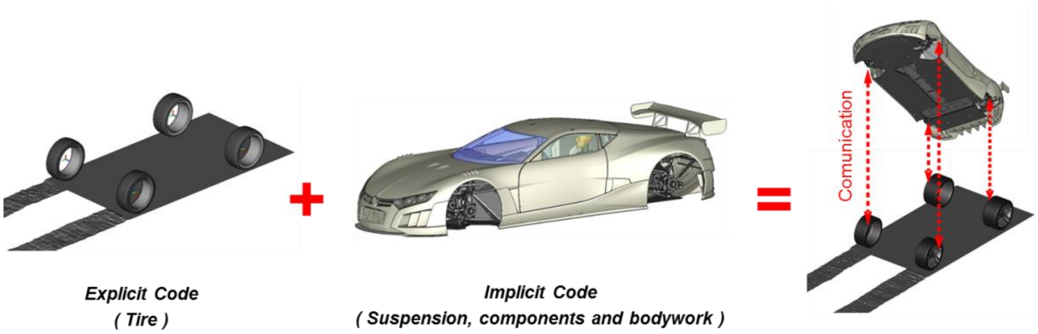

Figure 13. Assembling the model

5. Boundary conditions

The Pave fatigue road is commonly used from car manufacturers for the structural evaluation of suspension components and car body. The simulation test is carried out with a constant velocity of 35 km/h. The Volare, FEM model car has been considered for the present study.

From the simulation point of view, during explicit-implicit analysis, the following initial conditions have been imposed on the model:

- Initial velocity v=12500 mm/s , in X car direction applied to car.

- An angular initial velocity field ω = 36.4 rad/s for front tyres and ω = 36.3 rad/s for rear tyres.

The following loads and boundary conditions are applied:

- Gravity load applied to the entire vehicle model, in order to guarantee the continuity with the static equilibrium.
- Nodes of the fatigue road are totally constrained on all their 6 d.o.f with a rigid body.
- Tyres, front and rear, will be initially preloaded in order to support the total mass of vehicle. An initial step for tyre inflation (performed for the four wheels) at 0.29 MPa , a second step to define the contact enforcement between the wheels and the road surface and finally a third step with the preload force applied to front tyres Fz= 2930 N and 3914N for rear tyres.

6. Results

Due to the large dimensions, the complete model analysis is limited to a 100-metre fatigue road. The mises stress history distribution of the finite element results, obtained from flexible bodies for front and rear suspension and internal car structure, will be reported.
For front suspension components, the maximum Von Misses stress level is located on lower control arm link support (figure 10).

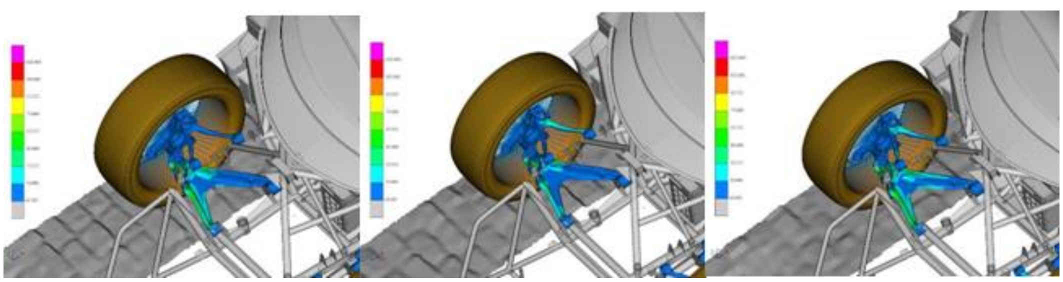

Figure 10. Front suspension mises stresses at 0.2, 0.3 and 0.4 seconds

For rear suspension components, the maximum Von Misses stress level is located on upper control arm support (figure 11).

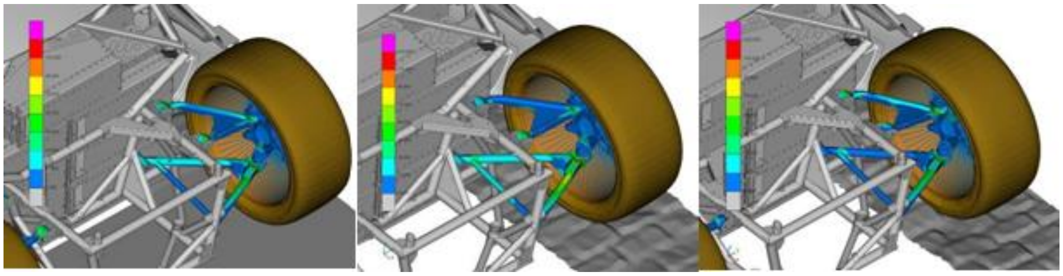

Figure 11. Rear suspension mises stresses at 0.2, 0.3 and 0.4 seconds

Finally, through Virtual Lab software the fatigue life of the components will be estimated by finding the life in cycles that corresponds to calculated stress amplitude by the intersection on the SN curve.

References

1. Dixon, J.C.,"Tires, suspensions and handling". Cambridge University Press, England, 1991.
2. D.F. Socie, M.R. Mitchell, and E.M. Caulfield, "Fundamentals Of Modern Fatigue Analysis"; College of Engineering, University of Illinois, 1977.
3. E. Duni, G. Toniato, "Vehicle Fatigue Load Prediction based on Finite Element TIRE/ROAD Interaction implemented in an Integrated Implicit - Explicit Approach". FIAT Group Automobilies , 2008.
4. E. Duni, G. Monfrino, R. Saponaro, M. Caudano and F. Urbinati. "Numerical Simulation of Full Vehicle Dynamic Behaviour Based on the Interaction Between Abaqus/Standard an Explicit Codes". FIAT Research Center, Orbassano (TO), Italy.
5. H.O. Fuchs; R.I. Stephens, "Metal Fatigue in Engineering"; Wiley,1980.
6. Reimpell, J., and H. Stoll, "Automotive Chassis-Engineering Principles", Arnold, England,1996.
7. Abaqus User´s Manual, Version 6.12.Dassault Systèmes Simulia Corp., Providence, RI, USA.

Acknowledgments

We would like to express our gratitude to the employees of Abaqus Technical support from Principia Spain for their professional assistance and patience in helping us to solve several technical questions.