

# VEHICLE DYNAMICS AND DURABILITY SIMULATIONS USING ANSA AND ABAQUS

<sup>1</sup>Harish Surendranath

<sup>1</sup>Dassault Systemes SIMULIA Corp., USA

KEYWORDS –

Co-simulation, Substructures, Dynamics, Suspension, Tires

ABSTRACT –

ANSA is widely used in the automotive industry as a pre-processor to generate detailed finite element models for crashworthiness, NVH and durability simulations. High performance computing (HPC) is enabling engineers to increase the fidelity of these simulations while reducing the turnaround time. Traditionally engineers resorted to either Implicit or Explicit finite element technique to solve dynamics of systems. Recently a co-simulation approach has been implemented in Abaqus that combines the strengths of Implicit and Explicit solution techniques by solving different regions of the same model using different solution techniques. This co-simulation approach combined with HPC provides a powerful and efficient tool to analyze systems with varying degrees of dynamics in space and time.

This paper describes a methodology in ANSA to generate models for subsequent implicit-explicit co-simulation using Abaqus. The vehicle body and chassis are solved using implicit solution technique. Substructures (superelements) are used to represent the body as well as some of the chassis components. The tire-road interaction is solved using an explicit solution technique.

TECHNICAL PAPER -

## 1. INTRODUCTION

As fuel efficiency standards become more stringent, it is expected that there will be a greater emphasis on lighter vehicles, making it necessary to improve the durability of parts while reducing overall vehicle weight. In order to effectively fulfil this requirement, engineers need to rely on advanced materials and innovative designs. By resorting to accurate and efficient simulations, the vehicle design process can be significantly accelerated, reducing the overall time to market.

### Vehicle Durability Workflow

A rough schematic of the vehicle durability workflow is shown in Figure 1.

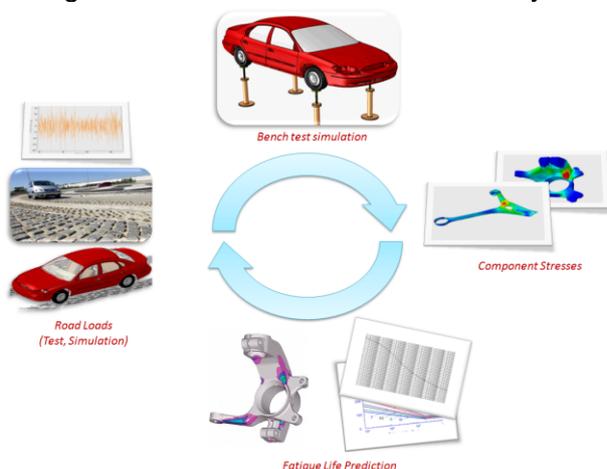


Figure 1: Vehicle Durability Workflow

---

The starting point for the workflow is the road loads acting at the wheel centres. However, at the beginning of the design cycle, physical prototypes are not available and load data from previous prototypes has to be used. This data might not be very representative of the vehicle under development. Since building prototypes and executing long-duration trials on fatigue reference roads and other scenarios is expensive and time-consuming, there is growing emphasis on a simulation-based approach to obtain the road load data [1]. The load data is used to simulate the bench test using a vehicle model comprising primarily of substructures for vehicle components. This simulation yields the component loads and stresses, which are used in fatigue calculations along with material S-N curves to predict the fatigue life of components.

### Implicit vs. Explicit Finite Element Solvers

The direct-integration dynamic procedures utilizes either implicit operators for integration of the equations of motion or the central-difference operator, commonly referred to as explicit dynamics. In an implicit dynamic analysis the integration operator matrix must be inverted and a set of nonlinear equilibrium equations must be solved at each time increment. In an explicit dynamic analysis, displacements and velocities are calculated in terms of quantities that are known at the beginning of an increment; therefore, the global mass and stiffness matrices need not be formed and inverted. This means that each increment is relatively inexpensive compared to the increments in an implicit integration scheme. However, the size of the time increment in an explicit dynamic analysis is limited, because the central-difference operator is only conditionally stable. On the contrary, the implicit operator can be unconditionally stable and, thus, there is no such limit on the size of the time increment that can be used for most analyses. In fact, the time increment size is controlled only by solution accuracy.

The stability limit for the central-difference method (the largest time increment that can be taken without the method generating large, rapidly growing errors) is closely related to the time required for a stress wave to cross the smallest element dimension in the model. Therefore, the time increment in an explicit dynamic analysis can be very short if the mesh contains small elements or if the stress wave speed in the material is very high. It can handle rapid changes in contact state as well as material failure much more effectively than the implicit dynamics scheme.

### Implicit-Explicit Co-simulation

The implicit dynamics scheme in Abaqus/Standard as well as the explicit dynamics scheme in Abaqus/Explicit is formulated using a uniform time-step for the entire mesh. For full vehicle simulations, the range of element sizes in a mesh can vary over a few orders of magnitude. With high performance computing becoming commonplace, the desire and the ability to use finer and finer meshes is growing. Using an explicit or an implicit scheme exclusively with a uniform time-step, for such problems, is computationally very inefficient. If one were to use an explicit scheme, the time-step would be restricted by the size of the smallest element in the mesh and it would take a very large number of increments to compute the response of the structure for the desired interval of time. On the other hand, using an implicit scheme with a large time-step, one would not be able to capture, accurately, the response in regions of the mesh with high gradients with respect to time. Furthermore, the implicit scheme can be inefficient in cases where large scale contact changes or material failure is involved. An intuitive solution to this problem is to integrate different parts of the mesh with time-steps and rendezvousing schemes that are appropriate for the requirements of stability and accuracy within the particular regions of the mesh as shown in Figure 2. The methodology used here divides a large finite element mesh into two parts, one to be solved with the implicit technique and the other with explicit. The decomposition is done to ensure that each region contains elements with similar requirements with respect to stability and accuracy. The two parts are then solved as independent problems separately and the individual solutions are coupled together to ensure continuity of the global solution across interface boundaries between the parts. Domain decomposition can be used on to divide the implicit and explicit parts further

---

into subdomains and processed in parallel. The subdomains using the implicit solution technique all use the same time step. The same is true for the subdomains using the explicit solution technique.

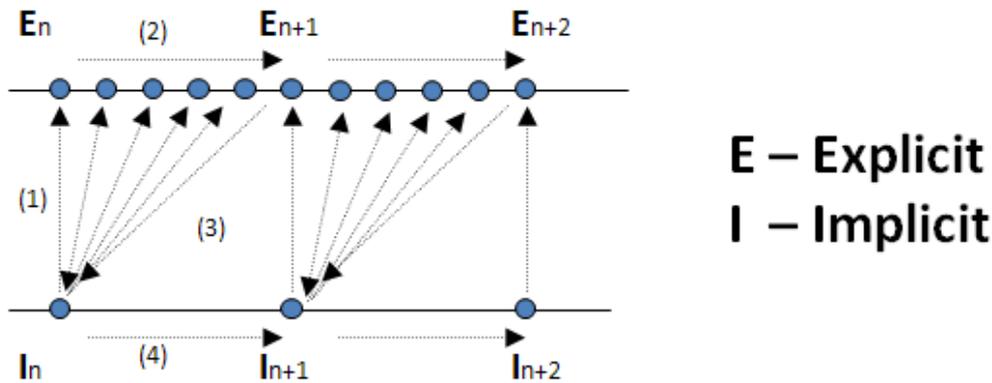


Figure 2: Implicit-Explicit co-simulation time stepping scheme

## 2. MODEL SETUP IN ANSA

In order to set up the model for co-simulation, one needs to first identify the regions to be simulated with Abaqus/Standard and Abaqus/Explicit. The Abaqus/Standard portion of the model can optionally contain substructures. Substructures represent the linear response of a set of elements in terms of the degrees of freedom of a few nodes, thereby increasing the computational efficiency.

The newly introduced **Co-simulator** tool along with enhanced substructure generation methods in ANSA is used to set up the model. The following is a description of the steps involved.

### Importing and assembling the model

Often times, the full vehicle model used for durability does not come from a single source. For example, the body model may come from a NVH simulation in NASTRAN. The suspension model may come from an Abaqus crash simulation and so on. In this example five separate models are used as input to ANSA. They are:

- a) The body in trim model, converted from NASTRAN to Abaqus
- b) Right control arm model, converted from NASTRAN to Abaqus
- c) Left control arm model, converted from NASTRAN to Abaqus
- d) Suspension and steering models from an Abaqus model
- e) Tire models from an Abaqus model

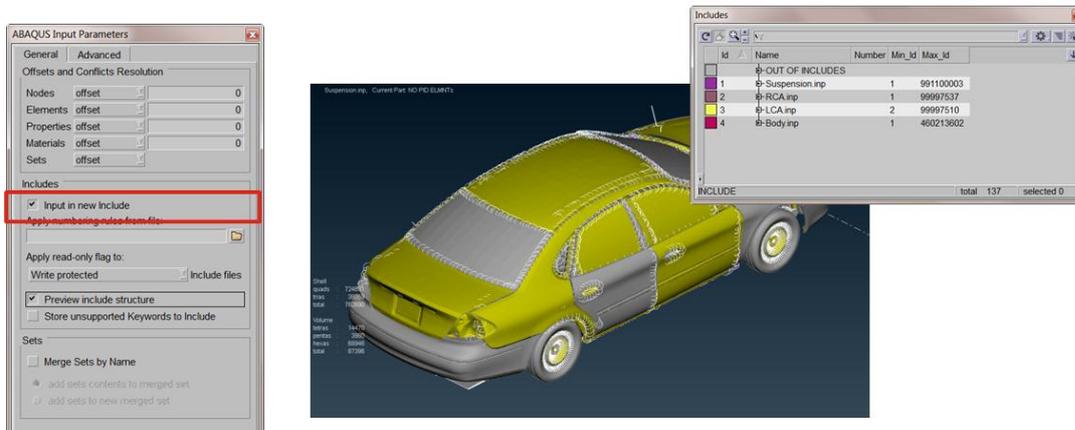


Figure 3: Abaqus inputs and creation of "Includes" during input

During the model input, each is stored as a separate include. Storing them as separate includes simplifies the process of substructure generation (described in the following section). Figure 3 shows the model as well as the Includes Manager. Separate includes have been created for the body, suspension and the controls arms. The tires are placed in the “Out of Includes” section, meaning that they are not part of any includes.

Using Co-simulator to define co-simulation regions and steps

The newly introduced Cosimulator functionality is used to define the implicit and explicit regions of the model as well as the steps involved in the co-simulation. A new co-simulation is created. The co-simulator window provides options to define the implicit and explicit regions and steps in the model.

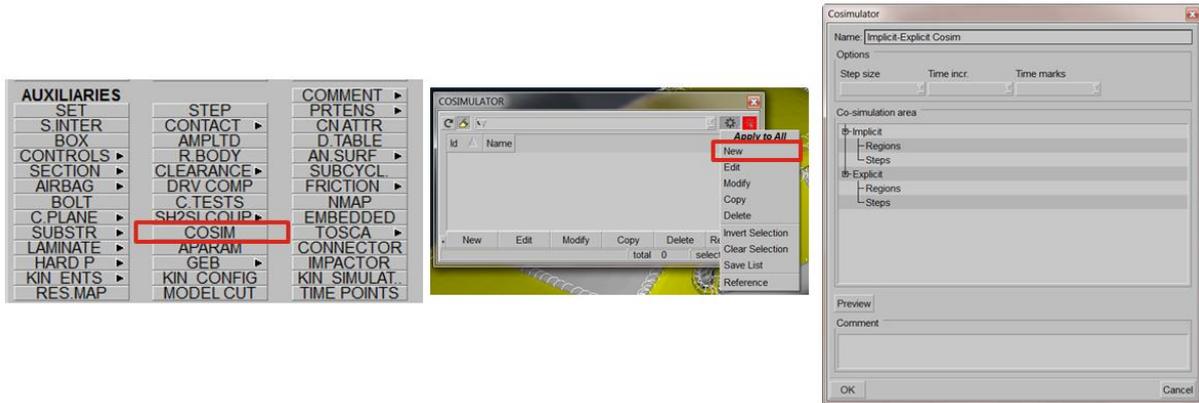


Figure 4: Using the co-simulator to create a co-simulation

By right clicking on the “Regions” option under implicit, we can introduce a new substructure into the implicit domain. Once a new substructure is created, it needs to be edited to add a frequency extraction step in order to extract the eigenmodes of the substructure. Also, the substructure generation step needs to be modified in order to retain the generated eigenmodes using the “Select Eigenmodes” option.

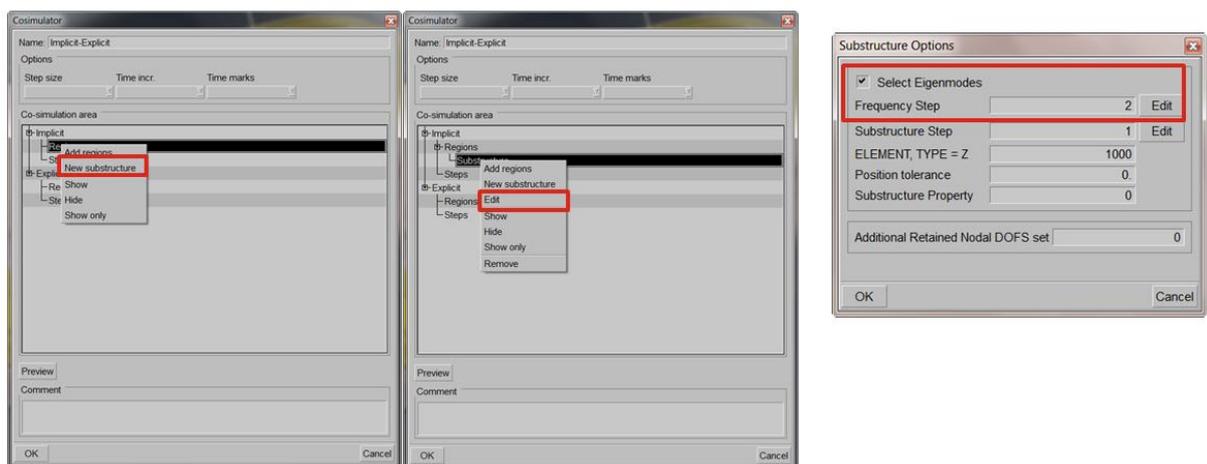


Figure 5: Adding a new substructure to the implicit domain and editing it

In the next step, we add the regions corresponding to the substructure by choosing the corresponding region from the Includes menu. In Figure 6, the region corresponding to the body of the vehicle is chosen to be included in the substructure. Similarly, substructures corresponding to the left and right control arms are also generated.

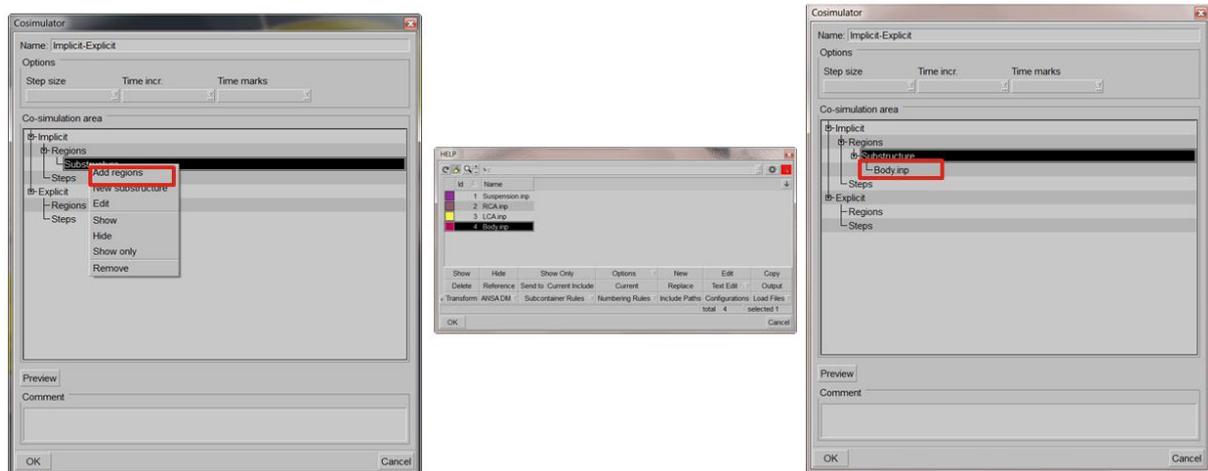


Figure 6: Assigning a region to be made into a substructure

Once all the substructures have been added to the implicit domain, the remaining implicit region, namely the suspension, should also be added. Once again, this can be accomplished by using the “Add Regions” from under the “Regions” option in Implicit.

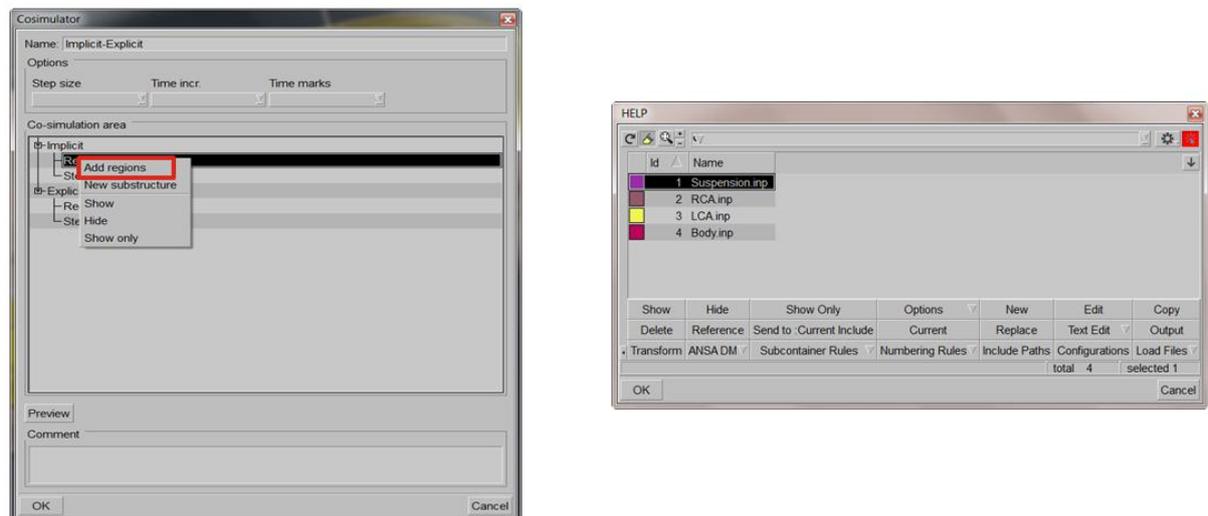


Figure 7: Adding non-substructure regions to the implicit domain

Note that we did not assign any regions under the Explicit domain. Once the implicit regions are assigned, ANSA can automatically group the rest of the model as the Explicit region. Once the region assignment is complete, the next step is to assign the implicit and explicit steps to the corresponding regions. These steps should be marked for co-simulation by right clicking on the corresponding steps and choosing the “Mark for co-simulation option”. Also, we need to set the rendezvousing scheme for the time incrementation. From under “Options”, select “SUBCYCLE” for the time incrementation scheme.

Note that ANSA will automatically create the retained nodes of the substructure based on its connectivity to the rest of the model. Similarly, ANSA will automatically identify the co-simulation interface so that the user does not have to specify the nodes explicitly. Once the cosimulator definition is complete, the model can be written out and submitted for simulation.

### 3. CO-SIMULATION

The simulation sequence is shown in Figure 8. It consists of two stages. In the first stage, we set up a gravity settling simulation using the implicit solution scheme entirely. In this simulation, the following sequence of steps is used:

- 1) Fix the wheel centers; apply gravity load to the vehicle
- 2) Fix the wheel centers; inflate tires
- 3) Fix the wheel centers; bring the road in contact with the tires by applying a vertical load corresponding to the vehicle weight
- 4) Release the wheel centers

The second stage is the co-simulation. The full vehicle model from the previous stage is now separated into implicit and explicit parts as we set it up in the previous section. The tires are imported into Abaqus/Explicit from the Abaqus/Standard simulation mentioned above. Since import between Abaqus/Standard and Abaqus/Explicit is currently not supported in ANSA, we use manual editing on the models written out by ANSA to set up the complete workflow involving the two stages of simulation described above.

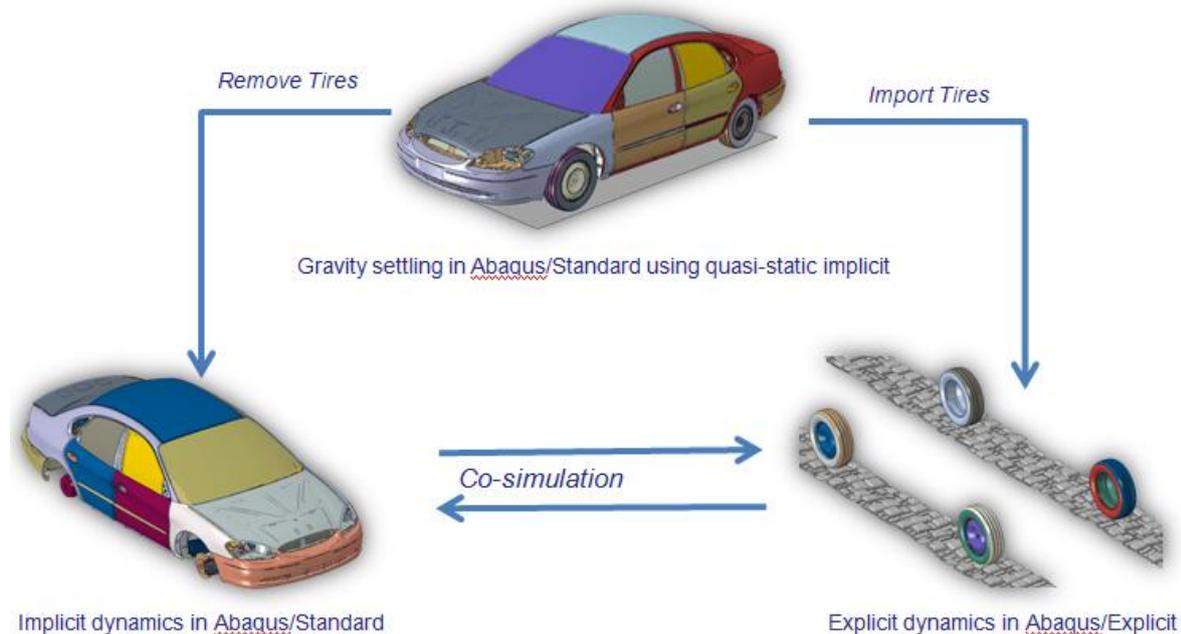


Figure 8: The simulation sequence

The vehicle model is then subjected to different load cases like pothole impact, straight travel on roads paved with Belgian blocks etc.



Figure 9: Pothole Impact simulation and the stress on the control arm

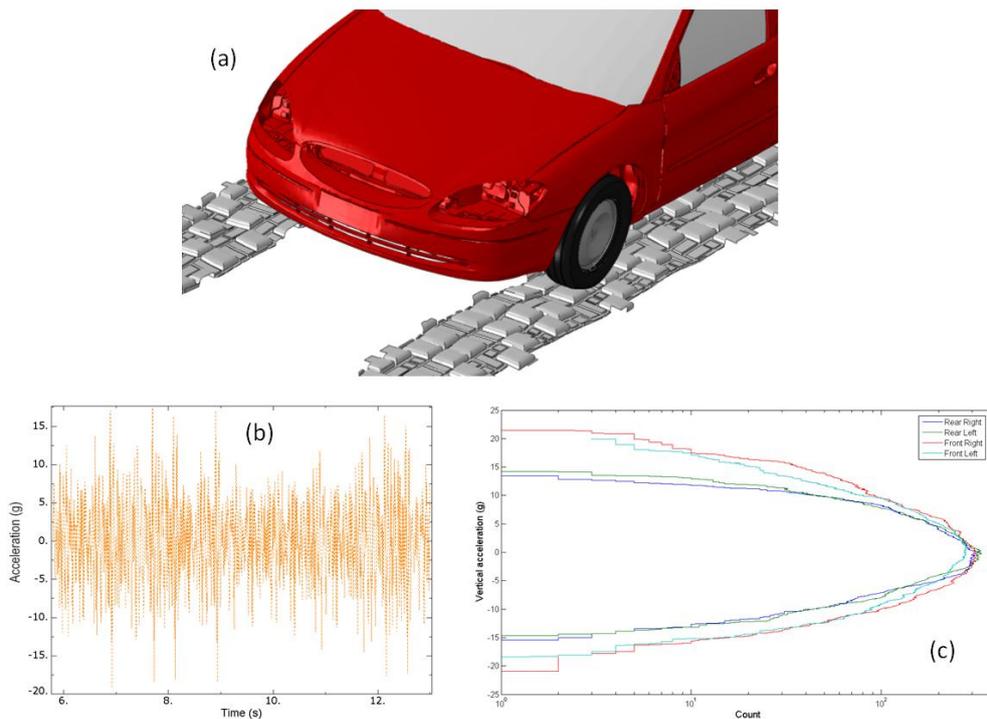


Figure 10: (a) vehicle on a road paved with Belgian blocks (b) acceleration time history at wheel center (c) level crossing plot for wheel center accelerations

#### 4. CONCLUSIONS

The new Cosimulator functionality introduced in ANSA provides a convenient way to generate Abaqus/Standard-Abaqus/Explicit co-simulation models. The user needs to only define either the implicit or the explicit region of the model. If the user has defined the implicit region, ANSA can automatically assign the rest of the model as the explicit region. The co-simulation interface is determined automatically. Substructure related enhancements make it easy to define and use substructures as part of the implicit region. Similar to the co-simulation interface region, the retained nodes of the substructures are also identified automatically. Some manual editing to the input files written out is required at this time so that import between Abaqus/Standard and Abaqus/Explicit can be performed.

#### REFERENCES

- (1) Duni, E, Toniato, G., Smeriglio, P., Puleo, V., and Saponaro, R., "Vehicle Dynamic Solution Based on Finite Element Tire/Road Interaction Implemented through Implicit/Explicit Sequential and Co-Simulation Approach", SAE Paper 2010-01-1138, April 2010.
- (2) ANSA version 12.1.5 User's Guide, BETA CAE Systems S.A., July 2008
- (3) Abaqus Analysis User's Manual, Dassault Systemes, 2010.