

Full Vehicle Durability Using Abaqus/Standard to Abaqus/Explicit Co-simulation

Summary

Finite element simulation of full vehicle behavior can provide significant savings of time and money in the automobile development cycle. Realistic analyses offer valuable insight into vehicle durability and can eliminate costly design changes.

When a vehicle traverses road irregularities or obstacles, it may experience highly dynamic loading, and the different subsystems respond with varying degrees of nonlinearity. High fidelity system-level simulation requires a broad range of analysis functionality such as nonlinear material behavior, contact interactions, and mechanisms with complex kinematic constraints.

In this Technology Brief, a co-simulation approach between Abaqus/Standard and Abaqus/Explicit is used to predict vehicle behavior on durability test tracks. The vehicle body and suspension, which typically respond in a mildly nonlinear manner, are solved using the implicit solution technique. The tires, which experience impact loading and rapid changes in contact state, are solved using the explicit solution technique. The co-simulation approach allows for an efficient analysis by capitalizing on the strengths of both solution techniques simultaneously.

Background

An automobile is a complex system consisting of various mechanical, electronic, and logical subsystems. The vehicle experiences a wide range of service loads: road hazards, wind, thermal cycles, etc. From a mechanical strength and durability perspective, the loads of primary concern are those caused by irregularities and obstacles on the road.

A schematic of the vehicle durability workflow is shown in Figure 1. The starting point is the road loading acting at the wheel centers. At the beginning of the design cycle, physical prototypes are not available and in the absence of test data, load information 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 obtaining road load data [1].

Finite Element Analysis Approach

Finite element (FE) software is widely used in vehicle durability programs. While the use of FE software has



Key Abaqus Features and Benefits

- Co-simulation capability combines the unique strengths of Abaqus/Standard and Abaqus/Explicit to more effectively perform complex simulations
- General contact capability in Abaqus/Explicit allows for handling of rapid changes in contact conditions between the tire and the road
- Substructures in Abaqus/Standard provide a cost-effective way to model the linear response of the body for long duration events
- Connector elements for modeling complex kinematic connections in the suspension and steering systems

traditionally been limited to component level simulations, rapid progress in the area of high performance computing is enabling its use in full vehicle simulations.

The equation solution phase of an FE analysis can use an implicit or explicit technique. The implicit technique is ideally suited for long duration problems where the response is moderately nonlinear. While each time increment associated with the implicit solver is relatively large, it is computationally expensive and may pose convergence challenges.

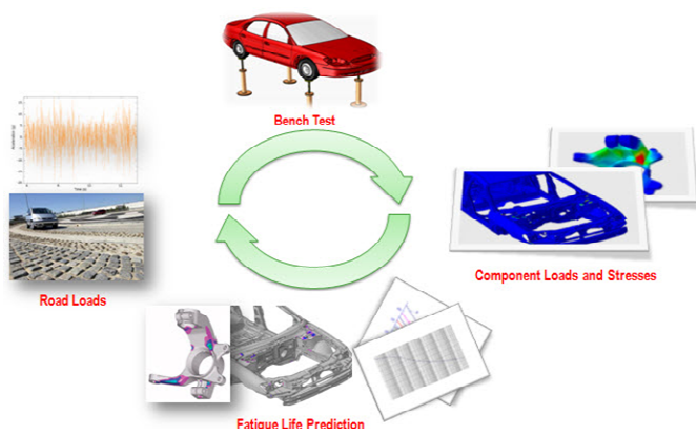


Figure 1: Durability workflow

The explicit technique is very robust and ideal for modeling short duration, highly nonlinear events involving rapid changes in contact state and large material deformation. While each time increment associated with the explicit solver is relatively inexpensive, numerical stability requires that it be very small. A large number of increments is thus required for typical durability scenarios, imposing a very long simulation time.

When a vehicle travels over an obstacle, the tires are impacted and can experience large, rapid deformations. The body, because it is isolated from the impact by the suspension, responds in a linear or mildly nonlinear way relative to the tire. When taken as a whole, the vehicle exhibits a range of responses to the road load; some are ideally suited to Abaqus/Standard, some to Abaqus/Explicit.

Abaqus/Standard to Abaqus/Explicit co-simulation allows the complete finite element mesh to be strategically divided into two parts, one to be solved with the implicit technique and the other with the explicit technique. The two parts are solved as independent problems and coupled together to ensure continuity of the global solution across the interface boundaries. A detailed description of the coupling algorithm can be found in [2].

Validation of Co-simulation Model

The accuracy of the Abaqus co-simulation scheme is first demonstrated using a full vehicle model running over an obstacle 80 mm high at 30 km/h (Figure 2). The vehicle body and suspension are solved using the implicit dynamic solution technique in Abaqus/Standard, whereas the tires and their contact with the road are simulated in Abaqus/Explicit.

Four co-simulation interface nodes are used, corresponding to the four wheel centers. For simplicity and computational performance, the vehicle body and the suspension components are assumed to be rigid. The results obtained from co-simulation are compared against those from a standalone Abaqus/Explicit simulation modeling the full vehicle. The vehicle body and the suspension components are assumed to be rigid in the standalone Abaqus/Explicit simulation as well.

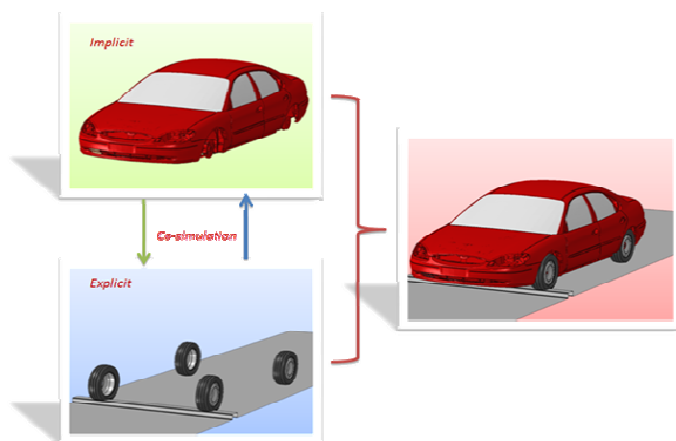


Figure 2: Co-simulation analysis of a vehicle traveling over an obstacle

Fatigue Reference Road Test

The co-simulation scheme is then used to study the behavior of a full vehicle running on a section of test track laid with Belgian blocks. The vehicle moves at 30 km/h for a total distance of 65 meters. The event duration is 10 seconds. The loading scenario is of moderate severity: the strains experienced by the vehicle body and suspension components are not severe enough to cause plastic strains. Under these conditions, the body as well as the suspension components can be represented using substructures. Substructures in Abaqus/Standard allow a collection of elements to be grouped together and all but the retained degrees of freedom eliminated on the basis of linear response within the group. The dynamic behavior of the substructure is improved by including the eigenmodes of the system (Figure 3). Converting the body and suspension components into substructures reduces the overall model size by a few orders of magnitude, thereby reducing the turnaround time drastically.

The kinematic joints, bushings, springs and dampers in the suspension are modeled using connector elements. The contact interaction between the tires and the road is modeled directly in Abaqus/Explicit.

A schematic diagram of the simulation process is shown in Figure 4. The first step in the workflow is to perform the tire inflation and gravity settling of the vehicle. This loading scenario is performed quasi-statically in Abaqus/Standard using a loading rate slow enough to eliminate the effects of inertia.

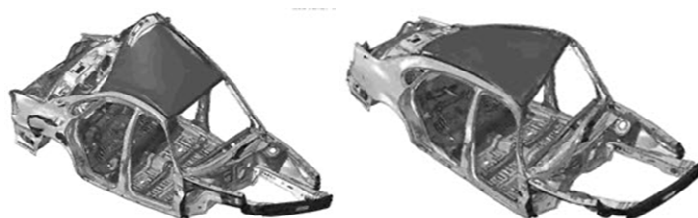


Figure 3: The dynamic response of the sub structure is enhanced by including eigenmodes

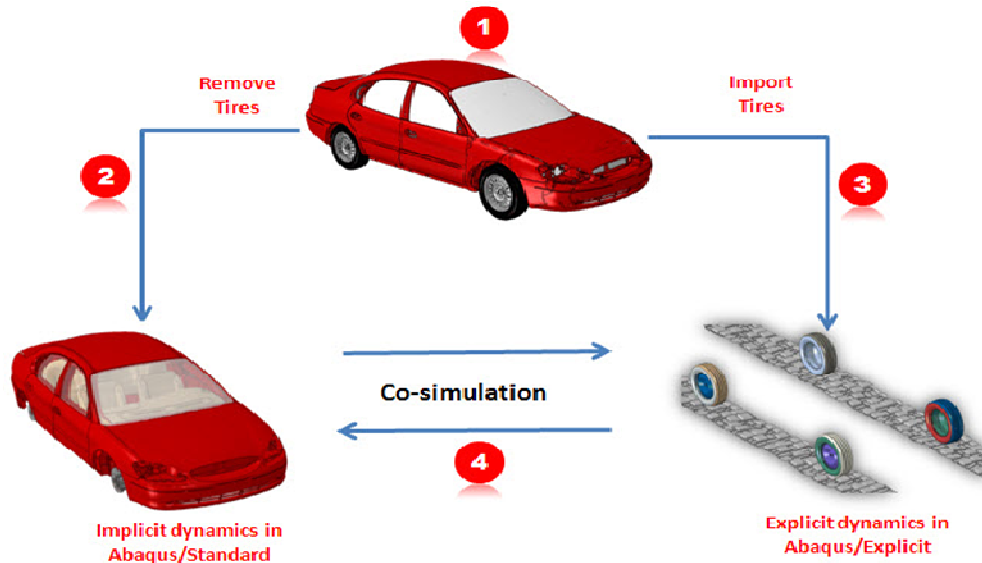


Figure 4: Fatigue reference road simulation workflow

The second step is to remove the tires from the vehicle model in Abaqus/Standard and transfer them to Abaqus/Explicit, leaving the body and suspension components for analysis in Abaqus/Standard. The finite element model of

the test track is added to the Abaqus/Explicit analysis and contact interactions between the tire and the road are defined. The final step is to perform the co-simulation analysis between the two models.

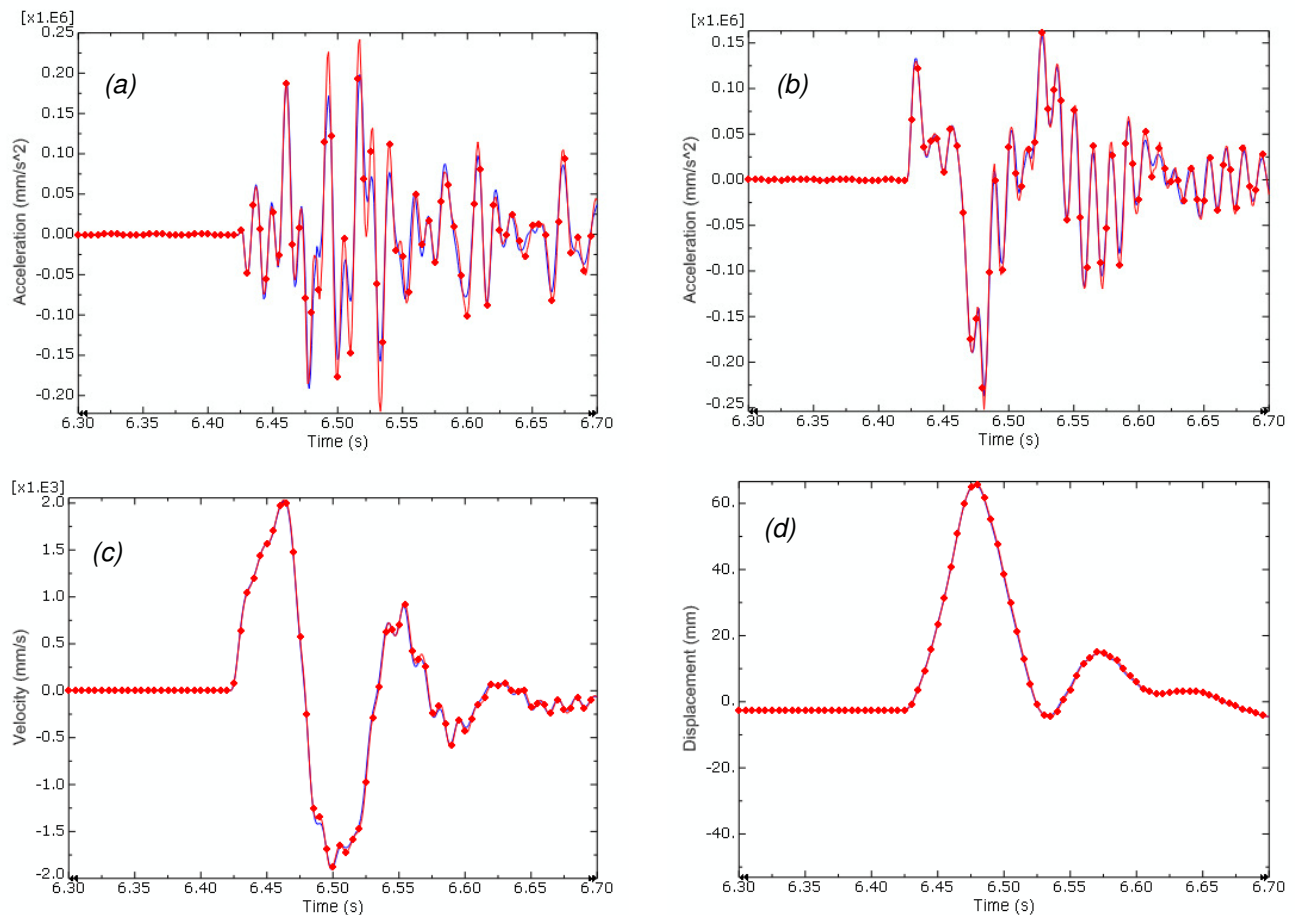


Figure 5: Comparison between Implicit-Explicit co-simulation (blue) and standalone Explicit simulation (red) at the left wheel center: (a) Longitudinal acceleration (b) Vertical acceleration (c) Vertical velocity (d) Vertical displacement

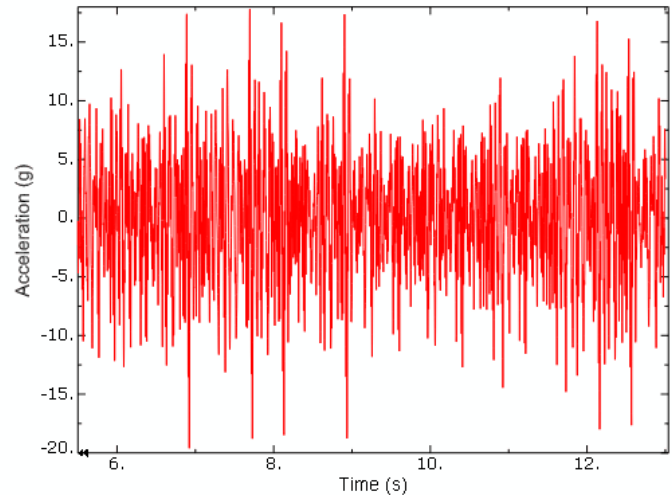


Figure 6: Vehicle on fatigue reference road (left), vertical acceleration of the right wheel center (right)

Results

In Figure 5 the vertical and longitudinal accelerations as well as the vertical velocity and displacement of the front left wheel centre are compared for the co-simulation validation test. It can be seen that the results predicted with co-simulation at the interface nodes match very well with the results from the standalone Abaqus/Explicit simulation that uses a uniform time step across the whole model.

As an example of the type of information that this simulation can provide, the accelerations at the front right wheel centre from the fatigue reference road simulation are shown in Figure 6. These loads, experienced by the vehicle from the road through the tire, can then be used as

inputs to bench tests in the laboratory or in additional simulations later in the design cycle. In addition, loads and stresses on individual components of the vehicle can be retrieved from the fatigue reference road simulation directly or from subsequent bench test simulations.

Conclusions

The Abaqus/Standard to Abaqus/Explicit co-simulation technique, in which the two programs are simultaneously run on complementary sections of a model, is a powerful tool to efficiently model the response of a full vehicle to highly dynamic loading. By coupling the strengths of the respective programs, vehicle durability workflows are enhanced with the ability to compute road load data.

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. Gravouil, A., Combescure, A., "Multi-time-step explicit-implicit method for non-linear structural dynamics," *International Journal for Numerical Methods in Engineering*, Vol. 50, pp. 199-225.

Abaqus References

For additional information on the Abaqus capabilities referred to in this document please see the following Abaqus 6.13 documentation references:

- Analysis User's Guide
 - "Co-simulation," Section 17.1
 - "Substructuring," Section 10.1

About SIMULIA

SIMULIA is the Dassault Systèmes brand that delivers a scalable portfolio of Realistic Simulation solutions including the Abaqus product suite for Unified Finite Element Analysis, multiphysics solutions for insight into challenging engineering problems, and lifecycle management solutions for managing simulation data, processes, and intellectual property. By building on established technology, respected quality, and superior customer service, SIMULIA makes realistic simulation an integral business practice that improves product performance, reduces physical prototypes, and drives innovation. Headquartered in Providence, RI, USA, with R&D centers in Providence and in Velizy, France, SIMULIA provides sales, services, and support through a global network of over 30 regional offices and distributors. For more information, visit www.simulia.com

The 3DS logo, SIMULIA, Abaqus and the Abaqus logo are trademarks or registered trademarks of Dassault Systèmes or its subsidiaries, which include Abaqus, Inc. Other company, product and service names may be trademarks or service marks of others.

Copyright Dassault Systèmes, 2011

