

Applications for MBS-FEM-coupling with MpCCI using automotive simulation as example.

Pascal Bayrasy¹, Michael Burger², Carsten Dehning¹, Ilja Kalmykov¹ und Michael Speckert²

¹ Fraunhofer-Institute for Algorithms and Scientific Computing (SCAI), Sankt Augustin, Germany

² Fraunhofer Institute for Industrial Mathematics (ITWM), Kaiserslautern, Germany

Abstract A coupled MBS-FEM-simulation approach is presented. The used programs are Adams and Abaqus. MpCCI (Multi-Physics Code Coupling Interface) is used for a coupled simulation. As example passing of a road hole by the left front wheel is considered. The corresponding tire is modelled with FEM approach. Position as boundary condition is provided by multibody model of a car in MSC.Adams. Both programs simulate parallel and communicate with MpCCI server for exchange of data.

Keywords:

MSC.Adams, Abaqus, MpCCI, coupled simulation, multibody system, finite element method

1 Motivation

Analysis of multibody systems (MBS) and finite element method (FEM) are both well established practices in computational engineering. Simulation of multibody systems is mainly used for analysis of mechanisms consisting of rigid components connected with joints. Examples of appropriate systems for this approach are industrial robots or, as considered in this paper, car chassis. The mathematical description results often in a system of differential algebraic equations (DAEs).

The finite element method allows much more detailed investigation of the system behaviour. In this case partial differential equations describe the models of the considered systems. However FEM also increase the simulation times.

On the one hand, due to different mathematical methods, the addressed problems from MBS and FEM are well separated. On the other hand, with advancing complexity of developed systems goals of computational engineering are increasingly shifted to coupled problems. An example for such an application are plastic or large deformations of automotive components integrated in a complex rigid body model of a car. Main fields of interest are parts which inherent takes deformation due to functional purpose, such as tires.

This article introduce an example of coupled MBS-FEM analysis. The passing of a road hole by a car is modelled. Due to high loading, the deformations of the tire are considered with FEM approach. A detailed model, provided by ITWM in Kaiserslautern, is simulated with Abaqus. Simultaneously, a MBS calculation of complete vehicle is made by MSC.Adams/Car.

The coupled simulation (co-simulation) exchanges the position of wheel hub from MSC.Adams to Abaqus and reaction force due to deformation of tire from Abaqus to MSC.Adams. Co-simulation setup is done by MpCCI. Both codes run in parallel and communicate over MpCCI server as clients.

2 Used models

2.1 Rigid body model of a car in MSC.Adams

MSC.Adams is developed by MSC software corporation. Application field is simulation of multibody systems. This software is well established for simulation in research and development. The extension module Adams/Car provides further capabilities for enhanced analysis of vehicles and is one of the main tools in the automotive industry.

An MBS model of the complete vehicle is simulated by MSC.Adams/Car. A modified example of MDI_Demo_Vehicle has been used. Changes include deactivation of left front tire (see picture 1) and all dependent elements. A standalone simulation of the modified model is possible, especially if appropriate forces or constraints are used to replace the removed wheel.

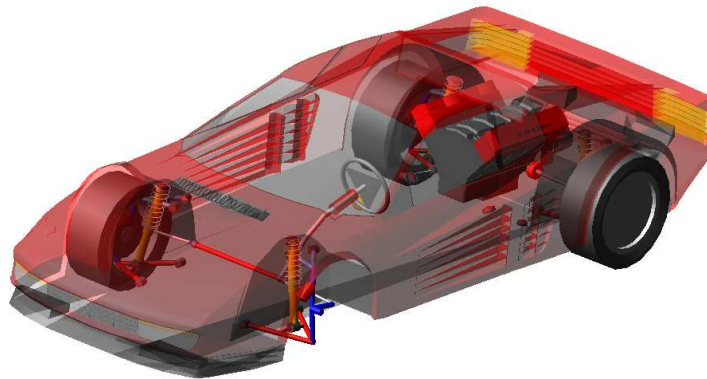


Figure 1: Modified MSC.Adams/Car model

The deactivated tire is replaced by a force element (GFORCE) acting on wheel hub (see picture 2). In a coupled simulation its value is provided by reaction forces acting on middle point of the tire in Abaqus.

The modelled car moves straightforward (straight line event in MSC.Adams/Car). After a short time a road hole is passed by the front left wheel.

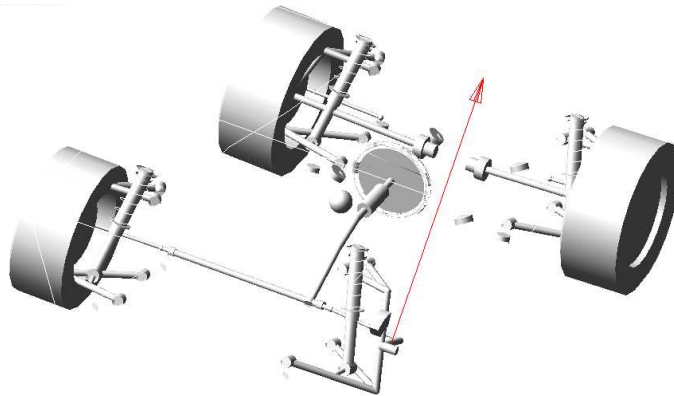


Figure 2: Chassis of the rigid body car model with applied coupled GFORCE element (red arrow)

2.2 FEM model of tire in Abaqus

Abaqus is a well known application for analysis using finite element method. This software is developed by Dassault Systèmes Simulia Corp. and is very common in industry and research.

In the considered example Abaqus simulates the tire. This model is provided by ITWM in Kaiserslautern (see figure 3). First a 2D analysis for a tire inflation is taken. DSLOAD on the inner side simulates the pressure. The tire is preloaded with 0.2 bar.

After the 2D analysis the SYMMETRIC MODEL GENERATION keyword is used to construct the 3D model of the tire. A revolution about the node in the middle is taken with partition size of 10 degrees.

An analytical surface in conjunction with a rigid body models the road. Definition of the surface is taken with TYPE=CYLINDER and uses LINE and CIRCL segments. The reference node of the rigid body is constrained in all degrees of freedom. A contact pair with TYPE=SURFACE TO SURFACE is used for the contact definition between the tire and the road.

The resulting model contains more than 12000 nodes. Used elements are C3D8R, C3D6 and SFM3D4R on the surface. The material models used are ELASTIC and HYPERELASTIC with NEO HOOKE.

With the COUPLING keyword, using the KINEMATIC option, the region of the tire, which contacts the rim, is connected to the middle point of the wheel (see figure 3). This node is also used as a coupling region to MSC.Adams/Car. The position of wheel hub in MSC.Adams provides boundary conditions for this point.

The solution process on the FEM side uses Abaqus/Standard. Both, dynamic and quasi-static simulations, are considered. The dynamic analysis uses the HHT-MD (Hilber-Hughes-Taylor with moderate numerical damping) as time integrator. The scheme of the step size control is CONSERVATIVE, which maximize the solution accuracy. The

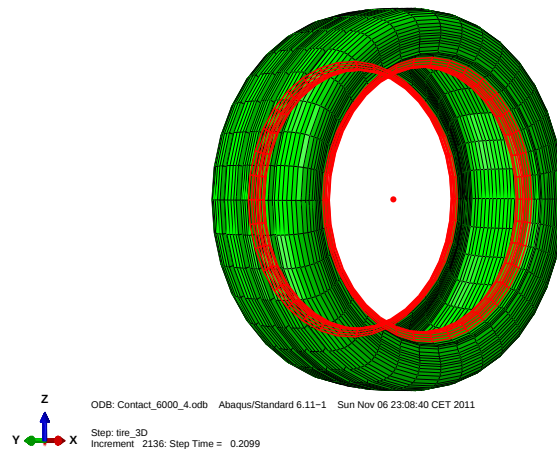


Figure 3: The coupling node in the middle of the tire and kinematic constraints to the rim contact surface

maximum step size is $1.0\text{e-}4$. In the STATIC case the STABILIZE option is used to increase the solution accuracy. The maximum step size for the STATIC calculation is set to $1.0\text{e-}3$.

2.3 Coupled simulation setup

The coupled simulation setup is done with the MpCCI - software developed by the Fraunhofer institute SCAI. This program makes usage of common interfaces of the simulation software, like user subroutines, to provide data exchange capabilities. MpCCI uses the client - server architecture (see picture 4), so coupled programs are running simultaneously and exchanging data with the MpCCI server.

The modification needed for MSC.Adams/Car is the deactivation of the left front tire (see section 2.1). In Abaqus the central point of the tire has to be defined in a node set containing only one node.

Next, a setup of coupled simulation project in MpCCI is considered. First, the user chooses the simulation programs, MSC.Adams and Abaqus. Appropriate model files define the subsystems. For MSC.Adams this are an .acf- and .amd-files. In Abaqus an .inp-file have to be defined. Next, the user selects the coupled elements. MpCCI provides functionality to automatically find possible coupling regions. This is done by pattern matching applied to model definitions. The user has to choose coupling components from a suggested list and select quantities to be exchanged. In the considered example this is the GFORCE-Element acting on left front wheel in MSC.Adams and NSET consisting of the central point of the tire in Abaqus. Exchange quantities are position and torque sent from MSC.Adams to Abaqus and force, sent in the opposite direction.

An example screen shot of setup configuration is shown in picture 5.

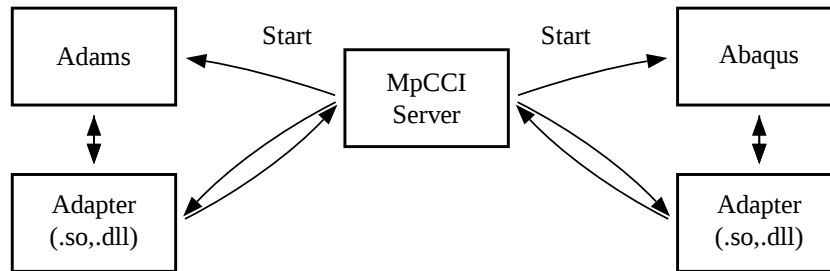


Figure 4: MpCCI architecture: MpCCI uses Perl scripts and shared libraries linked to a simulation software to provide data exchange. The coupling simulation is configured as client-server communication with support of codes running parallel.

Those are the main steps to configure a coupled simulation. Further option affecting representation of data and co-simulation methods can be also changed.

3 Co-simulation

The exchange of data includes the GFORCE element on MSC.Adams side and NSET in the Abaqus model. The modification of models are described in 2.1 and 2.2, respectively.

Both programs are running simultaneously and are connected to MpCCI server as clients. On both sides a transient simulation is considered. There are no constraints on time step, size or synchronization time points. Each simulation code uses its own error estimation and time step size control algorithms due to local model and the provided coupling values. In case of MSC.Adams it is the force and partial derivatives (for more information see section 3.2). Abaqus receives the nodal position of the wheel hub as constraint in the static case and acceleration for the dynamic simulation.

MSC.Adams uses the implemented integration control algorithms to control the step size. Those are well developed and extensively tested due to practical experience of software vendors. MpCCI provides also a capability of using a constant coupling step size. Though this is not used because of simulation performance reasons.

During the simulation each client sends exchanged data to MpCCI server. Then, in general, interpolation in time takes place to provide requested information from co-simulation partner. The solution process is only controlled by local setup of the simulation codes such as, e.g., minimal and maximal step size or error tolerance of the integrator, done by the user.

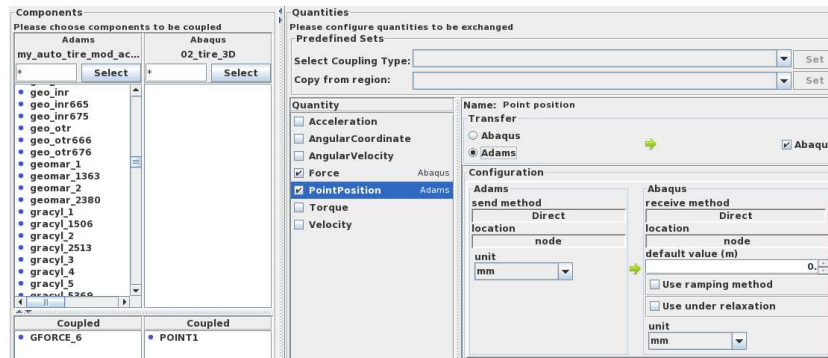


Figure 5: MpCCI setup. On the left side GFORCE element in MSC.Adams and POINT1 node set in Abaqus are selected as coupling elements. On the right side the selection of quantities to be exchanged is presented.

The main effort of the co-simulation is taken by the solution of the FEM tire. Simulation of one time step by 24 Intel Xeon cpu X7560 with 2.27 GHz takes approximately 9 seconds of total cpu time, so a simulation of 1 [s] of physical time takes about 25 hours. The selection of the communication time step size shows large effect on the solution quality, especially for the dynamic solution on the Abaqus side. In the considered example the FEM simulation, due to main effort of the problem, takes greater time step size to reduce calculation time. Those time steps are also taken for the data exchange. This solution has an advantage of more frequent communication for critical points of the simulation such as start phase or impact on the edge of the road hole. Basically a communication step size of $1.0e-4$ for dynamic and of $1.0e-3$ for static solution on the Abaqus side is taken. Smaller step sizes in the dynamic case caused by large forces due to acceleration which are not presented in the static case.

3.1 Support of iterative solution process in MSC.Adams

Data exchange between MSC.Adams and MpCCI server is done once for a single time point. Main reasons for this approach are interface restriction of the software and performance. Though internal solution algorithms are in general iterative. As consequence, there are variations of local state of both programs at single time point. The used semi-implicit approach introduces time history buffer for exchanged quantities to interpolate received values for internal state changes.

Semi-implicit methods are known, e.g. from numerical methods for solution of ordinary differential equations. For the implicit methods a system of non-linear equations arises. With semi-implicit methods a linearisation of this equations system is considered. This results in a simplified approach with lower computation costs. In the case of the co-simulation the processes also create simplified models of the partner, based on provided data.

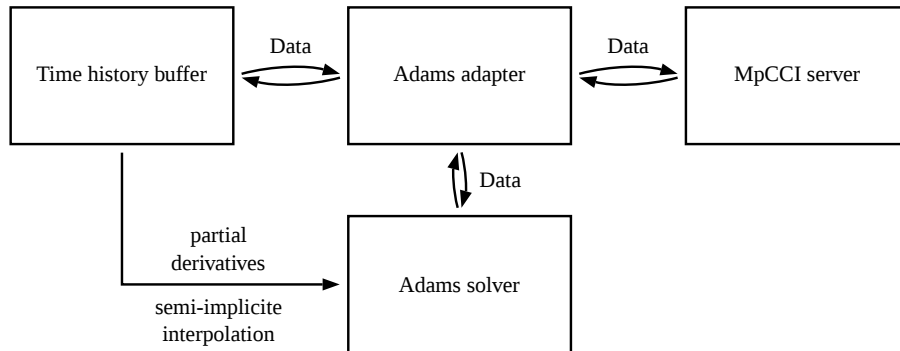


Figure 6: Scheme of co-simulation adapter for MSC.Adams illustrating implementation of partial derivatives and semi-implicit interpolation.

Currently semi-implicit approach is used only for MSC.Adams because of extended interface capabilities. The MpCCI adapter makes usage of CBKSUB subroutine, which is called for different events in the solution process. Especially `ITERATION_BEGINS` event is used to update reaction force received from Abaqus due to internal changes in the state of wheel hub, such as position, velocity and acceleration, made for current iteration. Next, the modified values are sent to MSC.Adams. The semi-implicit approach improves the stability of simulation on the MSC.Adams side, as it provides updated information on force values in each iteration of the solver.

3.2 Provide of partial derivatives in MSC.Adams

In MSC.Adams a predictor-corrector-approach is implemented to solve the equation of motion. Simulation algorithms use partial derivatives with respect to generalized coordinates for solving non linear equations, which arises during numerical time integration. In the considered example of coupled simulation this includes be also derivatives of force with respect to displacement.

An option to support partial derivatives is the so called `DFLAG`. MSC.Adams provides this flag to the subroutine to inform the user, that it calculates Jacobian matrix. For this calls small disturbances of state variables, used in the subroutine, are considered. The user is suggested to provide corresponding output values so MSC.Adams can calculate differential quotient.

In case of coupled MBS-FEM-simulation this approach means an evaluation of FE-model for a new state, which is rather expensive. In case of the used example this means simulation of a tire model with new position constraints, which is also not supported by data exchange capabilities of Abaqus.

A different approach is to save sent and received state values over time history and use this buffers to calculate derivatives. In this case the calculation of partial derivatives is made local on the MSC.Adams side without starting new calculation for Abaqus. In addition to it, special MSC.Adams subroutine SYSPAR is used. It can provide calculated partial derivatives direct to the MSC.Adams solver, so no calls of user subroutine with DFLAG occurs. The picture 6 illustrates the used approach to provide partials derivatives to MSC.Adams.

4 Results

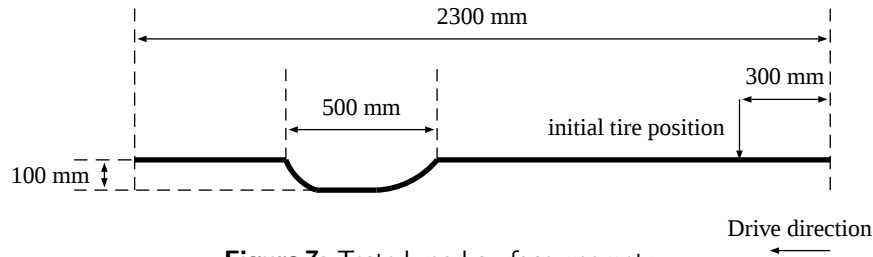


Figure 7: Tested road surface geometry

In this section first results of coupled simulation are presented. Local models are considered in sections 2.1 and 2.2. On the MSC.Adams side initial velocity for passing the road hole is prescribed. The road surface passed has an overall length of 2 meters. The geometry of the road hole is presented in picture 7. It is 10 centimeters deep and 50 centimeters long. Sides in the driving direction are rounded.

In the figure 8 reaction forces due to tire deformation are presented. The coordinate of suspension in the driving direction in millimetre is to see on abscissa. The x axis of the coupled model is defined against the driving direction. The forces [N] in vertical direction (dashed blue line) and the driving direction (red solid line) are presented on the ordinate.

In the first section between 0 and 950 millimetre settling process can be seen. The tire on Abaqus side contacts the road surface and deforms. Due to this reaction forces are applied to the node in the middle of the tire (see section 3). MpCCI provides this reaction forces to the MSC.Adams and prescribes the values of GFORCE-element in the multibody model. MSC.Adams takes account of the reaction forces and moments by the time integration and provides new position and angle to Abaqus on the next coupling time point.

In the second section between 950 and approximately 1300 millimetres the tire on the Abaqus side drives into the road hole. The forces in the vertical direction decreases as it passes the edge of the road hole and reaches zero for the loose of the contact. Simultaneously the force in the driving direction accelerates. This is due to the deformation of the tire on the edge of the road hole. As contact to the road disappears this force also goes to a zero value.

In the third section the tire reaches other edge of the road hole. The reaction forces in the vertical and the driving direction increases as it deforms. Now the force in the driving direction breaks the car. Then a settle process can be seen again. In the time interval without contact vertical position of the car decreases so the reaction forces of the tire after the road hole are higher due to increased suspension spring loading and therefore higher deformations.

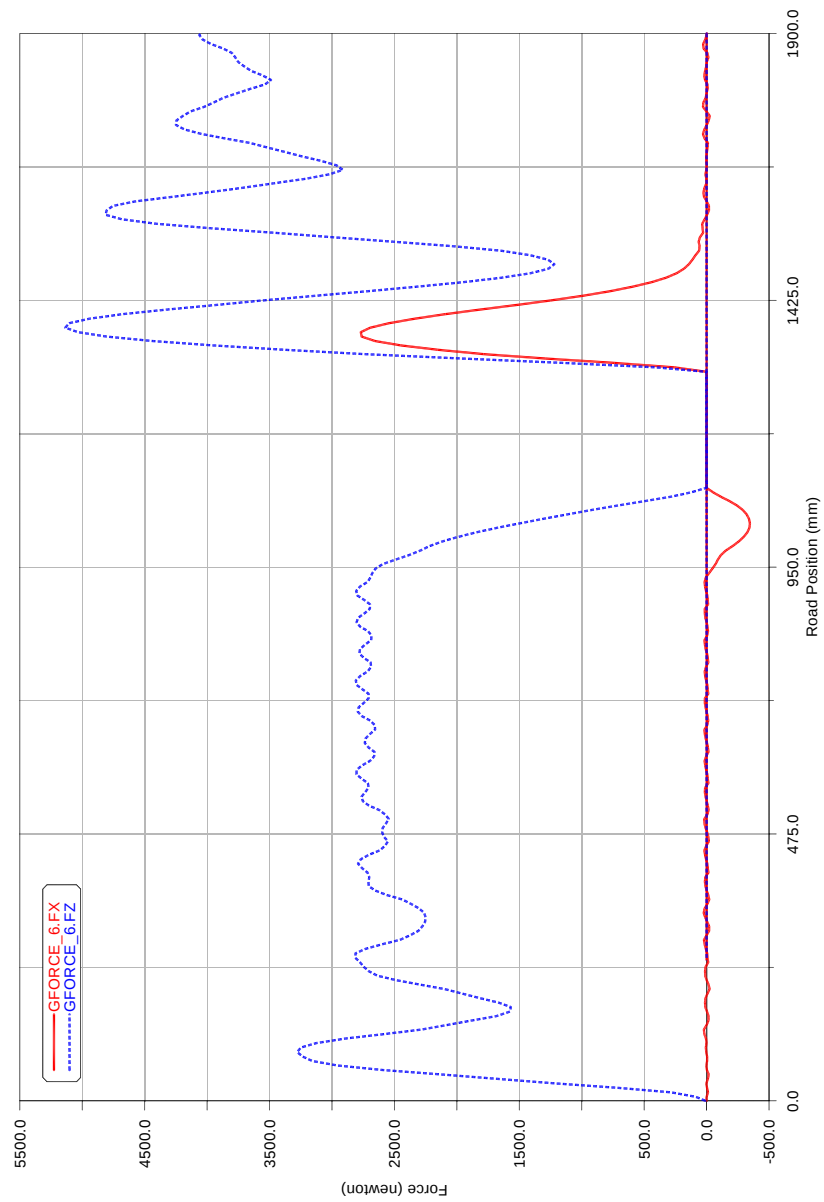


Figure 8: The forces [N] in vertical direction (dashed blue line) and the driving direction (red solid line) on the ordinate and coordinate [mm] of suspension in the driving direction on the abscissa by passing the road hole with a velocity of 10 [m/s]. Quasistatic solution on the Abaqus side.

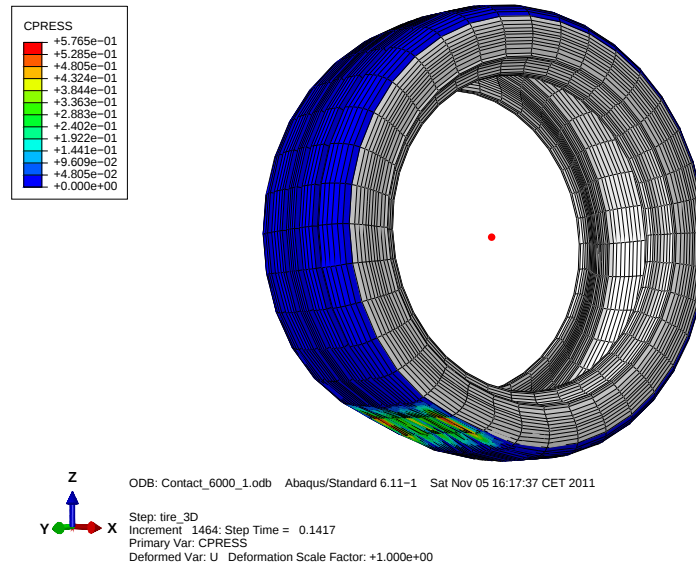
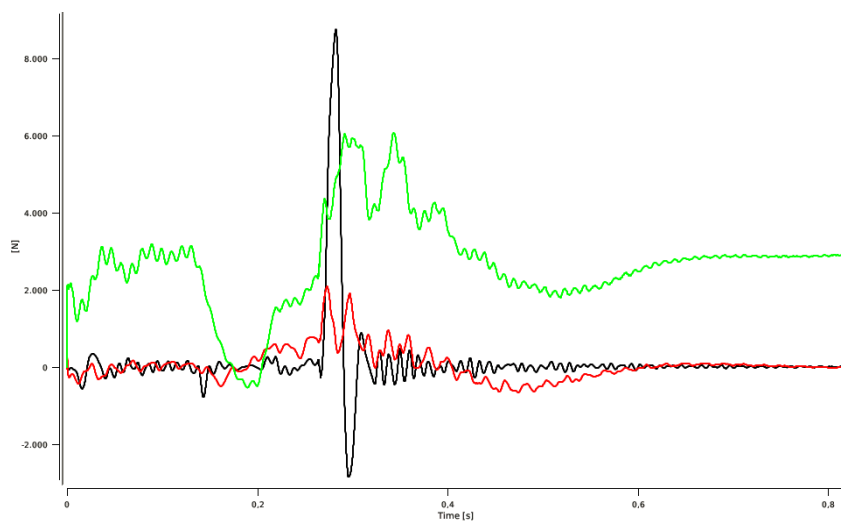


Figure 9: Deformation and contact pressure (CPRESS) of the tire in Abaqus for the time 0.1417 [s] for the quasistatic analysis.

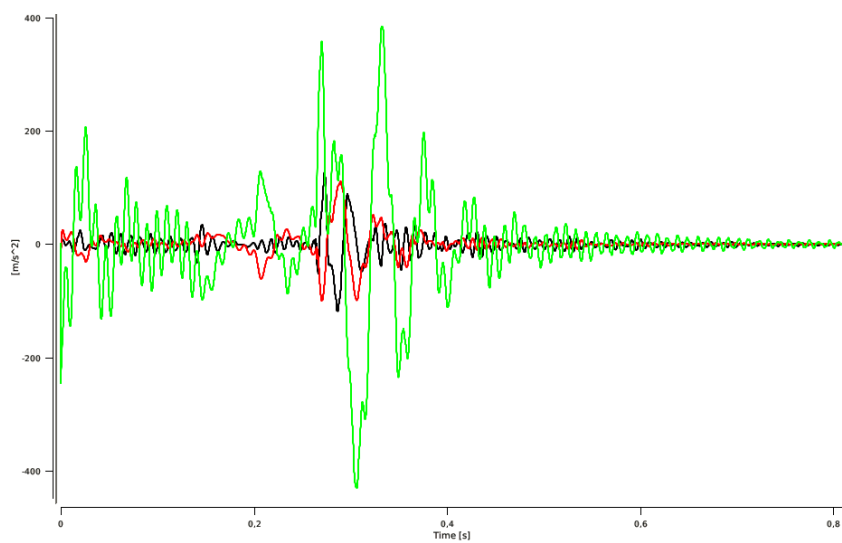
In the figure 9 deformation of the tire due to road contact is shown. The picture is taken for the time 0.1417 [s]. At this time the contact to the rigid surface (see section 2.2) is re-establishing. The color scheme represents the contact pressure between the tire and the road surface. The peak value is approximately $5.765 \times 10^{-1} \text{ [N/mm}^2\text{]}$.

In the figure 10 the values of the reaction forces and accelerations of the wheel hub are presented. Time from 0 to 0.8 [s] is shown to demonstrate a complete simulation. Accelerations and forces for the area of the road hole are presented in figures 11 and 12 for 7 [m/s] and 10 [m/s] respectively.

Compared to the static solution on the Abaqus side, a great effect of the acceleration on calculated reaction force can be considered. The suspension system on the MSC.Adams side has a stiffness of 125 [N/mm] , which cause very large accelerations for the wheel hub and therefore large reaction forces calculated by Abaqus. E.g. reaction force in the driving direction by the contact re-establishing is larger for the dynamic calculation, because of acceleration of the wheel hub is also considered. For the dynamic simulation on the Abaqus side the mass of the system increase to approximately 16.39 [kg] compared to 1.39 [kg] for the static simulation, for which only the mass of the wheel hub on the MSC.Adams side is considered. This cause in a larger amount of energy is saved by the complete suspension system and results also in higher oscillation amplitude.

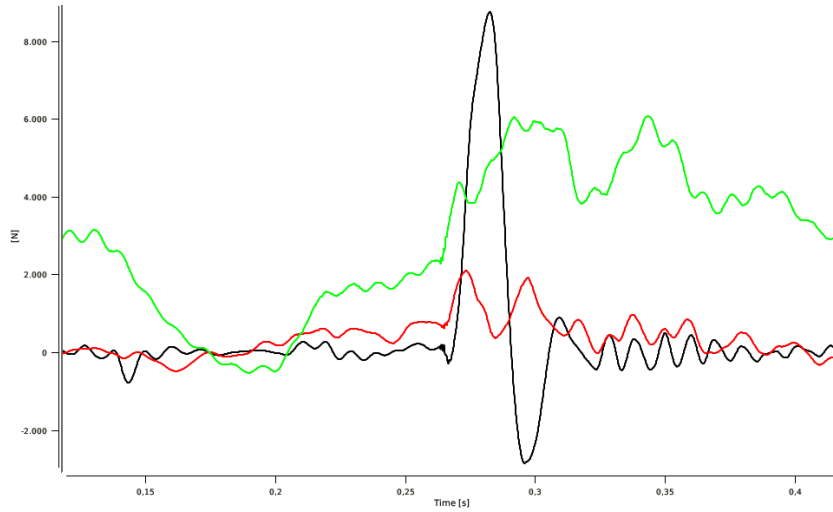


(a) The forces [N] on the ordinate for travel (black line), vertical (green line) and lateral (red line) direction over time {s} on the abscissa.

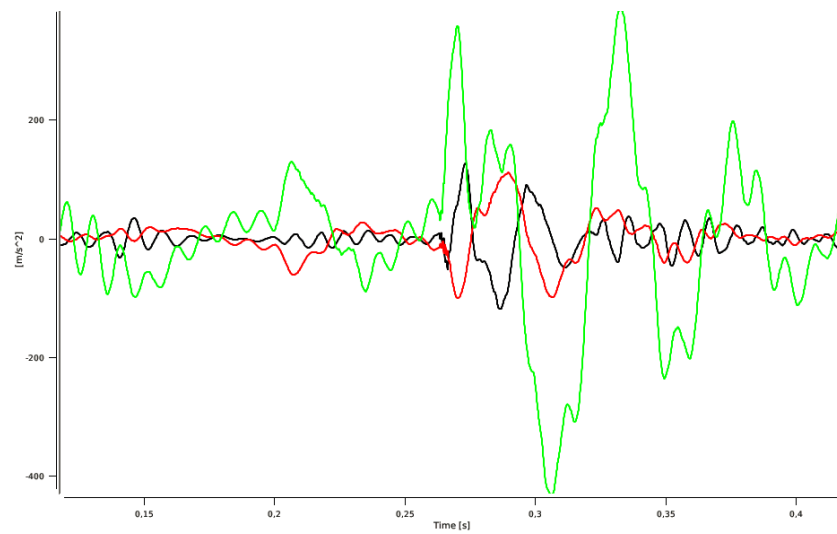


(b) The acceleration of the wheel hub $[m/s^2]$ on the ordinate for travel (black line), vertical (green line) and lateral (red line) direction over time [s].

Figure 10: Simulation output by passing the road hole with a velocity of 7 $[m/s]$ with a dynamic solution on the Abaqus side.

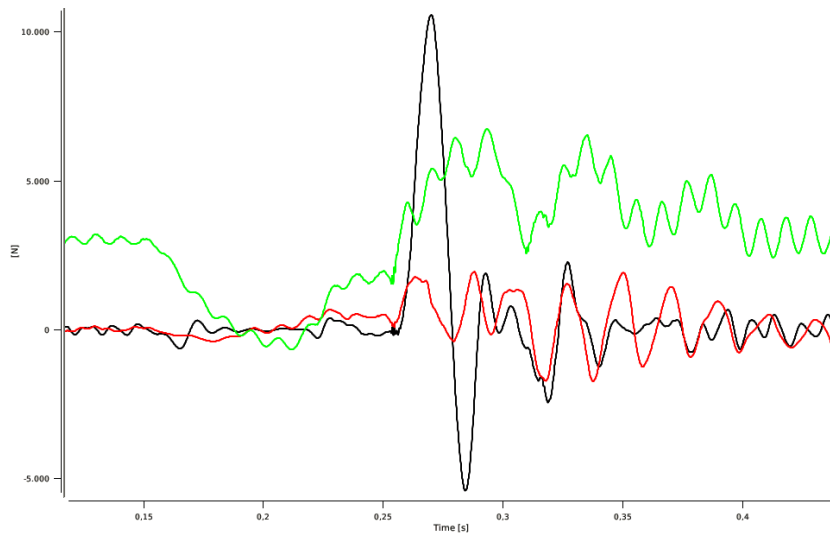


(a) The forces [N] on the ordinate for travel (black line), vertical (green line) and lateral (red line) direction over time [s] on the abscissa.

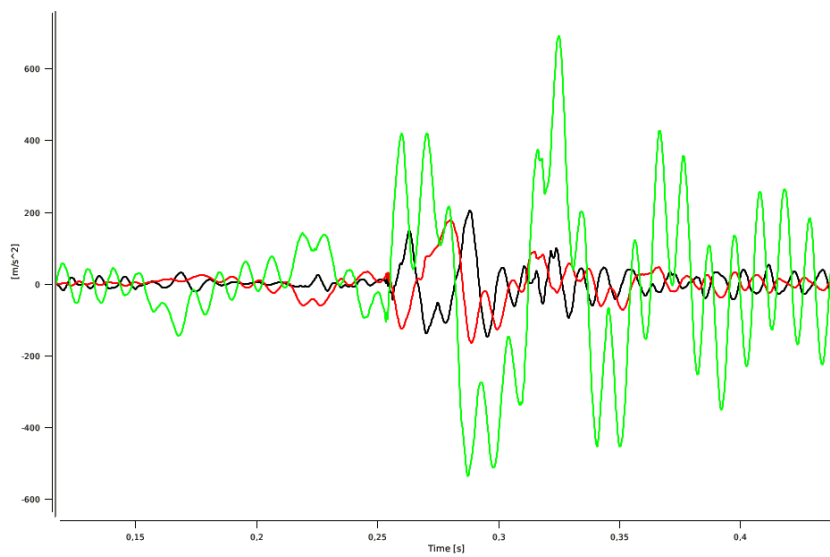


(b) The acceleration of the wheel hub $[m/s^2]$ on the ordinate for travel (black line), vertical (green line) and lateral (red line) direction over time [s].

Figure 11: Simulation output by passing the road hole with a velocity of 7 [m/s] with a dynamic solution on the Abaqus side. Zoom for the road hole area.



(a) The forces [N] on the ordinate for travel (black line), vertical (green line) and lateral (red line) direction over time [s] on the abscissa.



(b) The acceleration of the $[m/s^2]$ on the ordinate for travel (black line), vertical (green line) and lateral (red line) direction over time [s].

Figure 12: Simulation output by passing the road hole with a velocity of 10 [m/s] with a dynamic solution on the Abaqus side. Zoom for the road hole area.

5 Conclusion and outlook

A coupled MBS-FEM-simulation with usage of MpCCI is presented. The considered example is the passing of a road hole by a car. Due to large deformations, the tire is modelled with FEM approach in Abaqus. The remaining vehicle is modelled in MSC.Adams/Car as a multibody system. With the coupled simulation, deformation and forces of the tire can be considered in conjunction with complex dynamic boundary conditions provided by associated multibody system.

First simulations results are presented. Reaction forces provided by Abaqus and displacement provided by MSC.Adams show both complex and nonlinear behavior, which motivates further studies of coupled simulation, especially in case of MBS-FEM combination.

Next planned steps include quantitative comparison of coupled to a simulation done by a single program. So better separation of appropriate application field for coupled simulation is possible.

In the case of the considered example more simulations are planned to exploit complex behaviour and important parameters and effects. In addition better local modelling is aimed. Especially further cooperation with automotive OEM's and Fraunhofer ITWM are planned. In this context also appropriate verification of co-simulation results is to be made.

References

1. Holger Wendland, Scattered Data Approximation Cambridge University Press, 2005
2. Eich-Soellner, E., Führer, C., Numerical methods in multibody dynamics B.G. Teubner Stuttgart, 1998
3. Schwarz, H. R., Köckler, N., Numerische Mathematik (6., überarb. A.) Vieweg+Teubner, 2008
4. Busch, M., Entwicklung einer SIMPACK-Modelica/Dymola Schnittstelle Universität Halle, Institut für Informatik - diploma thesis, 2007
5. Busch, M., Schweizer B., MBS/FEM co-simulation approach for lubrication problems. Proc. Appl. Math. Mech. (PAMM), 2010 10:729-732
6. Dassault Systèmes Simulia Corp., Abaqus manual, 2011
7. MSC.Software, MSC.Adams manual, 2011
8. Fraunhofer SCAI, MpCCI manual, 2011