DEEP WATER WADING SIMULATION OF AUTOMOTIVE VEHICLES

Dr.-Ing. Prashant Khapane, Uday Ganeshwade, Jagan Senapathy
(Jaguar Land Rover, United Kingdom);
Dipl.-Ing. Ilja Kalmykov, Ing.-Inf. Pascal Bayrasy, Klaus Wolf
(Fraunhofer SCAI, Germany);

1. Abstract

In automotive engineering, vehicle wading refers to a situation where vehicle traverses through water at different speeds. Vehicle water wading is a standard test procedure for JLR vehicles. Test procedure consists of set of combinations with different depths of water level and entry speeds in to the wading trough. The different failure checks are the key indicators for the physical test scenario.

Lack of CAE capability for the wading test resulted in late detection of failure modes with unknown reasons, leading to expensive design changes potentially affecting the vehicle program timing. The need to find these reasons affecting the failure mode motivated JLR to develop a CAE method.
JLR has developed a state of the art method which is patented to simulate the physical test procedure virtually using CFD tool, STAR-CCM+. The current method is employed effectively to design and analyses the under floor and engine bay components.

Existing method has also got some limitations. One of the major limitation is computing the inertial field of the vehicle while wading. To address this limitation of the hydrodynamic forces due to the inertial field. A new method of co-simulating CFD and MBD is being employed. JLR and Fraunhofer Institute SCAI together have developed this method employing co-simulation engine developed by Fraunhofer named MpCCI (Multi-Physics Code Coupling Interface). This co-simulation communicates the MBD calculated instantaneous positions and velocity to the CFD model. The flow field is resolved calculating the effective fluid forces and torques. These forces and torques are feed back into MBD model. This two way communication is enabled with MpCCI.

Keywords: Computational Fluid Dynamics - Multiphase and phase change, Multi-Body Simulation - MBS, FE, and the co-simulation of both, Multi-physics - Coupled Simulations, Multiphysics - Fluid-Structure Interaction, Multi-physics - Multi-physics, Multi-scale, Multi-fidelity Modeling

2. Introduction

Vehicle wading at different depths of water and at different vehicle speeds is an important test procedure for a vehicle development program at Jaguar Land Rover (JLR). The test procedure looks at functional integrity of various components in the vehicle such as bumper, engine undertray, and transmission scoop, radiator, plastic sills etc. when traversing through water. Virtual method developed by JLR to simulate the physical test scenario is based on CFD code. In CFD code the motion was modelled with a rigid body motion. The CFD code was calculating the pressure field with only CFD rigid body motion. The pressure field captured was not considering the inertial field of the vehicle. In real life conditions when the vehicle is wading the inertial field play’s prominent role. As the vehicle positions change in vertical direction and the vehicle decelerates. These two conditions change the entire pressure field on the floor of the vehicle.

Few limitations in the current CFD modelling

Splash modeling;

Splashing of water which is a complex behavior to model and it is observed during high speed wading, we are currently exploring new ways to model splash during our simulation to bring it closer to reality.

Hydrodynamic force;

Hydrodynamic inertial field of force is generated due to the rotation of wheels and the impact of the vehicle floor on water surface. As of now this effect is not considered in standalone CFD simulation.

Vehicle suspension behaviour;

When the vehicle is traversing through water at different depths and speeds due to force acting on the vehicle components, it is understood that there is definite movement in suspension components affecting the traction of the vehicle. Ride height of a mid-sized SUV driven through the wading trough profile for 8 [km/h] is as shown in Figure 1, this illustrates that dynamic behaviour of the suspension is an important factor which needs to be considered during virtual simulation of wading.

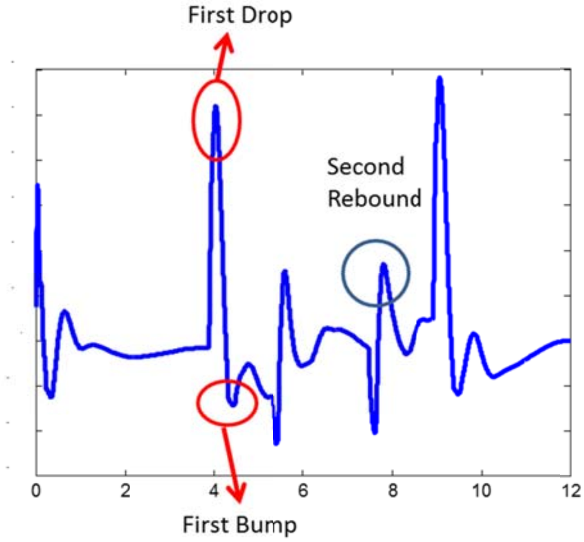

Figure 1 dynamic ride height of mid SUV driven through wading profile

Our continuous quest to improve the existing simulation method and to address two of the most important limitations in our method namely dynamic force and vehicle suspension behavior as described above, we came to conclusion to simulate wading using multi physics approach. Approach was to do a coupled simulation between **CFD** and **MBD**. Software's employed were **S**impack for Multi-body simulation and **STAR-CCM+** for **C**omputation **F**luid **D**ynamics

3. Co-simulation methodology

During the coupled computation both programs run in parallel. MpCCI acts as a server and controls the co-simulation. The data is exchanged on the synchronization time points, which are in general loose and not equal for the co-simulation clients.

MpCCI tries to affect the set-up of the coupled programs as little as possible. The most important co-simulation parameters, e.g. the synchronization time step, co-simulation scheme, job parameters, can be specified in the MpCCI GUI. However no parameters, especially the solver set-up (e.g. which solver is used, error tolerance, maximum time step), is not affected by the MpCCI or the co-simulation implicitly. Each simulation code uses its own error estimation and time step size control algorithms due to local model and the provided coupled values. The algorithms to control those parameters are very well developed and tested due to practical experience of software vendors.

MpCCI provides a number of physical and auxiliary (e.g. time-step-size) quantities which can be exchanged. The quantity must be supported by the simulation program. For the considered co-simulation between STAR-CCM+ and SIMPACK we exchange the kinematic quantities of angular and translational velocities and reaction torques and moments. SIMPACK provides the kinematic quantities and receives the forces and torques from STAR-CCM+ on the other side as depicted in **Figure 2**.

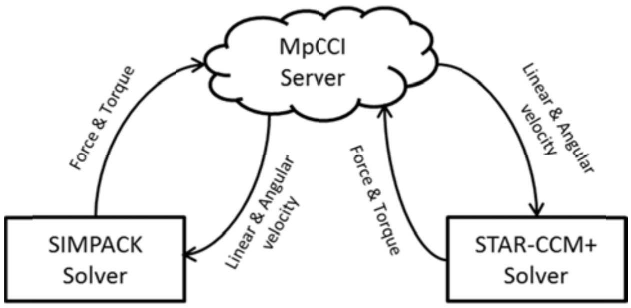

Figure 2 Coupled physical quantities for co-simulation

To improve the performance, adaptive time steps are used on both sides. During the simulation each client sends exchanged quantities to the MpCCI server. Then interpolation in time takes place to provide requested information.

SIMPACK

In SIMPACK, full vehicle multi-body model with car body, suspension components and tires is driven through the wading trough profile. In Simpack for co-simulation purposes an additional force element is written by Fraunhofer using sub routines. This force element exchanges physical quantities with MpCCI as depicted in **Figure 2** i.e. we can extract the velocities/angular velocities and forces/torques needed to drive the co-simulation.

STAR-CCM+

In STAR-CCM+, full vehicle model with detailed underbody components is driven through the wading profile with water. Motion to the car body is imparted employing mesh overset method. The movement and current orientation of the car are calculated in the MBS code SIMPACK.

Computation time/power for solving CFD simulation is much higher compared to MBS. To reduce the total simulation time, we use an adaptive time step size for STAR-CCM+. The time step size is controlled by the CFL number. This improvement is rather easy to implement, but very important for the practical simulation on the other side.

Advanced co-simulation method: Semi-implicit approach

Data exchange between the simulation code and MpCCI server is done for every single time step. Main reasons for this approach are interface restriction for client software and performance. Though internal solution algorithms are in general iterative.

In SIMPACK a predictor-corrector-approach is implemented for the stiff-solvers to evaluate the equation of motion. This algorithm makes usage of partial derivatives for solving nonlinear equations, which arise during numerical time integration. In the considered example this would be derivatives of force with respect to velocity. The solver has to calculate the Jacobian matrix. This is done with the finite difference scheme. As consequence, there are variations of local state at single time point, which have to be considered. In the case of coupled MBS-CFD simulation this means evaluation of the CFD-model for a new state. This is rather expensive. In our example this means simulation of the STAR-CCM+ model with new kinematic constraints. This is also technically sophisticated. One has to use the restart capabilities of STAR-CCM+. The other approach is to introduce an approximation for exchanged quantities. Those then can be used to interpolate received values for internal state changes due to iteration at single time point.

Currently the semi-implicit approach is used only for SIMPACK because of extended interface capabilities. MpCCI SIMPACK adapter saves the solution history of the co-simulation and calculates an approximation of the behavior of the exchanged quantities. This approximation can be then used for the iterative solving process in SIMPACK. Semi-implicit approach improves stability of the co-simulation, especially on the SIMPACK side, as it provides updated information about the force values in each iteration of the solver.

4. Current Approach

To validate the theoretical understanding of co-simulation, we simplified the model in Simpack and Star-CCM+. In MBS domain model chosen is a simple block with four wheels, with suspension i.e. springs directly connected to the block driven over standard wading trough profile as depicted in Figure 3.

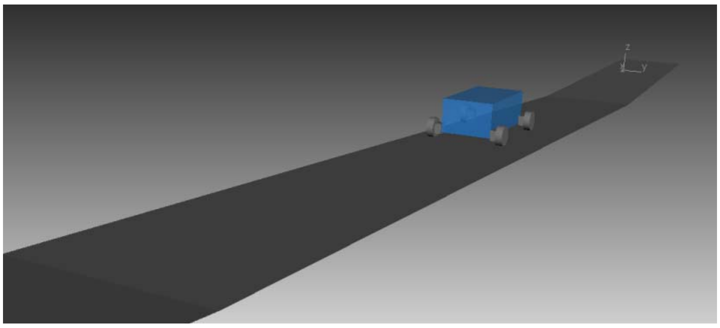

Figure 3 Simplified block travelling through wade trough in Simpack

In CFD domain model chosen is a simple box traversing through the trough with water as depicted in Figure 4. Schematic diagram with physical quantities that are being exchanged between Simpack and Star-CCM+ via MpCCI is depicted in Figure 5

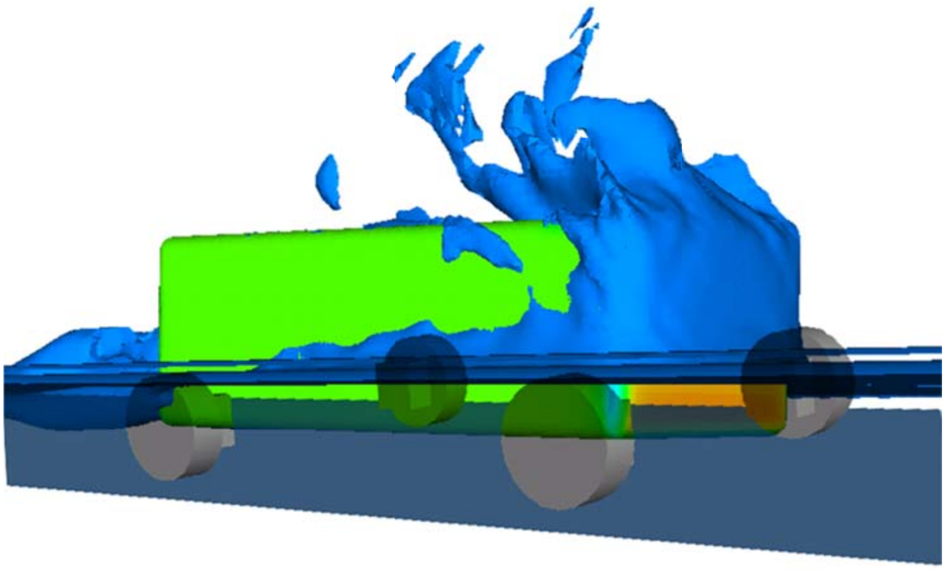

Figure 4 Simplified block travelling through wade trough in Star-CCM+

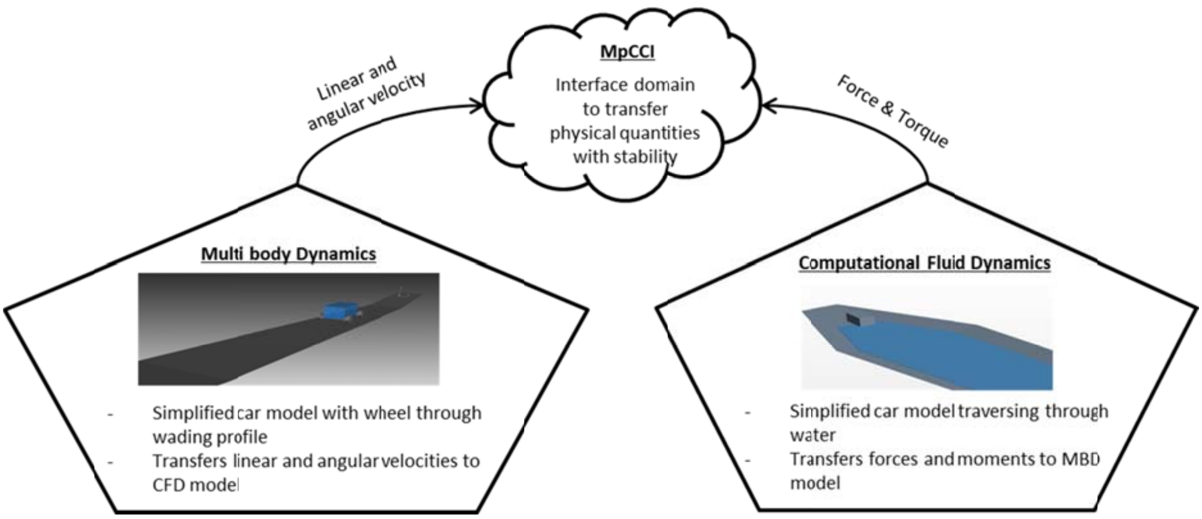

Figure 5 Schematic illustrating co-simulation

5. Preliminary results

During the co-simulation, two important phenomena were noticed; one being the damping effect of water on forces generated/observed on different components of the simple block i.e. when the block is traversing through water in downward slope of wading trough due to the presence of water around the block there was reduced amount of force acting at the components. Forces acting at tire and spring (force elements) components are depicted in Figure and Figure , amount of force generated in co-simulation is less relative to that from standalone Multi-body simulation since the surrounding water is acting like a dashpot.

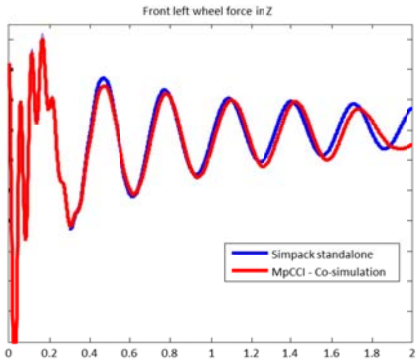

Figure 6 Front left wheel vertical (z) force

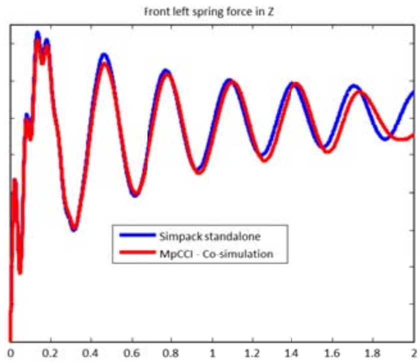

Figure 7 Front left spring force in vertical (z) direction

The second phenomenon comes into picture just after the first phenomenon. As the vehicle goes deeper in to the water trough, contact area of the block surface with water increases, here vehicle/block tries to push the water but water being incompressible and heavier (relative to air) generates a reaction force on the block. This reaction force is in the form of pressure acting on the block, the force acting on the block causes it to loose/reduce the traction on the surface of the profile resulting in aquaplaning. This phenomenon repeats itself until the reaction forces fall short of the gravitational forces. It is usually seen that vehicles at high speeds usually enter the trough with a couple of bounces.

Figure 6 and Figure 7 illustrate the second phenomena wherein, just after two seconds simulation time, the block is lifted up due to hydrodynamic force acting on it. It is observed that after this impact, solvers on both ends were getting unstable due to high amplitude of forces/torques generated in Star-CCM+. We are currently trying to resolve this issue by different ways one of them being better modeling of block where the edges of the block are smoothened (rounded).

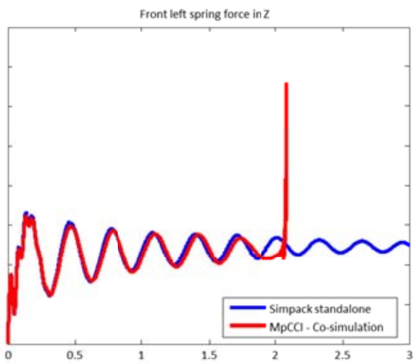

Figure 6 Front left spring for in vertical (z) force direction

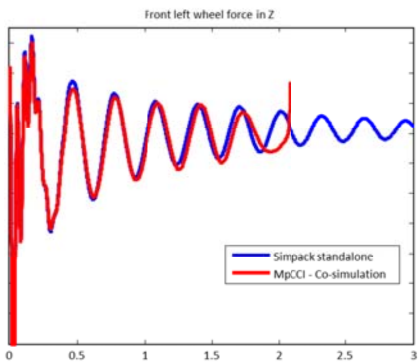

Figure 7 Front left wheel vertical force in Z

Due to discrete communication time points and different solver time steps in SIMPACK and STAR-CCM+, the co-simulation can in general increase the impact of the errors of the subsystems on the results. Therefore we intend to get a smoother reaction from STAR-CCM+ by using a model with smoothened edges. It is observed that the effect of CFD solution on co-simulation is larger than MBS< especially in the first phase of wading, we are working on improving the STARCCM+ model.

Another approach is to increase the numerical stability of the co-simulation. E.g. the impact of the communication time steps, exchanged quantities or solver used in by the clients. Here a robust method, with little impact from the iterative process on the convergence of the solution, would be preferred.

Central idea is to overcome the initial obstacles as described above and to validate the method for a simple block, then carry forward this method to afull vehicle model and demonstrate the capability of co-simulation. We intend to discuss results from on-going investigations during the conference.

Central idea is to validate the method for a simple block and apply this method to a full vehicle model and demonstrate the capability of co-simulation. We intend to discuss results from on-going investigations during the conference.

6. Conclusion

Load cases like vehicles wading through deep water have lead to a new approach with a coupled simulation setup. Whilst all the detailed effects of water waves and splashes can be simulated in a standard CFD code the jumping behaviour of the car due to hydrodynamic forces and vehicles reaction when diving into water trough is calculated in a MBS code.

In this paper an approach for the vehicle wading simulation, based on coupling of multiple codes, is presented. Due to the complex physics of the simulation process several effects have to be considered in detail. For this reason, different codes for the simulation process will be used in a coupled approach. The focus of this paper will be on the technical issues to realise such a new type of co-simulation. This application is a kind of fluid-structure-interaction where the structural analysis has been replaced by a simpler but faster elastic MBS. The MpCCI coupling environment provides some schemes to calculate the integrated forces from the wet CFD surface regions and map them on single bodies in MBS. Multiple coupling regions can be used to calculate the different loads onto under-hood parts, wheels, etc.

7. References

Khapane. P, Ganeshwade. U (2013). “Wading Simulation – Challenges and Solutions – NAFEMS 2013”

Khapane .P, Ganeshwade .U (2014). “Wading Simulation – Challenges and Solutions – SAE International 2014”