Fluid-Structure Interaction of Racing Car Spoilers

Bettina Landvogt (scapos AG, Germany)

1 Introduction

In all racing contests – most prominently in Formula 1 – there is a huge and close competition between the different teams and drivers. Minor technological advancements might lead to the crucial advantage to win the competition.
On the other hand, the design of racing cars – covering all aspects from the construction of the car body to the engine – is strictly regulated to ensure a safe and just competition. To create better performing cars within the range of regulations, engineers have to rely not only on extensive tests in wind tunnels and on race tracks, but also on numerical simulation.

Many different features can be investigated using numerical simulation: the engine or the exhaust systems, the external aerodynamics of the car or braking systems. Just like wind tunnel hours or the amount of money to be spent on a car, Formula 1 regulations restrict the CPU time to be used for these numerical investigations.
Therefore, the performed simulations should be as realistic as possible and incorporate as many of the physical effects as possible and feasible at once. One key aspect of realistic, multiphysical simulations are fluid-structure interactions.
Particularly front, rear or side spoilers can deform while the car is driving at high speeds and thereby influence the aerodynamic properties. As an example, a simple racing car model will be used here to investigate the fluid-structure interactions occurring at the rear spoiler and the influence on the general aerodynamic properties of the car.

First, the respective Computation Fluid Dynamics (CFD) and Finite Element Method (FEM) stand-alone simulations – for the external flow of the full car and only the rear spoiler, respectively – will be presented. The CFD simulations are performed by STAR-CCM+ or OpenFOAM; Abaqus is used for the FEM simulation. Finally, the coupling – using Fraunhofer SCAI's independent code coupling tool MpCCI – is described. Results for different coupled simulations are presented.

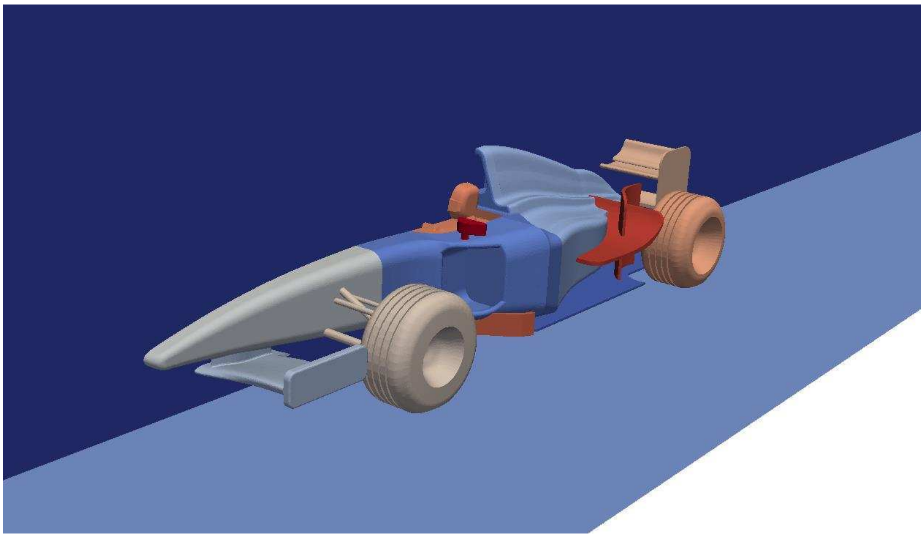

Fig. 1: Symmetric half model of simple racing car

2 CFD Simulation

The used simple racing car model can be seen in Fig. 1. The symmetric half model of the car is 8 m long and is positioned in a channel of a length of 30 m.
Different meshes are investigated: tetrahedral and hexahedral (trimmed) meshes with or without prism layers. The number of cells is – depending on the used configuration – between 9 and 15 million.

Multiple inlet conditions reflecting different driving speeds of the car are investigated. The turbulence is modelled using either the k-epsilon or the k-omega model. The air around the car is assumed to be incompressible.
Using steady state simulations the general aerodynamic performance of the car can be investigated. To take time varying effects – like overtaking, braking and steering – into account, transient simulations are performed. Both simulation types can be integrated in coupled FSI simulations later on. OpenFOAM (version 2.2.1) and STAR-CCM+ (version 11.02.009) have been used for the CFD simulations.

For the optimization of a racing car CFD simulations are a very important and well investigated topic. Many different settings and numerical possibilities can be tested and adapted to the desired configuration. When coupling with FEM, the CFD simulation usually is the significantly more complicated problem, being responsible for most of the running time of the coupled simulation.

3 FEM Simulation

The Finite Element Model is usually a lot less complicated than the CFD model and consists only of certain parts of the car. In this case the rear spoiler section of the simple racing car will be used. The FEM stand-alone model contains the whole rear spoiler (confer Fig. 2), but only the top two spoilers will be used as a coupling surface for the fluid-structure interaction in the next section because the pressure load on the top two spoilers is considerably higher than on the third spoiler.

The complexity of the FEM model depends on the exact construction and design of the layout of the spoilers and on the used materials. For the simple racing car model, the spoiler is assumed to be made uniformly of a linearly elastic material. For more realistic models, the beam structure inside the spoiler could be integrated into the FE model.
The density of the material is assumed to be 1000 kg/m^3. The Young's modulus of the employed material is 900 N/mm^2 and the Poisson number is 0.3.

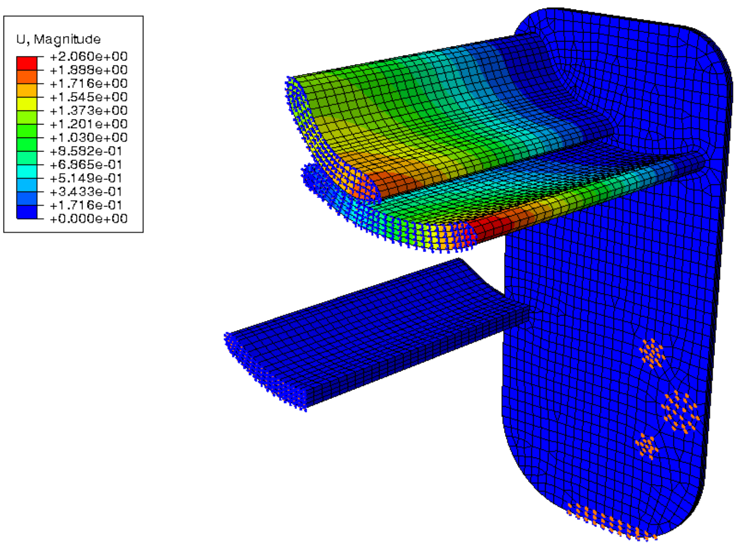

Fig. 2: Abaqus model of the rear spoiler with deformation results of static load tests (according to Formula 1 regulations). Maximum deformation of 2 mm is computed. Symmetry and fixed boundary conditions (on the side panel) are visible.

These material properties have been selected such that the spoiler deformation complies with Formula 1 regulations. The regulations specify several tests for the top two spoiler elements: when horizontal or vertical loads of a specified size are applied, the spoilers are only allowed to deflect in a certain range. These static load tests are emulated in Abaqus with different material properties to find a setting where the biggest still allowed deformation is reached. A result of such a computation for the rear spoiler can be seen in Fig. 2.

Symmetry and boundary conditions are set up (confer Fig. 2) and – just like for the CFD part – different types of analysis can be investigated during the FSI simulation: static or dynamics steps with any desired incrementation schemes. Abaqus (version 2016) is used for the FEM simulation.

4 Coupling FEA and CFD simulation with MpCCI

As a starting point for a coupled FSI simulation, the stand-alone CFD and FEM models need to be setup and tested independently.
The following coupling setup can be done with the MpCCI GUI: the coupling surfaces need to be defined – in this case the surface of the two top rear spoilers (confer Fig. 3) – and the coupling quantities need to be selected. As this is a classic fluid-structure interaction the wall force (or pressure) will be sent from CFD to FEM, and the new updated position will be sent in the other direction.

Furthermore different selections concerning the coupling algorithm can be made: two transient, two steady-state or one steady state and one transient simulation can be coupled using MpCCI. Further possibilities include ramping or relaxation of the exchanged quantities or the choice between a serial and a parallel coupling scheme. Depending on the coupling algorithms, both codes might need to subcycle, i.e. to compute a fixed (or adaptive) number of steps without exchanging data to achieve a converged interim solution.

Because of the FSI coupling the CFD simulation code needs to adapt the mesh using the newly computed Abaqus deformation. When using STAR-CCM+ this is automatically handled by the available morphing tools in the CFD code.
For coupling with OpenFOAM, the MpCCI Mesh Morpher tool is used. This spring-based morphing tool can be used to incorporate the deformations in the OpenFOAM mesh. MpCCI Mesh Morpher will not work on the whole CFD mesh but only on a much smaller area, lying close to the coupled surfaces. This reduces the memory and CPU time requirements significantly.

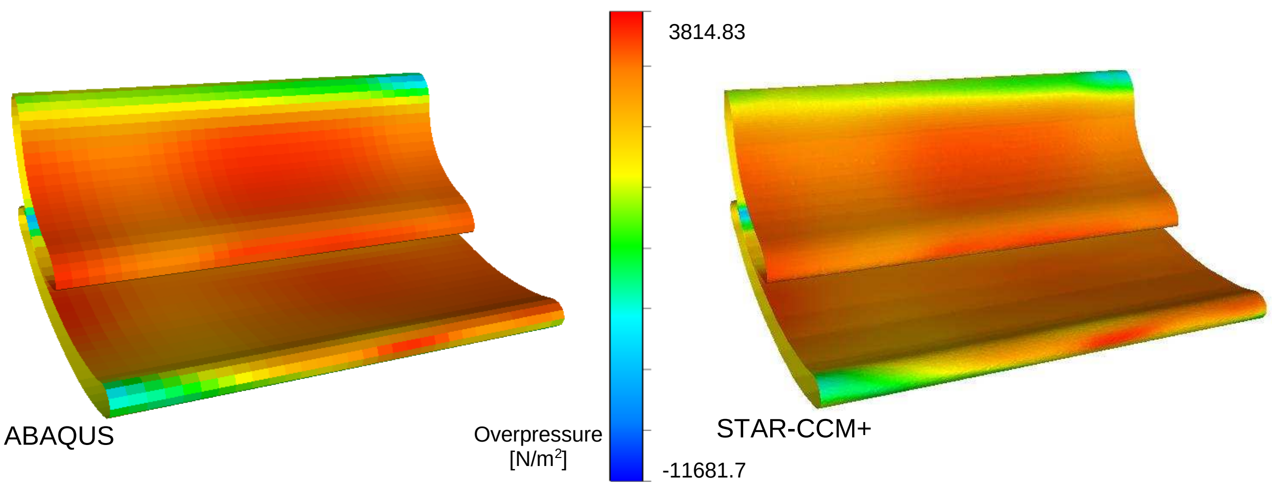

Fig. 3: Exemplary result of an FSI simulation, using Abaqus and STAR-CCM+. Overpressure is plotted as contour on both meshes. STAR-CCM+ uses a very fine trimmed mesh discretization. Abaqus a much coarser hexahedral mesh. Inlet velocity in STAR-CCM* is 80 m/s.

Fig. 3 shows an exemplary result of a coupled FSI simulation. In this case STAR-CCM+ and Abaqus were used. Both simulation codes were computing a steady state solution and the inlet velocity in STAR-CCM+ was 80 m/s. The contour shows the “overpressure” on the two simulation models. For this case, a very significant maximum deformation of more than 6 mm was observed.

This shows that aerodynamic properties of the racing car – in particular the drag or lift coefficients – are influenced by fluid-structure interactions. To find an optimal layout for a car which is driving at a high speed an FSI simulation is necessary because the geometric changes due to the CFD load on the car are big enough to have an influence on the aerodynamic performance of the car. A spoiler with a very good performance in a CFD stand-alone simulation might not be the best choice when taking the deformation computed with FSI simulations into account.

5 Conclusion

To summarize, FSI simulations for critical parts of racing cars provide interesting insights into the car’s aerodynamic performance. Especially rear spoilers or front wings deflect under the pressure load from the surrounding air. This can only be simulated with coupled CFD and FEM simulations.

With MpCCI the stand-alone CFD and FEM simulation models can be used almost without further modifications and many different CFD and FEM simulation codes are supported. Most computational time is spent on the CFD model which is usually quite large. To achieve optimal performance, the whole coupled simulation – including the CFD and FEM codes and MpCCI– can be integrated in a cluster workflow.

Other aspects of modern racing cars could also be simulated using coupled analyses: investigations of driver-adjustable bodywork, like the currently used “Drag Reduction System”, might be interesting. The mechanism to move the flap and thereby increase the top speed might, for example, be simulated using a Multibody Simulation software and then coupled with the CFD and FEM simulation.