Fluid Structure interaction of a spoiler on the DrivAer car model

James Dilworth, Ben Ashby, Peter Young

Arup

Abstract

Fluid structure interaction problems appear in a wide range of industries, including automotive, marine and aerospace. In the automotive industry, the drive to make components lighter can also reduce their stiffness, causing them to deflect significantly under aerodynamic loads. The deflections can affect the aerodynamic properties of the vehicle, cause dynamic fluctuations that are visible to the driver, or even lead to failure.

The Incompressible Computational Fluid Dynamics (ICFD) solver in LS-DYNA® is well suited to simulating fluid structure interaction as the code provides a range of robust and easy to use coupling algorithms and both solid and fluid solver can be readily accessed from within the same simulation environment. In this study, a lightweight spoiler has been added to the Technical University of Munich (TUM) DrivAer car model [1]*, and the deflection of the spoiler under aerodynamic load has been studied. The pressure profile on the car without the spoiler has been compared to experimental results. The spoiler is found to deflect significantly under aerodynamic loads due to turbulent eddies shed from the rear window of the car.*

Introduction

Understanding the effects of Fluid-Structure Interaction (FSI) on structural dynamics is becoming increasing important in the automotive industry. Car manufacturers are trying to find ways of reducing the weight of their vehicles in order to reduce carbon emissions and meet legislative targets. Reducing the weight of components does not come without a cost, however, as lightweight components are more likely to deflect or vibrate under aerodynamic loads. This is particularly true if when travelling in the wake of another vehicle, as the turbulent eddies from the leading car will cause an increase in pressure fluctuations on the aerodynamic surfaces of the following vehicle.

In recent years there have been a number of instances of production cars which are affected by bonnet flutter. It appears that these issues have occurred when the vehicles were driving at moderate to high speeds (60-80mph). Bonnet flutter of this sort is clearly distracting and unnerving for drivers, so a number of manufacturers are making efforts to analyse FSI phenomena pre-production to avoid the risk of flutter becoming apparent in production vehicles.

Fluid-structure interaction phenomena can be difficult to analyse for a number of reasons, but particularly due to the need to simulate both a solid and a fluid domain. In the automotive industry, it is very common to use different software packages for fluid simulation and solid mechanics simulation. This creates a problem for simulating FSI, as the outputs from each software package have to be mapped back and forth between the two solvers, in the hope that they will converge to a steady state. This does not allow for realistic coupling as computational constraints do not allow for regular transfer of information between the two solvers. A much preferable approach is to use one software package to solve both the solid and fluid domain simultaneously.

LS-DYNA has been widely used in the automotive industry for many years for many different applications including crashworthiness simulation and occupant and pedestrian safety analysis. It is well validated for a diverse range of applications and is used throughout the world to solve many complex real world problems.

LS-DYNA contains many multi-physics solvers, including an incompressible flow solver, ICFD, which can be fully coupled with the solid mechanics solver to simulate Fluid Structure Interaction (FSI). The ICFD solver has been validated on a number of well-known test cases and it has also been used on a number of automotive industry problems [1] [2]. The ICFD solver and Dyna's wide ranging solid mechanics capabilities means that it is the ideal environment to study fluid structure interactions.

DrivAer geometry

The DrivAer body was developed by the Technical University of Munich [3] as an improved geometry for the investigation of vehicle aerodynamics, providing a more realistic test case for numerical and wind tunnel testing than its predecessors such as the simplified Ahmed and SAE bodies. This geometry has been extensively used for validation of numerical models [4]. Wind tunnel experiments have been carried out at TU Munich and TU Berlin to provide data including pressures profiles and global forces (drag and lift). There are a number of different configurations for the model, with different levels of geometry simplification. Figure 1 and Table 1 show the configuration used for this study.

Table 1 DrivAer configuration used in this study

Top geometry	Notchback
Underbody geometry	Smooth Underbody
Mirror configuration	With Mirrors
Wheel configuration	Without Wheels

Figure 1 DrivAer geometry used for validation study

Validation study

In order to demonstrate the accuracy of the ICFD flow solver in standalone mode, simulations have been carried out of the standard DrivAer car model which has no spoiler. The results from these simulations have been compared to the equivalent wind tunnel results from TUM.
Figure 2 shows the computational domain used for this validation study. Only half of the car is modelled and the boundary along the centreline of the car modelled as a slip wall. The dimensions of the domain have been made much larger than the car in order to minimise blockage effects. The volume mesh for this case was generated using the built in ICFD automatic mesher, with a refined volume box added around the car and boundary layers were added to the car surface using the *ICFD_MESH_BL cards. In total, the mesh contained just over 2 million cells.
The same setup was used with a number of different turbulence models. The lines in Figure 3 show the results from a steady state RANS k-ω simulation and also results from an unsteady LES

simulation where the pressures have been averaged over the 5 second simulation time. The RANS simulations used standard turbulent wall functions on the surface of the car.

In addition to the CFD simulations using ICFD, a steady state RANS k-ω simulation was carried out using OpenFOAM. The same geometry was used for the OpenFOAM simulations, however instead of the tetrahedral grid used for ICFD, a hexahedral-dominant grid was generated. The setup in terms of the turbulence models and wall functions should be the same for both the ICFD and OpenFOAM simulations.

Figure 3 shows the plots of the pressure profiles on the centerline of the car, comparing the experimental results to the results from both ICFD and OpenFOAM. Both ICFD and OpenFOAM showed very good agreement with the experimental results around the front of the car. There is a more noticeable difference in the pressure profiles on the roof of the car, however the pressure profile in the experimental results for this region are likely to be affected by a model support, which held the car in place from its roof in the wind tunnel test. Previous studies using the DrivAer car model have noted this difference between the experimental and numerical pressure profiles in this region when the support geometry is not included in the model.

Towards the rear of the car, the slope of the rear window causes the boundary layer to experience an adverse pressure gradient, which makes it more unstable and separation is more likely. Figure 4 shows a comparison of the velocity profiles from a steady state RANS simulation and a transient LES simulation. The RANS k-ω ICFD simulation was carried out using the steady state solver, whereas the LES simulation was run transiently. The steady state simulation averages out any turbulent eddies which are shed from the rear window, however in the LES simulations, these eddies are clearly present. In the LES simulation, the flow is separating from the top of the rear window at the outboard section (y=0.4m), whereas it stays attached along the centerline of the car. These comparisons show that unsteady LES simulations are required to accurately simulate the flow field around the rear of the car.

Previous studies of the DrivAer car model have shown that the flow field around the rear window of the car can be sensitive to the choice of turbulence model and whether the geometry of the wind tunnel test section is replicated or not [4].

Table 2 shows a comparison of the computational resources used for each of the three simulations described in this section. The ICFD grid contained many more cells than the OpenFOAM grid as the OpenFOAM grid used Hexahedral cells, wheras the ICFD grid was made up of tetrahedral cells. This partly explains the difference in run time between the ICFD and OpenFOAM k- ω simulations.

The comparison of the presssure profile on the roof of the car shows that the ICFD solver is capable of simuating the flow around the DrivAer car model. Unlike many simple validation test cases, the DriveAer model is not simplified and is an accurate representation of a a generic road car. The validation study has captured the complex flow around the rear of the car, with flow attached along the centreline and attached further outboard.

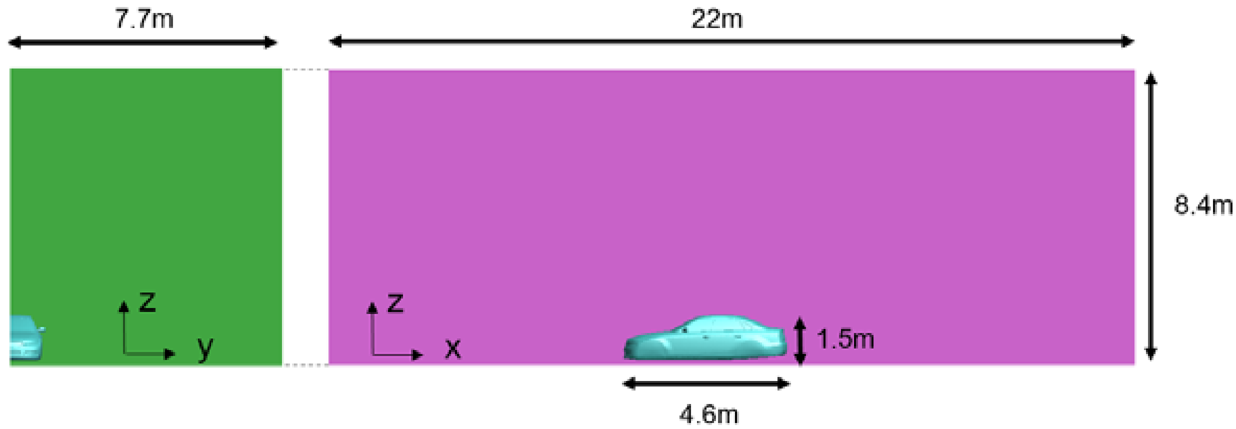

Figure 2 Computational domain for validation study

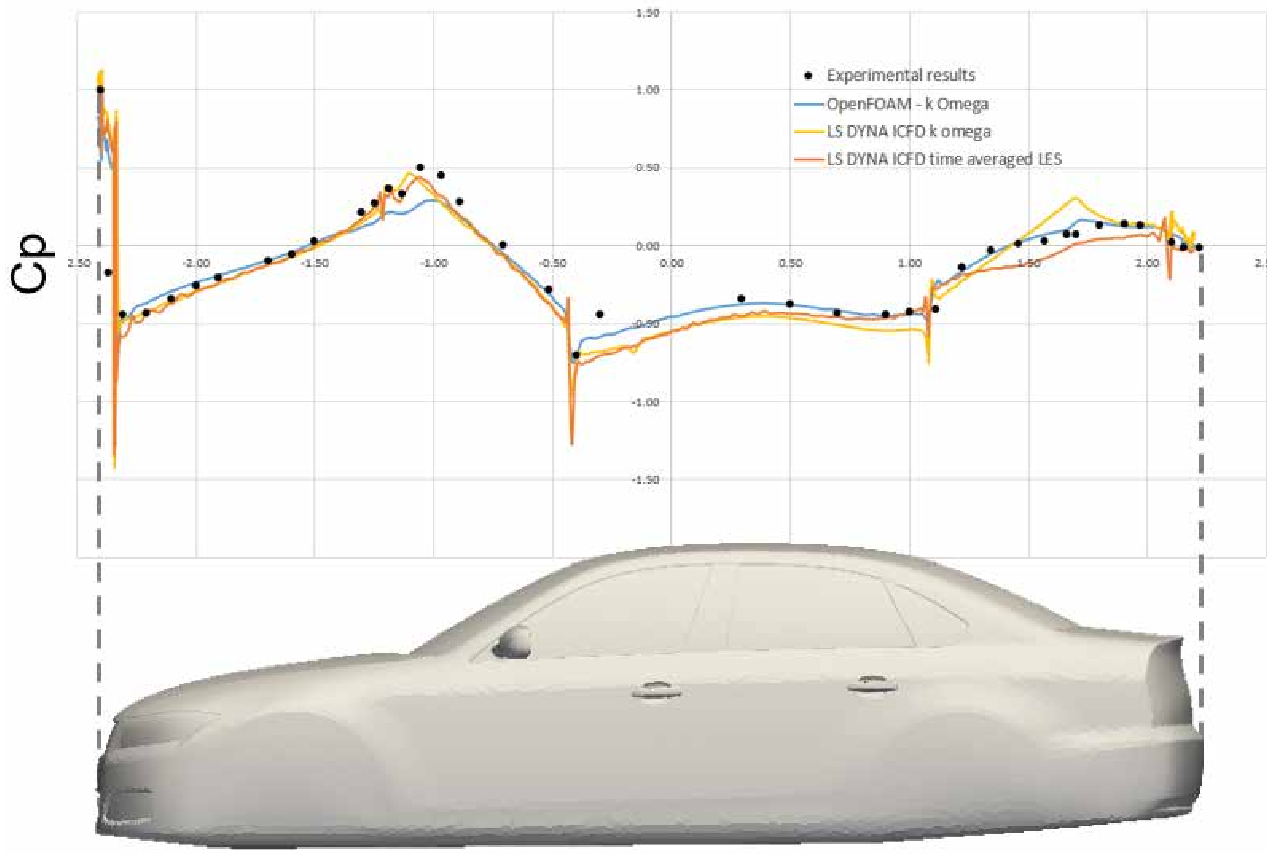

Figure 3 Pressure profile along the centreline of the DriveAer car

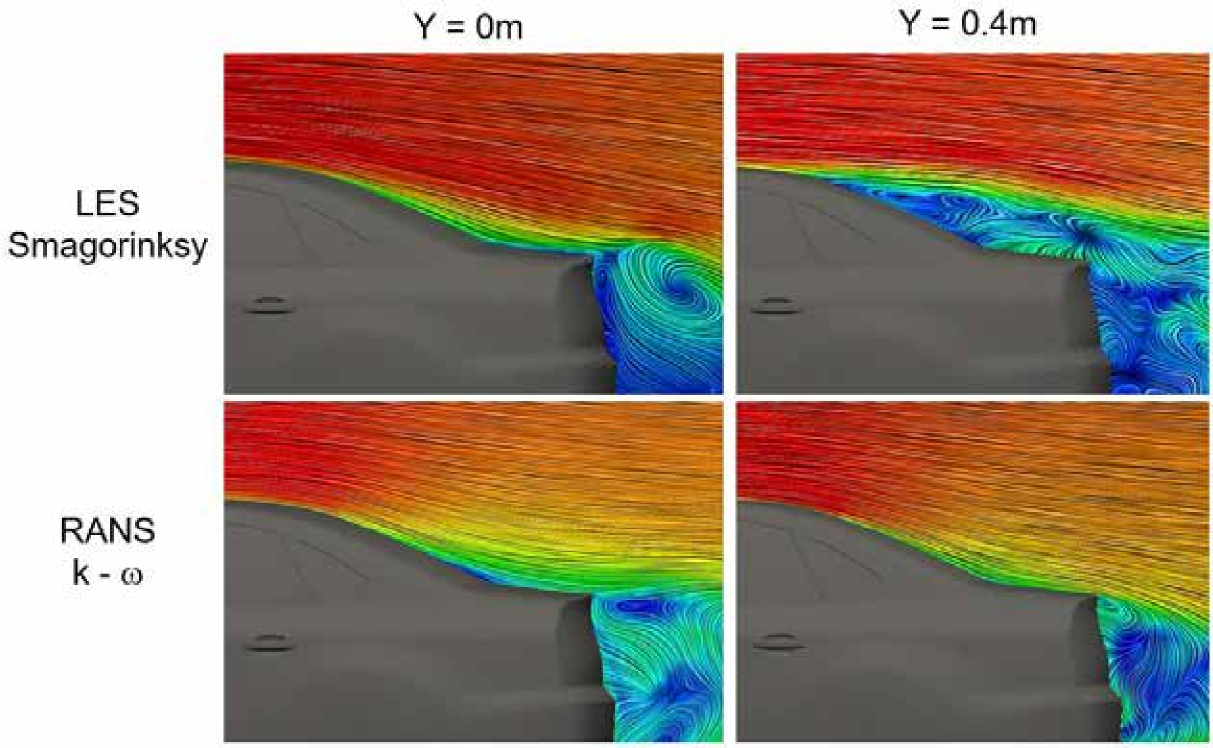

Figure 4 Velocity profiles around the rear window from the RANS and LES simulations

Table 2 Comparison of computational efficiency for validation cases

	OpenFOAM k- ω	**ICFD k-ω**	**ICFD LES**
Number of cells	1.3 million	3.4 million	3.4 million
Number of processors	4	16	16
Number of iterations	8000	8400	-
Simulation time	-	-	5s
Run time	7 hours	21 hours	80 hours

CFD simulations with spoiler

A generic spoiler was designed and added to the back of the DrivAer car model. A NACA 0012 airfoil was used for the spoiler wing section and a simple end plate and support were added. Other studies have been carried out validating the performance of ICFD for predicting the aerodynamic properties of this airfoil. [5]

After initial simulations, it was found that the inboard section of the spoiler experiences greater downwash from the car compared to the outboard section of the spoiler. In order to reduce the likelihood of stalling, the profile has been rotated to reduce the angle of attack in the inboard section. The spoiler therefore has a swept profile, with the inboard section at a greater angle to the x axis compared to the outboards section. The spoiler geometry is shown in Figure 5 and Figure 6.

These simulations of with the spoiler used for the most part the same setup as the LES validation case without the spoiler, however the freestream velocity was increased from 17.9m/s to 30m/s. Figure 7 and Figure 8 show the velocity profile around the rear window of the car along the centreline of the car and at a section 0.4m outboard of this. The flow around the spoiler is very different at these two cut sections. Along the centreline of the car, the flow remains attached to the rear window and over the bonnet, meaning that the spoiler is in a region of steady flow. The higher velocity under the front of the spoiler shows that it is working effectively to generate downforce. At the outboard section however, the flow separates from the top of the rear window, and turbulent eddies are shed towards the spoiler. As a result of this, the flow appears to have also separated from the lower surface of the spoiler, and it is unlikely to be generating any downforce. The 3D flow field is shown using streamlines in Figure 9. In this figure, it can be seen that the flow is attached along the centreline of the rear window, and separated at the sides.

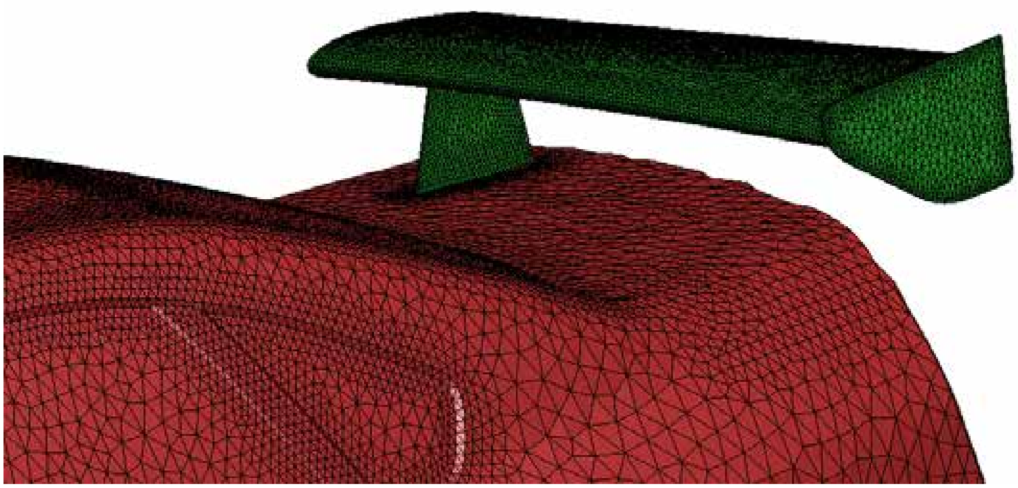

Figure 5 Spoiler geometry on DrivAer car model

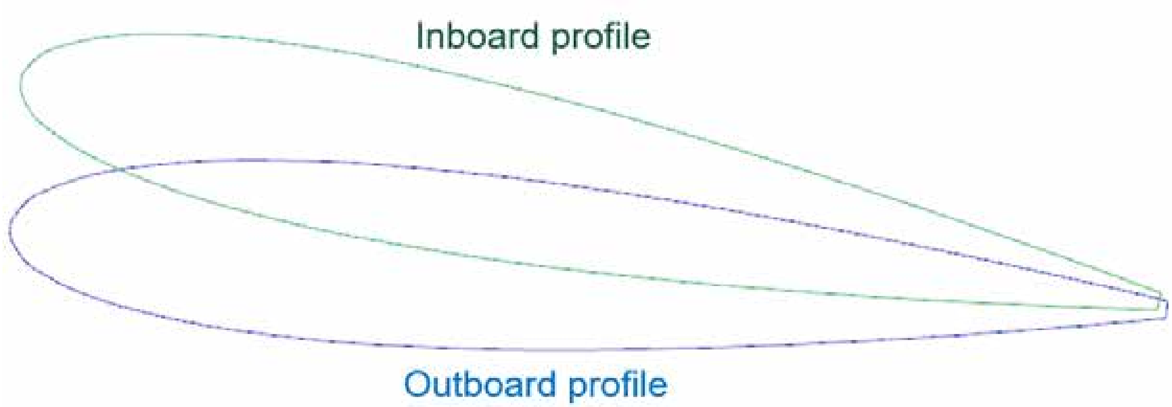

Figure 6 Comparison of the inboard and outboard profile on the spoiler.

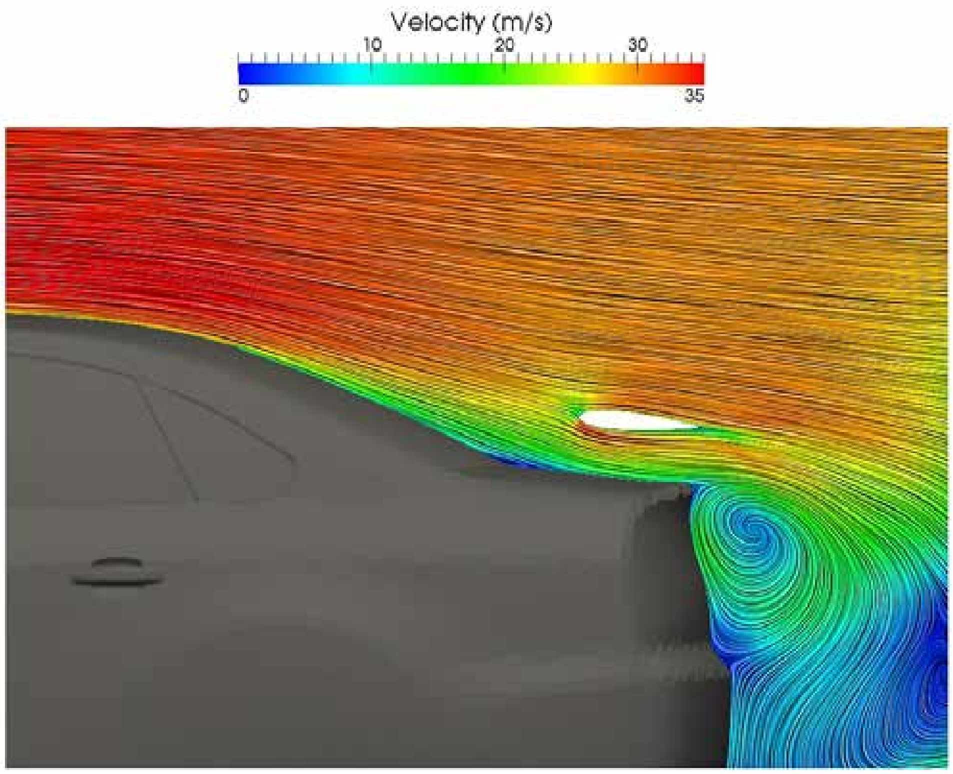

Figure 7 Velocity profile around rear window on centreline of car

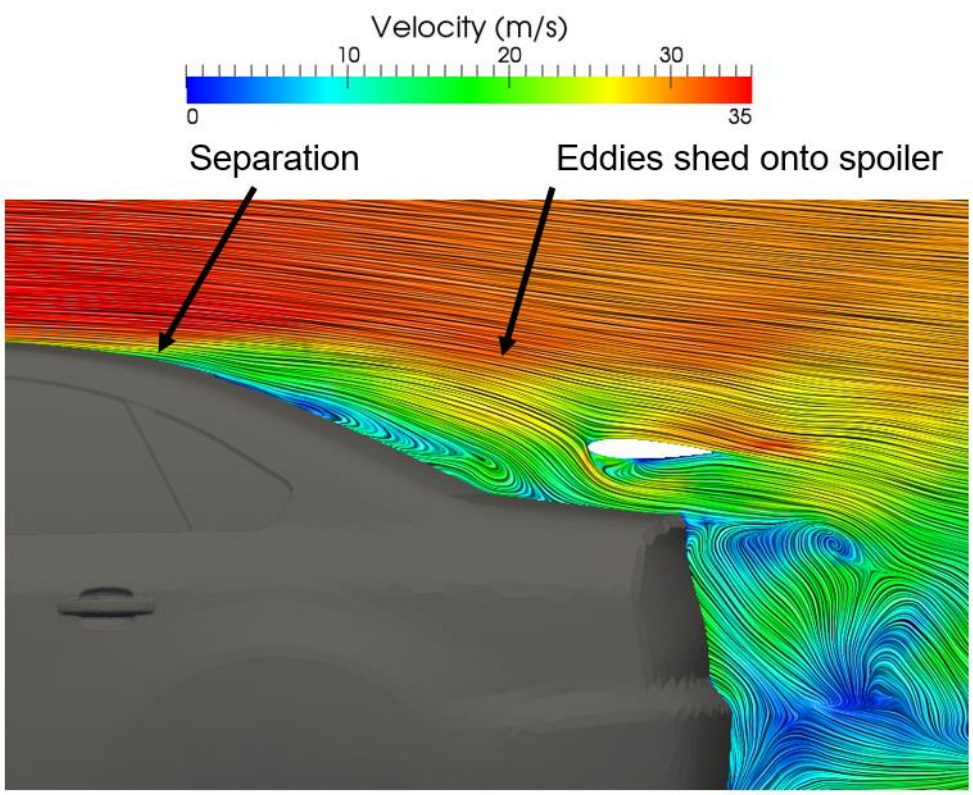

Figure 8 Velocity profile over rear window at y=0.4m

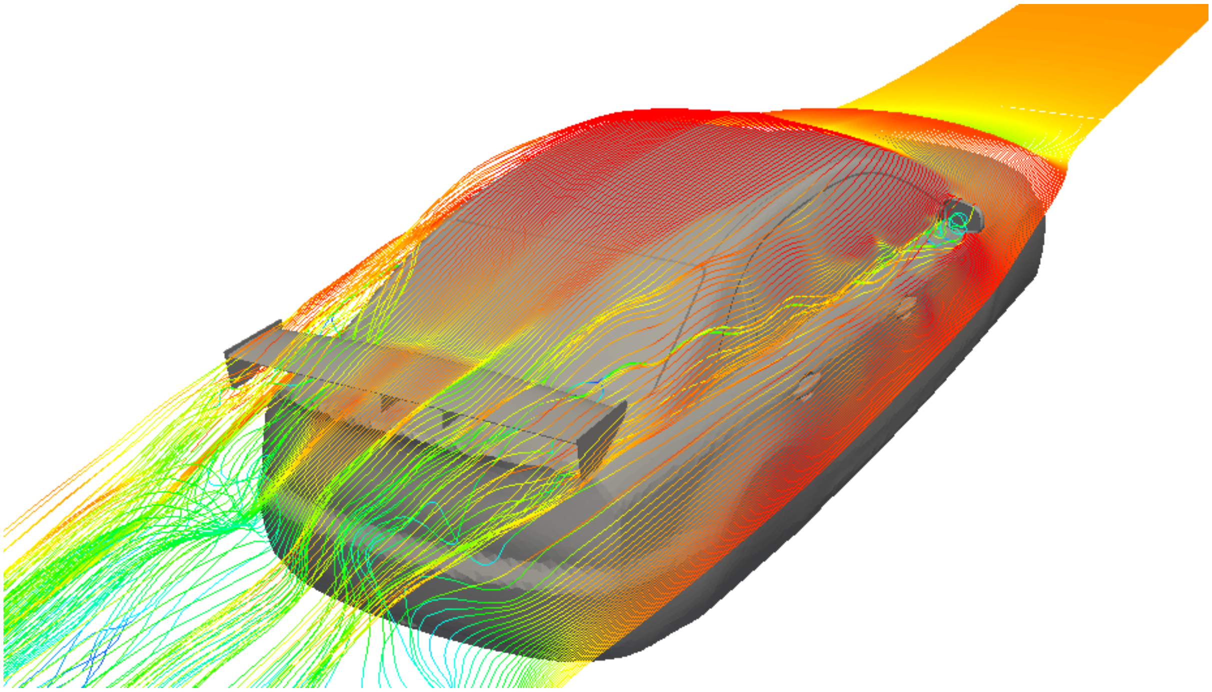

Figure 9 Streamlines around car with spoiler

FSI simulations with spoiler

The setup of the simulation in described in the previous section was modified so that fluid structure interaction phenomena could be modelled. A simple structural model of a spoiler was created, using the same surfaces as those which were used in the CFD model. The central spoiler and the endplate were model as rigid bodies, whereas the spoiler wing surface was modelled as 3mm thick ABS with a density of 1060 kg/m³ and a Young's modulus of 2 GPa.

Figure 11 shows the time history of the vertical displacement of the endplate. There is a visible step response from the spoiler due to the FSI coupling initiating at t=0.01s. Following this, the response of the spoiler is due to the unsteady forcing on it due to its location in the wake of the rear window.

Figure 12 shows the time history of the vertical force on the spoiler over time. It can be seen that the load on the spoiler varies significantly over time, with the net force on the spoiler varying between 0N and 30N. This unsteady load is caused by intermittent separation of the spoiler due to it operating in the wake of the rear window.

The vertical force on the spoiler from the CFD simulation (i.e. undeformed geometry) has been added to Figure 11. The aerodynamically excited movement of the spoiler does not cause a significant change in the vertical loading on the spoiler, however the magnitude of the fluctuations is large enough to be distracting and unnerving for the driver of this vehicle, or the driver of a following vehicle.

An additional advantage of carrying out the FSI simulation is that the stresses on the aerodynamic parts are calculated as part of the solution, and these can be analysed to check for yielding or possible fatigue issues. Figure 13 shows the profile of the maximum principal stress on the spoiler at one time step of the FSI simulation.

This study has highlighted the capability of LS-DYNA for accurately simulating the fluid flow around vehicles and for simulating fluid structure interaction of aerodynamic parts. The simulation of FSI in LS-DYNA could present numerous benefits for use in the design process of vehicles. Because the solid and fluid solutions are analysed in the same environment, LS-DYNA offers a range of strong and robust coupling between the two solutions, ensuring greater accuracy and quicker turnaround times for analysing different iterations of the design.

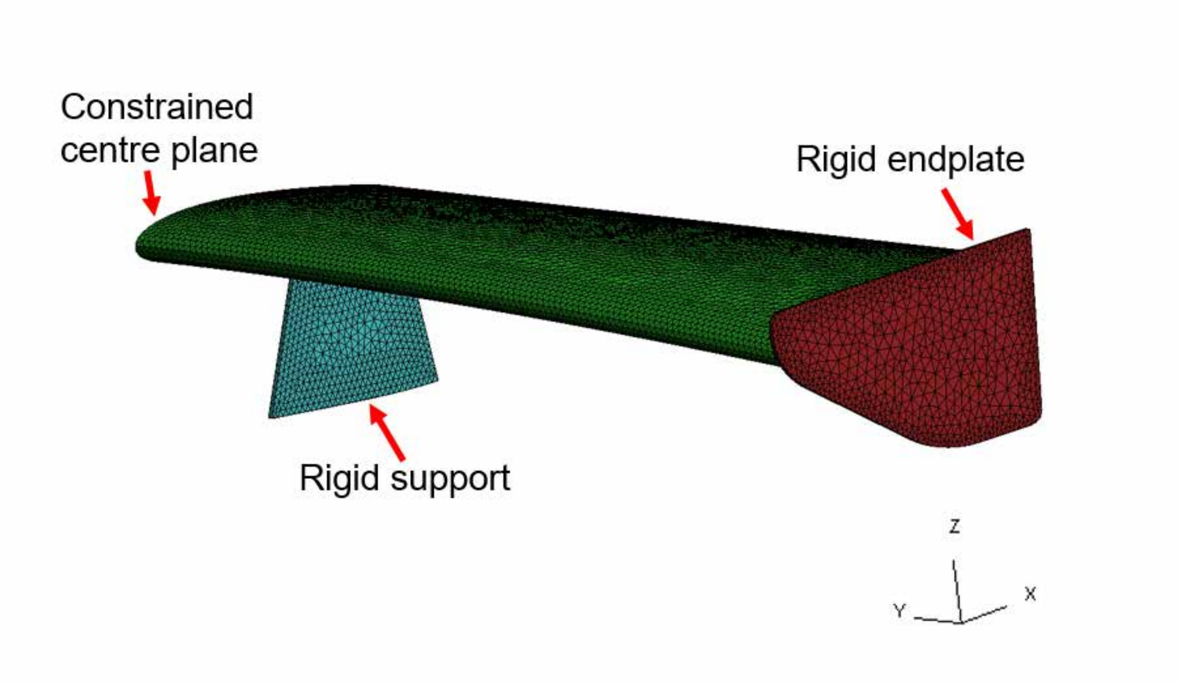

Figure 10 Structural model used in FSI simulation

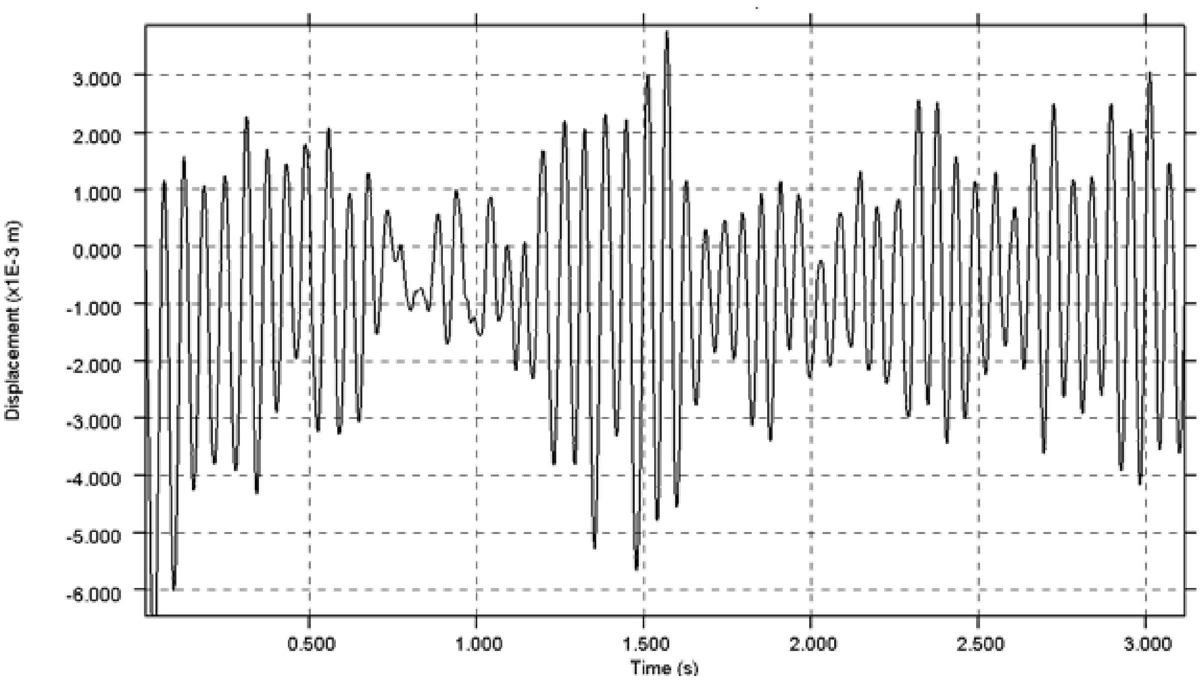

Figure 11 Time history of the vertical displacement of the end plate

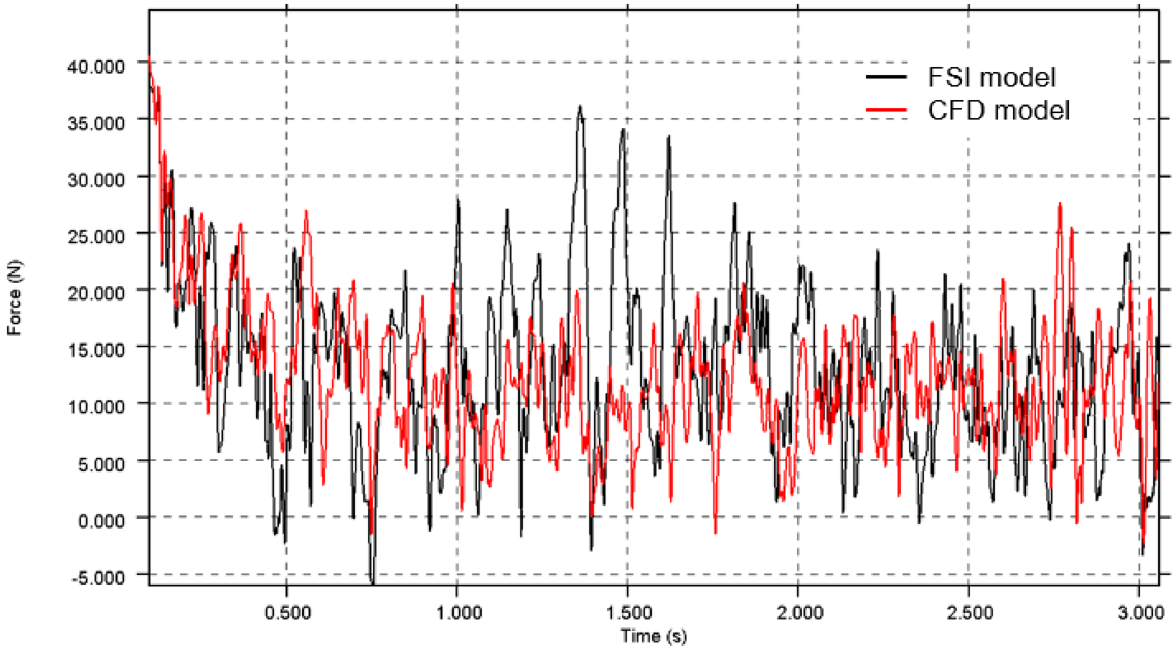

Figure 12 Time history of the down force from the spoiler

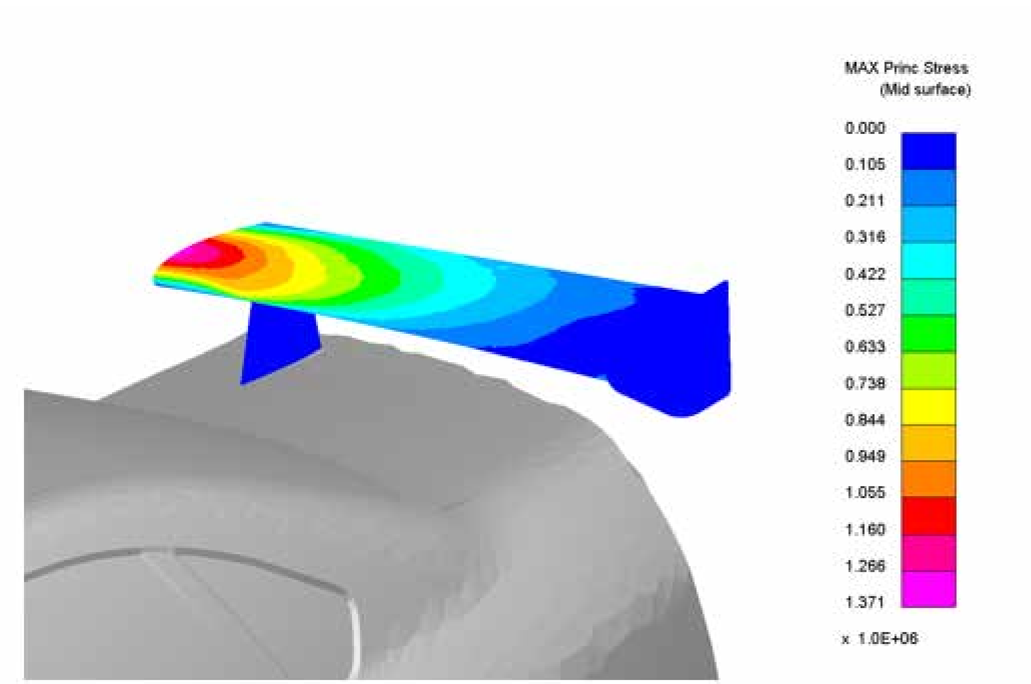

Figure 13 Maximum principal stress on spoiler during FSI simulation

References

[1] F. D. Pin, "LS-DYNA® R7: Strong Fluid Structure Interaction (FSI) capabilities and associated meshing tools for the incompressible CFD solver (ICFD), applications and examples," *9th European LS-DYNA Conference,* 2013.

[2] G. Wang, K. Gardner, E. DeHoff, F. d. Pin, I. Calcichoury and E. Yreux, "Applications of ICFD/SPH Solvers by LS-DYNA® to Solve Water Splashing Impact to Automobile Body," *11th European LS-DYNA Conference,* 2017.

[3] A. Heft, T. Indinger and N. Adams, "Introduction of a New Realistic Generic Car Model for Aerodynamic Investigations," in *SAE 2012 World Congress & Exibition*, Detroit, 2012.

[4] R. Yazdani, "Steady and unsteady numerical analysis of the DrivAer model," Chalmers University of Technology, Gothenburg, 2015.

[5] B. Perin, O.Verderel, P.Bordenave, E.Grippon, V.Lapoujade, H.elloc and I.Caldichoury, "Computational Fluid Dynamic of NACA0012 with LS-DYNA® (ALE & ICFD) and Wind Tunnel Tests," *14th International LS-DYNA Users Conference,* 2016.

[6] Dynamore, "Recent developments in LS DYNA," 30 August 2017. [Online]. Available: https://www.dynamore.de/de/download/presentation/dokumente/download-webinar-multiphysik/02-2016-04-dynamore-webinar-multiphysics-icfd.pdf.

[7] G. Wang, F. d. Pin, I. Caldichoury, P. Rodriguez, J. Tipple and S. Smith, "Applications of ICFD solver by LS-DYNA® in Automotive Fields to Solve Fluid-Solid-Interaction (FSI) Problems," *11th European LS-DYNA Conference,* 2017.