Fluid Structure Interaction Simulation of Hood Flutter

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. This paper shows some of the capabilities in LS-DYNA for simulating hood flutter, which is a known fluid-structure interaction problem. Hood flutter is affected by the turbulent wake from preceding vehicles, the hood opening mechanism and the opening up of seals. This paper considers the feasibility of commercially feasible simulation of this complex automotive FSI phenomenon through the creation of a series of models which display how the important physical features of hood flutter could be modelled.

Introduction

Automotive bodywork components can experience large unsteady deflections due to the fluid flow over the vehicle and aero elastic instabilities. Often bodywork vibration can be avoided by making bodywork stiffer, however as there are strong incentives to reduce vehicle weight and fuel consumption, bodywork panels are more likely to become lightweight enough for significant deflections to occur and for aeroelastic instabilities to develop. The position and large surface area of the hood means that it is an area of particular concern in this regard.

The term “hood flutter” is used to describe hood oscillations which are caused by a number of different aeroelastic phenomena. While the term “flutter” strictly refers to one type of aeroelastic instability caused by coupled translational and rotational models of an airfoil, the term is used in this paper in its wider sense here to cover any aeroelastic effect which causes hood vibrations. In order to describe these different effects which cause hood flutter, the classifications used by Naudacher & Rockwell [1] of Extraneously Induced Excitation, Instability Induced Excitation and Movement Induced Excitation are used here.

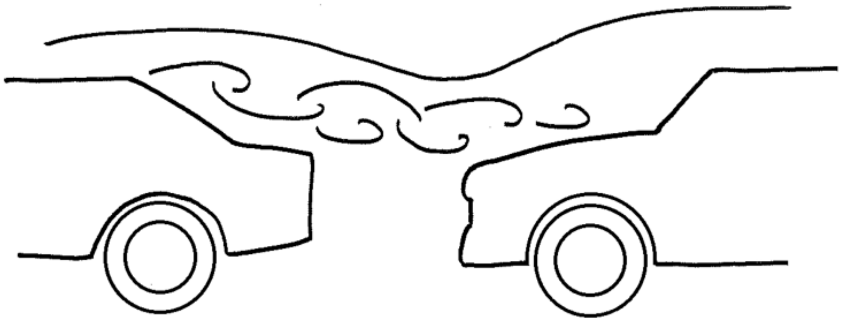
Extraneously induced excitation

Extraneously induced excitation describes vibrations which are caused by oscillations in the oncoming flow onto a body. In the context of hood flutter, this is may occur when a vehicle is travelling in the wake of another vehicle, either when following or overtaking, such as in Figure 1(a). The turbulent eddies in the wake of a preceding car may cause oscillating loads on the hood of the following vehicle. In a turbulent wake, there are many different sizes and frequencies of eddies, so many different modes of vibration of a hood may be excited.

Instability induced excitation

Instability induced excitations are caused by instabilities in the fluid flow over a structure. The most common form of instability induced excitation is vortex shedding, whereby the shedding of vortices causes periodic loads transverse to the direction of the oncoming air. Flow separation is another form of instability which can cause fluctuating loads. While flow separation is generally unlikely to occur over most hoods, some hoods with sharp leading edges may experience some separation as shown in Figure 1(b). This separation may be either intermittent or constant, and in either case will result in fluctuating loads on the hood which may cause harmonic responses.

(a)



(b)

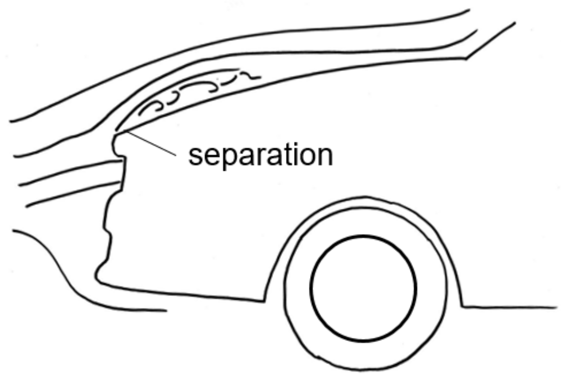


Figure 1 Extraneously induced excitation from the wake of a preceding car (a) and Instability induced excitation from separation at the front edge of a hood (b)

Movement induced excitation

Movement of the structure can in some cases cause instability. Classical flutter and gallop are two phenomena which fall into this category. Classical flutter describes the coupling of the translational and rotational degrees of freedom of an airfoil [2], and is unlikely to affect a hood, however it is possible that a hood may be susceptible to torsional gallop and other torsional instabilities. To illustrate torsional instability, Figure 2 shows a schematic of a hood which is supported on a damped spring. In reality most modern hoods are supported on 4-bar links, so the joint shown in Figure 2 is not an entirely accurate representation, but this simplification is useful for demonstrating the principle. The equation of motion for this body is given below:

$$J\ddot{\theta} + \left(2J\zeta\omega_n + \frac{1}{2}\rho U r A \frac{\partial C_M}{\partial \alpha}\right)\dot{\theta} + \left(k - \frac{1}{2}\rho U^2 A \frac{\partial C_M}{\partial \alpha}\right)\theta = 0$$

Where J is the moment of inertia of the hood about the pivot, ζ is the damping ratio, ω_n is the natural frequency, ρ is the fluid density, A is a reference area, C_M is the pitching moment coefficient, k the spring stiffness and α is the angle of attack of the hood.

The body shown can be unstable if the θ term or the $\dot{\theta}$ terms go to zero. The θ term can go to zero if the additional aerodynamic moment caused by a rotation is greater than the stiffness of the rotational spring. This type of instability is called torsional divergence, and it would cause the section to continue rising once a critical angle is reached. In the case of a hood, this instability would cause the hood to rise up until it is eventually held on the catch.

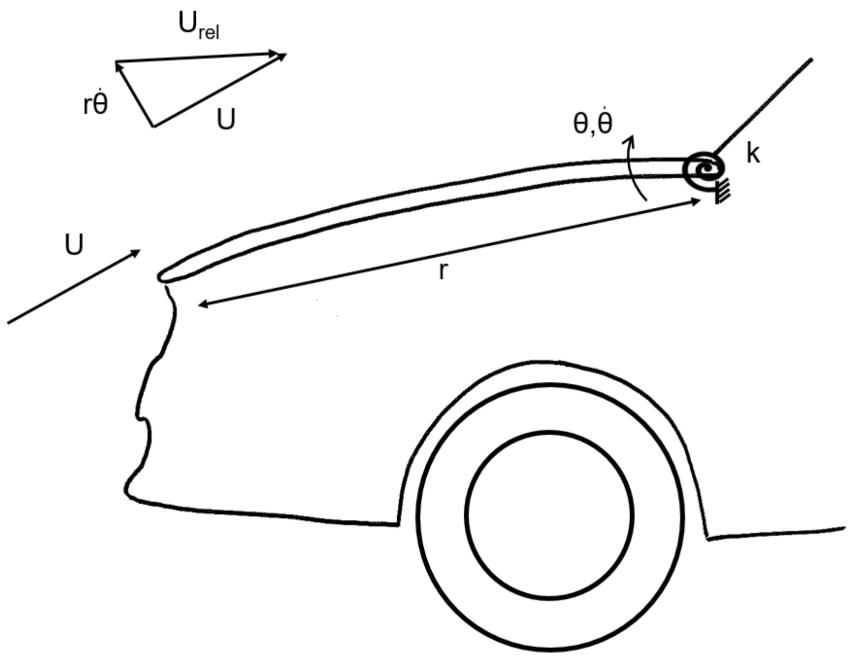


Figure 2 A hood mounted on a damped spring

The second type of instability that this section can exhibit is torsional galloping, which is caused when the $\dot{\theta}$ term goes to zero. This type of instability occurs because the rotational velocity of the section causes a change in the relative velocity of the flow onto the section. Figure 2 shows the onset velocity and the relative velocity onto the section when the section is rising. Due to the rotational velocity of the section, the effective angle of attack of the section is reduced. For sections like the sharp edged hood shown in Figure 2, this change in angle of attack can cause large changes in the aerodynamic moments, as the change in angle of attack can cause the boundary layer to reattach to the top surface, which serves to increase the lift force on the body and the moment about the pivot. This increase in moment when the section is rising is what causes the torsional galloping instability.

Analysis of Fluid Structure Interaction

The analysis of some fluid structure interaction problems can be simplified by simulating the fluid flow and the structural dynamics separately. Fluid dynamic loads can be extracted from numerical or experimental analysis and applied to a structural model to evaluate the response of the structure to the applied loading. This approach is generally appropriate for extraneously induced excitation and instability induced excitation as these instability mechanisms are not caused by the movement of the structure, however second order loads may become significant if the response of the structure is large enough.

For movement induced excitation problems such as torsional gallop and flutter, the stability can be assessed by assuming that the fluid dynamic loads which act on the structure are quasi-steady. Using this assumption, the loading on the static structure at different displacements is used as an approximation of the loads on the deflected structure when it is moving. This is a valid assumption for many structures, however in some circumstances, galloping and vortex shedding can interact non-linearly, and in some instances the vortex shedding can lock in to the natural frequency of the motion, which amplifies the motion further. As a result, quasi-static analysis of gallop and flutter is unreliable when the vortex shedding frequency is close to the natural frequency of the structure. When vortex shedding is coupled to the galloping response, and in other scenarios where movement can cause non-linear interactions, wind tunnel testing with an aeroelastic model is the most reliable method of aeroelastic stability analysis [2].

In the context of hood flutter, small movements may cause non-linear responses due to the way that hoods are held down. The hood sits on bump stops, its movement is retrained by a catch, and a seal around the outside prevents air movement around the hood. Small movements may cause the hood to lift off bump stops and seals, which may drastically change the stiffness and damping as well as the fluid flow around the edges of the hood.

The complexities of hood flutter as highlighted above mean that wind tunnel testing with an aeroelastic model is the most reliable method of testing. The purpose of this paper is to show whether coupled fluid structure interaction in LS-DYNA could potentially produce a reliable method of hood flutter analysis without the expense and long lead-times associated with aero elastic wind tunnel testing.

The following sections demonstrate the applicability of LS-DYNA for analyzing hood flutter through a series of simulation studies.

- A validation study has been carried out of the Incompressible Fluid Dynamics (ICFD) solver in LS-DYNA by simulating the flow around generic car body and comparing the results with those from wind tunnel testing.
- Unsteady loading over a hood due to the wake of a preceding vehicle has been simulated in ICFD.
- A large scale 3D fluid structure interaction problem of a spoiler on the back of a car has been simulated in ICFD. The spoiler is excited by the separated flow around the back of the car.
- A simulation has been carried out of a 2D cross section of a car with a movable hood. The hood is held down with springs, and an initial small gap under the edge of the hood can open up if the hood is lifted sufficiently.

These simulations studies are intended to show that ICFD has the capability to model many of the constituent parts of the problem of hood flutter.

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 3 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 3 DrivAer geometry used for validation study

DrivAer pressure profile 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 4 shows the computational domain used for this validation study. Only half of the car is modelled and the boundary along the centerline of the car modelled as a slip wall. The dimensions of the domain have been made much larger than the car in order to minimize 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 5 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 5 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 6 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 only 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.

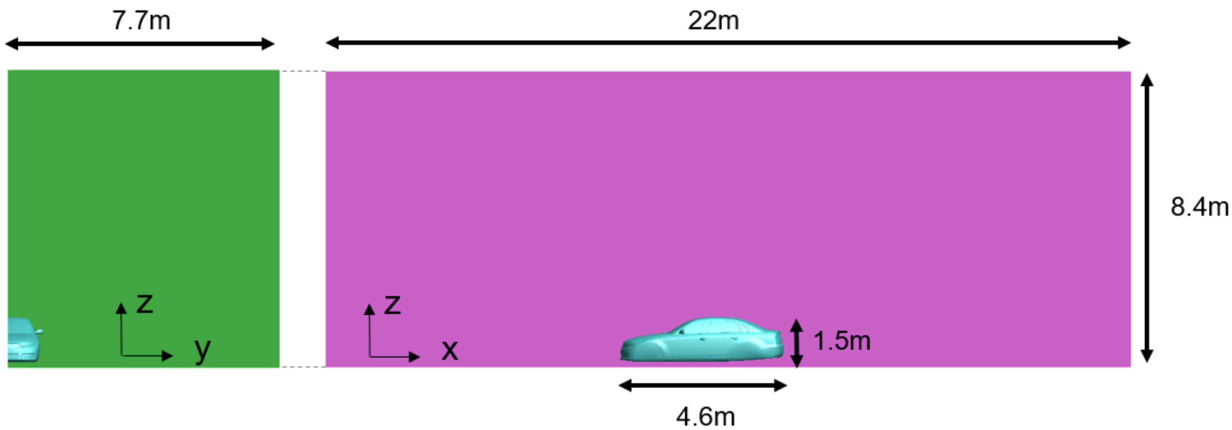


Figure 4 Computational domain for validation study

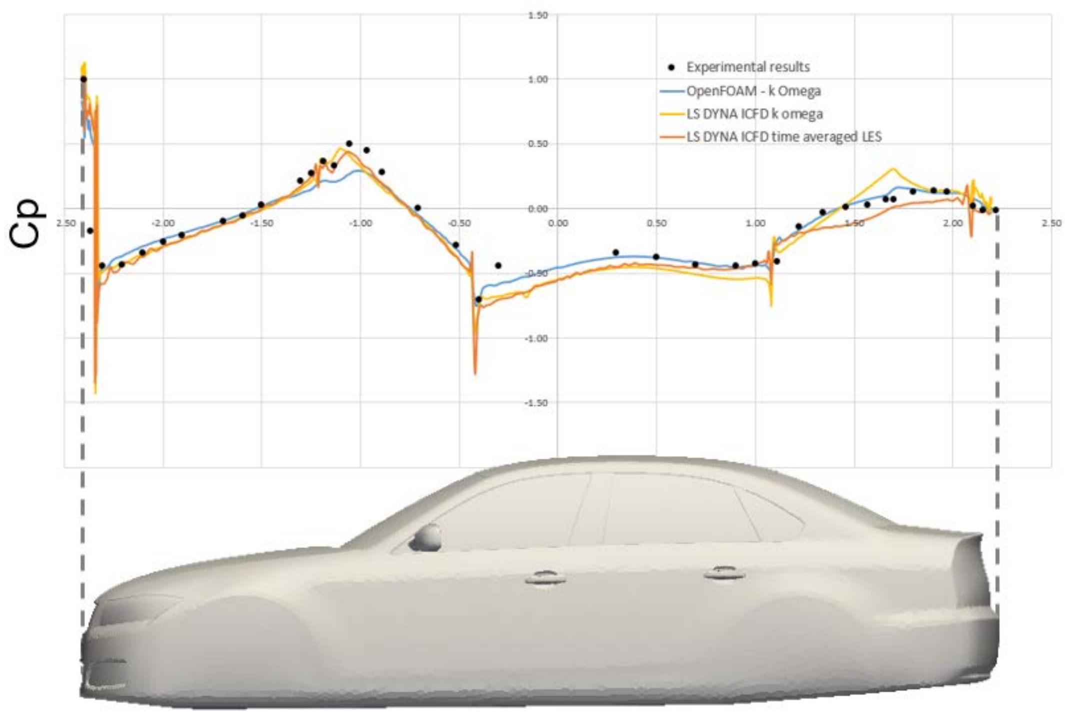


Figure 5 Pressure profile along the centreline of the DriveAer car

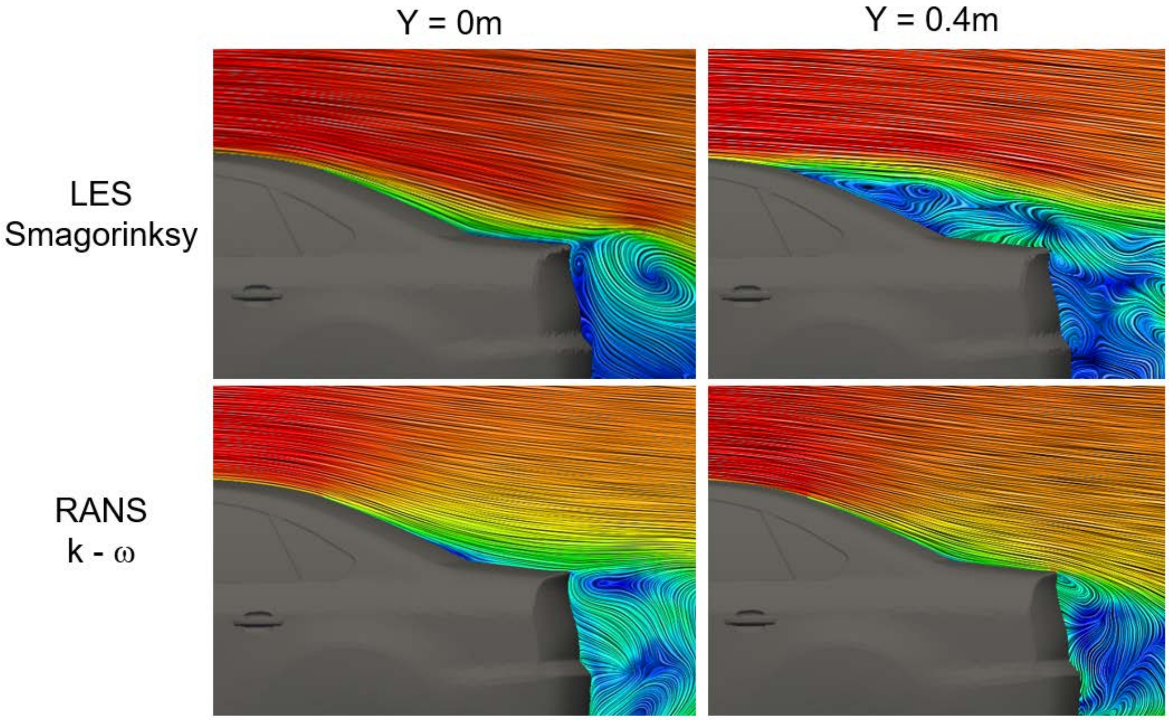


Figure 6 Velocity profiles around the rear window from the RANS and LES simulations

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

Simulation of unsteady loads from the wake of a preceding car

As noted previously, hood flutter can be caused by the unsteady flow field in the wake of a preceding vehicle. This type of excitation is most likely to occur when a vehicle is performing an overtaking maneuver, and this is clearly a dangerous time for a driver to be distracted by hood oscillations.

In order to illustrate the utility of LS-DYNA for modelling the transient flow field experienced by a vehicle during an overtaking maneuver, a model has been made with two DrivAer cars placed in close proximity as they would be during such a maneuver. The setup of the model is shown in Figure 7. Both of the two cars in this model are based on the DrivAer geometry, with the mesh of the front car coarsened slightly compared to the rear vehicle. The model setup is similar to that used for the validation of the pressure profile over the DrivAer car which is described in section. The LES Smagorinsky turbulence model is used, as this model gave good agreement with the pressure profile along the centerline of the car.

Figure 8 shows the velocity on a horizontal plane during the simulation. The slow moving air in the wake of the preceding vehicle is clearly visible, as are the vortices which are shed off the rear of the car. These vortices move the slow moving air in the wake, which periodically impinges on the hood of the following car.

Figure 9 shows a plot of the vertical load on the hood of the following car. The loading has a clear periodicity due to the regular shedding of vortices from the front car as well as some random fluctuations. The extracted video from the simulation shows how the pressure peaks start at the front edge of the hood and are swept up over the length of the hood to the bottom of the windscreen.

This is a pure CFD simulation with no structural model in place, but it is shown as a demonstration of the complex loading which could be simulated in an FSI simulation. Without a 3D simulation such as the model shown here, it would be impossible to obtain the same spatial and temporal profile of this aerodynamic load to apply to the structural model.

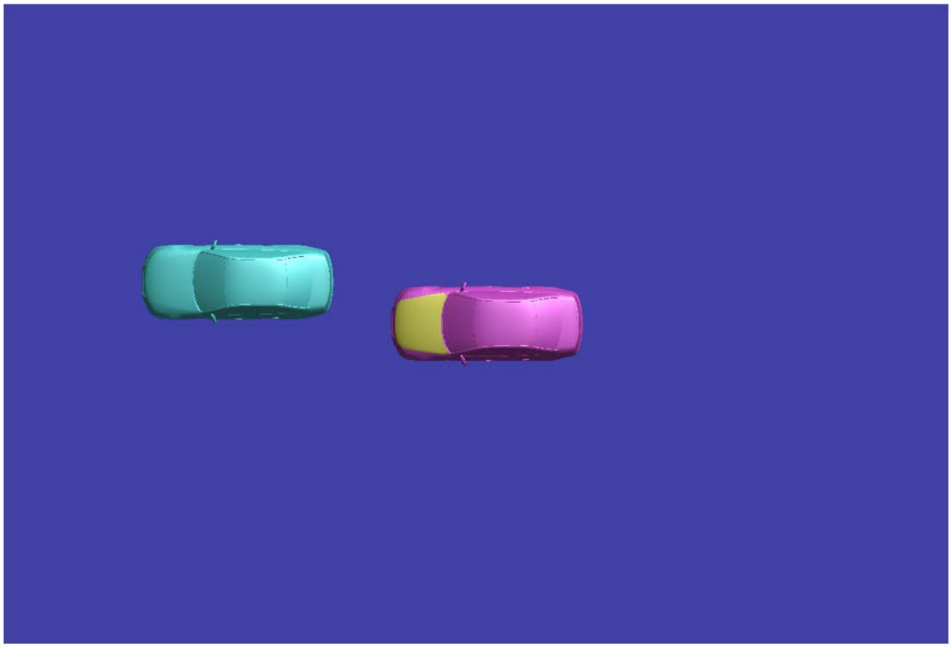


Figure 7 Model setup for overtaking simulation

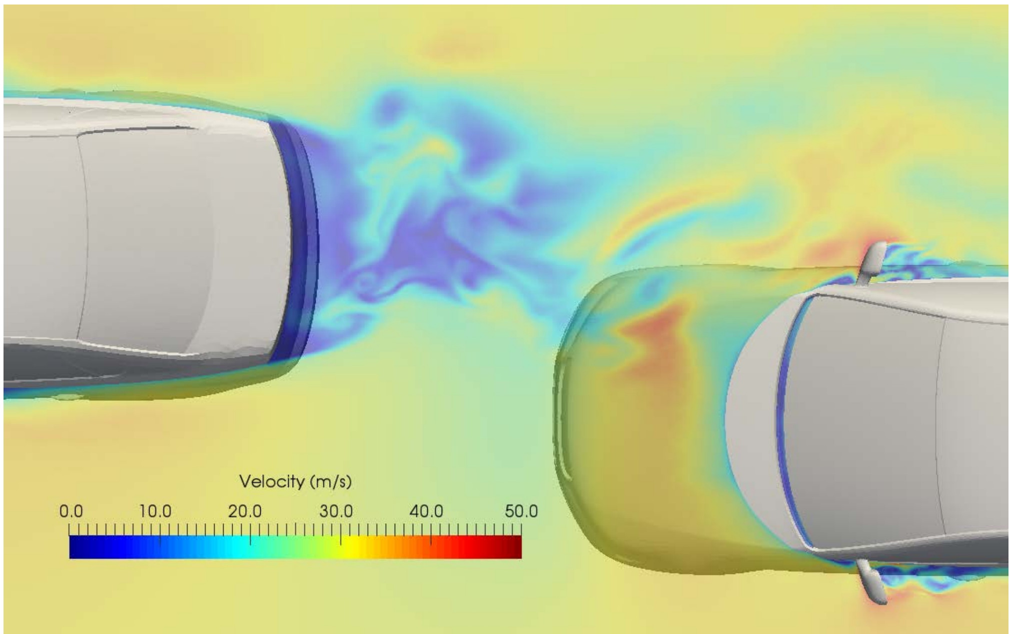


Figure 8 Flow field around the front of the following car

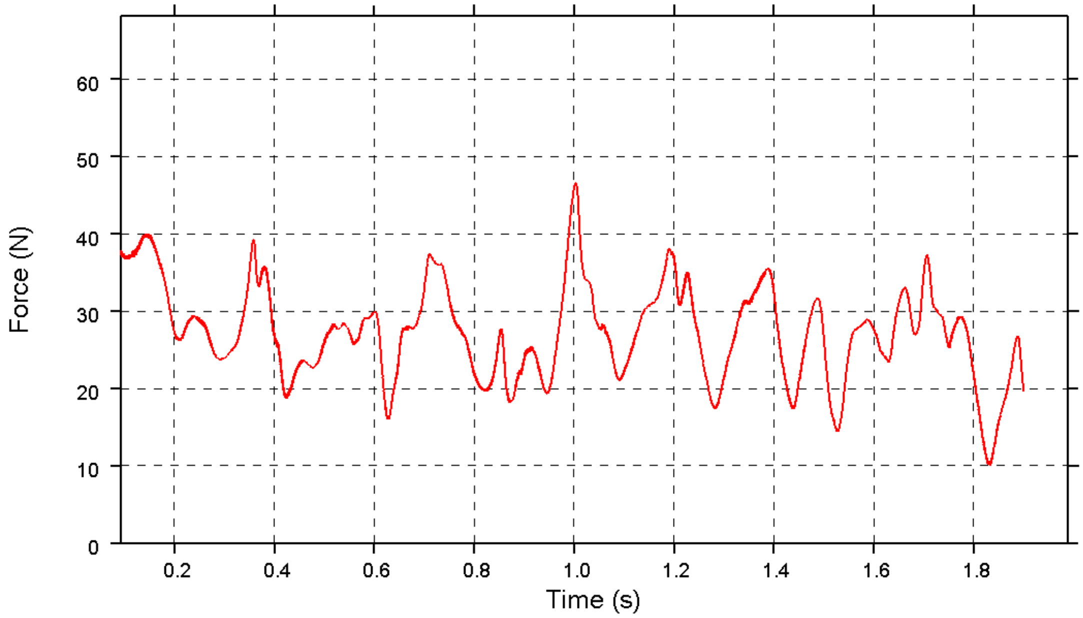


Figure 9 Time history of the vertical load on the hood of the following car

FSI simulations of a spoiler on the DrivAer car

The CFD model from the validation simulation was modified by adding a generic spoiler. A NACA 0012 airfoil was used for the spoiler wing section and a simple end plate and support were added. so that fluid structure interaction phenomena could be modelled. Figure 10 shows the simple structural model of the spoiler which 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. FSI simulations were carried out using a similar setup to the LES validation case without the spoiler, however the freestream velocity was increased from 17.9m/s to 30m/s.

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 analyzed 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.

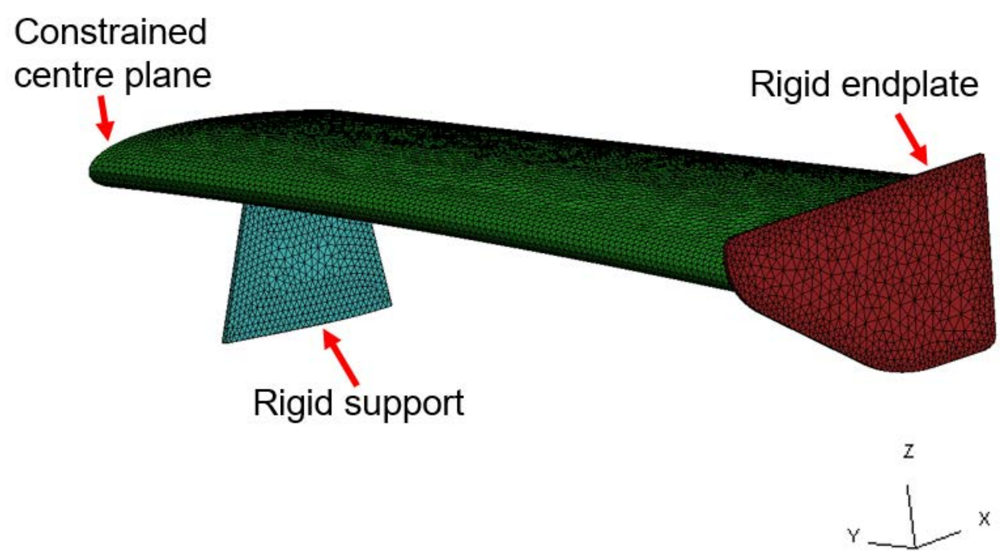


Figure 10 Structural model used in FSI simulation

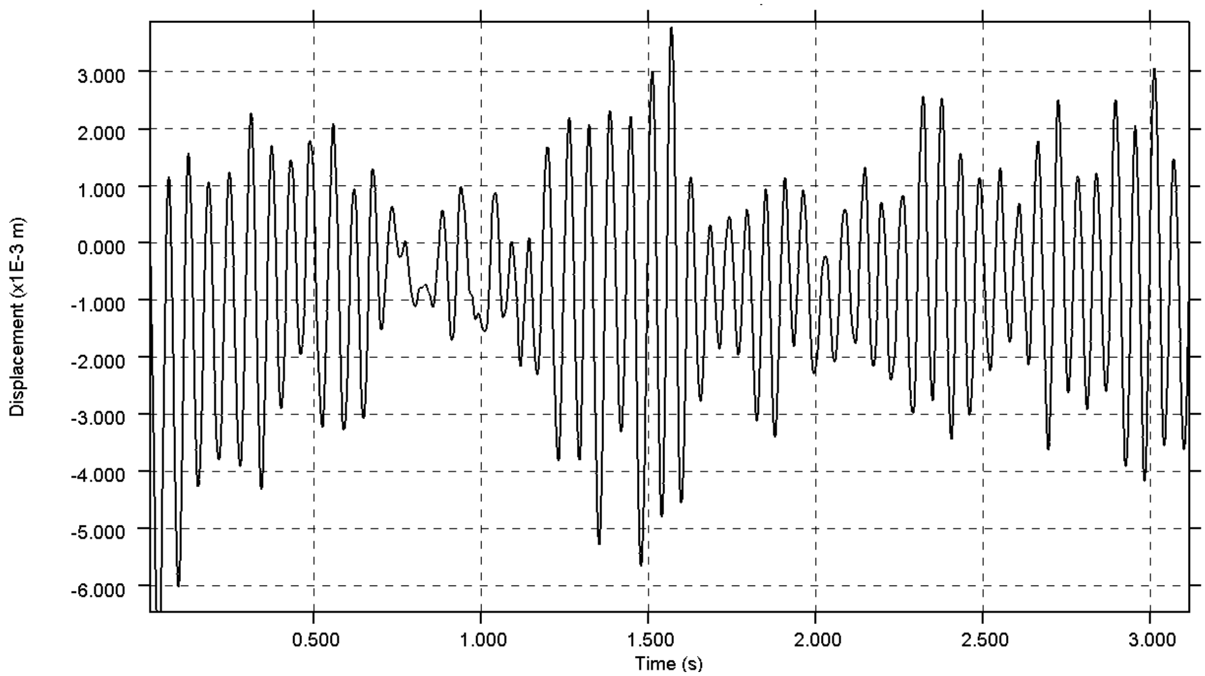


Figure 11 Time history of the vertical displacement of the end plate

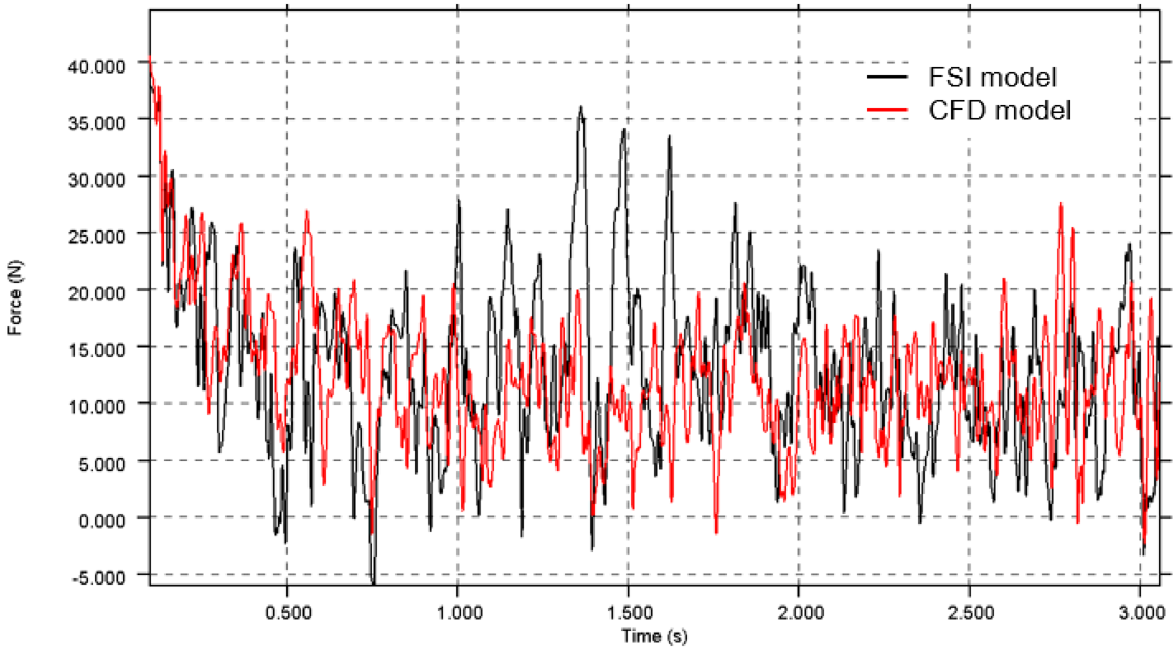


Figure 12 Time history of the down force from the spoiler

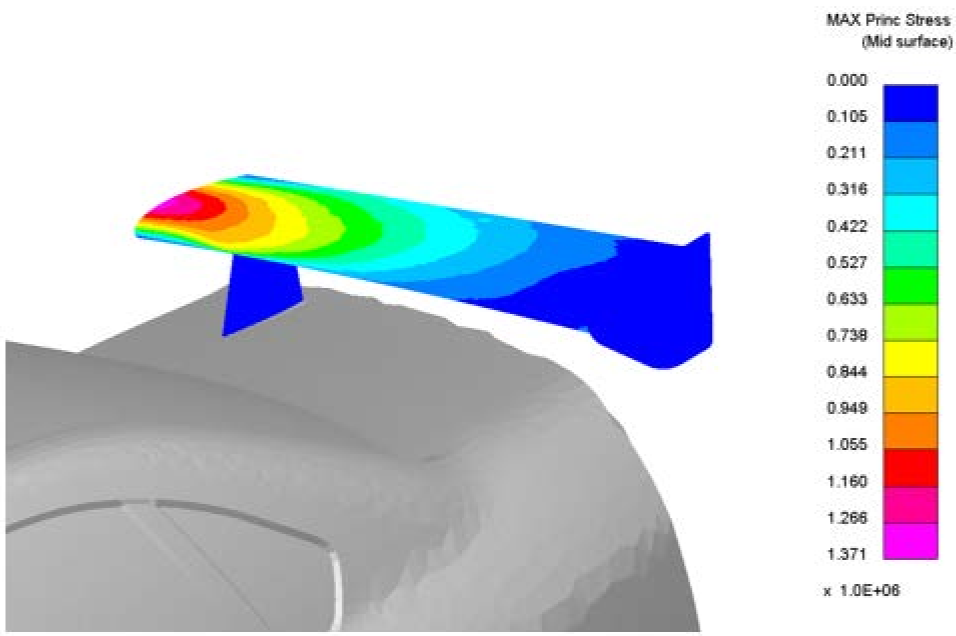


Figure 13 Maximum principal stress on spoiler during FSI simulation

2D FSI simulations of a flexible hood

The previous section demonstrated the use of ICFD for simulating FSI of a spoiler. In this example, FSI simulation of a hood is presented, demonstrating important features such as the opening up of gaps around the hood. To reduce simulation time, this model has been simulated in 2D. Figure 14 and Figure 15 show the model which is used for the FSI simulations. Figure 14 shows how the hood profile of the DrivAer car has been modified to provide a sharp leading edge which is more likely to cause flow separation. Figure 15 shows the small gap between the hood and the front of the car. This gap is intended to open up when the hood raises so that air can flow underneath the hood and into the engine bay. Ideally, this gap could be initially closed, and could open up when the hood lifts a certain distance, however the ICFD solver is not capable of modelling gaps opening up like this as the solver requires that connectivity between the surface elements stays constant over the course of the simulation. A non-linear spring has also been added to the very front edge of the hood to prevent the gap from closing over completely.

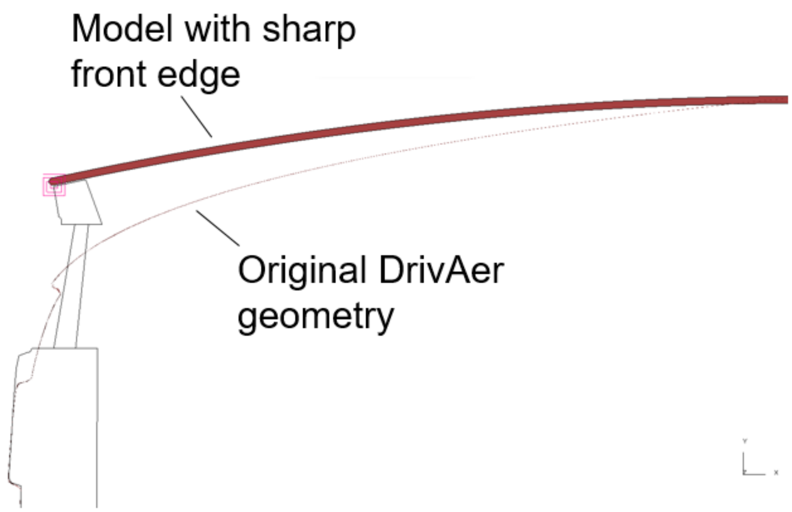


Figure 14 2D model of modified DrivAer car with a flexible hood

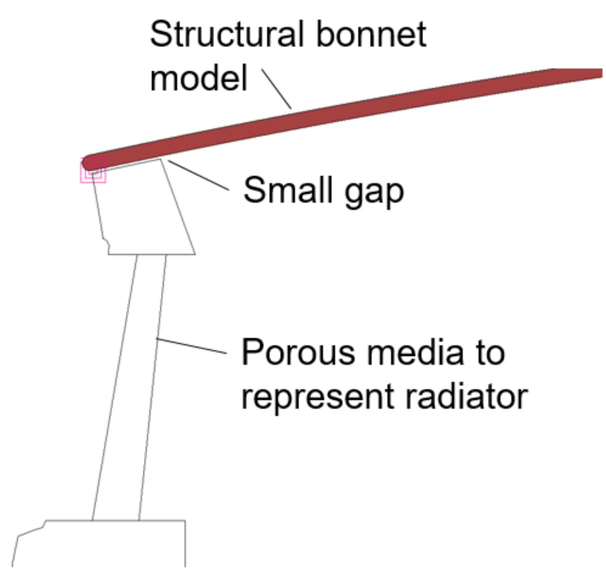


Figure 15 Close up of detail around the front of the 2D car model

An initial CFD simulation has been carried out to determine the loads on the hood when it is static. Figure 16 shows the contours of velocity from this simulation. At this flow speed of 10ms^{-1}, there is clear strong separation over the front edge of the hood, with vortices being shed from the edge. Figure 17 shows the time series of the vertical load on the hood over 10s of run time. This plot shows that after a 1 second initialisation, the force stays at an average level of around 150N positive upwards, with periodic fluctuations of up to 50N which are caused by the shedding of the vortices from the edge of the roof.

Further simulations have been carried out with the flexible hood shown in Figure 14. The top edge of the structural hood model is fixed so that the vibrations of the hood are due to bending modes about this point. The free stream velocity has been varied between 1ms^{-1} and 15ms^{-1} and the RMS hood deflections for each of these simulations are shown in Figure 19Figure 17. As an example, the time history of the deflection of the edge of the hood for the 15ms^{-1} velocity test case is shown in Figure 18. Flow velocities higher than 15ms^{-1} were tested, but these simulations crashed due to high frequency oscillations which appeared to be caused by re-meshing. This re-meshing takes place during the course of a FSI simulation in ICFD when the movement of the mesh causes cell quality to deteriorate.

Figure 19 shows how the response of the hood varies with increased wind speed. The steady increase in deflection would be expected due to buffeting from the separated flow over the hood and the increased forcing at higher flow speeds. Figure 16 clearly shows that vortices are being shed from the leading edge of the hood, and the response would be expected to peak at the vortex shedding period, however it appears that the natural period of the hood is not within the range of the vortex shedding frequencies corresponding to the flow velocities tested.

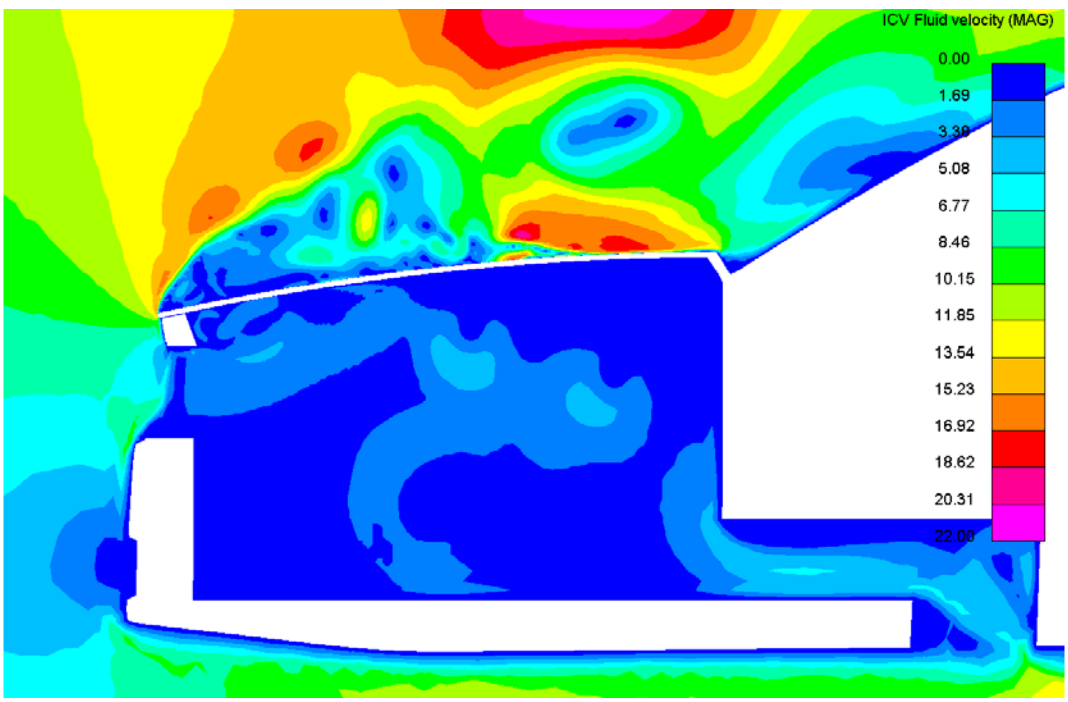


Figure 16 Velocity Contours from CFD simulation of sharpened hood profile

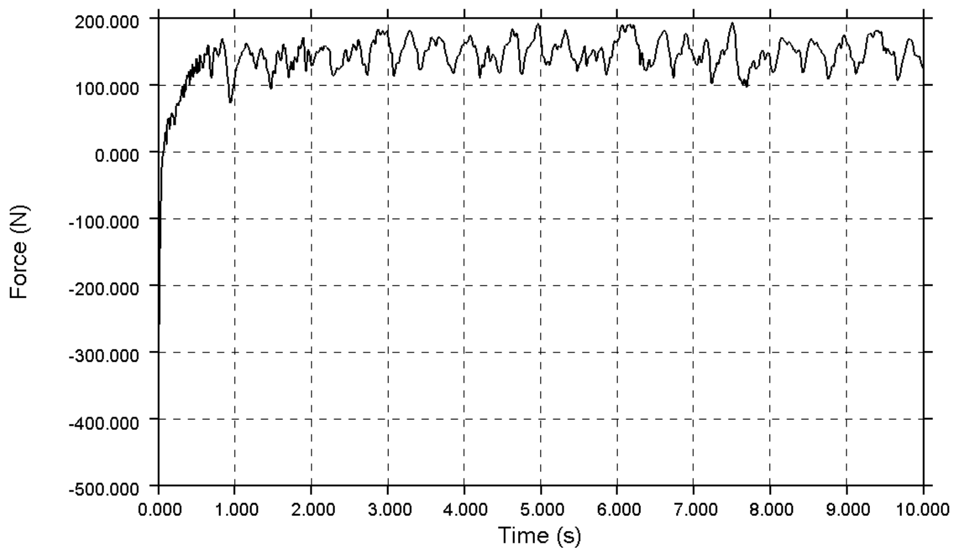


Figure 17 Time series of vertical load on the spoiler.

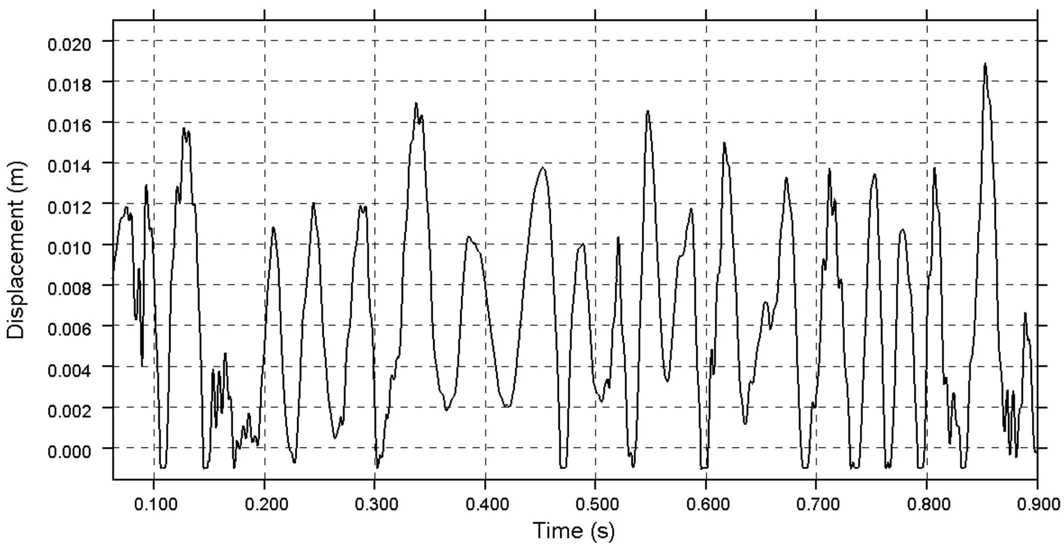


Figure 18 Time series of the deflection of the edge of the hood. Extracted from the simulation with a free stream velocity of 15m/s

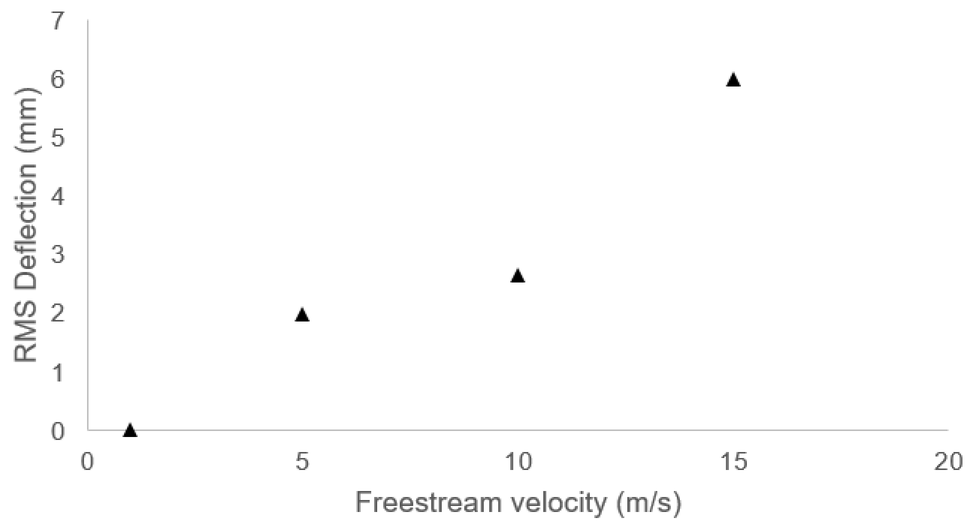


Figure 19 RMS deflection of the edge of the hood at a range of freestream velocity conditions

Conclusions

The objective of this paper was to consider the feasibility of simulating hood flutter at a commercial scale. A series of models have been created which display how the important physical features of hood flutter could be modelled. Through these simulations, the following conclusions have been made:

- Hood flutter FSI analysis requires the simultaneous simulation of a structural and fluid dynamic model, both of which may be large and complex. It may be necessary to carry out long simulations so that the stability of the hood can be reliably assessed. One of the key challenges for simulating hood flutter is reducing model complexity sufficiently so run times are not unacceptably long. Simplifications to the CFD model could include coarsening mesh in the far field and towards the rear of the car.
- One limitation of the use of ICFD for hood flutter is instability. On a number of the 2D hood FSI simulations, re-meshing caused high frequency oscillations which created unstable motions. These instabilities can be minimized to some extent by adding damping or mass to the model, but neither of these is an ideal solution, and it would be valuable for further research to be carried out in order to understand how these instabilities can be mitigated.
- The 2D example of hood flutter showed how hood opening could be modelled using an initially small gap which opens up as the hood is lifted. This is only a partial representation of a hood however, as in reality the edge of the hood would be sealed until the hood was lifted up a considerable distance. Additionally, the very small cells in the gap limited the time-step of the simulations, and is unlikely for this reason to be suitable for larger 3D simulations. Development of immersed fluid structure interaction in ICFD may provide a better way of modelling gaps opening up, however immersed FSI modelling using DEM is less reliable for modelling scenarios where the boundary layer profile is important. This may rule out its use for some hood flutter applications, as the boundary layer profile may strongly affect separation.
- The studies presented in this paper have been based on the DrivAer generic car model. Good comparison has been shown between the pressure profile from an ICFD simulation and wind tunnel results, but there is no equivalent testing data freely available for hood flutter. Further research could be focused on detailed validation using results from an aeroelastic wind tunnel model.

References

[1] E. Naudascher and D. Rockwell, Flow-Induced Vibrations: An Engineering Guide, Dover Publications, 2005.

[2] R. Blevins, Flow-Induced Vibration, Kreiger Publishing Company, 2001.

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

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

[5] 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.

[6] G. Wang, K. Gardener, E. DeHoff, F. d.Pin, I. Calcichoury and E. Yreux, "Applications of ICFD/SPH Sovlers by LS-DYNA to solve Water Splashing Impact to Automobile Body," in *11th European LS-DYNA Conference*, 2017.