Transient Fluid Structure Simulation of Ground Vehicles

Facundo Del Pin, Iñaki Çaldichoury, Rodrigo R. Paz, Chien-Jung Huang
Livermore Software Technology LLC

Abstract

Ground vehicle aerodynamics is an important stage in the design of a car. The field is currently well established, and the final goal is to decrease the lift to drag ratio functional by maintaining certain design constraints. Traditionally the studies are performed in wind tunnels but with the advance of hardware and software in the past two decades most of the design is now performed by simulation using Computational Fluid Dynamics (CFD). The automotive industry has embraced these techniques due to their low cost and high accuracy. But in recent year the tougher environmental regulations together with the natural evolution of technology have pushed the industry into new lighter materials, thinner panels and more compact parts distribution. These changes bring new challenges to the design process. The clay models traditionally used in wind tunnels cannot predict the response of the real structures subject to aerodynamic or thermal loads. Traditional CFD simulations are faced with similar limitations. Furthermore the internal organization of the CAE departments are not catching up fast enough to adapt to this new reality and the result is that last minute unexpected behaviors happen during drive test conditions forcing late modifications and the retooling of parts, greatly increasing the design cost. It is our goal to introduce in the design cycle intermediate steps where coupled Multiphysics simulations will be used to anticipate unexpected behaviors and correct the design before the tooling stage. In this work a real world model of a mid-size sedan is used as a showcase of the different possibilities that are available in LS-DYNA® to perform CFD together with Fluid Structure Interaction (FSI) to study the response of different structural parts of the vehicle subject to aerodynamic loads. One of the main advantages is that complex structural parts are "borrowed" from a LS-DYNA crash model and easily introduced into the CFD model greatly simplifying the process. All the material settings and geometry will be automatically ready for the FSI simulation.

Introduction

The final objective of this work is to provide a coupled solution between transient external aero simulations and structural analysis thus the discussion will start with the set up and validation of a transient external aero model with no FSI. Indeed this problem has some intrinsic challenges and thus building confidence in the CFD results is key before moving to a coupled simulation. In this work the well-known generic car model DrivAer [1,2,3,4] will be used. This model was developed by the Technische Universität München (TUM) in collaboration with two major car companies, the Audi AG and the BMW Group. The paper provides a brief explanation on how to set up the model as well as some best practices used in LS-DYNA ICFD to provide an accurate drag coefficient (C_d) result with the minimum effort possible.
After the DrivAer results a more realistic mid-size sedan model will be used for a coupled simulation where the transient CFD solver will be coupled to a time dependent structural solver in LS-DYNA for a FSI analysis thus replacing some of the rigid walls with flexible ones. To do this the CFD model will bring in an underbody panel from a structural model used for crash analysis. The results will provide insight with regards to the vibration frequencies of the panel when it is subject to aerodynamic loads.

The DrivAer model

The DrivAer is a modern generic car model that has wind tunnel experimental results for a large number of different configurations which are letter-codded. For instance the rear end can be fastback (F), estate back (E) and notchback (N). The underbody configuration has two different options: detailed (D) and smooth (S) (see Fig. 1).

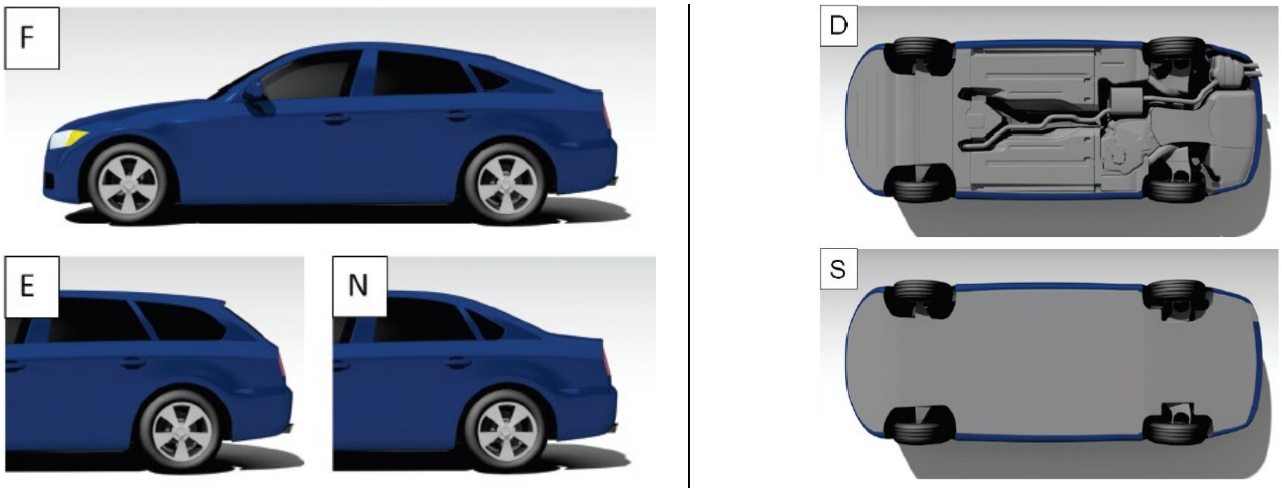

Fig. 1: Rear end configurations (left): fastback (F), estate back (E) and notchback (N). Underbody configuration (right): detailed (D) and smooth (S).

The vehicle may have mirrors (wM) or no mirrors (woM) and it may have wheels (wW) or it may not (woW) (see Fig. 2).

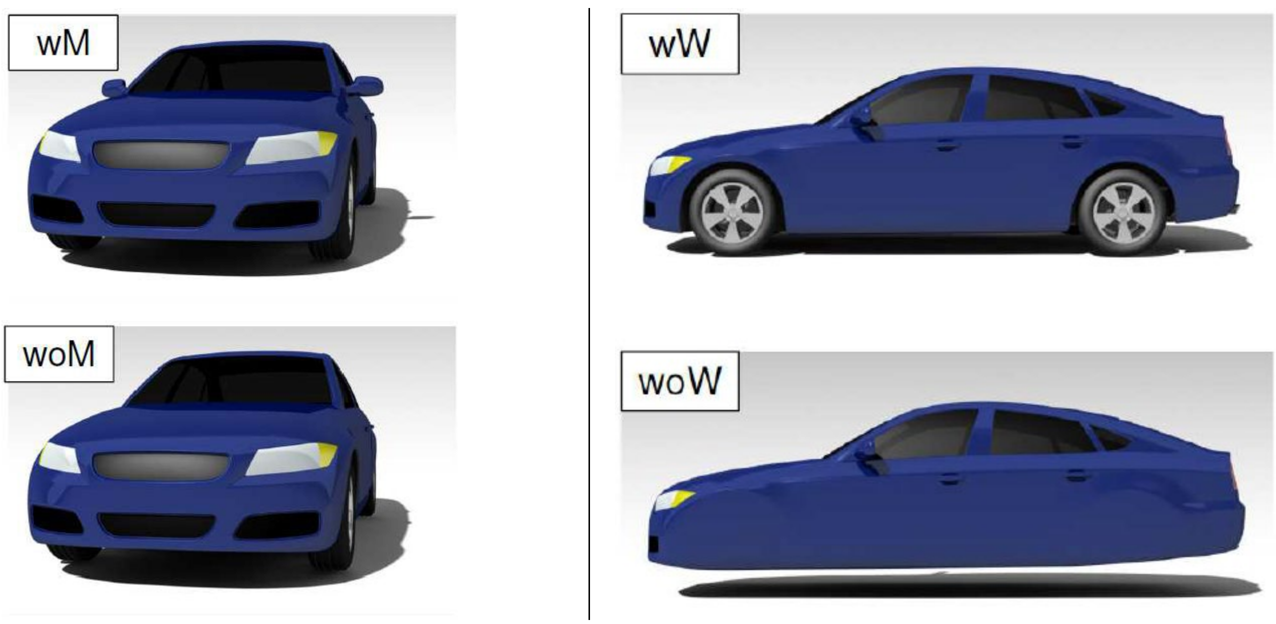

Fig. 2: On the left: vehicle with mirrors (wM) or without (woM). On the right: vehicle with wheels (wW) or wihtout (woW).

For the purpose of this paper the model to use will be F_D_wM_wW meaning fastback rear end with detailed underbody with mirrors and wheels. The geometry for all these configurations may be downloaded from the TUM website [5] in IGES format.

Model Set Up

The next step in the process was creating a surface mesh representation of the IGES geometry. For this the software package Ansa from Beta CAE [6] was used. To avoid the tedious process of CAD healing, etc. the powerful wrapping tool available in Ansa provided a CFD ready surface mesh with a simple framework easily accessible for a non-CFD expert. In fact, only the max/min element size were prescribed and then all the default options were used for a total pre-processing time of approximately three hours.

Since the ICFD solver does not require a volume mesh or boundary layer mesh to be created during the pre-processing step the meshing work is done. In Fig. 3 there are some details for the surface mesh size distribution. The mesh size ranges from 5mm in areas of highest curvature to 25mm elsewhere.

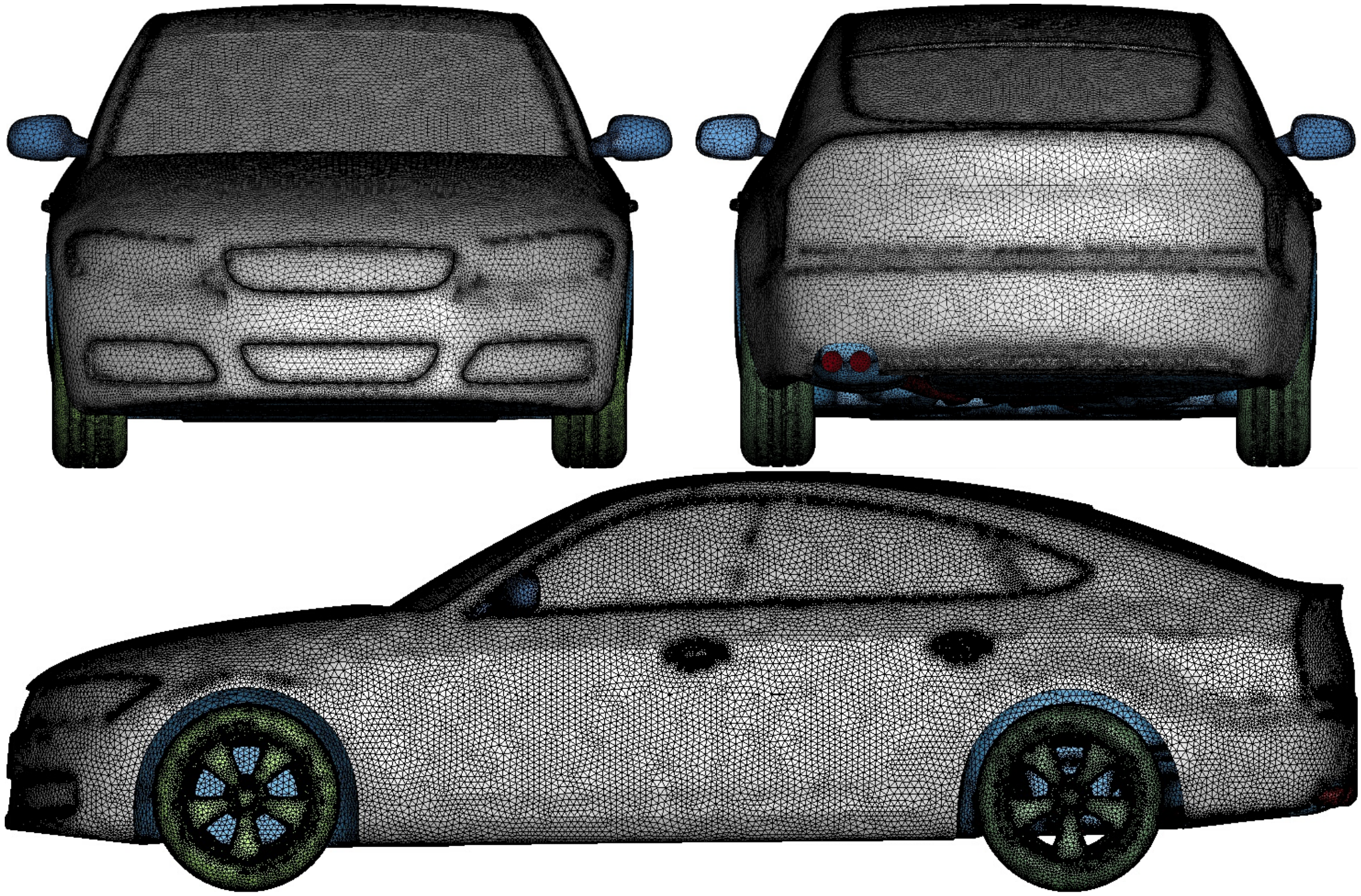

Fig. 3: Surface mesh for the DrivAer configuration F_D_wM_wW used in the CFD simulation. The mesh size ranges from 5mm in areas of highest curvature to 25mm elsewhere.

For the CFD analysis the vehicle is inserted in a wind tunnel domain that consists of a box with a higher mesh resolution on the ground closer to the vehicle as seen in Fig. 4. The walls and roof have symmetry condition. The ground could be moving or static. In the current analysis a free slip condition is used which means that the ground offers no viscous resistance. The inflow has a constant velocity and the outflow has a prescribed pressure. In the current analysis the following properties were applied:

Fluid density	1.2047 Kg/m^3
Fluid viscosity	1.7×10^{-5} Kg / (m s)
Inflow velocity	16 m/s
Frontal Area (A_{ref})	2.16 m^2

Using the length of the vehicle L=4.6m the Reynolds number becomes $Re=5.2 \times 10^6$ which is in accordance with the largest experimental Re number used. Larger Re number do not change the C_d significantly.
The drag coefficient is computed using the equation:

$$C_d = \frac{2\, F_x}{\rho u^2 A_{ref}} \quad (1)$$

Where F_x is the fluid forces in the x (frontal) direction and ρ, μ are the fluid density and viscosity.

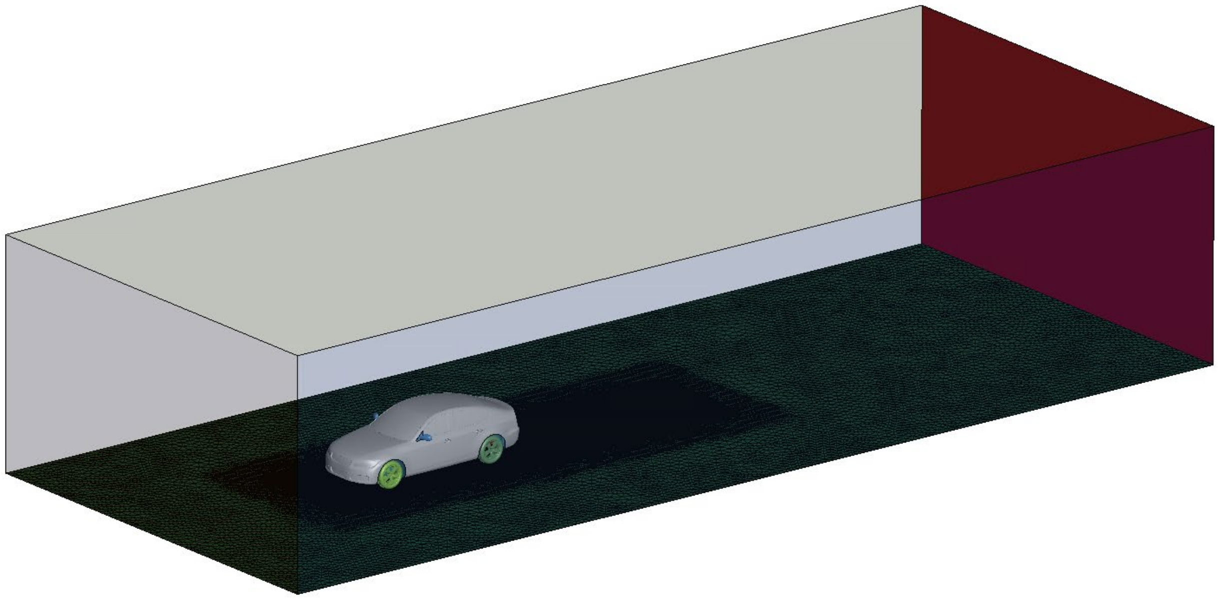

Fig. 4: Wind tunnel domain for the CFD analysis.

DrivAer results

At this point everything is set up to start the numerical analysis. All the controls for the volume mesh are prescribed in the input deck and executed at run time. There is a large array of command combinations for the user ranging from different turbulence models to meshing parameters, boundary layer mesh strategy, local element size definition, etc. In this work there will be a brief discussion of two different scenarios that a user may apply which could depend on skill level or familiarity with the problem, knowledge of the solver capabilities, computational resources available, level of detail required for the study, etc.

Case 1: default values

In the first scenario the user runs the model with default values. This means that the user only set up the material properties for air and the boundary conditions. This could be attributed to a novice user, somebody not familiar with CFD (or the solver) or an advanced user that needs a quick and dirty solution. The mesh size in this model is 6.2M tetrahedral elements.

Case 2: adding a turbulence model and a boundary layer mesh

With minimal effort the user can greatly improve the solution by adding a turbulence model and a boundary layer. To this end two keywords need to be added to the input deck: *ICFD_CONTROL_TURBULENCE and *MESH_BL. In this case an LES Smagorinsky turbulence model is added (option “2”) with default parameters and only one layer of anisotropic elements are added with the default strategy. So there is no tuning of the static Smagorinky parameter or the position and distribution of the elements in the boundary layer. The number of elements in this model is 10M tetrahedral elements. In this model $Y^+ \approx 100$ was used for the hood, roof and most boundaries with attached flow.

Case 1: Novice user / quick and dirty solution

Case 2: Advanced user: LES + boundary layer mesh

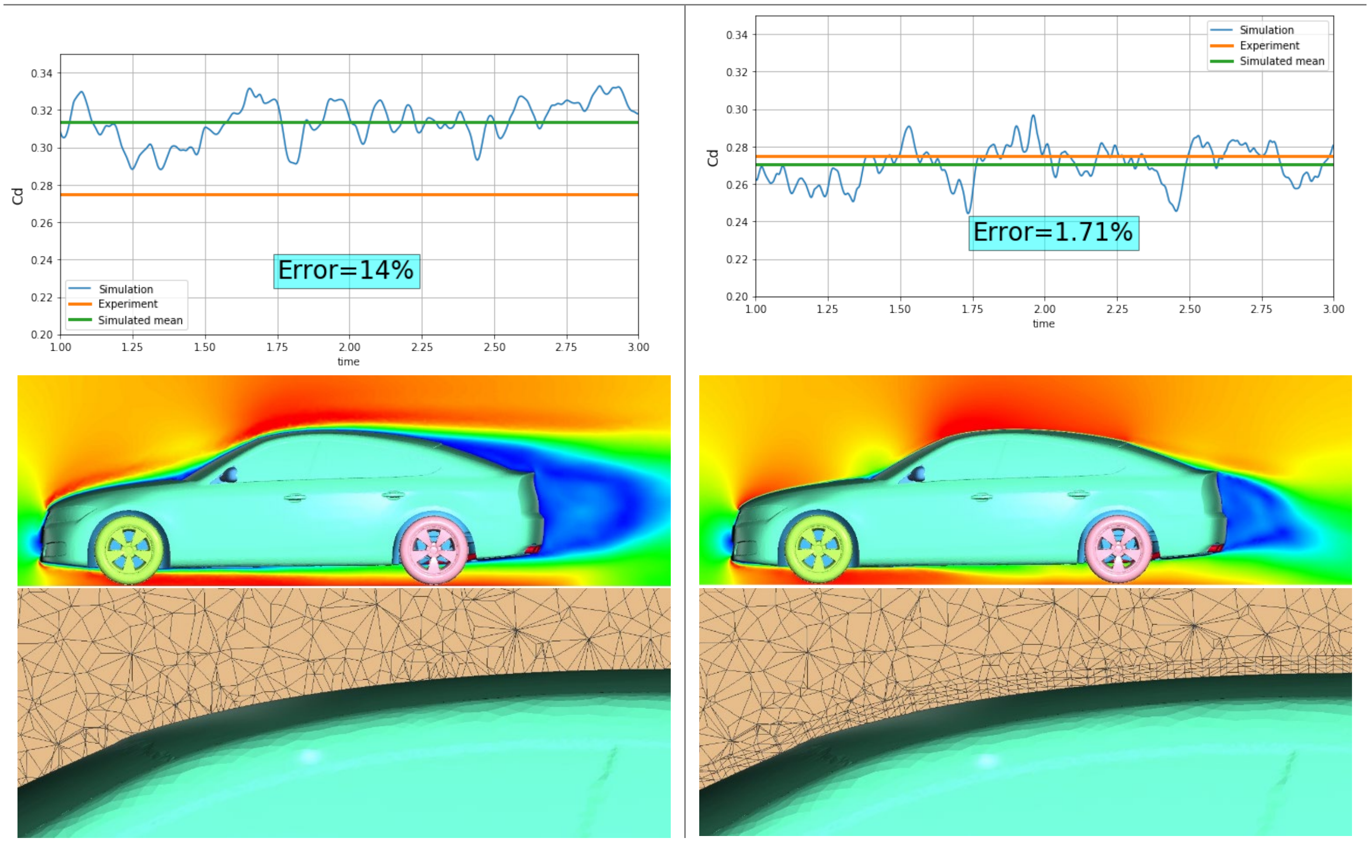

Fig. 5: comparison of solutions for Case 1 and Case 2. Top row shows the drag results and error, middle row shows the time average velocity and bottom row shows the mesh on the roof.

The result are presented in Fig. 5. Clearly the difference between simulation and experiment is large for Case 1 (left). But this is relative to the user and his objective. An expert wanting to simply identify a trend or the impact of a geometry change may actually be satisfied with the outcome. The other important fact to note is that the solver will run and provide an answer in a robust manner even when the settings are not ideal. On the right of the image is *Case 2* where the drag is in very close agreement with the experimental results.

The time average velocity for both cases are compared showing how the simplified *Case 1* model has a much larger bubble at the rear end of the vehicle that creates excessive suction increasing the drag. On the bottom of the figure it shows a detail of the mesh at the roof that shows the difference when a boundary layer mesh is used.

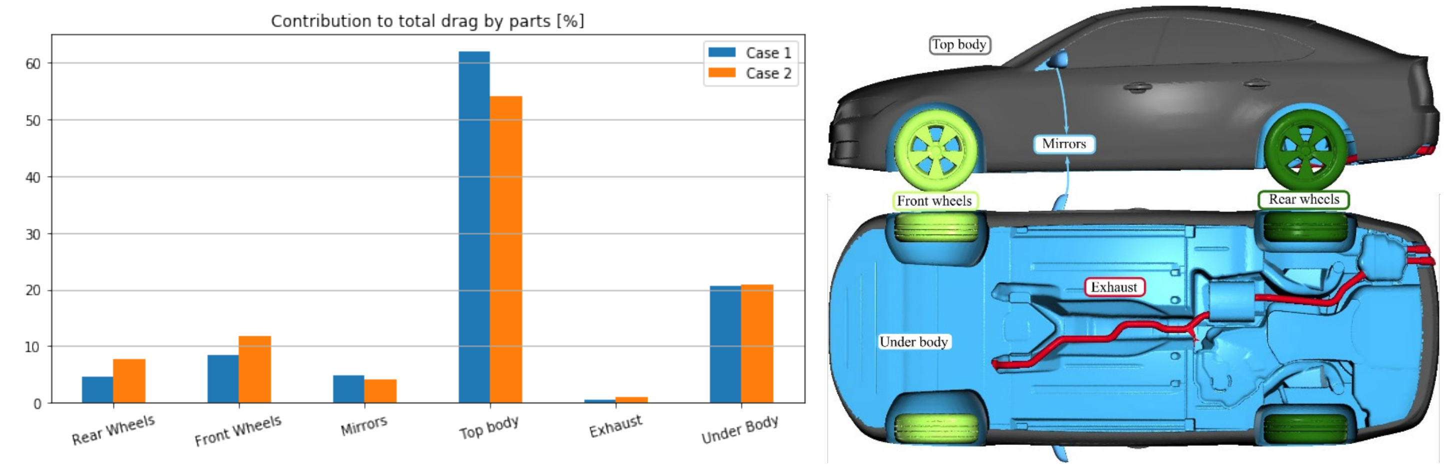

Fig. 6: contribution of the different components of the vehicle to the total drag as a percentage.

Finally, in Fig. 6 there is a comparison of the contribution of each part of the vehicle to the total drag for *Case 1* and *Case 2*. For *Case 1* the drag is over predicted on the top body while it is under predicted on the wheels. This simple example depicts the setup of a transient CFD analysis for ground vehicles in LS-DYNA ICFD. More advanced tools for mesh control, turbulence models, etc. are available to further fine tune the solver for models that require higher resolution.

A transient Fluid-Structure Interaction case

In this section a realistic model for a mid-size sedan vehicle with highly detailed underhood and underbody (see Fig. 7) will be used for the transient CFD analysis. The model also includes heat exchangers with porous media. The difference with the previous section is that a part of the vehicle will be flexible instead of rigid and will deform in time due to the transient fluid loads.

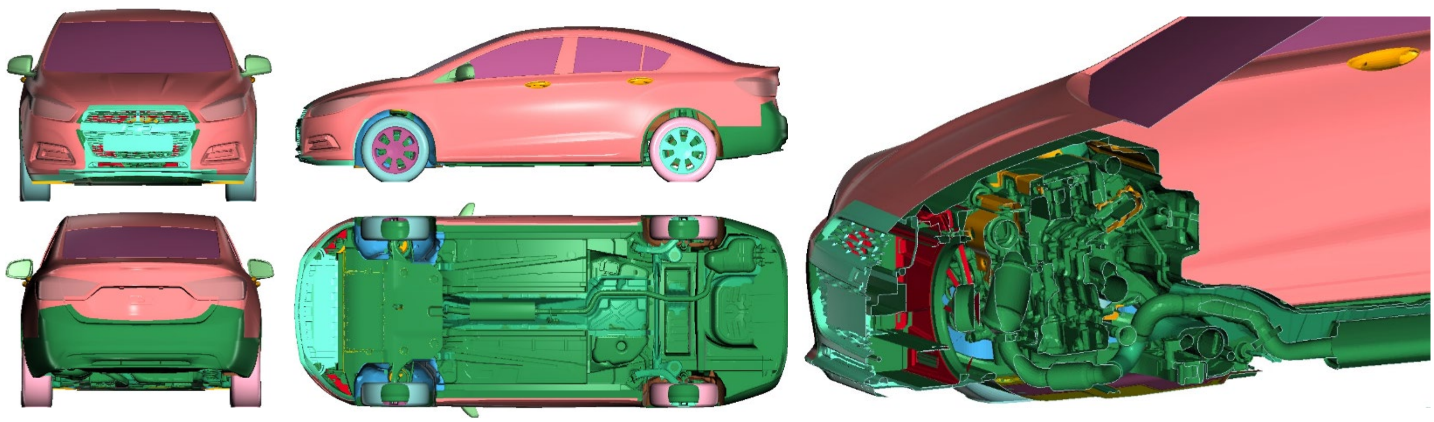

Fig. 7: mid-size sedan model used for the transient FSI simulation with detailed underhood and underbody.

The mesh resolution for this model is similar to that of the previous section (see Fig. 8) and much of the set up details will be skipped to focus on the FSI approach.

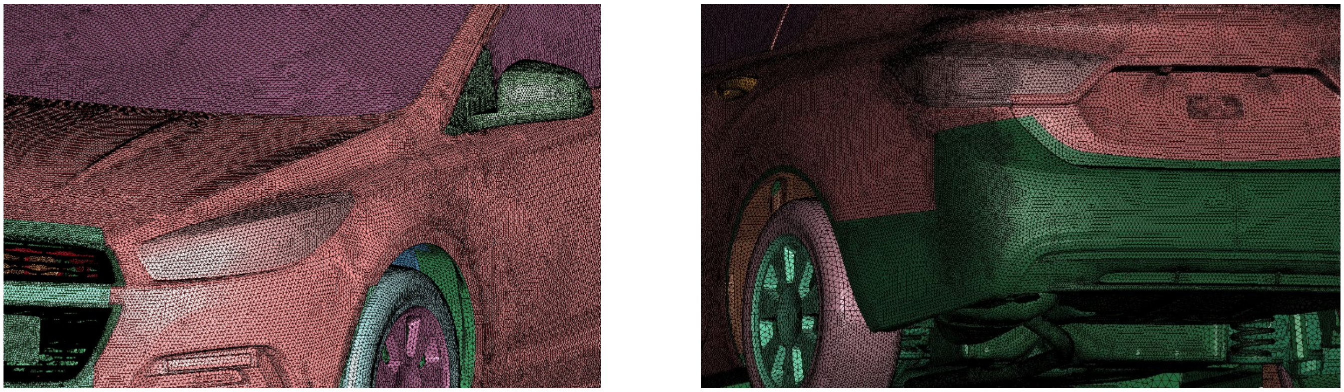

Fig. 8: mesh details in the front and rear end of the vehicle.

The main idea is to use a CFD model that is already working (possible to predict drag and lift) and convert some rigid parts into flexible by bringing in to the CFD model a structural part. In LS-DYNA this could be done by importing into the CFD input deck parts from crash analysis models which already have their material properties/models specified. In Fig. 9 the crash model for a side impact of a typical mid-size sedan is shown. This model has a large structural detail and some of their parts that are relevant for aero simulation may be imported into the CFD analysis for FSI. As an example the underbody panel that covers the engine compartment will be used. In Fig. 10 the part in the CFD model and the structural part are depicted.

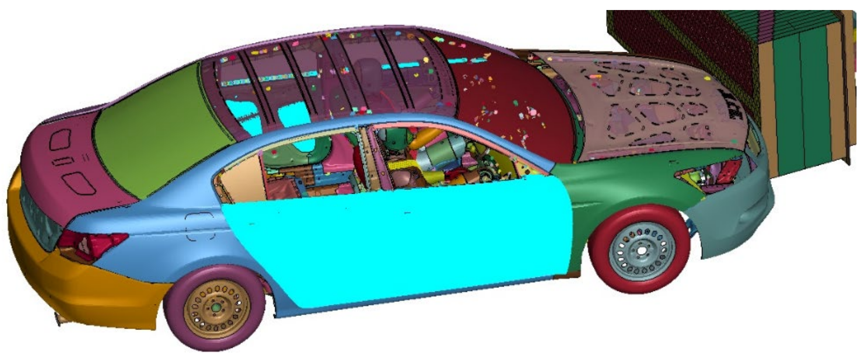

Fig. 9: crash model for a mid-size sedan vehicle.

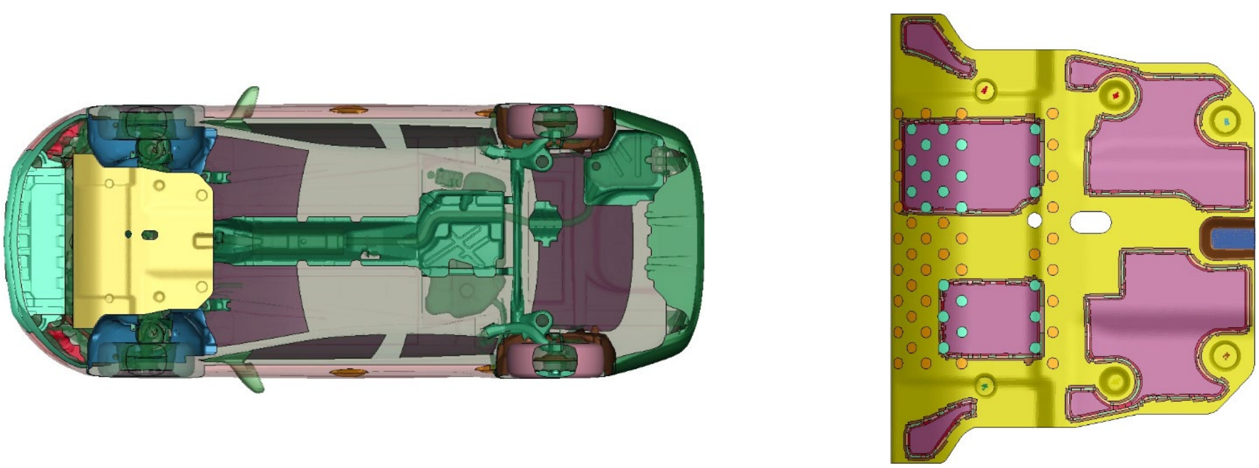

Fig. 10: underbody panel used in the FSI simulation. On the left the panel in the CFD domain. On the right the structural part. The colors indicate parts of the model with different material properties or material models.

What makes LS-DYNA so attractive for this kind of simulations is that only one new keyword is added to the CFD model to convert it into a coupled FSI simulation: *ICFD_BOUNDARY_FSI, which indicates the part IDs (PIDs) of the fluid domain that will be flexible during the FSI simulation.
The fluid properties, boundary conditions and description of the wind tunnel are inherited from the previous section with the exception of the inlet velocity which in this case is $u = 22m/s$. In Fig. 11 the CFD results for time average pressure as well as streamlines of the time average fluid velocity are shown.

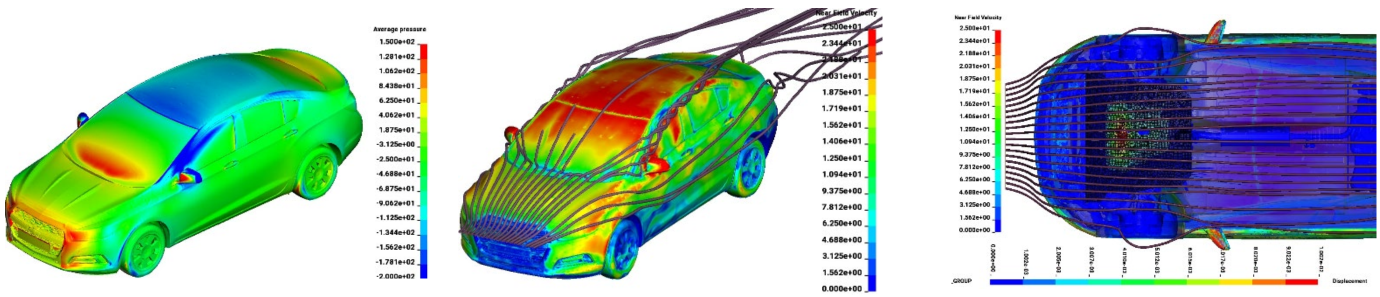

Fig. 11: CFD solution showing the time average pressure on the left, near field velocity (instantaneous) with streamlines computed using the time average velocity (center) and on the right the underbody panel together with the streamlines.

On the structural side the model can predict the vibration response of the structure under realistic excitation forces provided by the CFD solution for the case of a vehicle running at $22\ m/s$. A further analysis for flatter prevention should sweep the range of operational velocities of the vehicle which requires a more intensive study out of the scope of this work. In Fig. 12 the force excitation and their corresponding FFT are shown. It is observed that for F_x there are two main frequency modes one at $40Hz$ and the other one at $80Hz$. For the case of F_z the response is more chaotic.

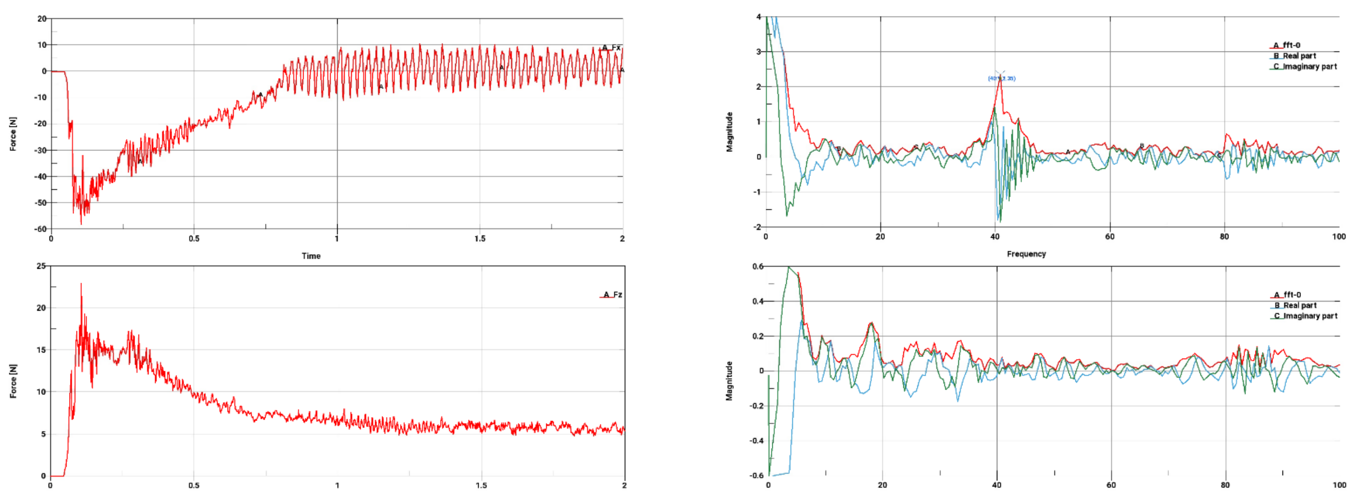

Fig. 12: On the right total excitation forces for F_x and F_z. On the left FFT to identify frequency modes that could be used in a flatter prevention or noise analysis.

These results are of great value in the design process to prevent potential excitations close to the structural flatter modes that can results in failure of the structure. In the case of underbody panels the results may not be catastrophic since the structure is close to the ground. In the case of rear aerodynamic spoilers a failure at high speed may result in pedestrian injuries or impact with other road vehicles. Although this analysis could be more computationally intensive than a simple CFD prediction they could be performed at the end of the design cycle to avoid last minute surprises that could result in costly last-minute re-designs and re-tooling.

Finally in Fig. 13 a map of displacements is shown for the underbody panel at a pseudo transient state.

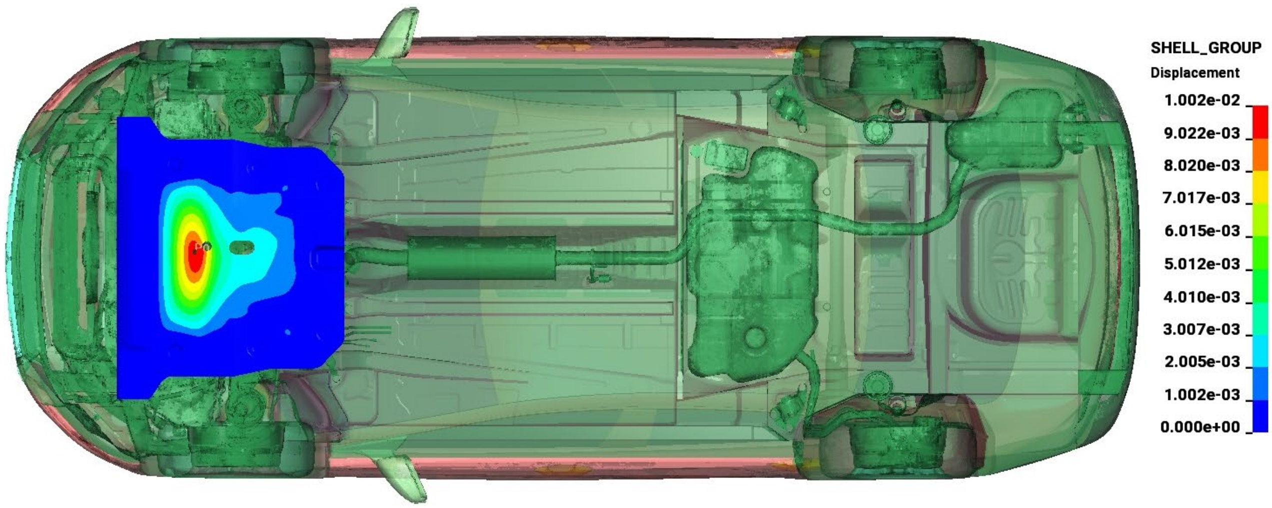

Fig. 13: Displacement field on the underbody panel at a pseudo transient state.

Conclusions

In this work the ICFD solver in LS-DYNA was successfully applied for the solution of transient simulations for CFD only and then coupled FSI approaches. In the first part of the paper an introduction to computing an accurate transient CFD solution was presented. It was shown that a basic model that uses only the pre-defined solver options produced a large error which for a novice user may indicate the failure of the solver while for a more advanced user it may show a qualitative quick and dirty trend of results. Although the result is not quantitative accurate the model runs robustly on a "first try". By adding a turbulence model and a boundary layer mesh the solution improved significantly which was regarded as the advanced user solution with the addition of only two keywords to the model: *ICFD_CONTROL_TURBULENCE and *MESH_BL. The results were compared to experimental results for the well-known DrivAer experimental benchmark.

In the second part of the paper the same principles were applied to a more realistic vehicle with detailed underbody and underhood. In this case instead of using rigid walls a part of the underbody was considered flexible. The structural model was brought in and included into the model from a crash input deck. A single keyword was needed to indicate the coupling: *ICFD_BOUNDARY_FSI. The transient FSI analysis allows accurate computations of frequency mode excitations that could be used in flatter or noise analysis to prevent last minute design surprises.

References

[1] A. Heft, T. Indinger, N. Adams: Experimental and Numerical Investigation of the DrivAer Model, ASME 2012, July 8-12, 2012, Puerto Rico, USA, FEDSM2012-72272.
[2] A. Heft, T. Indinger, N. Adams: Introduction of a New Realistic Generic Car Model for Aerodynamic Investigations, SAE 2012 World Congress, April 23-26, 2012, Detroit, Michigan, USA, Paper 2012-01-0168.
[3] J. Wojciak, N. Adams, T. Indinger, P. Theissen, R. Demuth: Investigation of Unsteady Vehicle Aerodynamics under Time-Dependent Flow Conditions, 29th AIAA Applied Aerodynamics Conference, June 27-30, 2011, Honolulu, Hawaii, USA, Paper AIAA 2011-3349.
[4] A. I. Heft, T. Indinger, N. A. Adams: Investigation of Unsteady Flow Structures in the Wake of a Realistic Generic Car Model, 29th AIAA Applied Aerodynamics Conference, June 27-30, 2011, Honolulu, Hawaii, USA, Paper AIAA 2011-3669
[5] https://www.mw.tum.de/en/aer/research-groups/automotive/drivaer/
[6] https://www.beta-cae.com/ansa.htm