Ground Vehicle Aerodynamics using LS-DYNA

Facundo Del Pin[1], Iñaki Caldichoury[1] and Rodrigo R. Paz[1]

[1]Livermore Software Technology Corporation

1 Abstract

The use of software for the simulation of airflow around ground vehicles plays an important role in the design of a vehicle. Traditionally companies within the automotive industry sector have a Computational Fluid Dynamics department dedicated to the analysis of drag and lift with the objective of improving fuel efficiency, safety, passenger comfort, cooling systems and heat exchangers and minimizing noise. In recent years the increased pressure from government regulators for dramatic improvements in fuel efficiency has pushed the automotive industry to start innovating in lighter materials and hybrid or full electric cars. This has created new challenges in the way that traditional companies operate since the mechanical response of new materials or batteries may be greatly influenced by the fluid temperature and fluid pressure loads which in turns may affect the fluid behavior. forcing companies to overlap efforts between a structural and fluid department. The current paper will show how the use of LS-DYNA may help in reducing the burden of moving results from a CFD / thermal solver to a structural solver, preventing the degradation of the results from mapping variables and the increased cost of licenses for more than one solver, simplifying a queuing system since potentially a single run could provide all the results at once and improving the accuracy since the full non-linear model is being solved. This in turn reduces the communication time between structural and fluid departments where the interaction is sometimes informal and based on “favors” more than following project guidelines. The paper will provide an overview of the tools that LS-DYNA has for CFD analysis and it will show some application examples.
Keywords: computational fluid dynamics, turbulent flow, multipysics, coupled problems, conjugate heat, fluid structure interaction.

2 Introduction

The study of ground vehicle aerodynamics provides insight to engineers and designers about the fluid forces that the air exerts over the vehicles. The forces are mainly of two types: the ones coming from normal stresses (pressure) and those from tangential stress (viscous forces). There is a large amount of research in the field of vehicle aerodynamics and the mechanics of the problem is very well understood. While engineers may prefer streamlined cars designers or stylists may prefer vehicles that appeal to consumers and finally a compromise between the two is reached.
In recent years the addition of new materials in the manufacturing of cars and the necessity of building lighter cars that reduce fuel consumption set new challenges for engineers. Indeed the use of thinner panels that can deform due to aerodynamic forces or materials whose properties may change when exposed to higher temperatures and pressure loads may results in behaviors that are not expected. Wind tunnel or pure CFD analysis will not correlate road tests in real driving conditions where the sum of all the variables play an important roll in the final result.
With the multiphysics capabilities of Ls-Dyna engineers can couple CFD analysis with a thermal or structural solver or both for a more realistic study of not only drag and lift but also the structural response (thermal and mechanical) that in turn may affect the drag and lift.
In the following sections an introduction to the incompressible CFD solver in Ls-Dyna is provided. Then different coupling possibilities are shows using examples for the thermal and mechanical problems.

3 The incompressible CFD solver in Ls-Dyna

In recent years LSTC has devoted big efforts in the development of a CFD solver for incompressible flows. The solver is specifically design to tackle coupled problems where low Mach numbers (M<0.3) are involved produceing a scalable parallel solution. Some of the main features of the solver involve:

1. implicit solver to allow larger time steps,
2. optimal MPP scalability,
3. automatic mesh generation including boundary layer mesh,
4. turbulence models,

5. free surface flows,
6. porous media flow,
7. coupled to the structural and thermal solvers.

Extensive validation has been performed to test the accuracy and robustness of the solver. The tests are documented and available thourgh our website.

4 Coupling possibilities

4.1 Thermal coupling

The incompressible CFD solver can be coupled to the heat equation to perform thermal analysis in the fluid. The heat equation for the fluid can also be coupled to the heat equation of the structural problem to perform conjugate heat transfer analysis. As shown in Fig.1 the properties of a typical plastic used in underhood and underbody parts of a vehicle have large variations as a function of temperature. This behaviour could be studied simultaniously with the CFD analysis in a full non-linar analysis.

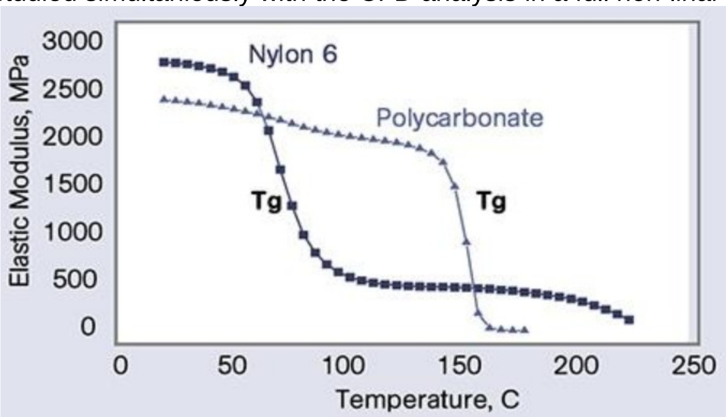


Fig.1: Modulus vs. temperature plot for unreinforced nylon 6 (semi-crystalline) and PC (amorphous). The materil properties change significantly with the temperature and this will affect the mechanical response and consequently the flow around it.

A thermal problem was solved using a model which was provided by GMC for an aerodynamic benchmark. The model includes the engine compartment, exahust, suspention, wheels, radiator, grill and mirrors. The model and mesh is shown in Fig.2. The aerodynamics analysis was augmented with a thermal analysis to predict the fluid temperature around the engine block and exahust pipes. The results are shown in Fig.3.

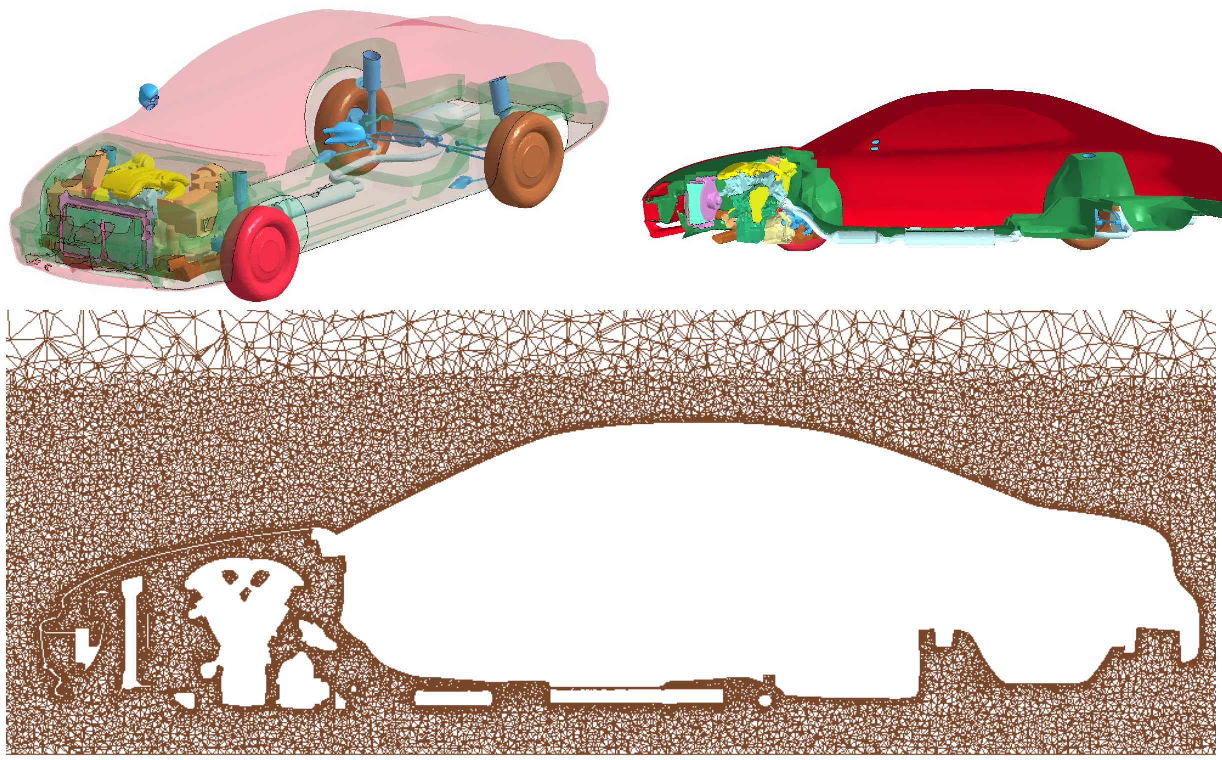


Fig.2: Geometry and cut plane of a GM Pontiac model used for benchmarking the CFD solver and coupling it to the thermal analysis.

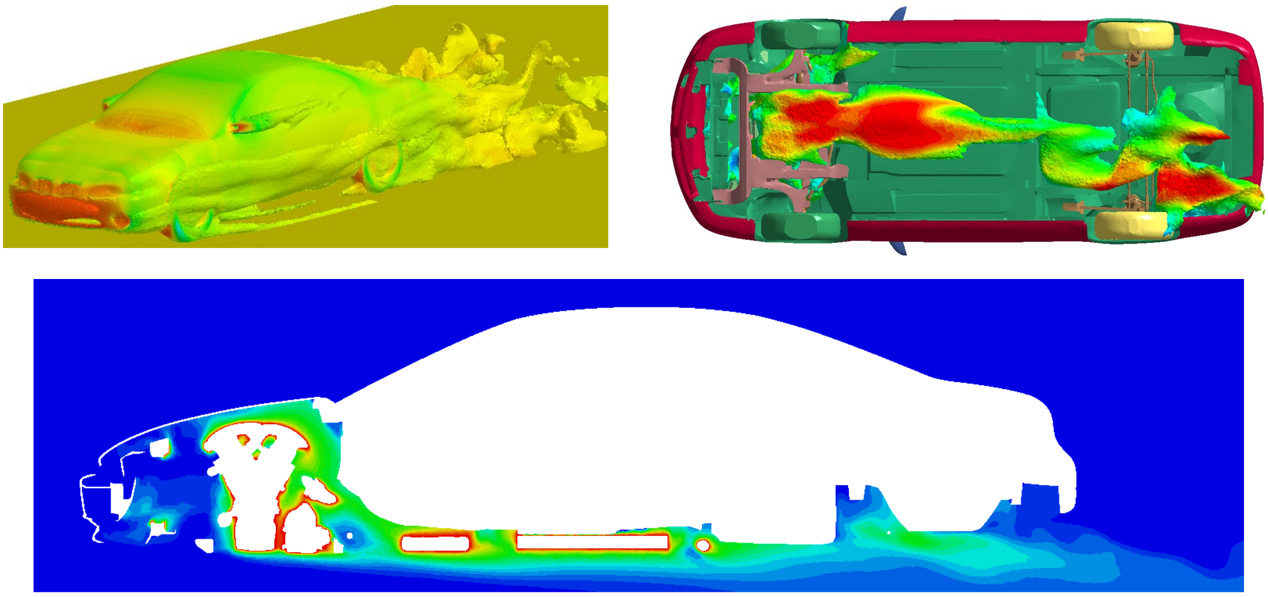


Fig.3: Results for the aerodynamics benchmark on a GMC vehicle. Top left shows the iso-surface of velocity, top right is iso-surface a fluid temperature and bottom is a cutplane with iso-contours of temperature showing the higher tempreature around the engine block and exahust.

4.2 Fluid structure coupling

The other coupling option which could be combined with the thermal coupling is the one that involves fluid structure interaction (FSI). Strong fluid/structure coupling ocurs when the structural and the fluid response are highly dependant of each other. A CFD model which is already working can be used for FSI. The structural part of the model needs to be defined using the standard keywords and format used for structural analysis. The meshes for the fluid and structural model do not need to match at the interfaces althought the geometry need to be close enough. Once this is ready a single added keyword on the CFD model will activate the FSI analysis.

In Fig.4 a CFD analysis of a GM Camaro vehicle is shown. After the CFD results where obtained an FSI analysis was requested to study the roof deformation due to the wind speed. Fig.5 shows the post-processing results overlapping CFD and structural deformation for a full non-linear analysis.

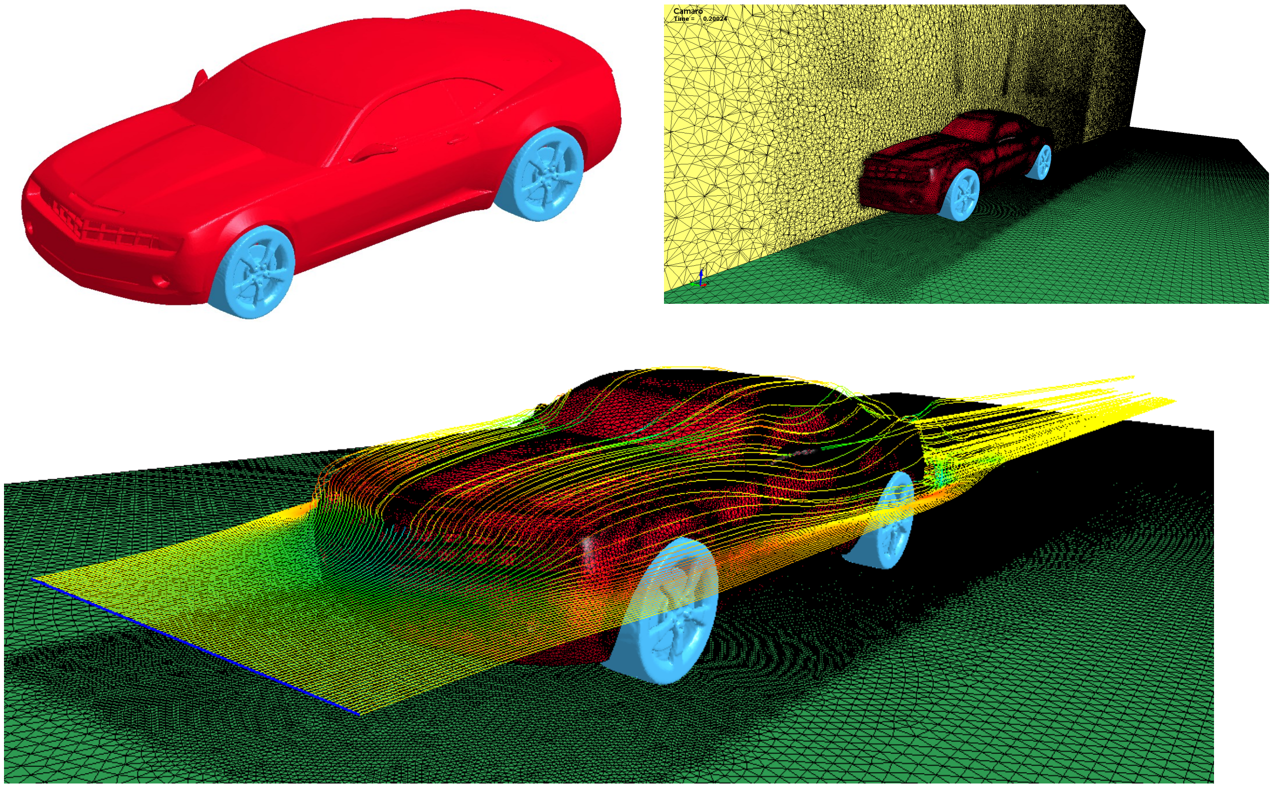
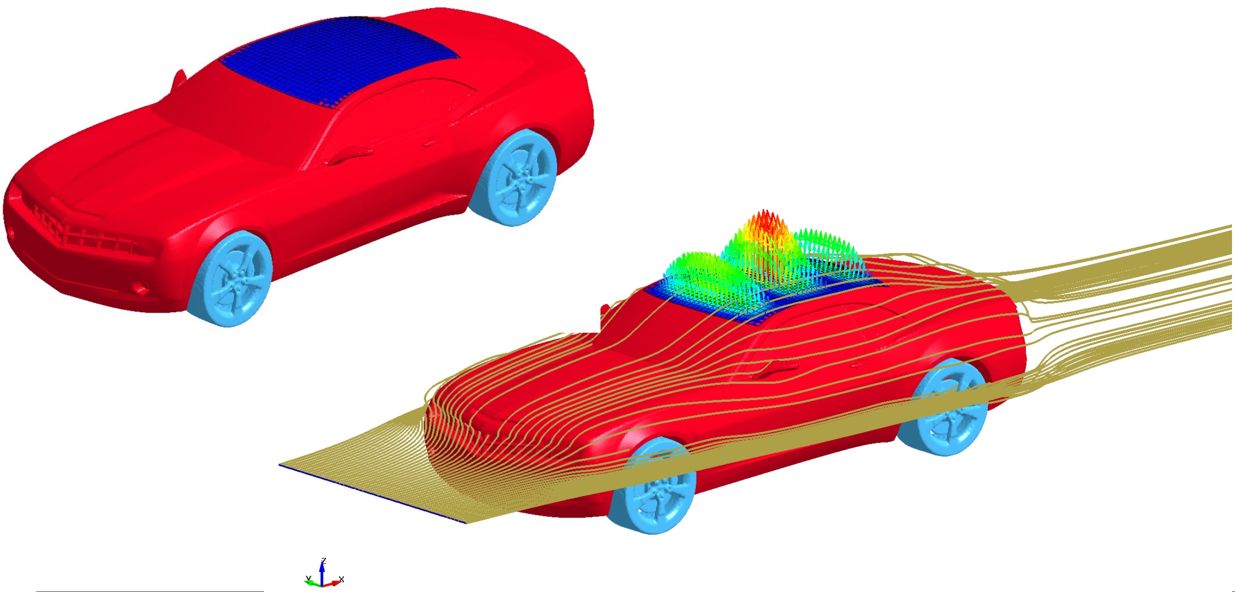


Fig.4: CFD analysis of a vehicle which will be used for FSI simulation.



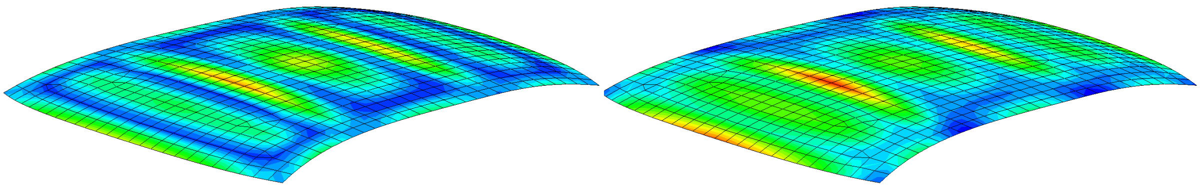


Fig.5: From top to bottom: structural mesh on top of the CFD model for an FSI analysis of the roof pannel, post-processing of the flow around the vehicle using streamlines and structural displacement of the roof, Presure and structural stresses for the roo panel.

5 Summary

This paper depicts the possibilities that Ls-Dyna offers in the area of multiphysics regarding CFD, structural and thermal analysis. It was shown that the behavior of turbulent air around a vehicle can be modeled and that the influence that the temperature or structural displacement may have over the flow can be predicted. This opens the possibility for more complete vehicle analysis with little extra cost added. In the case of FSI the extra computational time that is used in the non-linear coupling is much less than the time needed to run two separate analysis, one for CFD and then one for structural analysis using the CFD pressure mappings to achieve only linear coupling performance as it is normally done in the industry.