Multi-physics applications for ground vehicle aerodynamics: structural thermal radiation coupled to CFD for a more accurate temperature prediction

Facundo Del Pin, Art Shapiro, Iñaki Caldichoury, Rodrigo R. Paz and Julie Anton
Livermore Software Technology Corporation

Abstract

Thermal radiation is an important heat transfer mechanism which may greatly influence the mechanical behavior of structural parts of a vehicle when they are exposed to aerodynamic loads. This effect is even more evident in high performance vehicles where the engines are running at ever higher temperatures and the structure is built using lighter parts with less conventional materials. The radiated heat from the engine or exhaust system will increase the temperature in parts of the vehicle that are not directly in contact or even physically close. The increased temperature will soften the materials making it more prone to deformation in the presence of the fluid loads. The problem then requires the coupling of the radiation solver with a fluid mechanical solver to accurately predict the temperature of the mechanical part interacting with the air temperature and the deformation of that part when interacting with the air pressure. In the current work the first case will be studied. The conjugate heat transfer solver will be applied to the problem of predicting the temperature in parts of a vehicle when it is heated by radiation from the engine taking into account the cooling effect of the air.

Introduction

Multiphysics simulations are increasingly becoming more important in the design phase of ground vehicles. Traditionally simulation was only performed in the area of CFD, structural mechanics and thermal problems independently from each other. In recent years the use of new materials and new manufacturing methodologies has pushed the industry to try to predict the behavior of materials when fluids, structures and thermodynamics are coupled. A typical scenario is shown in Figure 1.

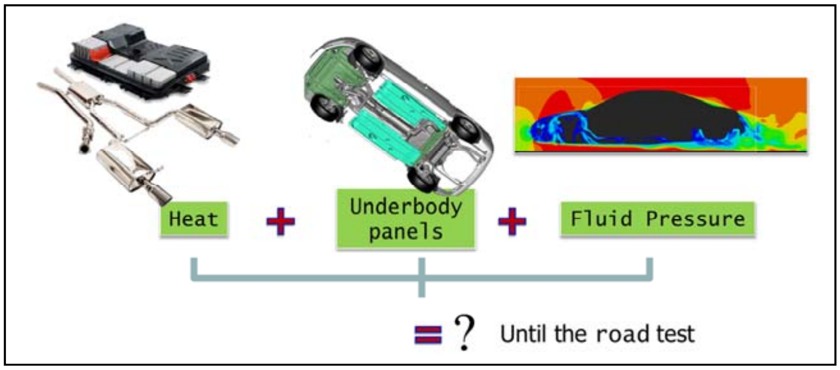


Figure 1 Schematic representation of what a fully coupled analysis should predict.

There are a number of reasons why the use of Multiphysics has not really penetrated the industry yet and the blame resides on both sides namely the automotive industry and the software providers. In the first case the industry has a rigid internal structure where a CFD group only resolves problems related to fluids/thermal analysis and the structural group does everything related to solids and structures and there are no intermediate department that helps in the interaction between the two groups. On the other side the simulation software is guilty of not providing a simple solution that would allow engineers to use a single model to deal with the fluid/thermal/structural problem. This approach would allow engineers to easily interchange models among different working groups without a major re-writing of the input decks. A typical approach is to use different software for each part of the model and then use an intermediate library to handle the transfer of information between the CFD, structural and thermal solver (see [1,2]). This process is costly, inaccurate and the results are very hard to reproduce when the configurations are changed.

In this paper a more compact methodology will be presented using only LS-DYNA® for the simulation. The paper will focus on the thermal analysis coupled to the CFD simulation. To further emphasis the point of using a single model a coupling with the structural solver will also be presented. The application chosen will attempt to predict the temperature at the hood of a full vehicle which is heated due to radiation from the engine block. The selected geometry is that of the GEO Metro reduced crash model from NCAC (http://www.ncac.gwu.edu/vml/models.html).

Model Description

The geometry chosen for the analysis has all the major components found in a ground vehicle (see Figure 2).

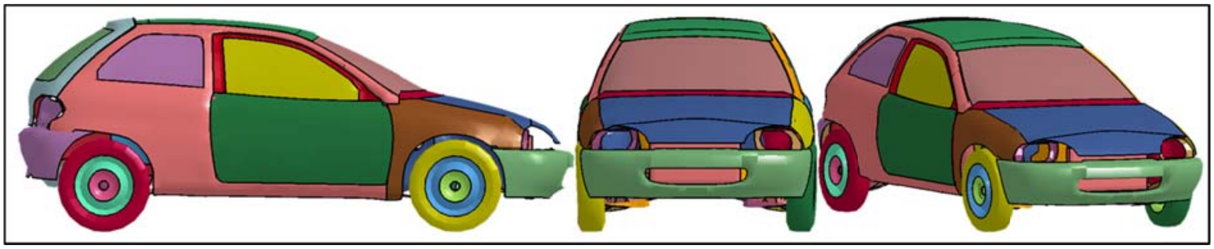


Figure 2 Views of the GEO metro reduced crash model.

Under the hood the engine block will provide the heat source where the temperature will be set to T=400K. The engine geometry has been simplified to allow for a faster simulation of the radiation solver. Figure 3 shows the engine and hood.

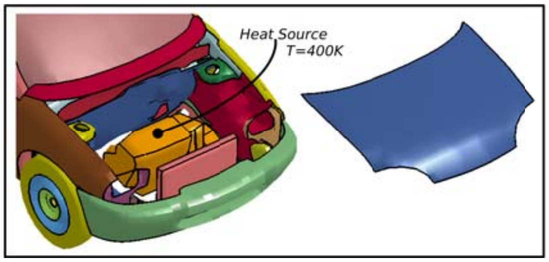


Figure 3 Engine compartment showing the engine block.

For the Conjugate Heat solution the vehicle will be placed inside a wind tunnel. Although the geometry of the vehicle is the same the mesh used on the CFD part of the solution will be refined increasing the resolution to capture more fluid features. The meshing tool ANSA was used to generate the fluid mesh. A comparison of the two meshes is depicted in Figure 4.

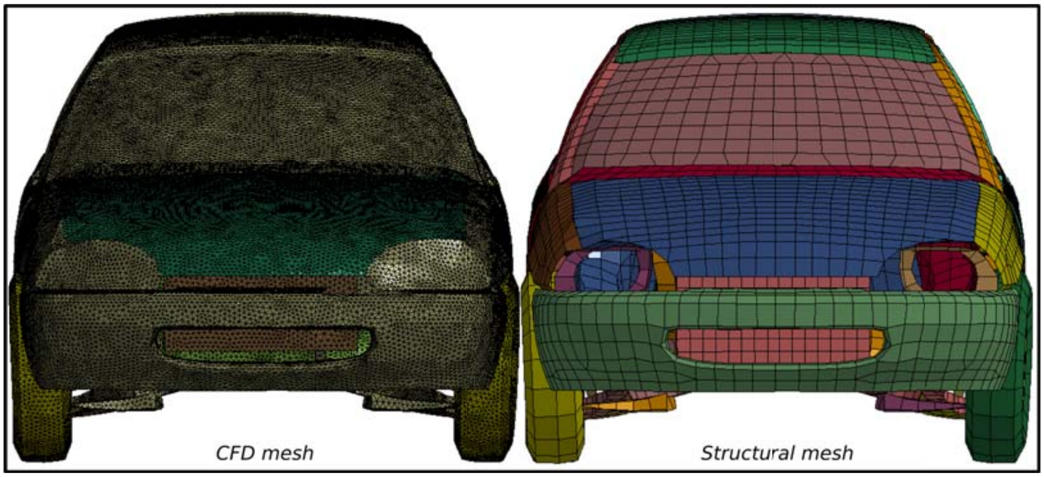


Figure 4 Comparison of the CFD mesh and the structural mesh.

Thermal Analysis

LS-DYNA® can model thermal radiation transport between gray diffuse surfaces using the *BOUNDARY_RADIATION_(option)_VF keyword. The suffix VF enables calculation of radiation view factors between surfaces. Two concepts govern LS-DYNA® thermal radiation calculations.

The first concept is the idea of an enclosure. The enclosure must be completely closed to achieve a perfect energy balance. Think about holding a flashlight while sitting in your office chair. Shine the flashlight beam on the various surfaces surrounding you. The beam of the flashlight represents photons striking these surfaces. Direct the flashlight beam out your open office door. The beam strikes surfaces beyond the door. If the temperature response of the external surfaces are not the objective of the model, we can simply close the door to capture these photons which otherwise would be lost. A modeling technique is to add fictitious surfaces to close holes in the model to prevent photons from escaping the enclosure. Escaping photons carry energy with them and thus lead to an inaccurate energy balance. These fictitious surfaces are assigned radiation properties and a black body temperature that corresponds to the rate at which radiant energy passes through them.

The second concept is that of a view factor. The view factor is a geometric entity and calculated by an algebraic equation for simple geometries (e.g., parallel opposing flat plates) or numerical integration for general geometries. The radiant flux leaving a surface will stream away from the surface into the entire hemispherical space above the surface. Suppose there is a second surface within this hemisphere. The view factor provides information on the fraction of radiant energy leaving one surface that arrives at a second surface within the hemisphere based on their geometric position.

The objective of this thermal model is to calculate the radiation heating of the front hood by the hot engine. The hot engine transfers radiant energy to the hood, radiator, fire wall, right and left fenders, and other engine bay components. Note that we can see the ground. This opening must be closed by a fictitious surface or a lower aerodynamic panel to form an enclosure.

The radiation model can be significantly simplified and thereby decrease the thermal code computation time by only considering radiation heating of the hood from the top surface of the engine. This is accomplished by adding the fictitious surfaces shown in Figure 5 (i.e., the blue funnel like structure sitting on top of the engine). The fictitious surfaces capture the radiant energy that would otherwise strike the fire wall, fenders, etc.

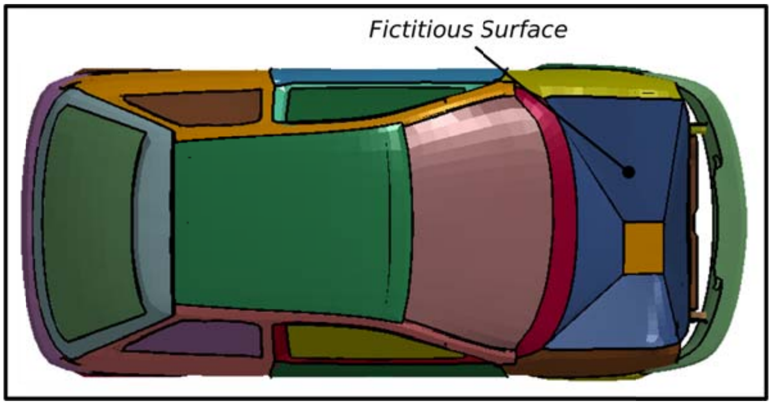


Figure 5 Fictitious surface used in the radiation solver.

Due to the non-linearity of the Stephan-Boltzmann law the thermal solver was run using the non-linear approach. The problem was run using the steady state thermal solver and the result is shown in Figure 6 .

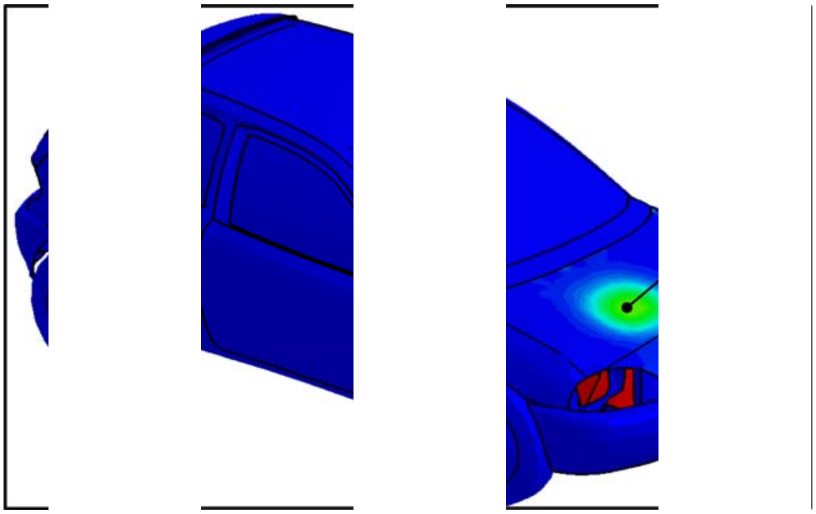


Figure 6 Thermal analysis solution showing the highest temperature on the hood surface.

For this analysis the structural thermal parameters were set as: thermal conductivity was set to $tc=46.0\text{x}10^{-3}$[W/mmK], the heat capacity hc=460[J/KgK] and the density $\rho=7.83\text{x}10^{-6}$ [Kg/mm^3].

Conjugate Heat Analysis

For the Conjugate Heat analysis a CFD model has to be defined. The ICFD solver in LS-DYNA® will be used for the prediction of velocity, pressure and for the coupling of the energy equations between the fluid and the structure. One of the advantages over approaches like the one in [1] is that the thermal coupling is performed in a monolithic way providing a robust and accurate scheme. A single system of equations is built and the temperature at the interface of the fluid/solid boundary is treated using linear constraints. As shown in Figure 4 the mesh does not need to match at this interface. A new iterative linear solver is available in version R9.0 which provides an alternative to the direct solver used up until now. The iterative solver has overall better performance in terms of scalability and speed. This new solver is accessed from the keyword **CONTROL_THERMAL_SOLVER* by setting the card *solver* to 17.

The vehicle is placed inside a numerical wind tunnel used for the CFD analysis as shown in Figure 7. An inflow velocity V=20m/s is set as boundary condition with an imposed temperature T=298K resulting in a Reynolds number of $Re=8.5x10^{-5}$ based on the length of the vehicle. For the fluid the thermal parameters were set as follows: thermal conductivity $tc=0.25x10^{-3}$ [W/mmK], the heat capacity hc=1000[J/Kg K] and the density $\rho=1.28x10^{-9}$ [Kg/mm^3].

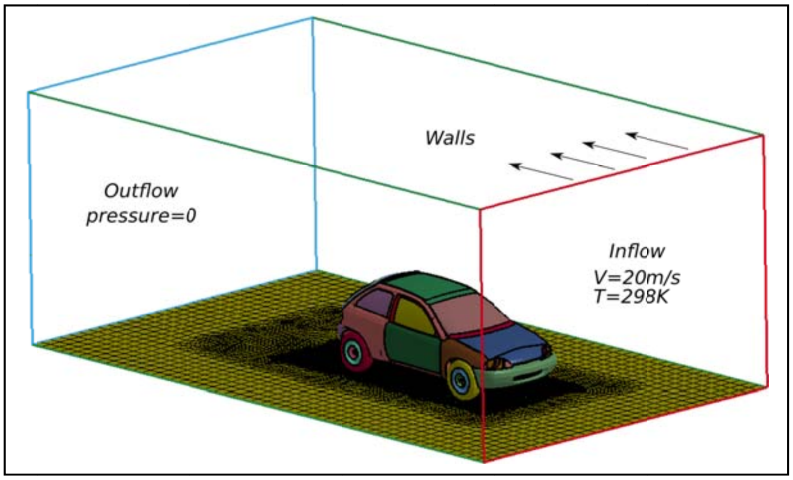
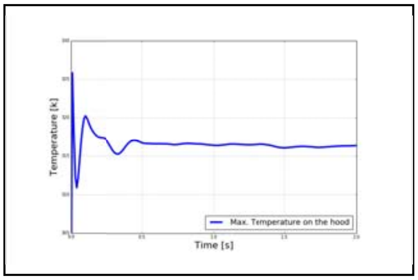


Figure 7 Wind tunnel and boundary conditions used by the CFD solver.

The CFD problem is run to steady state and the temperature is measured on the hood. Figure 8 shows some features of the CFD solution. The maximum temperature on the hood predicted by the conjugate heat solver and the time dependent value of the temperature on the hood is shown in Figure 9.

Figure 8 CFD solution showing iso-surface of velocity and streamlines going inside the engine compartment and under the vehicle.



The results show a temperature difference of δT=42K which is the due to the cooling effect of air over the hood and inside the engine compartment.

With a minimal extra effort the same model could run a full FSI/thermal analysis which may provide even more insight if softening of the structure due to heat is taken into account. The previous analysis was run in this way and the von Mises stress over the hood is shown in Figure 10.

A detailed description of the conjugate heat transfer solver with validation results can be obtained from [3].

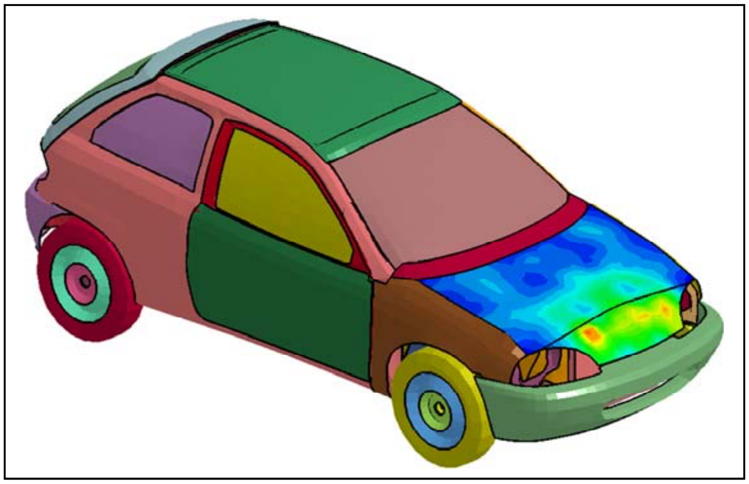


Figure 10 Von Mises stress computed by the FSI solver during the conjugate heat transfer simulation.

Conclusions

This paper provides some guidelines on how to use LS-DYNA® for the simulation of a coupled CFD/thermal and structural analysis. It was not the intent of the paper to validate the results but to show the feasibility of the solution and the capability of LS-DYNA® to handle a single model for three very different physical problems. This kind of Multiphysics approach will become a mainstream working methodology and LS-DYNA® is ready to deal with this complex scenario. Furthermore the simplification that arises from using a single input deck which could be easily shared and incrementally augmented to cover more physics among interdisciplinary groups is a big advantage in a real Multiphysics environment.

References

[1] Rauch, C., Hormann, T., Jagsch, S. and Almbauer, R.. 2008. *Advances in Automated Coupling of CFD and Radiation*. SAE Technical Paper. 10.4271/2008-01-0389

[2] Binner, T., Reister, T., Weidmann, E. and Wiedemann, J.. 2005. *Aspects of Underhood Thermal Analyses*.

[3] Validation of thermal problems: http://www.lstc.com/applications/icfd/test_cases/thermal