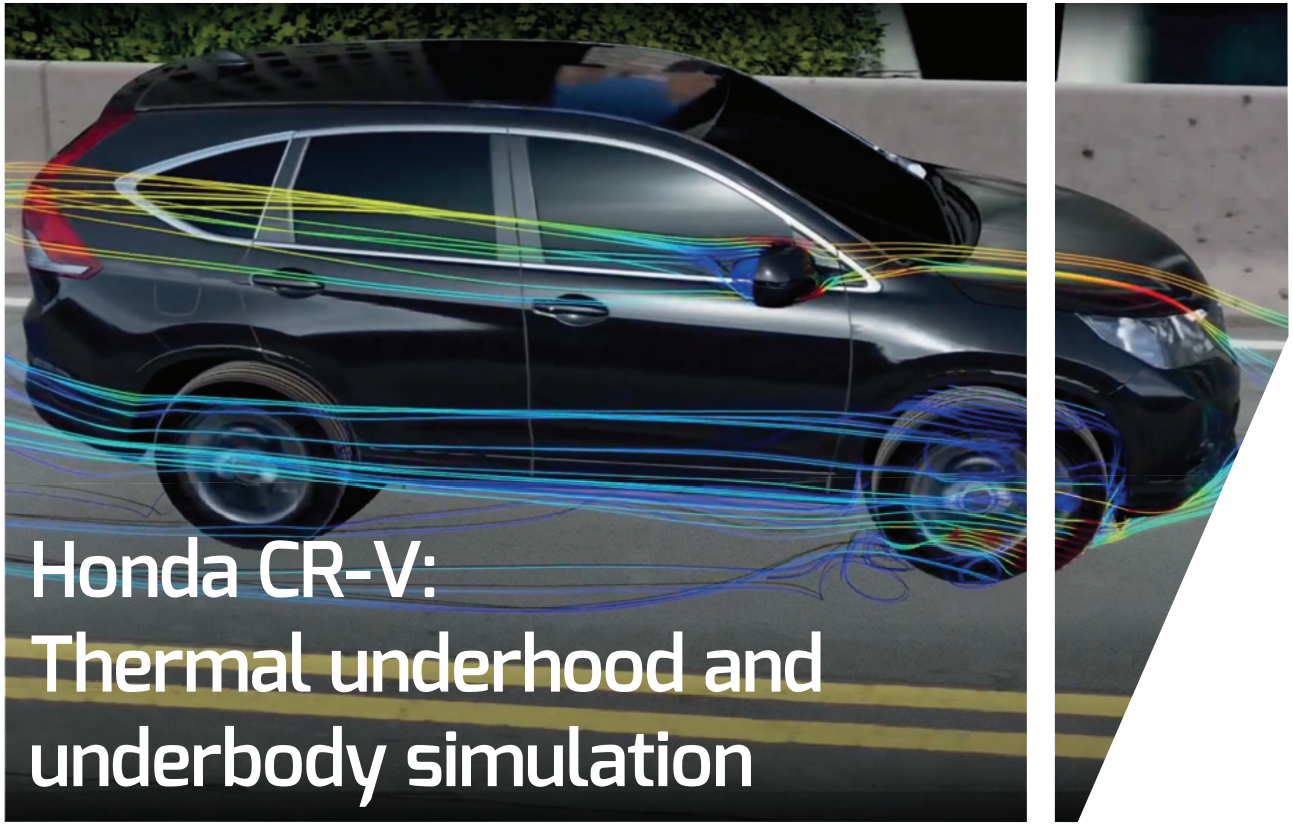


Honda CR-V: Thermal underhood and underbody simulation

By **Lohitasyudu Gorli**, *Aero, Multiphysics & Combustion Products & Applications Engineer* and **Kilian Claramunt**, *Multiphysics Head of Group and* **Yingchen Li**, *Openlabs & Adjoints Responsible, Cadence Design Systems.*

With the growing availability and cost-effectiveness of computational resources, computational fluid dynamics (CFD) simulations should be able to smoothly handle complex physics and their interactions. Moreover the development cycles in the automotive industry are constantly getting shorter, which drives the demand for reliable, automated simulation processes providing accurate results within a short time frame.

Against this background, Cadence presents a comprehensive toolchain for fully coupled simulations, addressing the thermal management of a complete vehicle, drastically reducing overall engineering time by using HEXPRESS™/Hybrid for meshing and FINE™/Open with OpenLAbs for the flow and thermal predictions.

FINE™/Open with OpenLabs allows large scale coupled simulations, taking into account the external flow, rotating components, porous media, conjugate heat transfer, heat exchanger modelling and radiation in one single simulation. The reliability of this fully coupled approach was proven by an industrial case of the Honda CR-V car. Special focus was put on the unstructured conformal multi-block meshing using HEXPRESS™/Hybrid, which represents a crucial starting point.

Project Description

A fully coupled thermal 3D-CFD RANS simulation of a comprehensive and detailed car model for the Honda CR-V SUV model is performed. The simulation mainly focuses on the thermal aspects within the under hood of the car. All relevant sources of heat are taken into account, including the engine, the exhaust system, the radiator and condenser, as well as the fans attached to it.

Meshing

This simulation requires a high quality mesh containing blocks for all parts of the car, which either act as heat sinks or sources or which play a significant role in heat conduction, convection or radiation.

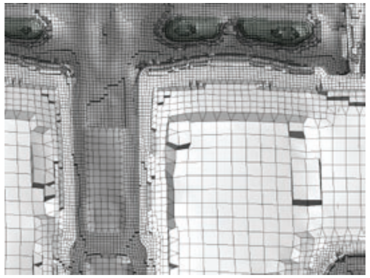
The definition of the adjacent mesh blocks and the corresponding interfaces is done automatically by the unstructured meshing tool HEXPRESS™/Hybrid, which reduces significantly the engineering time required to set up the computation. This also ensures conformal connections between all blocks, thus eliminating inaccuracies typically caused by the interpolation due to non-matching block connections.

HEXPRESS™/Hybrid directly addresses the demands of the automotive industry. The original, imperfect CAD data can be imported right away with reduced need for CAD pre-processing or manual adaptation of the input geometry again saving a significant amount of engineering time.

FIGURE 1 : Combined view of geometry and mesh of Honda CR-V model

The second kind of applications, called "closed thin wall" method, is used for multi-block/multi-material meshing. Typical examples are porous blocks inside a fluid, fluid/solid block meshing for conjugate heat transfer problems and rotating parts.

FIGURE 2 : Cut Section of the Engine Block with air inside the exhaust manifold and external air where the mesh is fully conformal



In total, the multi-block mesh contained 57 different blocks, all of which are connected through nodal-conformal interfaces. For the feasibility study, no viscous layers were inserted in the first attempt. Instead, wall functions were used to model the flow in the boundary layer. This approach led to a mesh size of 420 million Cells, covering fluid and solid domains, created in less than 9 hours on 32 cores.

Conjugate heat transfer

In order to obtain realistic and accurate heat transfer predictions, the different solid parts of the geometry are taken into account in the energy equation with the resolution of the heat conduction equation. The thermal properties of the solid bodies are characterized by their conductivity coefficient. At the solid-fluid interface, the heat flux is implicitly applied based on the gradient of the temperature between the solid and fluid bodies.

Surface-to-Surface Radiation

The exchange of radiative energy between the surfaces of the engine, exhaust pipe and other frames is virtually unaffected by the air flow and a surface-to-surface (S2S) radiation model is therefore chosen to simulate the radiative heat exchange between hot and cold components.

It is usual to assume that all surfaces are gray diffuse emitters (and thus, absorbers), as well as gray diffuse reflectors. With these assumptions, the radiosity-irradiosity method [2] can be applied, requiring the calculation of the view factors. In FINE™/Open with OpenLabs™, the Stochastic Ray Tracing method [3,4,5,6] is used to calculate these view factors.

With the high performance of the ray tracing algorithm, the view factor of the Honda car can be computed in one hour, with more than 200 processors, using 1,000 rays shooting per boundary facet.

The coupling of radiation with the flow can be applied to a selectable level. When the radiation is strong, it may need more cycles of the coupling: solving radiosity equations for radiative heat fluxes with current boundary temperature - applying boundary conditions with the computed radiative heat fluxes.

Heat exchanger model

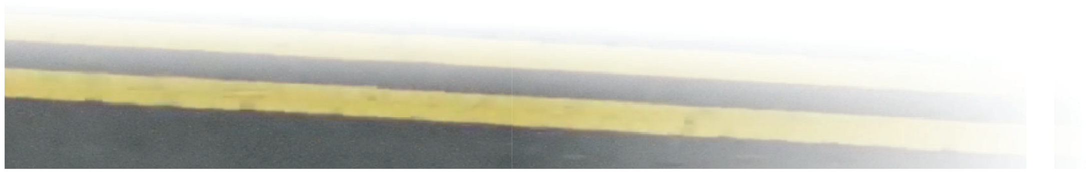
The radiator and condenser are modelled as porous media with isotropic pressure loss. OpenLabs™ is used to customize these two blocks so that the pressure drop across each block matches experimental data.

For the radiator block, in addition to the pressure drop, a coupled strategy between the steady-state CFD calculation and a thermal 1D calculation is defined. The coolant temperature varies throughout its flow path, with a non-uniform heat rejection from the radiator over the block. The heat exchanger subsystem is formed by the CFD mesh for the primary fluid, the air, and an overlapped 2D coarser mesh along the direction of the auxiliary fluid flow defined by the coolant. This approach of modelling the heat exchanger core by splitting it into macroscopic cells provides more realistic solutions for the heat rejection compared to a uniform heat source term.

Rotating Machinery

FIGURE 3 : Counter-rotating fans in front of radiator and condenser

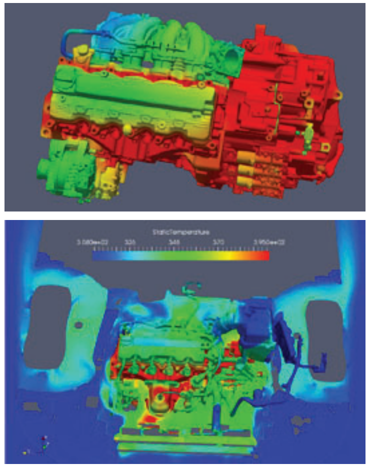
For the counter-rotating fans, two models are available in FINE™/Open with OpenLabs. The added momentum and energy could be introduced using an actuator disk model which doesn't contain the geometry, but only models the effect on the flow. The definition of the source terms would be possible through the programmable interface OpenLabs™. The other method is to build cylindrical blocks containing the fans and connect them to the surrounding domain using a rotor/stator interface. This second approach was chosen in combination with a frozen rotor interface. Although this represents only a snapshot in time, its advantage is the high robustness and low computational cost compared to a pitchwise averaging. The blocks containing the fans are conformally connected to the outer air domain.



Results

The fully coupled CHT simulation provides much more realistic results in both aerodynamic performance and thermal management prediction. Figure 4 gives the external aerodynamic view of the car, showing the pressure distribution and the streamlines around the car. Pressure distribution in front the wheel shows the complexity of the flow under the car, which has a strong effect on the thermal prediction of the underbody. Therefore a fully coupled CHT simulation is adopted here.

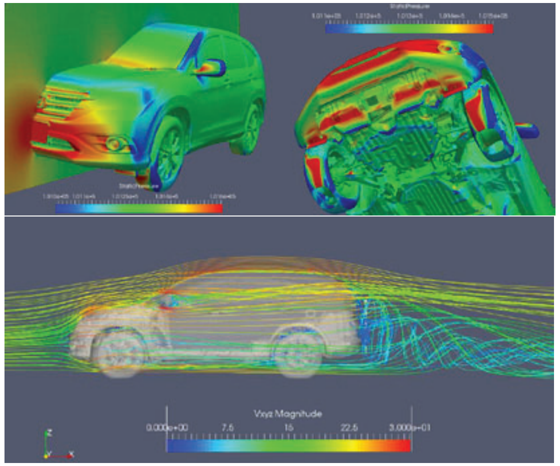
FIGURE 5 : Temperature of engine surface and the horizontal cutting plane around engine



As can be seen in Figures 5, a temperature distribution is visible on both the engine and muffler surfaces, showing that neither the temperature nor the heat flux can be assumed as constant along the surface. The thermal interactions in both the engine compartment and the exhaust system can therefore only be modelled accurately by the use of a coupled CFD simulation which captures the conjugate heat transfer effects.

Source: Posselt F., Gorli L., Claramunt K., Leonard B., Li Y. (2017) Thermal behavior of a SUV car with a fully coupled 3D-CFD CHT simulation. In: Bargende M., Reuss HC., Wiedemann J. (eds) 17. Internationales Stuttgarter Symposium. Proceedings. Springer Vieweg, Wiesbaden. https://doi.org/10.1007/978-3-658-16988-6_38

FIGURE 4 : Static pressure distribution in the front part of the car and external aerodynamics view



Figures 6 and 7 show the temperature distribution on the exhaust pipe and the underbody. The large temperature differences cause a strong radiation heat transfer. Thus, to achieve accurate thermal predictions, the direct coupling of CHT with radiation model has to be adopted.

FIGURE 6 : Temperature of the exhaust pipe

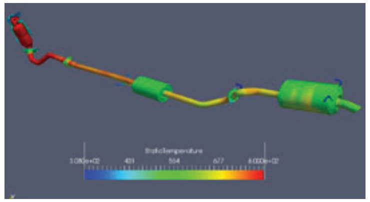


FIGURE 7 : Static Temperature on the exhaust system and flow structure at the underbody