

Experimental and Numerical Investigations of Thermal Soak

Ernst Peter Weidmann, Thomas Binner and Heinrich Reister

Daimler AG

Copyright © 2008 SAE International

ABSTRACT

This paper summarizes a common project of Mercedes-Benz and FKFS (Research Institute of Automotive Engineering) to apply numerical methods to thermal soak issues in a very early stage of the development phase of a new car. "Thermal soak" results from driving the vehicle at high load followed by shutting off the engine and a cool down phase. After stopping, the underhood flow is only driven by natural convection.

The thermal soak behaviour is discussed in principal and the numerical challenges are summarized. Four different issues are identified: the need for a transient computation including transient thermal load pattern, a method to compute natural convection in the underhood after the shutdown of the engine, the complex geometry and the lack of a single computational program to consider all three modes of heat transfer, which results in a coupled numerical approach. Based on these premises a numerical method is developed and validated for a simplified test case as well as for a full vehicle. Therefore experimental results are derived by measurements in the IVK / FKFS hot climatic wind tunnel.

INTRODUCTION

Today numerical simulation is an important part of the development of a new car. Mercedes-Benz has therefore divided the development process into a first phase with digital prototypes and a second phase with hardware prototypes. Following Breitling et al. [1] the maturity of the first hardware prototypes is significantly increased.

Tightly packaged engine compartments and strongly increased engine power demand extensive testing and special treatment to assure that no component gets too hot. Numerical methods are state-of-the-art in the engine cooling design and thermal protection, but limited to steady-state load cases for entire vehicle computations. In the last years several authors [2,3,4,5,6] have discussed the need and methods to assure thermal protection.

Thermal Protection is a classical fluid-structure interaction problem: Underhood component temperatures are sensitive to all three modes of heat transfer, conduction, convection and radiation. Due to very high temperatures next to the exhaust system, in particular, a thermal radiation analysis is required. In 2003 Reister and Maihöfer [7] proposed to split up the computation of component temperatures in an underhood by coupling sophisticated but specialized simulation codes. Today this approach is state-of-the-art in the development departments of many OEMs and suppliers, but limited to steady-state loadcases, like a slow trailer towing uphill drive or maximum velocity.

The prediction of component temperatures during soak requires a transient computation. Schuster [8] computed soak temperatures in a sub-model of an underhood using the conjugate heat option of a CFD Code and was able to rate the performance of different heat shields. Franchetta [9] described a computational procedure and validated numerical with experimental results from a simplified test case.

In the following paragraphs the thermal soak behaviour is reviewed and the main challenges, which make numerical computation difficult, are discussed.

THERMAL SOAK

"Thermal soak" results from driving the vehicle at high load followed by a shut-off of the engine and a cool down phase. Figure 1 shows the thermal behaviour of a temperature relevant component in the underhood for a 35 km/h uphill drive respectively for a maximum velocity drive and a following soak phase. In general, three phases in temperature behaviour are distinguished: the heat-up (phase 0, not shown in figure 1), the steady-state (phase 1) and the soak phase (phase 2). The temperature curves in figure 1 are non-dimensionalized referring all temperatures to the initial steady-state temperature of the uphill case. In conclusion the steady-state and the peak temperatures during soak are higher for the uphill drive for this component. The higher initial temperatures for the soak phase (end of phase 1) result in higher temperatures in the soak phase itself.

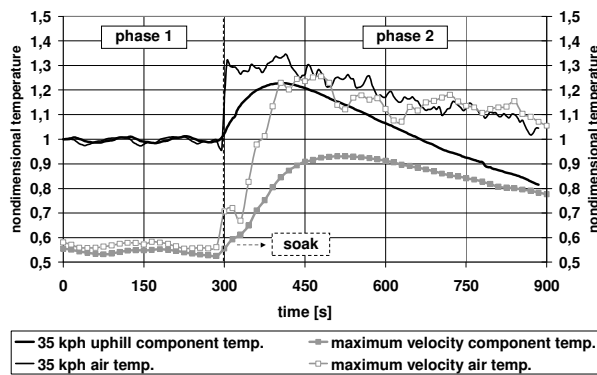


Figure 1 Thermal soak behavior for an underhood component after a 35 kph uphill drive and maximum velocity

After stopping, the underhood flow is only driven by natural convection. Temporary high temperatures result for components in proximity to hot exhaust pipes and also in regions where hot air plumes rise. The soak phase is characterized by the buoyancy driven flow, which yield smaller convective cooling or, even worse, convective heating in regions with high air temperatures. This phenomenon interferes with the cool down of the exhaust gas system. Measurements show a cool-down of an exhaust pipe of 300 K in a 20 minute soak phase [10].

Therefore, with regard to the physics, there are in principal two challenges to overcome in order to compute thermal soak: The transient behaviour and the strong coupling of the temperature and the velocity field in buoyancy driven flows. Computational issues add two further challenges: the complex geometry in an underhood, which results in millions of finite elements or volumes to be resolved, and the need for a coupled simulation caused by the lack of an efficient code, which is able to compute a multi-mode-heat-transfer analysis in an underhood of a full vehicle.

NUMERICAL METHOD

A coupled approach is used at Mercedes-Benz to compute component temperatures. The flowfield and air temperature are investigated by means of CFD with the commercial CFD Code StarCD [11] while for the thermal analysis a finite element code for full vehicles is used. We will use the term "thermal analysis" in this paper only for the component temperature analysis and not for the whole process.

COMPUTATIONAL FLUID DYNAMICS (CFD)

Although most CFD codes have the capability to solve the transient coupled thermal-fluid problem associated with underhood air flow, the computational requirements for solving such a coupled problem with millions of elements over time frames covering hundreds of seconds are prohibitive, even with today's massively parallel high performance computers. Therefore only

steady-state load cases are computed and the buoyancy driven flow is treated as a quasi steady-state flow.

Figure 2 shows a typical grid for a CFD analysis in an underhood. Heat exchangers are modeled using a porous media approach. The heat rejection is computed in a preceding 1D analysis [4]. The fan is treated with a MRF fan model.

For the computation of the buoyancy driven flow, the model setup is modified in order to use more realistic boundary conditions. The main difference is that no longer adiabatic wall boundary conditions are used. Most walls were set to fixed temperatures provided by the thermal analysis. A more detailed description of the CFD related issues in case of natural convection can be found in [4].

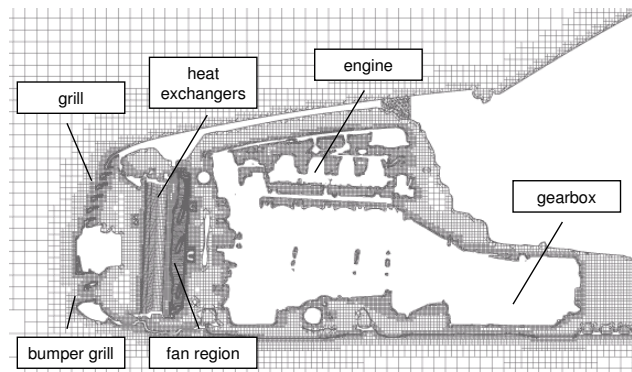


Figure 2 Numerical grid for an underhood flow analysis

THERMAL ANALYSIS

The transient thermal analysis is carried out using either RadTherm by Thermoanalytics Inc. [12] or the finite-element code PERMAS by Intes [13]. The first one performs an energy balance based on computing conduction, convection and radiation. Grey, multi-reflection radiation is computed based on viewfactors. The code just requires a surface mesh and can be run in transient mode utilizing a Crank-Nicolson implicit scheme for time discretization.

For a full vehicle analysis PERMAS is iteratively coupled to a radiation solver (Posrad). The finite-element code PERMAS solves the energy equation. The radiative heat fluxes are computed by Posrad [11]. Within the transient analysis a Newton-Raphson algorithm for the time-discretization is used. More details of this methodology can be found in [4]. The finite-element model covers the full vehicle and consists of approximately 16 million elements, while the POSRAD model contains only the surface with 7 million elements.

COUPLED APPROACH

Fluid-structure interaction is often computed using a partitioned approach [13,14]. For this purpose the coupling software MpCCI (Mesh based Code Coupling

Interface) [13] has been developed. MpCCI is a commercial software environment which enables the exchange of data between the meshes of two or more simulation codes at the coupling interface. Since the meshes belonging to different simulation codes are not compatible in general, this software package performs an interpolation. An additional interpolation tool [6] has been developed at DaimlerChrysler to assure physically reasonable values at each element, in case the coupling regions are geometrically different. Figure 3 shows the different discretized meshes for PERMAS and StarCD.

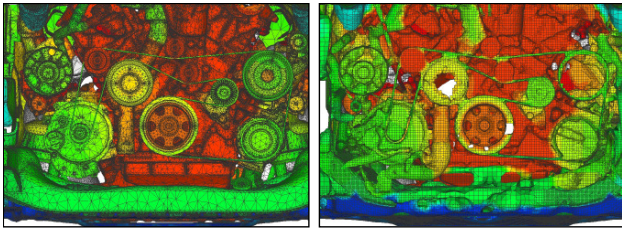


Figure 3 Mesh mapping on different discretized meshes for a finite element code (left) and a CFD code (right)

Different coupling schemes are used. The easiest way is a one-way coupling: Data is exchanged only once, which means that there is no feedback to the code which is sending first. This is suitable for load cases dominated by forced convection but it is not sufficient in case of natural convection. Therefore a “closed loop” or “two-way coupling” approach is necessary. MpCCI enables a full iterative coupling of codes. This feature is used for the fundamental testcase (testcase 1). At the moment this iterative process is not applicable to the complex geometry of a full vehicle, because of several process-related reasons: Many parts include “inner” geometry which is not part of the CFD mesh. Furthermore the geometrical slightly differing meshes force the user to check the mapped quantities, in particular in regions with high gradients, and to correct local errors. Finally it has shown that handling large cases on different computers and up to 32 parallel CFD jobs is challenging. Therefore, for this case data is exchanged manually only after fully converged runs.

NUMERICAL PROCEDURE FOR THERMAL SOAK

For accurate transient soak computations it would be necessary to couple the transient thermal analysis with a transient CFD run, which seems impracticable in the near future. Consequently, the current approach is to compute the air side “quasi-stationary” and to couple with the transient thermal analysis at pre-defined timesteps. In the long run this yields a fully automated, iteratively coupled solution for each time step as described in [14]. Nevertheless, the described methodology itself will be identical for a coupled transient thermal and transient flow computation.

Figure 4 illustrates the current numerical procedure. The initial condition is computed straightforwardly within a coupled CFD and thermal analysis with appropriate

boundary conditions (in this example 40 kph uphill). This temperature map is used to provide suitable boundary conditions to compute a quasi-steady CFD buoyancy driven flow solution. This is always done in a full vehicle model. The transient computation is carried out either for a sub-model or within a full vehicle analysis using the quasi-stationary natural convection CFD results as fixed boundary conditions for pre-defined time frames. Between these time frames the convective heat coefficients and the film temperatures can be constant or interpolated linearly.

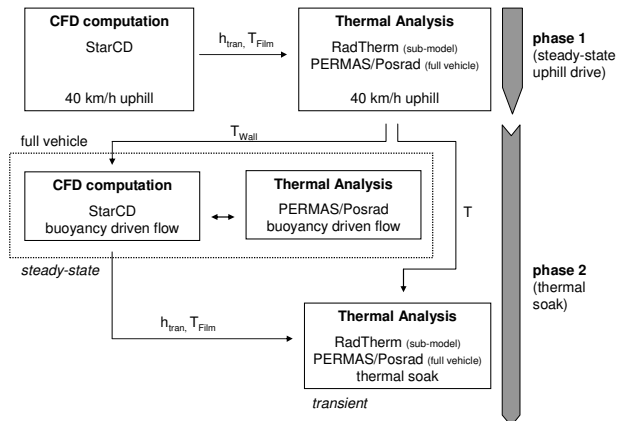


Figure 4 Numerical procedure for thermal soak

RESULTS

In this chapter the thermal soak calculation is validated for three different examples. First a fundamental test case is studied. Afterwards a single component, a rubber disc at the cardan shaft, is investigated within a sub-model of the tunnel region of a Mercedes-Benz sportscar. Results for a full vehicle (Mercedes-Benz S-class) conclude this chapter. All computations are validated against measurements in a special test rig or full vehicle measurements in the IVK / FKFS hot climatic wind tunnel, respectively.

TESTCASE 1: HORIZONTAL FLAT PLATE

A fundamental experiment with a flat horizontal plate in a special test rig is studied to have a closer look into the principal soak behaviour and to validate the numerical method.

Experimental Setup

The test rig is described in detail in [16]. Well defined boundary conditions are provided by locating 5 side walls of a copper box within a water bath. This is a well known practice in fundamental heat transfer experiments to provide a fixed temperature boundary condition. Air is blown into the test rig using a fan. The air mass flow is measured by an orifice. A special heater is located within the ground wall of the box. It consists of a solid aluminium block with three heat cartridges.