Abstract

Thermal Protection is of vital importance in the development of passenger cars. In the last years thermal issues have been increasingly investigated by numerical means. Computational fluid dynamics (CFD) is today an efficient way to optimize the aerodynamics of a new car or to investigate the underhood flow, especially the mass flux through the heat exchanger package. Additionally the computation of underhood flow field and air temperature distribution is today common practice. This allows, for example, an engineer to judge the performance of the engine fan and vents as well as look for hot air spots within the underhood. The computations of two standard steady state load cases – 240 km/h and 40 km/h hill climb – are shown and some results are briefly discussed in this paper.

The computation of component temperatures in the engine compartment requires a multi mode heat transfer analysis considering conduction, convective heat transfer and thermal radiation. Due to very high temperatures next to the exhaust manifold, in particular, a thermal radiation analysis is required. This paper describes an underhood thermal methodology based on coupling sophisticated and specialized tools. The advantages of this methodology are discussed.

This methodology is used to compute a new steady-state load case, briefly described as idle without operating fan. This load case is defined as a first step in order to compute thermal soak. Experimental and numerical results are compared in order to validate the commercial CFD code StarCD™ for buoyancy driven flows and the thermal analysis with natural convection.

10.1 Introduction

Thermal protection is increasingly important in the development process of passenger cars. Tightly packaged engine compartments and strongly increased engine power demand extensive testing and analysis. Traditionally thermal protection is tested in climate wind tunnels and road tests. Reduced time to market and high costs for prototypes result in the need for building the first prototype very close to its optimum design. The introduction of numerical methods allows the optimization of cooling requirements as well as thermal analysis of temperature sensitive components in a very early stage of the vehicle development process. Counter-measures such as heat shields can be designed several months before prototype
vehicles are built. Although road tests will also be necessary in the future to verify the numerical predictions, the parallel use of numerical methods helps to focus tests on the most critical points.

Vehicle thermal management is traditionally based on two or three steady-state load cases: maximum velocity, trailer towing uphill at 40 km/h and idle. Vehicles featuring a highly powered motorization are often restricted in performance when going faster than 240 km/h. Hence hill-climbing-drives at 40 km/h are often more critical, especially because air velocities within the engine compartment are lower despite the fact that the fan is operating. Chapter 10.4 compares numerical results for a Mercedes-Benz sports car travelling uphill at 40 km/h and at a constant 240 km/h.

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 manifold, in particular, a thermal radiation analysis is required. Several examples of multi-mode heat transfer computation can be found in recently published literature. Schuster [1] calculated temperatures in the exhaust tunnel of a car. He used the conjugate heat transfer option in the commercial CFD code StarCD™. Reister and Maihöfer [2] described in 2003 a procedure to import heat transfer data from a CFD calculation into a thermal analysis code. Different authors [3, 4, 5] published investigations of coupled CFD and thermal modelling methodologies to predict underbody temperatures of a vehicle. All papers deal with a car driving either at maximum speed or travelling uphill at 40 km/h.

10.2 Thermal Soak

In addition to steady-state load cases “thermal soak” is one of the most critical underhood operating conditions. “Thermal soak” or “hot-stop” results from driving the vehicle at high load followed by shut off the engine and a cool down phase (figure 10.1). After stopping, the underhood flow is only driven by natural convection and temporarily high temperatures result for components in proximity to hot exhaust pipes. Part 2 of this paper deals specifically with this flow condition.

There are in principal two challenges to overcome in order to run a multi-mode-heat-transfer analysis in case of thermal soak: The transient behaviour and the strong coupling of the temperature and the velocity field. Although the cool down phase is in general transient, a new steady state load case is introduced as a first step to validate the numerical methods for natural convection. 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 is prohibitive, even with today’s massively parallel high performance computers. Time dependent numerical analyses are therefore not subject of this paper.
Franchetto et al [6] and present authors [7] investigated multi mode heat transfer analyses in case of buoyancy driven flows in simplified engine compartments. While Franchetto compared the simulated flow to PIV results and found a good agreement, present authors described a numerical method based on the work of Reister and Maihöfer [2] and validated it with measured component temperatures which are affected not only by convection but also by radiation and conductive heat transfer. Experimental and numerical results were in good agreement. However both studies were based on relatively simple geometries with well known boundary conditions. The present study is aimed to validate StarCD™’s [8] ability to deal with this kind of flow in the complex geometry of an underhood region.

10.3 Theoretical Approach

In general three different modes of heat transfer - conduction, convective heat transfer and radiation - have to be taken into account.

Conduction is defined by

\[ \dot{q}_{\text{conduction}} = -\lambda \cdot \text{grad}(T). \]  

(1)

Convective heat transfer is known with the convective heat transfer coefficient \( h \) as

\[ \dot{q}_{\text{convection}} = h \cdot dT. \]  

(2)
Cooling due to forced convective heat transfer is the dominating phenomena in most areas of a vehicle. The flow is either driven by high vehicle speeds or an operating fan. Natural convection occurs in buoyancy-induced flows. The coupling between flow and wall temperature is significant for this kind of heat transfer. It is therefore important to mention that $h_{\text{nat.conv.}}$ is no longer independent of the temperature difference $dT$ and $\dot{q}_{\text{nat.conv.}}$ no longer rises in proportion to $dT$.

The dimensionless quantity, which describes natural convection, is the Grashof Number, relating buoyancy forces to the viscous forces.

$$Gr = \frac{g \cdot \beta \cdot dT \cdot L^3}{\nu^2}$$

(3)

with the coefficient of thermal expansion

$$\beta = \frac{1}{T_{\infty}}$$

(4)

The ratio of the buoyancy forces to the inertial forces can be written as the ratio of the Grashof and the Reynolds Number. If this ratio exceeds one, natural convection dominates [9].

$$\frac{Gr}{Re^2} \gg 1$$

(5)

The choice of normalization quantities is obviously difficult in complex geometries like an engine compartment. A rough estimate of length scales and temperatures yield a ratio $Gr/Re^2 = 0.01$ at maximum speed. The measurements with buoyancy driven flow described in chapter 10.5.1 are conducted for a ratio of $Gr/Re^2 = 10$.

Finally Stephan-Boltzmann’s law

$$\varepsilon = \varepsilon \cdot \sigma_S \cdot T^4$$

(6)

with

$$\sigma_S = 5.77 \cdot 10^{-8} \frac{W}{m^2K^4}$$

(7)

describes the emission $\varepsilon$ of a body with constant temperature $T$ and emissivity $\varepsilon$.

10.4 Standard Steady State Load Cases

Steady state load cases are often used to test the engine cooling and review temperature sensitive components. At Mercedes-Benz the relevant load cases for the sportscar SL 600 are uphill travel at constant speed of 40 km/h and driving 240 km/h.
which is approximately maximum velocity (in fact the car is electronically limited to 250 km/h).

The finite volume based code StarCD™, solving the three-dimensional Navier-Stokes equations, is used for the numerical simulation of the underhood flow. The flow is assumed to be turbulent and the k-ε model for turbulence is used with the algebraic "law of the wall" representation of flow and heat and mass transfer. For the temperature the energy equation is solved. Buoyancy effects are taken into account as source terms in the Navier-Stokes equations.

The external, as well as the underhood flow is simulated with StarCD™. The computational mesh can be seen in figure 10.2. Grill or wheel vent regions are refined in order to solve the flow accurately. The grid consists of 10.9 million fluid cells. Six heat exchangers (radiator, condenser, charge air cooler, engine oil cooler, power steering servo cooler and ABC oil cooler) are investigated using a porous media approach (figure 10.3). The fan is modeled using a multiple implicit rotational reference frame method (MRF) [10]. The car is simulated in a sufficiently large exterior domain. The flow at the inlet is fixed to either 40 km/h or 240 km/h. Ground and wheels are moving at corresponding speeds. Engine and exhaust system temperatures are boundary conditions with fixed experimental or estimated values. Additionally the heat emitted at the heat exchangers is calculated by a program system called KUEBER [11] and input data to the numerical analysis.

Figure 10.2: Computational domain with local grid refinement near the car

Figure 10.3: Underhood model with MRF grid, grill and heat exchanger package

Figure 10.4 compares the numerical results for air temperatures within the engine compartments for both load cases. The overall temperature distribution within the engine bay can be seen. In the case of maximum velocity the effective air temperatures within the engine compartment are approximately 10 K higher. This is in particular a product of the heat exchanged at the radiator: Because this sports car
is not offered with trailer coupling, the load and heat exchanged at the radiator is relatively low in the case of 40 km/h. Both cases are computed without working air conditioning.

![Figure 10.4: Underhood temperature distribution for 40 km/h hill climb and 240 km/h](image)

Nevertheless there are striking differences concerning the cooling air flow through the engine compartment. As an example the performance of the engine mount vents is investigated. In the 240 km/h case they perform well, but not at 40 km/h (figure 10.5). Geometrical variants can be designed and simulated in order to optimize these vents in case of hill climb. In both cases the heat shield above the engine mount effectively separates the high temperature air from the components, e.g. the engine mounts.

![Figure 10.5: Air temperature distribution around the engine mount for 40 km/h hill climb (left) and 240 km/h (right)](image)

Although flow structure and temperature distribution provide extensive data to optimize the thermal packaging in the engine bay, a full thermal analysis including radiation is required to simulate component temperatures. For this analysis the convective heat flux is an important result from the CFD simulations. Heat transfer coefficients and near wall air temperatures are mapped to the thermal analysis code. This will be shown as an example for the load case “Idle Without Fan” (IWF), which is briefly described in the next paragraphs.
10.5 Idle Without Fan Configuration

To validate the numerical analysis in case of natural convection an additional steady state load case is investigated: A car idling without operating fan. This configuration corresponds to the flow field of thermal soak after shutting off the engine but with constant thermal load. First measurements (figure 10.6) with the car idling with zero fan speed were investigated and yield the conclusion that it is not possible to reach thermal equilibrium at a temperature level below maximum allowable temperatures for coolant, engine oil and components. The heat released by the engine cannot be transported out of the engine compartment. Therefore the idea was born to use a separate external radiator to cool down the coolant. The next section gives a brief overview of the experimental setup.

![Figure 10.6: Temperature rise of coolant and underhood temperature in case of idle without fan](image)

10.5.1 Experimental Setup

A radiator and a small fan are provided as an additional heat exchanger operating outside the car. The radiator is built up in a chamber next to the test rig with the car. The coolant flow is diverged from the main stream by flexible tubes and a variable pump. T-junctions are used to set up two parallel fluid streams over the car’s radiator and the external one. A 1D study revealed that only a small part of the coolant actually passed through the car’s radiator for this experimental setup. Additionally the flow through this radiator is inverted. Nevertheless this setup is able to cool the engine and the whole engine compartment. Figure 10.7 shows schematically the experimental setup.
With this setup the sportscar idles approximately one hour until all recorded temperatures are steady. Recorded values are for example fluid pressures and temperatures (e.g. coolant, engine oil, ...), component temperatures, especially at the exhaust system and air temperatures within the engine compartment. Standard thermocouples type “K” are used for the temperature measurements. Four thermocouples in a vertical row are placed in three locations in the engine compartment in order to measure the temperature distribution in z-direction. The positions are marked in figure 10.7 with dotted lines.

The coolant temperature is regulated by the mass flux over the external radiator. Three different configurations are measured:

- 115°C coolant temperature,
- 105°C coolant temperature,
- 105°C coolant temperature, closed grill.

Variant 3 resulted from the idea that the warm air has to leave the engine bay through grill and wheel vents.

### 10.5.2 Numerical Analysis

Although surface temperatures were also measured, this study is aimed to validate the computed air temperatures. Therefore the first approach was to use a similar procedure as for the forced convection load cases described in chapter 10.4. The measured surface temperatures at the exhaust system and the engine are set as boundary conditions. The segments of the exhaust system and the applied temperatures for all three configurations are given in table 10.1.

As in the standard load cases adiabatic wall boundary conditions were applied to all other surfaces within the engine compartment. With this initial approach the computed temperature distribution was much too high. The adiabatic boundary condition and the relatively "weak" flow were not sufficient to release the heat from the engine bay. Reference [7] discusses in detail the numerical methodology used.
within a simplified underhood to compute such buoyancy driven flows. This methodology is reviewed in the following section and adapted for this special issue.

Table 10.1: Temperature boundary conditions for the simulation

<table>
<thead>
<tr>
<th>fixed temperature boundaries in [°C]</th>
<th>coolant 115°C grill open</th>
<th>coolant 105°C grill open</th>
<th>coolant 105°C grill closed</th>
</tr>
</thead>
<tbody>
<tr>
<td>cylinderblock</td>
<td>116</td>
<td>102</td>
<td>103</td>
</tr>
<tr>
<td>cylinderhead</td>
<td>116</td>
<td>102</td>
<td>103</td>
</tr>
<tr>
<td>crankcase</td>
<td>116</td>
<td>104</td>
<td>105</td>
</tr>
<tr>
<td>oil pan</td>
<td>106</td>
<td>95</td>
<td>96</td>
</tr>
<tr>
<td>gearbox</td>
<td>113</td>
<td>101</td>
<td>103</td>
</tr>
<tr>
<td>manifold</td>
<td>251</td>
<td>244</td>
<td>244</td>
</tr>
<tr>
<td>turbo turbine</td>
<td>243</td>
<td>240</td>
<td>238</td>
</tr>
<tr>
<td>turbo compressor</td>
<td>120</td>
<td>120</td>
<td>120</td>
</tr>
<tr>
<td>turbo bearing</td>
<td>116</td>
<td>102</td>
<td>102</td>
</tr>
<tr>
<td>exhaust before cat</td>
<td>173</td>
<td>163</td>
<td>171</td>
</tr>
<tr>
<td>catalyst</td>
<td>235</td>
<td>235</td>
<td>237</td>
</tr>
<tr>
<td>exhaust after cat</td>
<td>216</td>
<td>217</td>
<td>218</td>
</tr>
<tr>
<td>mufflers</td>
<td>100</td>
<td>100</td>
<td>100</td>
</tr>
</tbody>
</table>

A coupled CFD and thermal analysis methodology is used in this study. Fluid-structure interaction is involved in many problems. In particular the strong coupling of the velocity and temperature field in buoyancy driven flows requires a two-way coupled multi-mode heat transfer analysis. In this paper the computational fluid dynamics code StarCD™ and the finite element code PERMAS [12] are coupled sequentially after fully converged runs (figure 10.8).

Figure 10.8: Closed loop multi mode heat transfer analysis
For the thermal analysis the finite-element code PERMAS is used. Equation (8) describes the energy equation solved in PERMAS.

\[
\rho \cdot c_p \cdot \frac{\partial T}{\partial t} - \lambda \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right) = \dot{q}_i
\]

\[
C \cdot \dot{T} + K \cdot T = F
\]

(8)

with \( \dot{T} = 0 \) in steady-state calculations, \( C \) the capacity matrix and \( K \) the conductivity matrix. \( \dot{q}_i \) and \( F \) respectively describe the thermal load vector, consisting of radiative and convective heat fluxes.

Figure 10.9: Heat transfer at the interface boundary of fluid and solid

The thermal equilibrium at the interface boundary (figure 10.9) of fluid and solid parts yields to

\[
n^t \cdot \lambda \cdot \text{grad}(T) - \dot{q}_S + \alpha \cdot (T - T_{F Film}) = 0
\]

(9)

The net radiation heat flux \( \dot{q}_S \) is calculated in a third program named POSRAD [8]. POSRAD and Permas are iteratively coupled. Radiative heat transfer is considered as a thermal load and modifies in each coupling step the right side of equation 8. PERMAS volume mesh and the surface grid needed in POSRAD consist of approximately 10 million elements.

The convective heat transfer data such as \( h_{\text{tran}} \) and the near wall temperature \( T_{F Film} \) is provided by StarCD™ and is input data to equation 9. Although other turbulence model options are supposed to be more accurate [13], the \( \kappa - \varepsilon \) approach with wall function is chosen, because the authors believe that the complexity of the engine compartment will not, at least in the next few years, allow for using grids to resolve the boundary layer region with acceptable computational time and effort. More details about near wall treatment in StarCD™ are provided in [8].

The CFD mesh and the PERMAS grid are based upon the same CAD data but are generated with different meshing tools with different accuracy for the representation
of the surface. StarCD™ works most accurately with a trimmed mesh consisting of hexahedral volume cells in almost all parts of the computational domain. For PERMAS a tetrahedral grid, which is in most cases automatically created, is sufficient. The common interface of both codes is the surface of both meshes. Therefore a mesh mapping tool is needed to map the physical properties from one mesh to the other (figure 10.10). The commercial tool MpCCI™ [14] is used for the mapping. An interpolation tool [2] has been developed at DaimlerChrysler to assure physically reasonable values at each element, although the coupling regions are geometrical not identical.

![Figure 10.10: Mesh mapping on different meshes for StarCD and PERMAS](image)

After a first CFD run, providing the convective data for a first PERMAS / POSRAD run, the wall temperatures computed in PERMAS are mapped to StarCD™ as fixed temperature boundary conditions. Both – mapping and computations in PERMAS and StarCD - are very time consuming for a whole vehicle. Therefore only one further iteration is carried out.

10.5.3 Results

In this section some results of the additional load case “idle without fan” are presented. First the 115°C coolant temperature configuration is investigated. Additionally the modification with closed grill and a coolant temperature of 105°C is computed and compared to the open one.

As an example the temperature distribution in z-direction behind the engine, above the gearbox is considered (Figure 10.11 left side). Additionally a second position on the left side of the underhood is compared to the numerical simulation. The exact positions are marked in figure 10.7. The methodology including a closed loop between CFD and thermal analysis code shows a very good agreement between experimental and numerical results. The difference for the third thermocouple is most likely a result of direct radiation on the thermocouple in the measurement.
The difference between open and closed grill variant is compared for numerical and experimental results. Again the air temperature distribution in the z-direction above the gearbox is plotted in the left side of figure 10.12. It can be seen that the difference caused by the geometrical modification can be computed by the multi mode heat transfer analysis. The right side of figure 10.12 shows the temperature difference between closed and open grill. In the top passage the temperature stays nearly the same. In the case of the closed grill the hot air cannot leave the engine compartment and the temperature in the bottom passage rises. This can be computed quite well. However because of the very large gradients in these regions in the case of open grill it is very difficult to compare exactly the experimental and numerical results.

The velocity field for an idle without fan is in striking difference to the velocity fields computed in the standard load cases: Hot air rises above the gearbox and the engine...
and leaves the engine compartment through the upper passages of the grill. The temperature field shows the expected gradients in the z-direction.

10.6 Conclusions

Nowadays the numerical analysis of the underhood flow is a vital part of the development process of passenger cars. Additionally today thermal protection is increasingly a subject of numerical investigation. This requires a multi mode heat transfer analysis considering conduction, convective heat transfer and radiation. This can be done by coupling three sophisticated and specialized simulation codes.

In particular two steady state load cases are under investigation: a trailer tow uphill at 30 km/h and maximum speed. Additionally thermal soak is an important issue. Although thermal soak is transient by nature, a new steady state load case is introduced to validate the CFD simulation in the case of natural convection in an engine compartment. This is considered as a first step in simulating thermal soak in the near future.

Because of the strong coupling of temperature and flow field a coupled multi mode heat transfer analysis is carried out to compute both air temperature and component temperatures. The air temperature distribution which is compared to experimental results shows promising results including differences between geometrical variants as well as absolute numbers.

Nevertheless not all measurement points agree well with experimental results. Especially in regions with high temperature gradients deviations are very likely to occur. Fortunately, temperatures of components normally do not show large temperature gradients. So it will be much easier to compare component measurements and numerical results.

Acknowledgement

This research was funded by DaimlerChrysler AG, department EP/SAE. The authors would like to thank Dr.-Ing. Raimund Siegert and Dipl.-Math. Walter Bauer for their support.
Nomenclature and Abbreviations

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>$c_p$</td>
<td>[J/KgK]</td>
<td>specific heat</td>
</tr>
<tr>
<td>C</td>
<td></td>
<td>capacity matrix</td>
</tr>
<tr>
<td>$e$</td>
<td>[W/m²]</td>
<td>emitted energy</td>
</tr>
<tr>
<td>g</td>
<td>[m/s²]</td>
<td>acceleration due to gravity</td>
</tr>
<tr>
<td>Gr</td>
<td>[-]</td>
<td>Grashof Number</td>
</tr>
<tr>
<td>$h$</td>
<td>[W/m²K]</td>
<td>convective heat transfer coefficient</td>
</tr>
<tr>
<td>K</td>
<td></td>
<td>conductivity matrix</td>
</tr>
<tr>
<td>L</td>
<td>[m]</td>
<td>characteristic length</td>
</tr>
<tr>
<td>Re</td>
<td>[-]</td>
<td>Reynolds Number</td>
</tr>
<tr>
<td>$T$</td>
<td>[K]</td>
<td>temperature</td>
</tr>
</tbody>
</table>

Greek symbols

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Unit</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ε</td>
<td>[-]</td>
<td>emissivity</td>
</tr>
<tr>
<td>$\lambda$</td>
<td>[W/mK]</td>
<td>conductivity</td>
</tr>
<tr>
<td>$\nu$</td>
<td>[m²/s]</td>
<td>Kinematic viscosity</td>
</tr>
<tr>
<td>$\rho$</td>
<td>[Kg/m³]</td>
<td>density</td>
</tr>
<tr>
<td>$\sigma_S$</td>
<td>[W/m²K⁴]</td>
<td>Stefan-Boltzmann’s constant</td>
</tr>
</tbody>
</table>

Abbreviations

<table>
<thead>
<tr>
<th>Abbreviation</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>CFD</td>
<td>computational fluid dynamics</td>
</tr>
<tr>
<td>MpCCI</td>
<td>Mesh-based parallel Code Coupling Interface</td>
</tr>
<tr>
<td>nat. conv.</td>
<td>natural convection</td>
</tr>
</tbody>
</table>

References


[14] Fraunhofer Gesellschaft SCAI, MpCCI user guide, St. Augustin, 2004