

Experimental and Numerical Study of Heat Transfer at the Underbody of a Production Car

Simon Huber, Thomas Indinger, and Nikolaus Adams
Technische Universität München

Thomas Schuetz
Audi AG

ABSTRACT

The optimization of the flow field around new vehicle concepts is driven by aerodynamic and thermal demands. Even though aerodynamics and thermodynamics interact, the corresponding design processes are still decoupled. Objective of this study is to include a thermal model into the aerodynamic design process. Thus, thermal concepts can be evaluated at a considerably earlier design stage of new vehicles, resulting in earlier market entry. In a first step, an incompressible CFD code is extended with a passive scalar transport equation for temperature. The next step also accounts for buoyancy effects. The simulated development of the thermal boundary layer is validated on a hot flat plate without pressure gradient. Subsequently, the solvers are validated for a heated block with ground clearance: The flow pattern in the wake and integral heat transfer coefficients are compared to wind tunnel simulations. The main section of this report covers the validation on a full-scale production car. A specially developed heated electronic component dummy mounted to the underbody of the car introduces heat into the flow field. Again the soundness of the underlying methodology is appraised by comparing the CFD simulations to wind tunnel experiments. Attention is paid to lift and drag coefficients as well as to flow and temperature patterns around the car and the heated dummy at the underbody. With the new method, concepts can be thermally investigated in early product stages, leading to a reduced time to market.

CITATION: Huber, S., Indinger, T., Adams, N., and Schuetz, T., "Experimental and Numerical Study of Heat Transfer at the Underbody of a Production Car," *SAE Int. J. Commer. Veh.* 7(1):2014, doi:10.4271/2014-01-0582.

INTRODUCTION

Car manufacturers are forced to further decrease fuel consumption due to finite petroleum resources, customer claims and legal requirements. In addition to optimizing the driving resistances, it is necessary to come up with new and unestablished drive concepts, such as hybrid and electronic vehicles. The aerodynamic performance plays a key role and, due to the ability to recover, becomes significantly more important than the rolling and acceleration resistance for efficient vehicles, [18]. Furthermore, thermal management faces new challenges on account of various new components: electric engine, power electronics, high voltage battery, auxiliary heating, etc. These components can be randomly placed inside the vehicle and require sufficient cooling.

The aerodynamic investigation starts at early design stages employing well validated numerical methods, whereas the thermal dimensioning is strongly based on experience and first validated on prototypes in road trials. Components of new drive concepts cannot be covered with this method, since there are

no empirical values. Design changes in late project phases are always costly and often show detrimental effects on the aerodynamic performance of the vehicle.

Heat Transfer in Turbulent Flows

Heat transfer problems can generally be divided into three different mechanisms: Convection, conduction and radiation. Radiation is heat transfer between bodies with different temperatures via electromagnetic waves. For this report, the assumption of gray bodies is used: The emission ratio and absorption ratio is considered to be equal. Under this assumption the heat loss via radiation can be calculated, [2]:

$$\dot{q} = -\sigma \varepsilon (T^4 - T_{\infty}^4) \quad (1)$$

In equation (1), $\sigma = 5.67 \cdot 10^{-8} [W/m^2K^4]$ is the Stefan-Boltzmann constant and is the hemispheric total emissivity. Even though for underhood thermal management problems, radiation is a significant heat sink, [3] it is neglected at this stage for the car test case. Conduction is the heat transfer within one material

due to a temperature gradient. Heat transfer via free electrons is the dominant mechanism for conduction in aluminum. The heat flux is directly proportional to the temperature gradient and can be calculated with Fourier's law, [2]:

$$\dot{q} = -\lambda \frac{\partial T}{\partial x} \quad (2)$$

Here, λ is the thermal conductivity, which is only slightly temperature dependent for gaseous fluids. The third, and for our applications most important, mechanism for heat transfer is convection. Convection is heat transfer via energy transported by fluid movement. Consequently the fluid field, namely temperature and velocity components next to the hot surface, are of crucial importance. The temperature of the hot surface is of minor influence on the heat transfer coefficient. The heat flux via convection from solids to surrounding flow can be calculated by:

$$\dot{q} = -\alpha(T_w - T_\infty) \quad (3)$$

In equation (3), $\alpha = [W/m^2K]$ is the convective heat transfer coefficient (HTC). T_w is the local surface temperature and T_∞ is the ambient temperature of the surrounding flow. In the viscous sublayer of aerodynamic boundary layers, heat can solely be transported by conduction within the fluid, due to the no-slip boundary condition on solid walls. With known temperature field, the heat transfer coefficient can be calculated with equations (2) and (3):

$$\alpha = -\lambda \frac{\left(\frac{\partial T}{\partial y}\right)_w}{T_w - T_\infty} \quad (4)$$

For a given geometry the heat transfer coefficient is directly proportional to the fluid velocity next to the solid:

$$\alpha = f(Re^{0.8}) \quad (5)$$

A distinction is made between forced and free convection. If the fluid motion is driven mainly by density differences, due to different temperatures, it is called free convection. A forced convection is present if the free fluid flow is dominant. In order to distinguish between the two convection schemes, the Grashof-number can be used:

$$Gr = \frac{g\beta(T_w - T_\infty)L^3}{\nu^2} \quad (6)$$

If the Grashof-number is small compared to the square of the Reynolds-number, buoyancy forces are small compared to inertia forces: forced convection is present. In equation (6), β is the constant thermal expansion coefficient and g is the gravitational acceleration.

The dimensionless heat transfer coefficient can be expressed with the Nusselt-number:

$$Nu = \frac{\alpha L}{\lambda} \quad (7)$$

The Nusselt-number is considered to be the dimensionless temperature gradient on the wall. There are some empirical correlations for base bodies. For a profile cylinder in cross flow, an integral Nusselt-number can be calculated from the laminar and turbulent Nusselt-numbers, [2]:

$$Nu_m = 0.3 + (Nu_{m,lam}^2 + Nu_{m,turb}^2)^{0.5} \quad (8)$$

with

$$Nu_{m,lam} = \frac{2}{\sqrt{\pi}} Re_x^{0.5} \varphi_1(Pr) \quad (9)$$

and with

$$Nu_{m,turb} = \frac{0.037 Re^{0.8} Pr}{1 + 2.443 Re^{-0.1} (Pr^{2/3} - 1)} \quad (10)$$

The constant $\varphi_1 = 0.519$ is a function of the Prandtl-Number. The correlation is valid for a wide range of Reynolds-numbers, which applies in this case. This correlation is used to validate the generic test case.

Turbulent Boundary Layers

As known, a no-slip boundary condition generally applies for fluid flows on solid walls. The fluid velocity on the wall is zero and increases in the typical boundary layer profile towards the ambient flow speed. For surfaces with a temperature, that differs from the fluid temperature, a similar profile develops. The wall distance y , where

$$|T_{\delta_T}(y) - T_w| = 0.99 |T_\infty - T_w| \quad (11)$$

is valid, is called the thermal boundary layer, [4]. The composition of the thermal boundary layer is similar to the aerodynamic boundary layer, [11]. For fluids with $Pr \approx 1$ the thickness of both boundary layers are of similar magnitude. For the logarithmic layer of the thermal boundary layer, an analytical solution can be given:

$$\frac{\theta}{\theta_\tau} = \frac{Pr_\tau}{\kappa} \ln \frac{y U_\tau}{\nu} + A_H(Pr) \quad (12)$$

Herein θ_τ is the friction temperature, Pr_τ is the turbulent Prandtl-number, κ and A_H are constants, U_τ is the friction velocity ν and is the kinematic viscosity. As the thermal

boundary layer is crucial for convective heat transfer, [equation \(12\)](#) is used to compare it with the results in the inner layer of the thermal boundary layer.

Computational Fluid Dynamics

For decades numerical methods have been an inherent part of aerodynamic development, [14]. However, standard Reynolds Averaged Navier Stokes (RANS) turbulence models cannot resolve the complex flow structures associated with detachment and reattachment around a typical vehicle in a satisfying manner. Hence, more sophisticated turbulence modeling is often used. The present approach relies on a Detached Eddy Simulation (DES) model, which can be considered a hybrid between a Large Eddy Simulation (LES) and a RANS model in the vicinity of solid walls. Philosophy of the LES model is the assumption that spatial large structures are crucial and will be calculated directly. The influence of the RANS model on the solution quality is inferior. The one-equation Spalart Allmaras Model (SA) is used for the RANS region. The method is imbedded in a fully automated preprocessing, solving and post processing tool and validated for a broad spectrum of automotive shapes, [9]. It delivers robust results for adiabatic flow dynamics. This well established process will be extended to account for temperature transportation. Three different procedures are approached: First, the incompressible method will be extended by a scalar transport equation for temperature. Second, the incompressible method with transport equation for temperature will be extended with the Boussinesq approach [7] accounting for buoyancy. Third, a fully compressible solver is used. This paper presents current results for the two incompressible methods. Results for the compressible solver are presented for a generic test case as well.

In the simplest approach a scalar transport equation is included to the solved system of equations:

$$\frac{\partial T}{\partial t} + \nabla(UT) = \nabla \left(\frac{\lambda_{eff}}{\rho} \nabla T \right) \quad (13)$$

The temperature transport occurs completely decoupled from the flow field, so that the temperature distribution does not affect the flow field. In [equation \(13\)](#), λ_{eff} is the effective thermal diffusivity, which is calculated from fluid and turbulence properties. This method is considered to be reliable up to a temperature difference of about $\Delta T \approx 10K$. In the following, this method will be called scalarDES. In the second approach the same scalar transport [equation \(13\)](#) for temperature is used, but now density is not constant everywhere. The incompressible momentum equation is extended by a hydrostatic pressure term, where the density is a function of temperature, but independent of pressure:

$$\rho_k = 1 - \beta(T - T_{ref}) \quad (14)$$

Apart from that, density is constant in all other terms. Thus temperature only effects the term for buoyancy. In this approach the flow field (ρ , U) is not decoupled by the temperature. The full set of equations with derivation is given in [7]. The second approach will be called buoyantDES in the following. This method is considered to be correct within an error of 1% for a maximum temperature difference of about $\Delta T \approx 15K$, [6]. The third approach is fully compressible. In addition to the compressible Navier-Stokes equations, the following enthalpy (h) transport equation is solved:

$$\begin{aligned} \frac{\partial \rho h}{\partial t} + \nabla \cdot (\rho U h) \\ = \frac{\partial P}{\partial t} + \nabla \cdot (\lambda_{eff} \nabla h) - \frac{\partial K}{\partial t} - \nabla(UK) \end{aligned} \quad (15)$$

In [equation \(15\)](#), U is the fluid flow velocity and K is the kinetic energy. The density is modeled as an ideal gas. This third approach is called compressibleDES in the following.

NUMERICAL SIMULATION OF HEAT TRANSFER IN TURBULENT BOUNDARY LAYER ON A FLAT PLATE

The development of the aerodynamic and the thermal boundary layer is essential for the correct prediction of heat transfer problems. Hence a simulation has been performed to test the accurate formation of the boundary layers. Hoffmann and Perry [8] investigate the development of turbulent thermal layers on homogeneously heated flat plates in an extensive way. Two hydraulically smooth plates with a length of 1.52m are steam-heated to an uniform temperature difference of 12K. The analysis of the virtual origins of the aerodynamic and thermal layers is the basis for the conducted simulations. The virtual origin of a turbulent boundary layer is where a fully turbulent boundary layer without laminar turbulent transition would start.

[Figure 1](#) shows the numerical and geometric setup of the heated flat plate test case. The geometry is a 2D channel with friction walls on the lower side. Wall 1 is inviscid and adiabatic. The length of Wall1 corresponds to the virtual origin of the turbulent boundary layer. Wall2 and Wall3 are the heated friction walls. The inflow conditions are consistent with the Experiments. The averaged wall resolution on the friction side is $y^+ = 0.5$. This satisfies the requirement that it must be below unity for the applied low-Reynolds Spalart - Allmaras Model. The dimensionless wall distance y^+ is defined as:

$$y^+ = \frac{y \sqrt{\frac{\tau_w}{\rho}}}{\nu} \quad (16)$$

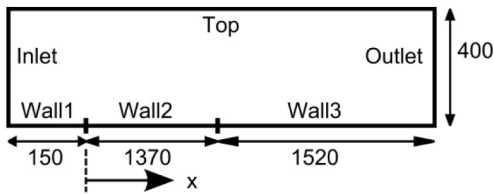


Figure 1. Numerical and geometric setup of heated flat plate test case. (Units in millimeters)

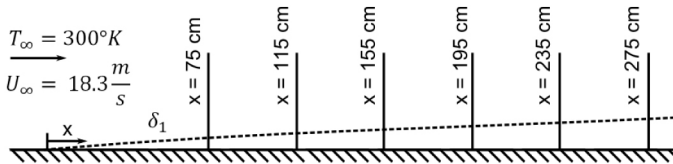


Figure 2. Lines for analysis of the numerical simulations. (Plotted with the displacement thickness.)

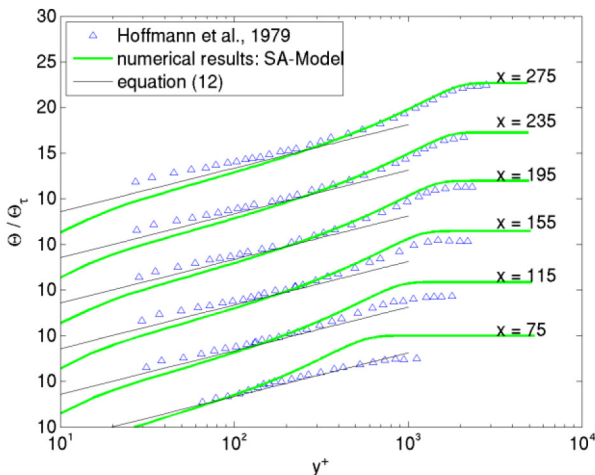


Figure 3. Analysis of thermal law of the wall: Simulation, experiment and analytical solution [equation \(12\)](#).

A grid study exposes the sufficient mesh resolution. The boundary called "Top" is modeled as symmetry. As the case is 2D, the boundary conditions on the front and back are corresponding. For the numerical examination the OpenFOAM® CFD Toolbox from ESI Group [5] is used. The software is Open Source and combines various numerical solvers for continuum mechanic problems as well as tools for mesh generation and pre- and post-processing. Due to the fact that it is written in C++ one can adjust solvers for specific problems. In later simulations the DES (detached eddy simulation) model [16] will be used for turbulence modeling. In RANS regions, which is next to solid walls, the DES model relies on the Spalart Allmaras one equation turbulence model [15]. Therefore the flat plate validation case is simulated using the SA model. The discretization of the convective and diffusive fluxes is second order accurate. The discretization in time is carried out with a first order bounded Euler scheme. The simulation time is 2.2 seconds to reach a developed flow with an averaging of 0.1 seconds. The soundness of the simulations is assessed using the values along the lines shown in [figure 2](#).

[Figure 3](#) shows the gradient of the temperature next to the hot wall. As expected the simulation deviates from the log law towards lower temperature for small values y^+ in the range of the viscous sublayer. Departing from the wall the simulation deviates towards the defect law. Only the profile for $x = 75\text{cm}$ does not show good agreement with the experiment. A possible explanation is that the boundary layer is not fully developed here. All in all the simulation is in fair agreement with the experiments and with the analytical solution, [equation \(12\)](#). Eventually, the reproduction of the boundary layer is assumed to satisfy the needs for simulating the heat transfer on more complex geometries.

HEAT TRANSFER ON A BLOCK WITH GROUND CLEARANCE

As a second proving stone, a heated block with ground clearance is investigated. The key aspects are the temperature and velocity distribution in the near wake flow of the block, as well as the integral heat transfer coefficient. The block is made of solid aluminum. Four evenly distributed cartridge heaters are used to heat up the block with a constant electric heat flux of $\dot{Q} = 1.5\text{ kW}$. The maximum core temperature is $T = 150^\circ\text{C}$. The Reynolds number build with the overflow length [2] is $Re_L = 2.9 \cdot 10^5$ and The Grashof-number, [equation \(6\)](#) for the block is $Gr = 1.9 \cdot 10^9$. With these values the heated block is well in the flow regime of forced convection. The flow fields above and behind the block are measured using a multipurpose probe rake. The experimental results are used to validate the simulations performed with the code OpenFOAM employing the above explained approaches for temperature transport.

Dimension and Configuration

The block is mounted on 4 pins in order to realize a ground clearance. 4 Teflon washers are used between the block and the pins. To reduce the conductive heat flux into the wind tunnel bottom. A direct connection across the 4 M6 screws is left. Hence the conductive heat loss via the stilts is small. The physical dimensions of the block are: $L = 350\text{mm}$, $W = H = 200\text{mm}$. [Figure 4](#) gives an overview over the wind tunnel setup.

The heated block was investigated in wind Tunnel C at the Technical University of Munich. The cross section of the wind tunnel is $W \cdot H = 2.7\text{m} \cdot 1.8\text{m}$, the measurement section is $L = 21\text{m}$, since the wind tunnel is designed for investigating wind loads on buildings and wake flows of airplanes. More Information about the subsonic closed circuit wind tunnel can be found in [1]. The blockage factor of 1% is relatively small. The block is located 1.95m behind the nozzle. The turbulence intensity in the test section is $Tu_x = Tu_y = Tu_z < 0.5\%$. The inflow velocity is chosen to be 8.3m/s , this is the estimated velocity an electronic component in the underbody of the car would experience. The inflow temperature is 25°C averaged over all experiments. With a coefficient $\delta/z = 0.75$ of the block is out of the aerodynamic boundary layer.