

The Effects of Cooling Air on the Flow Field around a Vehicle

Timo Kuthada, Felix Wittmeier, Benjamin Bock, Christoph Schoenleber, and Alexander Link
FKFS

ABSTRACT

Cooling air flow is an important factor when it comes to vehicle performance and operating safety. In addition, it is closely linked to vehicle aerodynamics. In recent years more and more effort is being spent to optimize the losses generated by the flow through the vehicle. Grille shutters, better sealing and ducting are only some examples for innovations in this field of work, resulting in a lower contribution of the cooling air flow to overall drag.

But cooling air not only affects the internal flow of the vehicle but also changes the flow around it. This paper will show changes in the flow field around the generic DrivAer model resulting from cooling air flow, especially in the wake behind the car and in the region around the front wheels. The results were gathered using PIV measurements, multi-hole-probe measurements and pitot tube measurements in the 1:4 model scale wind tunnel of IVK University of Stuttgart. Unless stated otherwise, all the measurements were conducted with rotating wheels and ground simulation. Additionally, CFD simulations were performed using the Lattice-Boltzmann solver Exa PowerFLOW. These simulations allow an even closer look at the flow field around the vehicle, especially in the under-floor region that is hardly accessible in measurement.

CITATION: Kuthada, T., Wittmeier, F., Bock, B., Schoenleber, C. et al., "The Effects of Cooling Air on the Flow Field around a Vehicle," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(2):2016, doi:10.4271/2016-01-1603.

INTRODUCTION

With increasing engine power and tighter packaging in the engine compartment, sufficient cooling of all components with thermal loads is indispensable. Therefore air is typically taken from the area of the stagnation point in the vehicle's front and guided through one or several heat exchangers as well as through the engine compartment. This cooling air is then released into the wheel houses or into the under-floor region of the vehicle resulting in interference effects and flow losses.

On modern cars, the amount of cooling air is not constant, but can often be varied through grille shutters or the amount of power put into the radiator fan. These variations not only affect the cooling capacity, but also affect the aerodynamic properties of the vehicle. These aerodynamic effects can not only be found in the front of the vehicle, but, for example, can also be seen in the wake of the vehicle.

In this paper, the effects of variations on the cooling air system are investigated experimentally and numerically. In the model scale wind tunnel of IVK University of Stuttgart different measurement techniques, like e.g. a 3D-PIV system and a multi hole probe, were used to measure the flow field around the vehicle.

STATE-OF-THE-ART

The following section gives a short introduction to cooling air flow and corresponding research work on reference models.

Aerodynamic Coefficients of a Cooling Air Flow System

The losses related to the cooling air flow, together with momentum losses at the in- and outlets in combination with interference effects of exiting air and surrounding flow are determined by a differential method. The cooling drag coefficient c_{DC} is derived from the difference between the measured drag of a vehicle with cooling airflow minus the drag of the same model with closed air inlets, the so called mock-up:

$$c_{DC} = c_D \text{ with cooling air} - c_D \text{ mock-up} \quad (Eq. 1)$$

All results presented for mock-up are deduced with open exit regions. The values for lift are derived by the same method.

The cooling air volume flow coefficient:

$$c_{VR} = \frac{\dot{V}}{v_A \cdot A_R} \quad (Eq. 2)$$

is the quotient between the cooling air volume flow and the product of radiator area A_R and free stream velocity v_A , which is equivalent to the maximum possible volume flow through a given radiator [1]. c_{VR} is equivalent to the non-dimensional mean velocity through the radiator v_R .

Interference Effects with Surrounding Flows

The flow through the underhood compartment interacting with the surrounding flow results in interferences at cooling air in- and outlet areas. Garrone and Masoero [2] determined with a vehicle model with integrated balances for vehicle surface and air ducting the contributions of vehicle upper- and underbody as well as of the air flow system. They showed the cooling air drag c_{DC} was smaller than the drag on the air flow system determined by the balance. This indicates, that the interference effects at in- and outlet cause a reduction of the total drag compared to the measurement without cooling air flow.

Examples for these interference effects are e.g. a changed approach flow angle towards the front wheels due to the cooling air flow, as it has been reported by Wiedemann and Settgaß [3]. This results in an increased drag of the front wheels. The influence of engine compartment under-shields and different outlet openings was investigated by Herbig [4]. He could achieve a reduction of cooling air drag of 60% by minimizing the momentum losses and disturbances at the cooling air exit area.

A full scale passenger car was investigated in detail by Kuthada [1]. It was shown that most of the changes in drag related to cooling air flow are located in the underhood region. Leakage flows around the radiator are strong contributors to the cooling air drag. Tests with a sedan and an estate vehicle with the same underhood package demonstrated that the cooling air drag depends on the vehicle rear end configuration as well as the rotation of the wheels. While the estate vehicle showed for fixed wheels only half of the cooling air drag than the notchback vehicle, this difference was equalized by testing with rotating wheels. By total pressure measurements to the side and behind of the vehicles, this effect was attributed to the interference effects at the wheel house interacting with the vehicle base pressure.

Generic Models for Cooling Air Flow Investigations

Generic models have a long history in aerodynamic research. They can be subdivided in two classes: simple bodies and basic car shapes. A comprehensive overview of the different available models was presented by Le Good and Garry in 2004 [5]. The majority of these models have been defined in the last century, before ground simulation techniques became available in automotive wind tunnels and are therefore either placed on struts or equipped with fixed wheel stubs only. In addition, even the basic car shaped models do only feature a limited set of details and cooling air flow is neither considered.

Models with internal flow have been presented by Barnard [6], and Kuthada [7]. However, these models were still based on simple box shape models like the SAE Reference Body and were therefore missing a level of detail which is preferable for today's aerodynamic research.

This shortcoming was addressed when Wittmeier and Kuthada [8] introduced an open grille geometry of the DrivAer model, presented by Heft et al. [9], in order to enable basic investigations of cooling air flow aerodynamics. The model geometry is freely available for download [10] and is the starting point for the investigations presented here.

EXPERIMENTAL SETUP

All experiments have been carried out in the model scale wind tunnel of IVK University of Stuttgart. It is capable of measuring models in 20% and 25% scale and is equipped with state-of-the-art ground simulation techniques including a five belt system and a boundary layer control system. With boundary layer control and moving floor, the wind tunnel features a block profile. Without boundary layer control the thickness at the vehicle center is 45 mm [11]. The maximum test speed is 80 m/s. The free stream turbulence intensity is below 0.3% [12]. The measurements were conducted both with and without ground simulation using the DrivAer body as test vehicle at a test speed of 50 m/s equal to a Reynolds number of $3,6 \cdot 10^6$. All measurements are presented without the application of any correction methods.

DrivAer Body

The DrivAer model is a detailed generic vehicle based on the Audi A3 and the BMW 3-series. It was developed by TU Munich [9] to account for the need of having a freely available generic vehicle geometry, which is detailed enough to investigate a large variety of topics in the field of aerodynamics. It can also be used to compare different wind tunnels and CFD simulations with each other.

The model can be equipped with different rear ends: A notchback, an estate back and a fast back are available. Except for the rear ends, the model stays completely the same, which helps to investigate the influence of the rear shape.

In 2014/2015 cooling air and a detailed engine compartment was added to the DrivAer model with input from the ECARA CFD Subgroup [13], allowing an even wider use of the DrivAer model. [Figure 1](#) shows the 25% DrivAer model in the IVK model scale wind tunnel.

The cooling air, entering the vehicle through the two grille openings, passes through a radiator simulator, representing the same pressure drop as the full cooling package of a production car (2001 Ford Mondeo). Following the radiator simulator a simplified 4-cylinder engine is installed in the vehicle. The cooling air then leaves the underhood compartment either through large openings in the front wheel houses or through the tunnel into the under-floor region. [Figure 2](#) shows a rendering of the underfloor configuration of the used DrivAer Model. The large wheel house openings, displayed in [Figure 3](#), have been chosen to facilitate a larger degree of freedom for geometry changes on the model once hardware models have been built. However, this feature results in about $\frac{3}{4}$ of the cooling air exiting into the wheel arches [8].

In comparison with the results in [8], the pressure loss of the radiator simulator had to be slightly increased as it turned out that the originally used perforated sheet resulted in a too low pressure loss compared to the Mondeo reference values. The resulting curves are plotted in [Figure 4](#). The data for the reference curve can be found in [Table 1](#) in the [appendix](#).

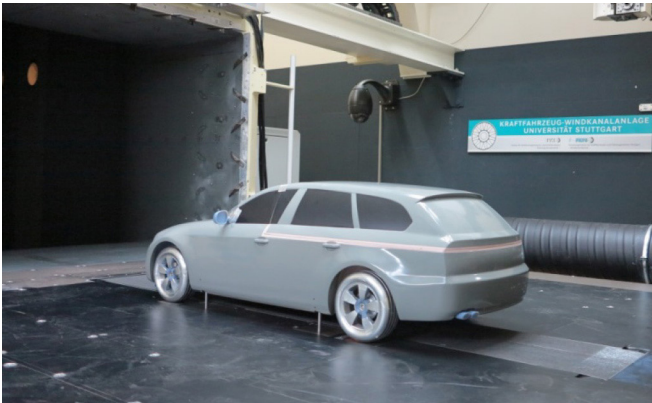


Figure 1. Estate back DrivAer model (quarter scale) in the IVK model scale wind tunnel

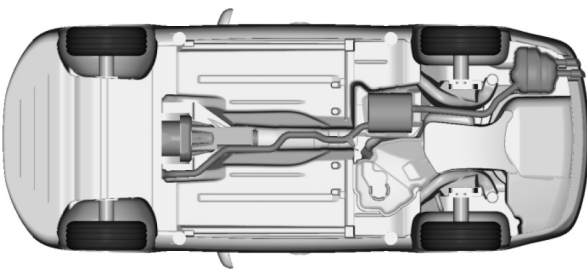


Figure 2. Rendering of the underfloor of the FKFS DrivAer model

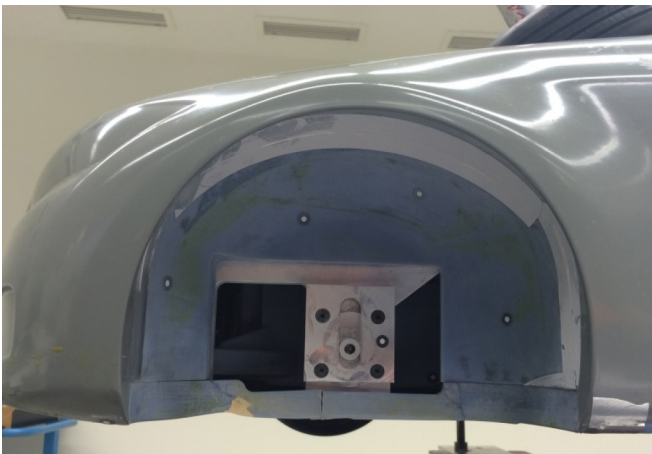


Figure 3. Wheel house openings of the FKFS DrivAer model

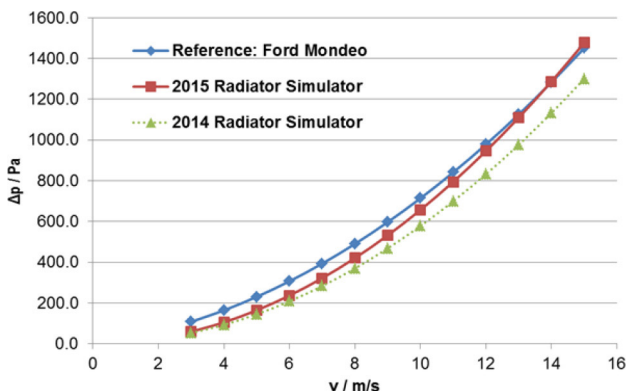


Figure 4. Measured pressure loss curve of the radiator simulator before and after the adjustment

PIV System

Stereo particle image velocimetry (PIV) was used to investigate the flow in the wake of the vehicle. The system was built by LaVision and the setup includes a 10 Hz fixed frequency, dual cavity Nd:YAG laser, with a maximum energy of 850 mJ per pulse and a wavelength of 532 nm, and two 16 bit CCD cameras with a resolution of 2560×2160 pixels. The whole system is mounted to the traversing system of the model scale wind tunnel, which can be precisely positioned via plc (refer to [Figure 5](#) and [Figure 6](#)).

To direct the laser beam, optical mirrors were used that were also connected to the traversing system. This enables measurements of planes of various locations with only one calibration. The laser beam was expanded to a light sheet with a thickness of approximately 2 mm, perpendicular to the y-plane or z-plane, depending on the configuration.

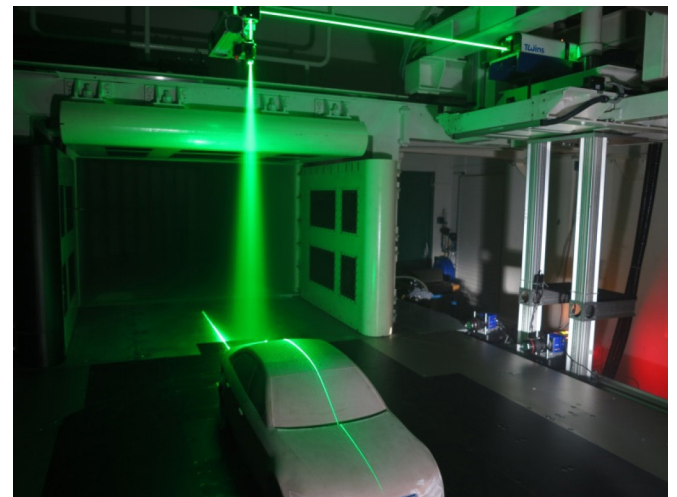


Figure 5. Setup of the PIV system for measuring x/z-planes in the wake in the vehicle. Laser and cameras are connected to the traversing system of the model scale wind tunnel

For the measurement of the wake volume 24 x/y planes in a range between 5 and 300 mm above the ground with distances of 5-15 mm were measured. Each measured volume consists out of 460,000 measurement points.

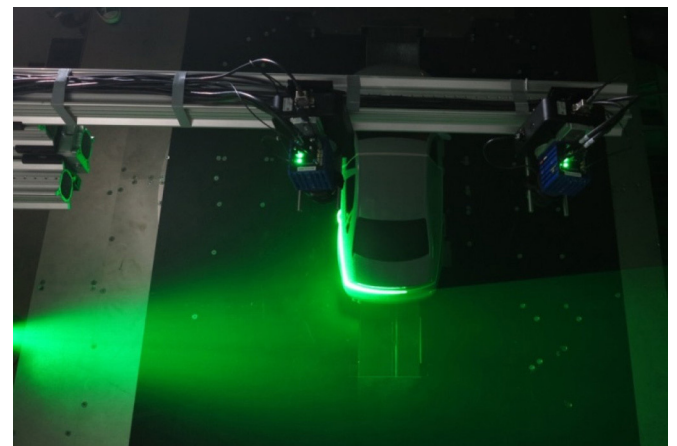


Figure 6. Setup of the PIV system for measuring x/y planes in the wake in the vehicle. Again, the whole setup can be moved with the traversing system.