

Manipulation of the Aerodynamic Behavior of the DrivAer Model with Fluidic Oscillators

Dirk Wieser, Henning Lang, Christian Nayeri, and Christian Paschereit
Technische Universität Berlin

ABSTRACT

The effect of an active flow control method is investigated on a 1:4 scale realistic vehicle model called "DrivAer" with notchback geometry. The wind tunnel experiments are conducted at a Reynolds number of $Re=3.0 \cdot 10^6$. Fluidic oscillators are applied at the c-pillars and at the upper rear edge of the window. The actuators are installed inside the hollow designed model emitting a high frequency sweeping jet. The spacing of the actuators, the mass flow rate, and the position of actuation are varied. The effect of the active flow control on the car is investigated with force and surface pressure measurements. The surface trace pattern is visualized with tufts for the active flow control cases and the baseline case. A tuft algorithm analyzes provides statistical data of the flow angles. Moreover, particle image velocimetry measurements are performed in the plane of symmetry for $\beta=0^\circ$ to capture the flow field at the rear end and the wake. The results indicate a significant pressure increase at the backlight and the upper trunk for increasing mass flow rates of the actuators. An increased pressure level of $\Delta C_p \approx +0.35$ is found compared to the non-actuated case. Furthermore, the aerodynamic forces show a decrease of the net drag of approximately $\Delta C_{D,corr} \approx -3.5\%$. The downforce varies between $\Delta C_{L,corr} \approx -25\%$ and $+38\%$ depending on the actuation case. The tuft visualizations indicate a modification in the surface trace pattern at the rear end section of the notchback. The asymmetrical vortical structures at the backlight are typical on notchback rear ends and become smaller or vanish. The PIV measurements show a changed wake structure with a shifted location of the free stagnation point. Hence, the application of oscillating jets demonstrates a high potential to change the aerodynamic behavior of a realistic road vehicle design.

CITATION: Wieser, D., Lang, H., Nayeri, C., and Paschereit, C., "Manipulation of the Aerodynamic Behavior of the DrivAer Model with Fluidic Oscillators," *SAE Int. J. Passeng. Cars - Mech. Syst.* 8(2):2015, doi:10.4271/2015-01-1540.

INTRODUCTION

The exterior vehicle shape is mainly guided by customer requirements and designer aesthetics. Hence, the capability for aerodynamic modifications on the exterior geometry is very constricted. Only certain non-visible areas such as the underbody of the vehicle offer freedom of design. However, an improved aerodynamic design is important regarding the reduction of fuel consumption, increased performance, and driver comfort requirements. Intensive wind tunnel tests and computational fluid dynamics have been conducted in the past decade to meet these demands. Nevertheless, a bluff body is dominated by flow separations at the rear end section. Hence, the aerodynamic manipulation of the flow field has enormous potential to change the overall performance of the vehicle.

Wieser [2] demonstrated the potential of the flow manipulation at the rear end of a realistic car model with vortex generators as a passive flow control method. The downforce of the car is increased significantly due to an increased pressure recovery at the rear backlight and upper trunk. Though, the overall drag is slightly increased due to the additional drag penalties induced by the vortex generator. However, this approach may not be desirable for stylistic reasons. Internal active flow control (AFC) devices are a smarter method to manipulate the outer flow field with less drag penalties.

In the past, several AFC methods were developed and implemented on generic car models [1]. The main challenge by using AFC is to achieve low power consumption combined with a high actuator output such as actuation strength and the penetrated region. One potential method is offered by fluidic oscillators which are integrated into the surface of a realistic car model at selected locations. These devices emit an in-plane high frequency sweeping jet without the need for moving parts using an external pressure supply. Wosizdlo [13] and Schmidt [15] employed fluidic oscillators on a 1:10 truck model called G.E.T.S with base flaps (generic representation of a European long-haul heavy duty vehicle). The flow remains attached for high slant angles. The base pressure is increased and the net drag is significantly reduced up to $\Delta C_D = -15\%$. Ostermann et al. [1114] and Gärtlein et al. [10] investigated the used actuator design. Ostermann et al. examine and discuss various phase-average methods of the naturally oscillating flow field.

This study focuses on the potential use of fluidic actuators on the realistic car model (DrivAer). Therefore, a first actuator design was chosen and tested for two positions and various mass flow rates. However, since the DrivAer is already aerodynamically optimized, the expected gain in lift and drag may be lower than for a less optimized generic design (e.g., the G.E.T.S. model). Nevertheless, the

use of fluidic oscillators may change the aerodynamic design process and limits and may offer new design opportunities (e.g., higher slant angles without separations).

EXPERIMENTAL SETUP

Wind Tunnel Setup

The experimental investigations are performed in a closed-loop wind tunnel at the Institute of Fluid Dynamics and Technical Acoustics at the “Technische Universität Berlin” (Figure 1). The cross-sectional area of the closed test section is $2\text{m} \times \sqrt{2}\text{m}$ and the maximum freestream velocity is $c_\infty \approx 41\text{m/s}$ with a turbulence intensity of less than $Tu < 0.5\%$. The Pitot tube is located upstream of the model in the ceiling of the test section. The Reynolds number is $Re = 3.0 \cdot 10^6$ based on the length of the model. A thermocouple monitors the temperature in the wind tunnel. A splitter plate cuts off the boundary layer of the wind tunnel. Thus, the boundary layer effect at the model is minimized. No scoop or moving belt is available.

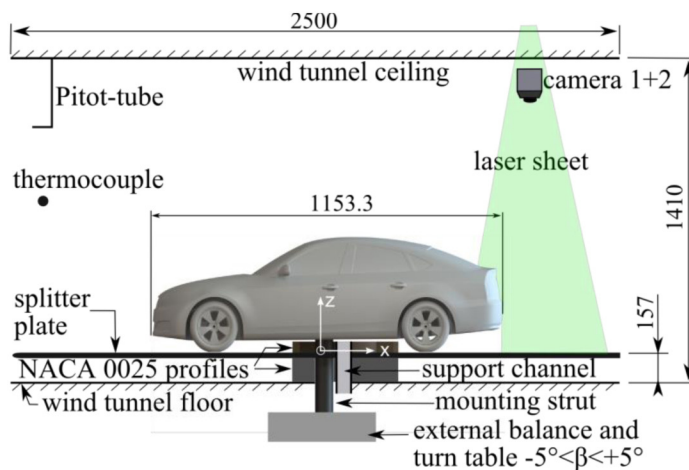


Figure 1. Wind tunnel setup with the 1:4 scale model. All values in mm.

The model is mounted on an external balance which is located below the test section and further enables yaw. Three distinct crosswind conditions are investigated: $\beta = 0^\circ$ and $\beta \pm 5^\circ$. The zero yaw angle $\beta = 0^\circ$ is determined as the angle where no side force occurred. A mounting strut holds the model and has no contact to the wind tunnel floor and the splitter plate. The mounting strut is covered with two NACA 0025 airfoils to avoid aerodynamic effects acting on the strut. The cord length of the NACA airfoil is 350 mm. A gap between the tip of the NACA profile and floor of the model is left to avoid force closure. A support channel through both NACA profiles allows the accessibility for the differential pressure sensor cables, the pressure reference tube, and the active flow control pressure supply. The yaw angle variations and the force measurements require a minor clearance between the horizontal cut tires and the splitter plate of $t_{\text{gap}} \approx 2\text{ mm}$. The blockage of the model ($A_{\text{DrivAer}} = 0.135\text{m}^2$) in the test section at $\beta = 0^\circ$ is $\varphi \approx 5.4\%$. A correction of all aerodynamic forces and moments is applied by correcting the freestream velocity for example for the drag coefficient:

$$C_D = \frac{F_D}{\rho/2 \cdot ((1 + \varphi) \cdot c_\infty)^2 \cdot A_{\text{DrivAer}}} \quad (1)$$

DrivAer Model

The DrivAer-model is a design hybrid of an “Audi A4™” and a “BMW 3 series™”. The geometry contains details of a realistic car such as rimmed wheels, windows, and side mirrors. The DrivAer model is modularly designed with several design options such as three different rear end configurations (fastback, notchback, and fullback) and a smooth or detailed underbody. Other studies upgraded the geometry with under hood flow phenomena or heat exchangers [8]. More detailed information regarding the design is provided by Heft et al., [9]. The exterior design of the DrivAer model is available to the research community.

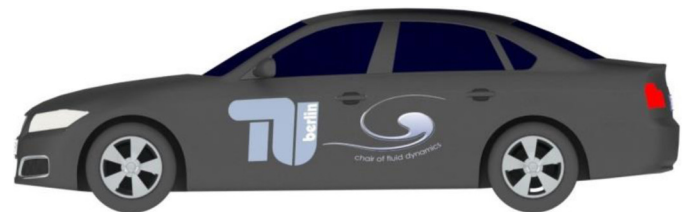


Figure 2. The DrivAer model in notchback configuration with smooth underbody and non rotating wheels.

In the present study a 25% scale DrivAer model in notchback configuration with the smooth underbody option is used (Figure 2). The outer dimensions of the model are $L = 1.15\text{m}$, the height $H = 0.354\text{m}$ and the cross-sectional area $A = 0.135\text{m}^2$. The wheels are fixed and at the lowest point slightly flattened.

The hollow model provides sufficient space for the pressure measurement sensors and the active flow control system such as fluidic oscillators, pressure distributors, and pressure piping. Strangfeld et al. demonstrate that the model exhibits the Reynolds independence at approximately $Re = 2.8 \cdot 10^6$, [6]. Previous studies [6], [3] confirm the results of the TUM [9].

Force Measurements

A six-component balance is located outside of the wind tunnel. Furthermore, the balance is able to turn about the vertical z-axis. The measureable range is 2200N for each force component. The maximum error of the balance for all measured forces is 0.1% full scale. The maximum error for the yaw moment is 0.1% and for the other moments it is 0.3% full scale. The maximum effective sampling rate is about 5 Hz due to the proportions of the balance. Hence, the force measurements are time-averaged values.

Pressure Measurement

143 pressure positions are available for the notchback rear end module. Furthermore, at the base and the rear fender are 117 additional pressure taps available (Figure 3). Most of the pressure taps are distributed on the left side of the model, which enables a fine

spatial resolution with fewer pressure sensors. Additional reference pressure taps located at the right side. Measurements are performed for positive and negative yaw angles and the results are combined in the post-process to gain the entire pressure distribution for both sides of the car. However, this procedure does not entirely capture the asymmetric flow field over the rear window and the upper trunk [3]. Furthermore, Nayeri [4] shows that the differences between the reference pressure taps and the corresponding taps on the other car side is about 10 Pa due to the asymmetric flow pattern at the backlight.

In the present study 60 pressure taps at the backlight, the upper trunk, and on the roof are used. Furthermore, six pressure taps are located at the base of the vehicle. The red markers in Figure 3 illustrate the positions of the used pressure.

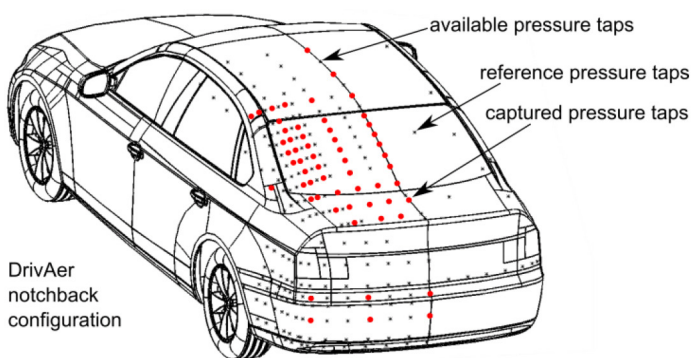


Figure 3. Pressure taps located at the DrivAer model. Red markers indicate the measured sensor positions.

The pressure is measured simultaneously with time-resolved differential pressure sensors. The reference pressure is given by the Pitot tube (Figure 1). The piezo-resistive sensors are installed inside the model and acquire a differential pressure range of $\Delta p = \pm 1000$ Pa. Each sensor has its own power supply and amplifier. The maximum error of the sensor is less than 0.5% full scale. All sensors are calibrated a priori with a given calibration error of 0.5% full scale. The sampling rate is $f = 4096$ Hz with a total measurement time of eight seconds. Distortions in frequencies and amplitudes are negligible due to the tubing length of approximately 40 mm between the piezo-membrane of the sensor and tap orifice on the surface, [6].

Particle Image Velocimetry

Stereo particle image Velocimetry (PIV) is employed to identify the effect of the fluidic oscillators on the rear end section and the wake of the vehicle, see Figure 2. Both cameras as well as the laser are mounted to a traverse system. Hence, only one calibration is needed for the measurements. Two x-z-planes in the plane of symmetry at $y = 0$ mm are measured with an overlap of approximately 150 mm. The results are merged afterwards. The air is seeded with silicon oil droplets of approximately 1 μ m in nominal diameter. The time resolution of the PIV system is $f = 5$ Hz. The dual cavity Nd:YAG laser shoots with 170 mJ per pulse and a wave length of 532 nm. The pulse distance is 20 μ s. The approximately 2 mm thick laser light sheet originates from above. Due to shading by the model, no velocity data is available under the model. The resolution of the 14 bit CCD-

cameras is 2048×2048 pixels. The viewing angle of the cameras to the laser light sheet is approximately $\pm 40^\circ$. 1100 pairs of images are acquired for each configuration to ensure convergence of the mean values. A background subtraction eliminates errors caused by reflections. The grid refinement has an initial size of 96×96 pixels and a final size of 48×48 pixels with an overlap of approximately 60%.

Fluidic Oscillators

Figure 4 depicts a CAD model of the used fluidic oscillators. The fluidic oscillator emits an in-plane sweeping jet with a sweeping jet angle of $\theta \approx \pm 40^\circ$. This is due to unsteady flow phenomena inside the main chamber. This angle has proven to be constant for various mass flow rates. However, the sweeping jet frequency depends on the geometry and the mass flow rate and varies between 10 Hz and 20 kHz.



Figure 4. CAD model of the CNC machined main plate of the fluidic oscillator and pressure tube connector.

The outer dimensions of the used actuator are $L = 22$ mm, $W = 16$ mm, and $H = 4.7$ mm (Figure 4). The area of the exit orifice covers $A_{\text{jet}} = 1.3$ mm $\times$ 1.72 mm. The actuators are CNC machined out of an aluminum plate and glued together with a thin cover sheet. The two clearance holes in the fluidic oscillator are used to attach the devices to the model. Each oscillator is connected to a 4 mm tube which supplies the system with an overpressure of 1-3 bar. This corresponds to a mass flow rate per actuator of $m = 0.2$ -3 kg/h.

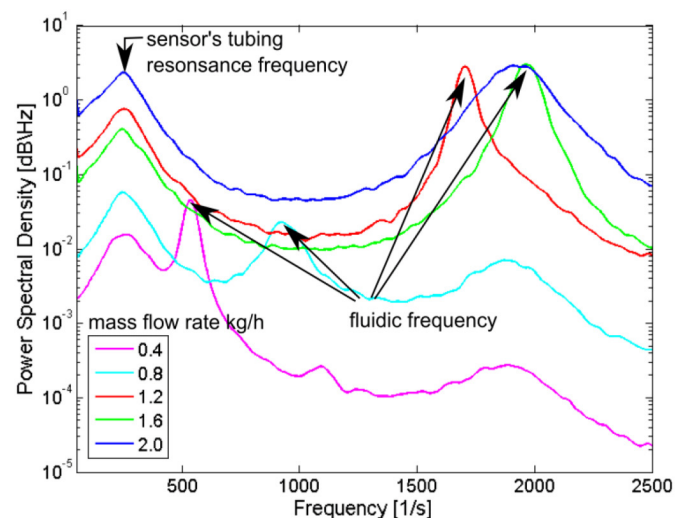


Figure 5. Oscillation frequency behavior depending on the mass flow rate.

The frequency of the sweeping jet varies due to the change in mass flow rate. Figure 5 illustrates the power spectral density of the used devices depending on the mass flow rate between $\dot{m} = 0.2\text{--}2 \text{ kg/h}$. The frequency of the sweeping jet increases from $f \approx 600 \text{ Hz}$ ($\dot{m} = 0.4 \text{ kg/h}$) up to $f \approx 1900 \text{ Hz}$ ($\dot{m} > 1.6 \text{ kg/h}$). Generally, the frequency is almost linearly proportional over a wide range of the mass flow rate. This frequency behavior is well known and documented in [12, 16]. A qualitative visualization of the sweeping jet located on the roof in a quiescent condition is presented in Figure 6. The jet affects a wide spatial area downstream of the actuator. Furthermore, secondary flow structures are induced by the jet. The literature suggests that the induced vorticity by the jet has a higher impact than induced kinetic energy [12, 13, and 14].

For the measurements, eleven fluidic devices are located in the DrivAer model at the edge between the rear window and the roof. Additionally, seven devices are installed at each c-pillar. Hence, overall 25 fluidic oscillators are fitted inside the model. A qualitative illustration of the position of the fluidic actuators is depicted in Figure 7. Adjacent actuators are spaced evenly $s = 20 \text{ mm}$. A doubled spacing of the fluidic devices is obtained by disconnecting every other one. Based on the length of the DrivAer the non-dimensional spacing is $s^* = 0.017$ and $s^* = 0.035$.

The actuators located at the edge of the roof are aligned approximately 30° to the inflow velocity c_∞ . Meanwhile, the actuators at the c-pillars are aligned towards the plane of symmetry. The angle between the emitted sweeping jet and the plane of symmetry is approximately $15^\circ\text{--}25^\circ$. The orientation of the actuators is limited due to manufacturing reasons such as convex surfaces, wall thickness, and pressure taps. However, the orientation of the fluidic oscillators is chosen in that way that the jet emits as much in the direction of the local flow as the constructional constraints allow.

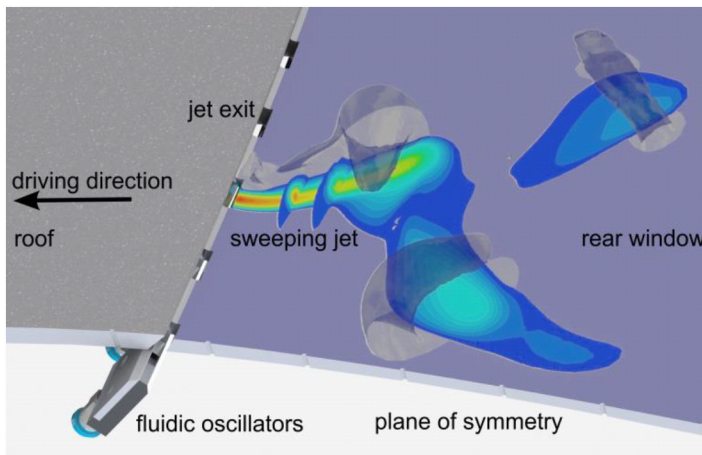


Figure 6. Visualization of the velocity distribution emitted by the fluidic jets and of the secondary flow structures.

Hose System

A schematic overview of the active flow control setup is shown in Figure 7. The necessary compressed air is supplied externally by a compressor. A digital mass flow controller with an operating range between $0\text{--}100 \text{ kg/h}$ controls the supply. The maximum error of the digital mass flow controller is 0.5% full scale. Furthermore, two Festo™ pressure proportional valves (max. error 0.5% full scale) are installed to distribute the mass flow separately into two equally designed hose systems. The regulation of the Festo™ proportional valves changes the pressure and hence, the mass flow distribution inside the hose systems. This hose setup offers the opportunity to change the actuation strength and position as an operational process without modifying the model. Two pressure distributors are located inside the model: One supplies the actuators located at the c-pillars meanwhile the other one supplies the actuators on the roof. All pipes inside the model are equally constructed in terms of tubing length ($L = 150 \text{ mm}$) and inner diameter ($\varnothing = 2.6 \text{ mm}$). Hence, the flow losses are similar for all tubes by neglecting individual tubing bending.

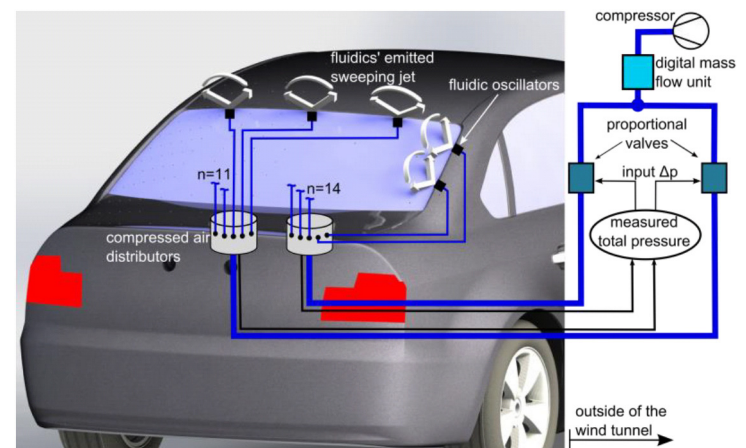


Figure 7. Schematic CAD- illustration of the external and internal hose system of the active flow control with the emitted sweeping jet of the fluidic oscillators.

Non-Dimensional Momentum Coefficient

For the calculation the momentum coefficient, the frontal area of the DrivAer model A_{DrivAer} and the freestream velocity of $c_\infty \approx 42 \text{ m/s}$ are used

$$C_{\mu, \text{act}} = \frac{A_{\text{jet}}}{A_{\text{DrivAer}}} \cdot \left(\frac{u_{\text{jet}}}{c_\infty} \right)^2. \quad (2)$$

The density of the jet at the orifice of the fluidic oscillator is assumed to be equal to the ambient conditions. Hence, the momentum coefficient has a quadratic relationship with the mass flow rate. The momentum coefficients for a single fluidic oscillator and various mass flow rates are listed in Table 1.

Table 1. Non-dimensional momentum coefficient for a single oscillator $c_{\mu,1act}$ calculated for the mass flow rates of 1, 2, and 3 kg/h.

Mass flow rate	$\dot{m} = 1.0 \text{ kg/h}$	$\dot{m} = 2 \text{ kg/h}$	$\dot{m} = 3 \text{ kg/h}$
momentum coefficient	$C_{\mu,1act} = 2.0 \cdot 10^{-4}$	$C_{\mu,1act} = 8.0 \cdot 10^{-4}$	$C_{\mu,1act} = 1.8 \cdot 10^{-3}$

All fluidic actuators are supplied with the same mass flow rate at any given time to achieve a coinciding frequency, jet velocity, and momentum coefficient per actuator for all actuators. However, the overall momentum coefficient $C_{\mu,all}$ varies due to the changing amount of total actuators per test configuration.

Therefore, the actuation only at the c-pillars with 14 fluidic oscillators has the highest overall momentum coefficient $C_{\mu,all} = 2.5\%$. Meanwhile the momentum coefficient for the actuation only on the roof is $C_{\mu,all} = 2\%$. A combination of both cases yields a value of $C_{\mu,all} = 4.5\%$.

Power Consumption of the Active Flow Control

The specific enthalpy h of a compressor for ideal gas is a function of the total air temperature T_t and the specific heat capacity $c_p = 1004.5 \text{ J/kgK}$:

$$\Delta h = c_p \cdot \Delta T_t = \frac{P}{\dot{m}}. \quad (3)$$

Thereby, the relation between the total air temperature T_t and the total pressure p_t for an isentropic flow is defined as

$$\frac{T_{t2}}{T_{t1}} = \left(\frac{p_{t2}}{p_{t1}} \right)^{\frac{\kappa-1}{\kappa}}, \quad (4)$$

with the adiabatic index of $\kappa = 1.4$. The energy efficiency of the compression process is assumed to have no losses. Table 2 lists the corresponding power consumption P for a single fluidic oscillator depending on the mass flow rate. The total pressure p_{t1} and temperature T_{t1} are simplified and approximately equal to the ambient conditions. The total pressure p_{t2} is measured during the experiment in the pressure distributors (Figure 7). This ensures that the flow losses between the pressure distributors and the orifice of the fluidic oscillators are taken into account.

Table 2. Calculation of the power consumption of an isentropic compressor without losses.

Mass flow rate	Operating pressure during the experiment	Energy consumption for a single fluidic oscillator
$\dot{m} = \frac{1 \text{ kg}}{h} = 2.8 \cdot 10^{-4} \frac{\text{kg}}{s}$	$p_{t2} = 1.5 \cdot p_0$	$P = 10 \text{ W}$
$\dot{m} = \frac{2 \text{ kg}}{h} = 5.6 \cdot 10^{-4} \frac{\text{kg}}{s}$	$p_{t2} = 2.65 \cdot p_0$	$P = 55 \text{ W}$
$\dot{m} = \frac{3 \text{ kg}}{h} = 8.3 \cdot 10^{-4} \frac{\text{kg}}{s}$	$p_{t2} = 3.85 \cdot p_0$	$P = 120 \text{ W}$

Table 2 shows that the power consumption for a single actuator increases significantly for higher mass flow rates. Moreover, a higher operating pressure is needed, if the mass flow rate increases. That corresponds to increased flow losses due to higher velocity in the entire hose system. The overall power consumption for the actuation at the c-pillars and on the roof for a mass flow rate of $\dot{m} = 3 \text{ kg/h}$ is $P = 3 \text{ kW}$.

Flow Visualization with Tufts

A novel method of tuft flow visualization is being developed at the Hermann-Föttinger-Institute (HFI). This new method offers the opportunity to extract the mean flow angle and the deviation angle of the tufts over a period of time. From this data, the surface trace pattern can be extracted. Furthermore, statistical analyzes as well as flow field decompositions are obtainable [17]. The application of tufts is a simple method without further contamination of the wind tunnel compared to other flow visualizations as e.g., oil-paint. This may be a huge advantage especially in modern wind tunnel tests.

The DrivAer model is equipped with a large amount of woolen tufts at the rear end. The tufts are carefully selected based on the tuft thickness, length, color, and stiffness. The length of the tufts is about 25 mm and the diameter is about 1 mm.

For the application of statistical tools, a sufficient amount of high resolution pictures per test case are recorded, processed and analyzed. The Figure 8 illustrates an example of a post-processed single image. The red dots shown are required markers for the algorithm to discover the starting point of the tufts. The algorithm analyzes each tuft for every picture and calculates the mean tuft angle with an interpolation onto an equidistant grid and performs a line integral convolution (LIC). Afterwards, the picture is projected onto the CAD-DrivAer model.

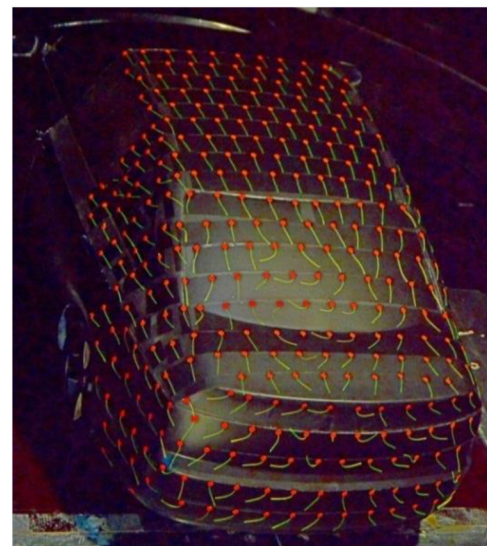


Figure 8. A single snapshot of the flow visualization with tufts at the notchback in the wind tunnel.