

The Aerodynamic Development of the New Audi Q5

Thomas Blacha and Moni Islam
 Audi AG

ABSTRACT

The aerodynamic development of the new Audi Q5 (released in 2017) is described. In the course of the optimization process a number of different tools has been applied depending on the chronological progress in the project. During the early design phase, wind tunnel experiments at 1:4 scale were performed accompanied by transient DES and stationary adjoint simulations. At this stage the model contained a detailed underbody but no detailed engine bay for underhood flow. Later, a full scale Q5 model was built up for the aerodynamic optimization in the 1:1 wind tunnel at Audi AG. The model featured a detailed underbody and engine bay including original parts for radiators, engine, axles and brakes from similar vehicles. Also the 1:1 experiments were accompanied by transient DES and stationary adjoint simulations in order to predict optimization potential and to better understand the governing flow. The strong coupling of experimental and numerical tools enabled a best-in-class drag coefficient to be achieved not only for a single optimized vehicle setup but for the majority of four cylinder diesel and gasoline engines in base configuration even in SUV ground clearance (as required for LDT). This is not only true if the vehicle is equipped with the aerodynamically optimized "aero wheel" but also a number of optional rim designs allow comparably low drag values. In order to reach such low aerodynamic drag values, a variety of design and technical measures were necessary. A description of their definition and function is subject of the present paper.

CITATION: Blacha, T. and Islam, M., "The Aerodynamic Development of the New Audi Q5," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(2):2017, doi:10.4271/2017-01-1522.

INTRODUCTION

During the last years the aerodynamic optimization of passenger cars has gained great importance. This is not only due to the fact that the maximum velocities of passenger cars have been increased but also to the fact that the topic of CO₂ driven climate change has become so prominent. In addition, the technological development of experimental techniques as well as computer based simulation tools have undergone great progress. On the one hand, automotive wind tunnels for aerodynamic drag and lift measurements have significantly increased in size to reduce wind tunnel interferences and they include a number of improvements as far as measurement systems are concerned. On the other hand, CFD has reached a state of accuracy where it significantly supports the aerodynamicist in understanding the flow field around the car.

Since highly accurate CFD calculations come along at a high computational cost, a great part of the improvement is due to an extreme increase of computer resource disposability during the last decades. Another reason for the improvement is the development of more sophisticated turbulence models. This includes the elaboration of various RANS, Reynold stress and LES models but also hybrid approaches such as zonal approaches, PANS and DES. In particular, hybrid LES/RANS models belong to the most promising approaches for industrial applications since on a given grid they offer a high resolution of turbulent scales at moderate computational cost.

Therefore, the modeling error of turbulent mechanisms is kept at a minimum. A detailed overview of the most commonly used hybrid LES/RANS models can be found in Fröhlich et al. [1].

The overall computational process to simulate the external flow around a passenger car includes a number of methods starting with mesh creation and ending with a post process of the results in order to visualize the data. The applicability of all of those methods in an industrial environment does not only depend on accuracy constraints but also on the overall turnaround time. Therefore, a fully automatized calculation process has been developed at Audi as presented by Islam et al. [2]. It is based on transient DES calculations on an automatically generated hexaeder dominated grid for the prediction of drag and lift using the OpenFOAM framework. In addition, an adjoint process has been elaborated as demonstrated by Blacha et al. [3, 4] in order to calculate drag and lift sensitivities on the vehicle surface. These sensitivities indicate the gradient of the corresponding aerodynamic force with respect to a surface normal displacement. Therefore, they represent a prediction of force gain or loss if the surface would be displaced in either direction. A detailed description of the underlying theory for the applied adjoint method can be found in Othmer [5].

A brief report on the aerodynamic properties of the new Audi Q5 was first presented in Blacha et al. [6]. For the aerodynamic development of this car a strong coupling of wind tunnel measurements and CFD-calculations including adjoint predictions has been applied. This allowed for the achievement of a class leading drag value of C_d = 0.31.

0.30 for most four cylinder diesel and gasoline engines even in SUV ground clearance (as required for LDT) if equipped with the aerodynamically optimized “aero wheel”. A number of other optional rim designs which increase the aerodynamic drag of the vehicle only by few drag counts will also be presented in this paper.

AERODYNAMIC OPTIMIZATION PROCESS

The aerodynamic optimization of the Q5 included a large number of development stages. At each stage a variety of parameters have been analyzed leading gradually to the final aerodynamically optimized vehicle shape. In the following, this optimization process will be described based on two exemplary development stages:

- The optimization of an early design will be described using CFD and a 1:4 scaled clay model in the model wind tunnel of the FKFS in Stuttgart, Germany.
- The further optimization of an advanced design, which was used for the first 1:1 scale clay model, will be demonstrated.

The evolution of the design throughout the vehicle history is displayed in [Fig. 1](#).



Figure 1. Evolution of vehicle design. left: design of 2012, middle: first 1:1 clay model, right: final car.

Early Design Optimization using CFD and 1:4 Clay Models

The first design of the new Q5 was presented by the Audi styling department in 2012. In order to obtain an aerodynamically optimized vehicle shape, it was very important to influence this new car design as early as possible.

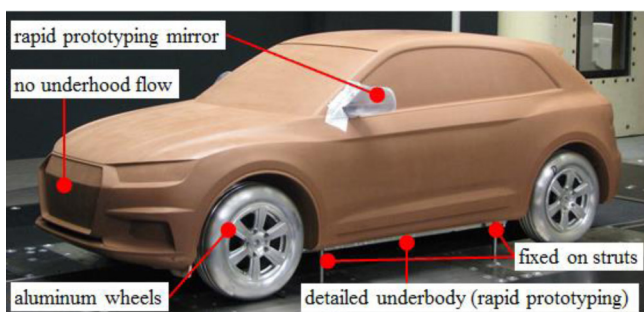


Figure 2. 1:4 scale clay model in the model wind tunnel of the FKFS

The first step in the development chain was the preparation of the CAD data. Based on this data first CFD-simulations were run in order to get an impression of the general flow field around the car. These simulations were then supported by 1:4 wind tunnel measurements at the model wind tunnel of the FKFS in Stuttgart (Germany) [7]. The models used were clay models with detailed underbody and rotating wheels but without underhood flow as shown in [Fig. 2](#).

One of the challenges of the aerodynamic optimization in the wind tunnel is the elaboration of drag reduction measures within a limited time frame. In order to improve the efficiency of this task, adjoint calculations have been performed before the wind tunnel testing. These were based on a time averaged transient DES primal solution and the stationary continuous adjoint approach as described in Blacha et al. [3, 4]. The resulting drag sensitivity maps could then be used for a faster definition of drag reduction measures. An example of these sensitivity maps is shown [Fig. 3](#) for the design which had to be optimized in early 2013. In order to reduce drag, blue areas need to be displaced inwards and red areas need to be displaced outwards.

White areas are indifferent and the black line indicates the line of zero drag sensitivity.

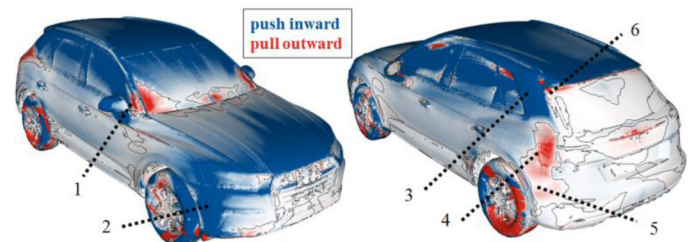


Figure 3. Sensitivity maps on Q5-model and identification of drag reduction measures.

The first mark in [Fig. 3](#) shows a red region in the base of the A-pillar suggesting that the A-pillar should be more rounded in this area. Also the blue regions in the upper part of the A-pillar concentrated around the edges support this finding. This measure showed a drag reduction potential of 2 counts ($\Delta C_d = -0.002$).

The second mark highlights a blue region around the front bumper. Apparently, the front bumper should be less rounded in order to obtain an ideal flow around the wheels. However wind tunnel measurements revealed that the bumper was already ideal. No improvement could be obtained in either way.

The third mark shows a dark blue region on the D-pillar suggesting that boat tailing might be very effective. Therefore, most of the wind tunnel time was spent to find an ideal D-pillar. Here, a drag reduction of 5 counts could be obtained even without touching the rear side windows in favor of the interior package.

The fourth mark concentrates on the rear lights where an outward displacement is supposed to improve drag. This can be realized by a rear light spoiler, for example. In the final Q5 a rear light spoiler was indeed included leading to a drag reduction of 2 counts.

Mark 5 indicates that an outward displacement behind the rear wheels may reduce drag. This finding was confirmed during the measurements with a drag reduction potential of 1 count. However, the required change was not accepted for the final shape due to styling reasons.

The small red region indicated by mark 6 suggests a better separation edge at the outmost left and right corners of the rear spoiler. This change has been realized and led to a drag reduction of further 2 counts.

Every 1:4 measurement campaign concerning the new Q5 was supported by a design representative leading to a very efficient and slim aerodynamic design optimization process. The aerodynamicist could explain his needs to the design engineer and the latter was able to directly suggest a styling in order to meet aerodynamic and styling constraints. In addition, all major measures with drag reduction potential were reported to the Q5 project management with special emphasis on measures where no compromise was found between aerodynamic and styling department. This strategy was used to enhance a critical review of the technical and design feasibility of all drag reductions measures. Based on the very first styling proposal a drag reduction of roughly 40 counts was realized during the 1:4 optimization phase.

In the course of the early vehicle development there is always a large number of design modifications that are continuously being released. In consequence, during the development of the new Q5 only a small selection of intermediate designs could be treated in 1:4 wind tunnel measurements. All other design modifications during this development stage were first visually inspected and, if needed, evaluated by CFD.

First Full Scale Clay Model with Detailed Underbody and Engine Bay

As the construction of the first full scale clay model with detailed underbody and engine bay was finished, most regions of the upper outer shape were fixed within a small millimeter tolerance range. Massive geometry changes were no longer possible. Therefore, the focus was now on the following topics:

- Fine tuning of rear spoiler and D-pillar separation edge
- Definition of rear light spoiler
- Fine tuning of side grills
- Rim design
- Underbody
- Underhood flow (including active grill shutter)
- Lift prediction



Figure 4. First full scale clay model.

In Fig. 4 the first full scale clay model is demonstrated. All aerodynamic optimization measures for this model have been elaborated in the Audi wind tunnel [8] with ground simulation (5 belt system with boundary layer suction). Again, these measurements were strongly supported by DES and adjoint simulations. In the beginning, the aerodynamic optimization concentrated on wheel deflectors, side grills, D-pillar and cooling air flow.

One of the most challenging tasks for CFD is an accurate prediction of the impact of rotating wheels on the aerodynamic forces. In the literature numerous studies on this topic can be found [9, 10, 11, 12]. Nevertheless, until today no complete method exists which is able to correctly capture all relevant aerodynamic mechanisms caused by wheel rotation. For the rotation of the rim good progress has been made starting with simplified models such as rotating wall boundary and ending with a complete mesh motion for the rim such as sliding mesh. However, a complete description would have to include tire deformation, tire ground contact and the impact of tire surface (e.g. tread or label) which is still subject to extensive research.

For numerical simulations within the current work the sliding mesh boundary was not available, yet. Therefore, the rotating wall boundary was used to model the rotation of rim and tire. In consequence, the numerical prediction of the flow field around the wheels is expected to show drawbacks in accuracy. MRF (moving reference frame) boundary conditions - often motivated by the work by Waeschle [13] - were not considered, since at Audi no gain in accuracy was found as compared to rotating wall boundary. In particular, it was found that aerodynamic forces significantly depend on the choice of MRF volume size which is to some extent arbitrary. The problem of additional uncertainties introduced by the MRF approach is also addressed by Landstroem et al. [14].

For these reasons, the aerodynamic optimization of the front wheel deflectors was mainly driven by experiments. Due to the high aerodynamic potential of front wheel deflectors, they have been aerodynamically optimized by a very detailed geometrical study in the wind tunnel. In total 20 hours of wind tunnel time were spent leading to a final front wheel deflector with a drag reduction of more than 10 drag counts as compared to the original geometry.

The original side grills of the model featured a deep pocket. This was requested by the styling department in order to obtain distinct three dimensional features. But, since the outer step led to a strong separation of the flow as shown in Fig. 5, it would have resulted in a drag loss of 13 counts as compared to perfectly smooth side grills. Therefore, the requested design feature was rejected and the objective of a maximum outer step size of 15mm was defined. This would allow the flow to remain attached until the front wheels without any additional drag loss.

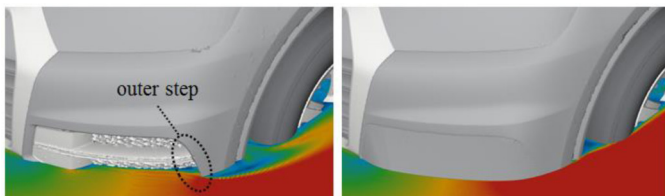


Figure 5. Computed velocity magnitude for original side grill styling request (left) and for perfectly smooth side grills (right). (blue to red: 0-40m/s)

A strong focus for the aerodynamic drag reduction was on cooling drag. First a tight target has been fixed for cooling air mass flow together with the department responsible for cooling systems. Then the flow through the underhood had to be optimized. Seals from the grill to the radiator allowing for more efficient cooling air use are commonly applied by the majority of passenger vehicle producers. In addition, air shields in the wheel housing were added to reduce the cooling air mass flow exiting through the wheel house. This leads to an elevated mass flow through the tunnel. For the optimal cooling drag, the cooling air should exit the tunnel tangential to the outer flow and the interactions with the outer flow should be kept at a minimum [15, 16]. Therefore, a tunnel cover was applied as demonstrated in Fig. 6. This cover also helped to reduce pressure peaks caused by underbody parts extending into the outer underbody flow. Nevertheless, it can be observed that the cover leads to increased tunnel pressures which in turn result in a reduced cooling air flow but also in elevated cooling air leakages. This is mainly due to the fact that the first muffler is partially blocking the cooling air flow in the tunnel. Consequently, the full potential of this concept could not be realized. Instead, the first muffler was moved further downstream and given a more streamline shape. The resulting position and shape of the exhaust system allowed for a further reduction in cooling air drag, as will be shown in the description of the final vehicle.

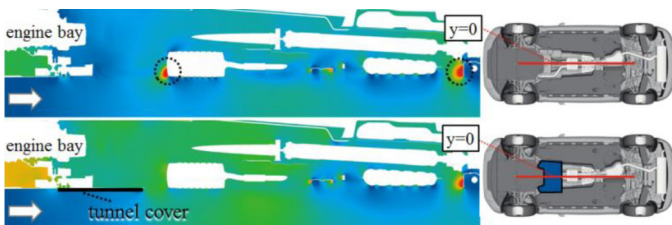


Figure 6. Cp distribution in $y=0$ plane of the tunnel for base configuration (upper) and with tunnel cover (lower). (blue to red: -0.3 to 0)

In order to optimize the flow around the rear underbody region a cover has been applied as demonstrated in Fig. 7. It is an important element of the rear diffuser guiding the flow from the tank to the final muffler. In the final vehicle this part is also used for stone chip protection.

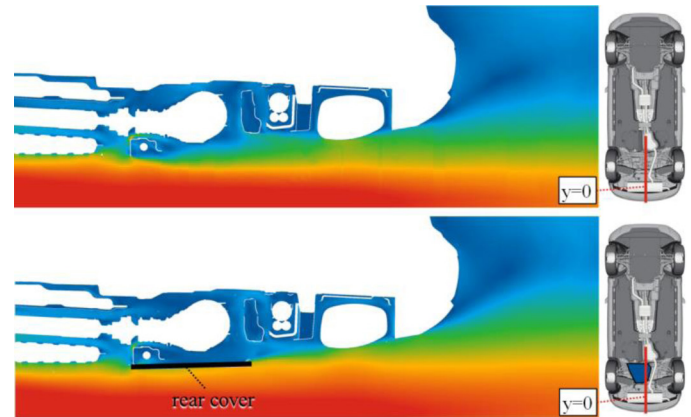


Figure 7. Distribution of velocity magnitude in $y=0$ plane in the rear for base configuration (upper) and with rear cover (lower). (blue to red: 0-40m/s)

Table 1. Drag and lift predictions for wind tunnel (WT) and CFD.

	ΔC_d		ΔC_{lr}		ΔC_{lr}	
measure	WT	CFD	WT	CFD	WT	CFD
cooling drag	0.018	0.015	0.065	0.040	-0.065	-0.054
side grill pocket	0.013	0.013	0.000	-0.017	0.008	0.000
tunnel cover	-0.004	-0.003	0.000	-0.001	-0.002	0.005
rear u.b. cover	-0.002	-0.001	0.001	-0.007	0.001	-0.005

A number of the above measures have been calculated by CFD and are compared to wind tunnel experiments in Table 1. Good agreement is found in drag predictions for all measures. Lift predictions are off for the side grill pocket. The reason for this discrepancy is not clear, but it may be due to interactions of the flow with the rotating front wheels further downstream. For all other cases the trend on lift forces is captured correctly (respecting that lift predictions by CFD feature higher uncertainties than drag predictions).

The aerodynamic optimization of the Audi Q5 is not only based on the two configurations shown above but on a great number of them. Even after the first prototypes appeared, important aerodynamic optimizations have been carried out and were fed to the project. The chronological development of the aerodynamic drag throughout the project is shown in Fig. 8. For the 1:1 scale clay model drag values smaller than 0.30 are found. These are due to the fact that the impact of gaps cannot correctly be represented in a clay model and that some aerodynamic demands could not be realized for the final vehicle.