

A Numerical and Experimental Evaluation of Open Jet Wind Tunnel Interferences using the DrivAer Reference Model

Christopher Collin, Steffen Mack, and Thomas Indinger
Technical University of Munich

Joerg Mueller
AUDI AG

ABSTRACT

The open jet wind tunnel is a widespread test section configuration for developing full scale passenger cars in the automotive industry. However, using a realizable nozzle cross section for cost effective aerodynamic development is always connected to the presence of wind tunnel effects. Wind tunnel wall interferences which are not present under open road conditions, can affect the measurement of aerodynamic forces. Thus, wind tunnel corrections may be required. This work contains the results of a CFD (Computational Fluid Dynamics) approach using unsteady Delayed Detached Eddy Simulations (DDES) to evaluate wind tunnel interferences for open jet test sections. The Full Scale DrivAer reference geometry of the Technical University of Munich (TUM) using different rear end shapes has been selected for these investigations. Experimental results from measurements of the 40% scaled wind tunnel model confirm the relevance of wind tunnel effects for different blockage ratios and test section characteristics at the wind tunnels of the TUM and Audi. The DrivAer geometry has been applied to CFD to simulate open road conditions in order to validate the aerodynamic forces with experimental data as well as set a reference for wind tunnel simulations. The simulation of the open jet test section including the DrivAer model has been applied to wind tunnels with nozzle cross sections between 11.0 m² and 89.6 m². The results are then compared to potential flow models by Wickern as well as Mercker and Cooper. Finally, conclusions are derived for the selected geometries in test sections with the plenum and the nozzle method.

CITATION: Collin, C., Mack, S., Indinger, T., and Mueller, J., "A Numerical and Experimental Evaluation of Open Jet Wind Tunnel Interferences using the DrivAer Reference Model," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(2):2016, doi:10.4271/2016-01-1597.

INTRODUCTION

Building upon the first investigations of wind tunnel wall interferences by Prandtl in 1919 [1] and later e. g. Garner [2] and Ewald [3], almost a hundred years of research has been focused to achieve better knowledge about the effects in both open and closed wind tunnel test sections. Likewise, today there are a number of methods to account for these effects in full scale open jet test sections for aerodynamic development of passenger cars. For example, the methods of Wickern et al. [4,5,6,7] and Mercker et al. [8,9,10,11,12] are based on potential flow models and partially include different interference effects between car and wind tunnel boundaries. Hence, different results can be expected for specific configurations of wind tunnel test sections. Nowadays, computational power has achieved a level which enables unsteady DDES of cars in open jet test sections at realistic Reynolds numbers.

Compared to a number of strongly simplified reference models [13], the DrivAer reference geometry [14] was designed in order to cover the geometric properties of a typical passenger car in a more detailed way. This work contains results of wind tunnel tests of the 40% scaled DrivAer model [15] in wind tunnels of the TUM [16] and Audi [17]. Due to differences such as solid blockage, static pressure

gradient and moving ground simulation, measurements in those wind tunnels sometimes produced different results in terms of aerodynamic forces. Based on the experimental results, the setup of the DDES using the full scale DrivAer geometry has been validated and developed further into combined wind tunnel and DrivAer simulations. This work focuses on interferences between the wind tunnel nozzle, open jet and car model including different rear ends. In order to isolate these effects, the test section geometry was set up to allow the assumption that influences of collector, static pressure gradient, moving ground simulation and plenum dimensions are negligible. However, for an adequate transferability to a real wind tunnel test section, a full modeling of all test section characteristics is required. Thus, this study provides a better knowledge of interferences between car, nozzle and open jet.

Since the proximity of the model with significant blockage ratios produces interference effects with the wind tunnel boundaries for both the plenum and nozzle method [18], the requirement of the application of wind tunnel corrections seems plausibly justified. Therefore, Wickern [7] proposes an approach for a self-correcting open jet wind tunnel by considering interference effects between nozzle and model which can be balanced in order to obtain the

correct drag coefficient. Wickern's potential flow model predicts the compensation of drag-increasing effects (nozzle blockage, model induced nozzle gradient) and drag decreasing effects (solid blockage) at different longitudinal model positions for wind tunnels operating with either the plenum or the nozzle method. To cover the physics of a "real" open jet test section in a more detailed way and to determine its effects on a realistic car geometry, CFD was applied to Wickern's approach for blockage ratios of $A_M / A_N = 0.024 \dots 0.197$. The results were compared to the two potential flow models of Wickern [7] and Mercker and Cooper [8]. An essential advantage of CFD is that the effects on lift forces can be determined as well. However, lift force corrections are typically not covered by classical potential flow theory since the applied models are based on duplex mirror images. Based on the results, some consequences for the design of open jet test sections have been derived. However, it should be noted that general conclusions covering the entire range of designs for automotive vehicles cannot be reliably drawn here since the results of the present study are strongly connected to the selected geometries.

EXPERIMENTAL SETUP OF DRIVAER MEASUREMENTS

The DrivAer Geometry

For the experimental investigations we used the DrivAer reference geometry which emerged from a collaboration between the TUM and industry partners [14]. The DrivAer geometry represents a generic, yet detailed vehicle geometry aiming to close the gap between very idealized vehicle geometries for principal investigations (like the Ahmed body [19] and the SAE body [20]) and very detailed vehicle geometries. It contains most of the features that dominate the flow field around a typical mid-sized mass production passenger car. The very modular DrivAer geometry includes three different rear ends - a fast back, a notch back and an estate back (station wagon) shape - and two underfloor geometries, including a completely smooth and a detailed version that features more geometrical details like an exhaust system. Figure 1 illustrates the geometrical set that can be obtained at the Institute of Aerodynamics and Fluid Mechanics of the TUM free of charge.



Figure 1. DrivAer geometry with three different rear ends: fast back (red), notch back (white) and estate back (blue) - as introduced in [14].

Since its introduction, the DrivAer geometry has been subjected to numerous published research projects performed by universities and within the industry throughout various countries.

The 40% Scaled DrivAer Wind Tunnel Model

The wind tunnel model that was employed in the present study offered the same geometrical flexibility as the basic DrivAer geometry (see also Mack et al. [15] for details). It was possible to exchange the rear end panel, the underfloor inset and the wheel/rim geometry without major effort. In combination with an underhood extension (not part of the basic DrivAer geometry) to also simulate cooling air flow, this provided a multitude of possible combinations. Figure 2 depicts a CAD scheme of this 40% scaled wind tunnel model in one particular configuration and of the top sting holding the model's body in place.

The outer shell of this wind tunnel model was made of carbon fiber mounted on an aluminum spine, which resulted in a fairly lightweight design. Underfloor inserts made of polyurethane provided additional stiffness without adding too much weight. Throughout the process of designing this wind tunnel model, special care was taken to provide plenty of space for the housing of measurement technology as well as space inside the engine compartment for adding a model radiator and ducts for cooling investigations. Even though the design of this wind tunnel model was very simple compared to wind tunnel models used in the industry, it greatly served and still serves its purpose in various research projects at the Institute of Aerodynamics and Fluid Mechanics at the TUM.

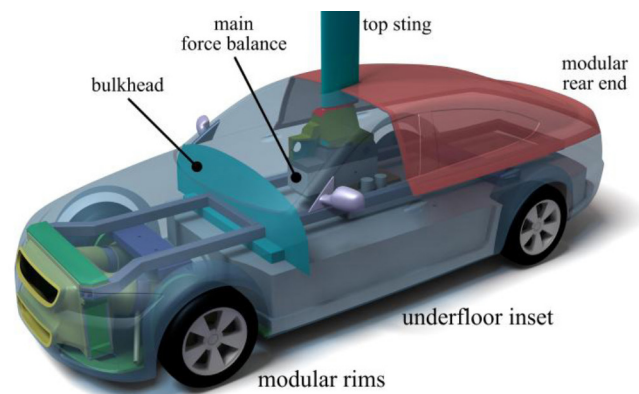


Figure 2. CAD scheme of the 40% scaled DrivAer model as introduced in [15].

In the present study, we picked model configurations with lower geometric complexity; we installed the smooth underfloor inset and closed the cooling inlets but exchanged the rear end shapes between the configurations fast back, notch back and estate back. Furthermore, the wheels used in this study had sealed rims surrounded by non-pneumatic and non-deformable tires made of carbon fiber. These simplifications led to a less complex flow field around and through the vehicle and, hence, significantly reduced the complexity of the numerical modeling.

Table 1 lists the most important geometrical properties of the wind tunnel model including the model's frontal area to determine its solid blockage. The exact value of the solid blockage depends on the chosen configuration (see separate contributions). The blockage ratio totaled to about 11.7% in the WTA and was well below the recommended value of 10% in the AAWK (see next section).

Table 1. Dimensions of the 40% scaled DrivAer wind tunnel model [15].

Size (Length, Width, Height)	1.8451 m, 0.7280 m, 0.5674 m
Wheel Base	1.1145 m
Track Width (Front, Rear)	0.6082 m, 0.6108 m
Wheels (Diameter, Width)	0.258 m, 0.091 m
Frontal Area (Without Rear End) Body + Wheels + Underfloor	342.75 10^{-3} m ²
Additional Frontal Area for Rear End: (Fast Back, Notch Back, Estate Back)	3.09 10^{-3} m ² , 3.61 10^{-3} m ² , 3.79 10^{-3} m ²
Additional Frontal Area due to Top Sting and Wheel Struts	141.43 10^{-3} m ²

Please keep in mind that the nozzle exit area of the WTA was reduced by the inlet area of the boundary layer scoop - 144 10^{-3} m² - that "cuts off" the lower part of the flow through the nozzle. Furthermore, the blockage effects of all struts were considered as well. Excluding the struts, the solid blockage would have totaled to about 8.3% in the WTA.

Table 2. Dimensions of the employed wind tunnel facilities as published in [1] and [16].

	WTA at TUM	AAWK at AUDI
Nozzle Size (Height x Width)	1.80 m x 2.40 m	2.79 m x 3.94 m
Cross Section of Nozzle Exit	4.32 m ²	11 m ²
Contraction Ratio	7:1	5.5:1
Length of Test Section	4.80 m	9 m – 10 m (variable)
Cross Section of Collector Inlet	7.54 m ² (max., variable)	37.5 m ²
Max. Inflow Velocity (3/4 Open Test Section)	65 m/s	83.3 m/s
Turbulence Level	< 0.4 %	< 0.3 %
Belt Dimensions (Length x Width)	4.53 m x 1.39 m	5.5 m x 1 m
Max. Speed of Belt	50 m/s	65.3 m/s
Material of Belt	Polyester-Based	Polyester-Based
Boundary Layer Treatment	Boundary Layer Scoop (adjustable)	Boundary Layer Suction System (adjustable)
Model Position (Center of Wheel Base)	2 m Downstream of Nozzle Exit	4 m Downstream of Nozzle Exit
Displacement Thickness (Center of Wheel Base, Symmetry Plane)	1.9 mm	< 1.0 mm
Boundary Layer Thickness (Center of Wheel Base, Symmetry Plane)	29 mm	23 mm

The model was equipped with various sensors and gauges to record aerodynamic properties and to log the attitude of the vehicle. The most important measuring instruments were the set of five force balances and the multi channel pressure transducer. We employed a six-component force balance of the RUAG 188-6 series to record the aerodynamic forces acting on the body of the vehicle. This "main" balance was accompanied by four one-component wheel load cells of

the RUAG 270-1 series to measure the drag forces on the wheels. Please note that the nature of this setup did not allow for the measurement of lift forces or side forces acting on the four wheels.

Surface pressures were recorded using a pressure transducer manufactured by Scanivalve. Three ZOC33 modules featured a total number of 192 ports that could be sampled simultaneously. The pressure taps (0.3 mm inner diameter) were evenly distributed over the top surface and the center line of the underfloor of the vehicle and were connected to the ZOC33 modules via hoses of 0.8 mm inner diameter.

The Employed Wind Tunnel Facilities

We conducted experimental studies in two inherently different wind tunnels: the wind tunnel at the TUM (WTA) and the aeroacoustic wind tunnel at Audi (AAWK). The most important difference was their sizes. While the wind tunnel at the TUM featured a nozzle exit area of about 4.32 m² - and hence was only suited for model-scale testing of passenger cars - the wind tunnel at Audi had a nozzle exit of about 11 m² and was almost exclusively used for full-scale testing of typical passenger cars. Both of these wind tunnels have closed-return airlines and a 3/4-open test section. Before addressing some details of these two facilities, a quick overview table of the major properties and dimensions of these wind tunnels is provided in Table 2.

Being able to test the DrivAer wind tunnel model inside these two substantially different wind tunnel facilities was ideal for validating the theories of the present study. Figure 3 and Figure 4 depict the setup of the 40% scaled DrivAer model inside the AAWK and the WTA. We used the same method to support the model in both testing series. It consisted of five struts - one top strut (or top sting) to hold the body of the vehicle in place and four additional struts to support each wheel separately. Fortunately, both wind tunnels were equipped with traverse systems that were stiff and sturdy enough to support the vehicle (including the additional aerodynamic loads), largely easing the positioning of the model inside the test sections. Please note that in this setup the wheels were not connected to the body of the vehicle but were held only by the wheel struts.



Figure 3. 40% DrivAer wind tunnel model (with estate back rear end installed) inside the AAWK.

In the same line of thinking, this measurement technology (all balances/load cells, pressure transducers, etc.) was also used in both wind tunnels. This combination of using exactly the same wind

tunnel model, the same support struts and the same measurement technology in both tunnels significantly eased the comparison of the obtained results.

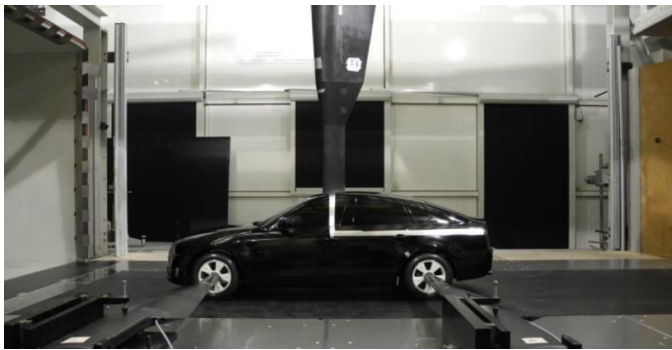


Figure 4. 40% DrivAer wind tunnel model (with fast back rear end installed) inside the WTA at the TUM.

Both of the mentioned wind tunnels were equipped with a rolling road system to simulate the relative motion between ground and vehicle and to drive the wheels of the vehicle. While the WTA was equipped with a so-called single belt system, the AAWK featured a five belt system used for full size testing. In this case, however, we only employed the long center belt - which was wider than the track width of the model - and covered the four wheel drive units.

Not only were geometrical parameters like the solid blockage associated with the differences in sizes of these two wind tunnels but also with the differences in their aerodynamic properties. For the present study, the most important aerodynamic property is the gradient of the static pressure along the center lines of the test sections as indicated in Figure 5. Furthermore, measurements of the static pressure gradient in direction of y and z were obtained in the plane $x = -1.0$ m in the WTA. The standard deviation of static pressure gradient $\Delta c_p(y, z)$ was determined to be 0.00094 within a range of $-0.4 \text{ m} \leq y \leq 0.4 \text{ m}$ and $0.02 \text{ m} \leq z \leq 0.97 \text{ m}$ respectively. Figure 5 also contains a sketch of the DrivAer model and the size of the test section of the WTA as a reference. The test section of the WTA covers the region between $x = -2.0$ m to $x = 2.8$ m (nozzle exit plane to collector inlet plane), while the much larger test section at the AAWK begins at $x = -4.0$ m and ends at $x = 5.5$ m. The position $x = 0$ m refers to the center of the wheel base of the 40% scaled DrivAer model in both tunnels. Please note that the values of the static pressure coefficients were adjusted (by a constant offset in c_p) such that they were $c_p = 0$ at the center of the wheel base of the vehicle in order to ease the comparison of the curves.

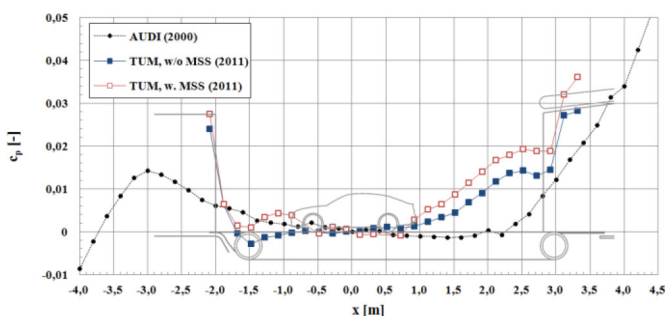


Figure 5. Pressure gradients inside the TUM model-scale wind tunnel and the full-scale wind tunnel at Audi as found in [1], [16] and [5].

Even though the pressure values plotted in Figure 5 were recorded at different z -positions¹ (at $z = 0.24$ m in the WTA and at $z = 0.60$ m in the AAWK), they illustrate the pressure distributions (or pressure gradients) present in both test sections. It is clear that the model support system (MSS) which held the model influenced the pressure gradient across the test section of the WTA. It is also apparent that the increase in static pressure by the collector started rather far upstream and was therefore likely to offset the measured drag values, producing smaller values.

In contrast to this, in the larger AAWK the pressure gradient was much flatter in the region where the model was mounted, and hence, the falsifying effect due to the so-called horizontal buoyancy was expectedly smaller. Figure 5 also particularly illustrates the effect of the boundary layer suction system that was used in the AAWK (see also the investigations by Wickern et al. [5]). However, since this happened quite far upstream of the wind tunnel model - with a pressure peak at around $x = -3$ m - its detrimental effect onto the static pressure distribution was largely reduced at the front of the vehicle model. For more details about the employed wind tunnel facilities, interested readers may refer to the publications of Wickern et al. [1] and Mack et al. [16].

Wind Tunnel Testing Procedure

Each wind tunnel run contained a “static” tare to set the pressure scanners and the force balances to zero and a “dynamic” tare at a belt speed of 5 m/s to account for the (mechanical) rolling resistance of the wheels. The use of non-deformable tires largely reduced the complexity of this procedure. For example, we did not experience a significant change in tire temperature nor the associated change in rolling resistance. Due to the arrangement of the wheel load cells that recorded the drag forces acting on the rotating parts, ventilation losses (notable as an additional drag component) were part of the measured drag values. Ventilation losses are essentially losses associated with the torque required to maintain the rotation of the wheels opposing those aerodynamic forces that slow down their rotational movement (e.g. also see Mayer et al. [21]).

Due to the design of the airline of the WTA, the properties of the free stream were heavily influenced by the surrounding weather conditions. Hence, we conducted tests at constant Reynolds numbers, which resulted in a certain variation of the free stream velocities. The Reynolds dependency of the measured drag value of the DrivAer fast back configuration is plotted in Figure 6. Although the standard testing Reynolds number (about $Re \approx 5.2 \cdot 10^6$) was not quite in the leveled-out region, we were not able to continuously test at significantly larger Reynolds numbers due to the limitations of the wind tunnel facility at the TUM.

In the WTA, we recorded the dynamic pressures using the plenum and the nozzle method. The measured values using the nozzle method were roughly 2% below those measured when applying the plenum

¹ The z -position of the static pressure recordings ($z = 0.24$ m in the WTA and $z = 0.60$ m in the AAWK) scaled by the scaling factor of the wind tunnel model. We did not perform additional investigations on the variation of the static pressure for additional z -positions within the symmetry plane.

method. In the standard cases, absolute forces and absolute pressures were normalized by the dynamic pressure measured using the plenum method.

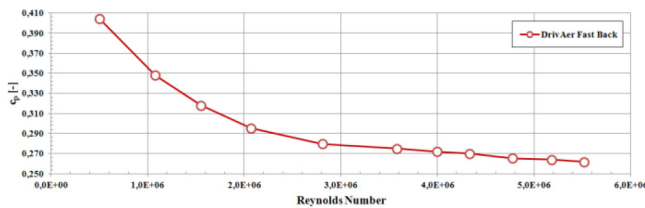


Figure 6. Reynolds dependency of aerodynamic drag acting on the DrivAer fast back measured in the WTA. Please note that this test was conducted with different wheels/rim geometries than the other tests of this study.

CFD SIMULATION OF DRIVAER MODEL IN 3/4 OPEN TEST SECTION

CFD Setup

Wind Tunnel Geometry

The basic idea of this approach was to perform CFD simulations of the full scale DrivAer reference model in open road conditions and open jet test sections for the purposes of charting interferences between wind tunnel and car model. Thereby, the investigated effects in this work are limited to those that affect the measurement of aerodynamic forces in a wind tunnel. It is well known that a number of further components such as boundary layer treatment, moving ground simulation, static pressure gradients and collector walls often influence the aerodynamic forces on the model in a wind tunnel as well. However, to isolate the interferences between model, nozzle and open jet, the applied CFD domains were set to be as illustrated in Figure 7.

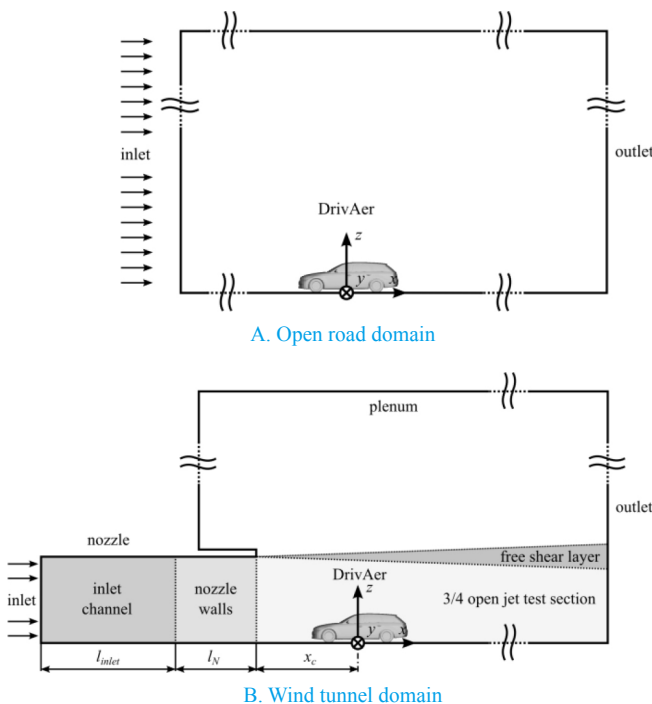


Figure 7. Schematic model of the domains for both open road and wind tunnel simulations with the full scale DrivAer reference model (here estate back). Due to illustration purposes, the plenum dimensions are adapted to fit the view (plenum dimensions: $B_p = 72$ m, $H_p = 36$ m and $L_p = 64$ m).

Figure 7 depicts the domains for the simulations of the DrivAer in open road and wind tunnel conditions. For open road conditions (Figure 7 A) a rectangular-shaped domain was created and its width $B_p = 72$ m, height $H_p = 36$ m and length $L_p = 64$ m were selected to achieve a blockage ratio of 0.084%, which was assumed to be negligible. The simulations in the open road domain set the reference to compare aerodynamic forces and to evaluate the interference effects inside the wind tunnel domain (Figure 7 B). In order to be less dependent on a specific nozzle shape, the nozzle has been approximated by a rectangular channel. To cover the boundary layer growth along the nozzle walls (side and top), this channel was split into two sections - an inlet channel with a slip wall boundary condition and nozzle walls including wall friction. The length $l_N = 3.74$ m was adjusted in order to achieve a realistic boundary layer thickness at the nozzle exit plane and was based on a selected reference nozzle geometry with $A_N = 22.4$ m². This nozzle area is considered to be of reasonable size for a potential future wind tunnel project. The length l_{inlet} of the total inlet channel was selected by $l_{inlet} + l_N = 10.0$ m in order to be large enough for the inlet boundary condition to not affect the velocity field near the model.

All floor boundaries were set to be slip walls for both open road and wind tunnel conditions, thereby eliminating any influence of possible boundary layers on the ground. This is essential since the longitudinal model position was varied within the present study. Hence, the effect of changing boundary layer profiles for different model positions had to be avoided. The plenum was considered to be infinite since its dimensions were the same as those for the open road domain. The dimensions mentioned above were regarded as constants for all nozzle sizes investigated. Since the motivation of this work is to determine longitudinal effects of the model location inside the test section, the distance x_c between nozzle exit plane and wheel base center was varied. This approach was applied to the full scale DrivAer geometry with different rear end shapes, smooth underbody, mockup and generic wheels as presented for the 40% scaled wind tunnel model by Mack et al. [15].

In addition to x_c , the second parameter to be varied was the nozzle area A_N in the range $11.0 \text{ m}^2 \leq A_N \leq 89.6 \text{ m}^2$. This was applied to evaluate interferences for different blockage ratios in the range of $0.024 \leq A_N / A_M \leq 0.197^2$. Based on the reference nozzle area of $A_M = 22.4 \text{ m}^2$ we picked two larger nozzles with a desired area factor of $A_N / A_{N,ref} = 1.5$ and $A_N / A_{N,ref} = 4.0$ respectively. Furthermore, a smaller nozzle of $A_N = 11.0 \text{ m}^2$ representing comparable blockage conditions of full scale cars in the AAWK was selected. Figure 8 illustrates this concept concerning nozzle size and blockage ratio A_M / A_N . The width-to-height ratio B_N / H_N of the nozzle was deemed constant with $B_N / H_N = 1.4$. All full scale DrivAer dimensions can be scaled from the 40% scaled model in Mack [15].

² The values were calculated from the estate back frontal area. The blockage ratio marginally changed due to the variation of the rear end shapes.

9 depicts the initialized flow field in proximity to the nozzle for the empty wind tunnel test section at the time 0. The problem here was that the potential flow solution did not produce an open jet, and hence, a longer starting period of about 6 s for the flow field to fully develop was necessary. To enhance performance and limit the overall computational time the simulation was split into two sections. In the first section, the simulations ran with a rather large time step of $\Delta t_{Start} = 1 \cdot 10^{-3}$ s. In combination with an upwind scheme, the simulation could be controlled to remain stable by accepting a loss of accuracy. However, this method was determined to be sufficient for developing the flow inside the test section. After 6 s the time step was reduced to $\Delta t_{Final} = 1 \cdot 10^{-4}$ s in order to gain a maximum Courant number of $Cou < 20$. The simulation end time was set to be 15.0 s, whereas the flow field was averaged between 8.0 s and 15.0 s (see b) in Figure 9 for the empty wind tunnel test section). Additionally, Figure 9 shows the time averaged velocity fields in c) for a simulation of the DrivAer with estate back in the test section (configuration: $A_N = 11.0$ m², $x_c = 3.984$ m) and in d) for the corresponding simulation in open road conditions. All images in Figure 9 show limited views from a restricted region of the domain in the symmetry plane $y = 0$. Therefore, Fig. A1 to A4 in Appendix A provide a more thorough visualization of the flow field in the entire domain. The period for time averaging was determined in order to achieve an accurate average of the unsteady aerodynamic forces of about ± 0.001 in lift and drag. Therefore, initial tests ran with an end time of 20.0 s in which the backward average provided a required averaging time range of 7.0 s (from 8.0 s to 15.0 s).

value and $\Delta C_i(t)$ for unsteady forces and standard deviation. For the drag coefficient the standard deviation was in the range of 0.012 and < 0.035 for the lift coefficient respectively. As stated above, the selected averaging period was between $t = 8.0$ s and $t = 15.0$ s. The plots in Figure 10 show that compared to drag, the averaging of the lift forces takes more physical time to converge within the same accuracy. For an overview of the most relevant simulation settings please see Table 4.

Table 4. Settings and properties for the simulation of DrivAer in open test section.

	Symbol	Value
Reynolds Number	Re	$1.1 \cdot 10^7$
Solver		pimpleFoam
Time Step Start	Δt_{Start}	$1 \cdot 10^{-3}$ s
Time Step Final	Δt_{Final}	$1 \cdot 10^{-4}$ s
Time Averaging	Δt_{Av}	7 s
Max. Courant	Cou_{Max}	16
Number of Cells		90 - 160 Mio.
Core Hours per Case		45.000 – 80.000

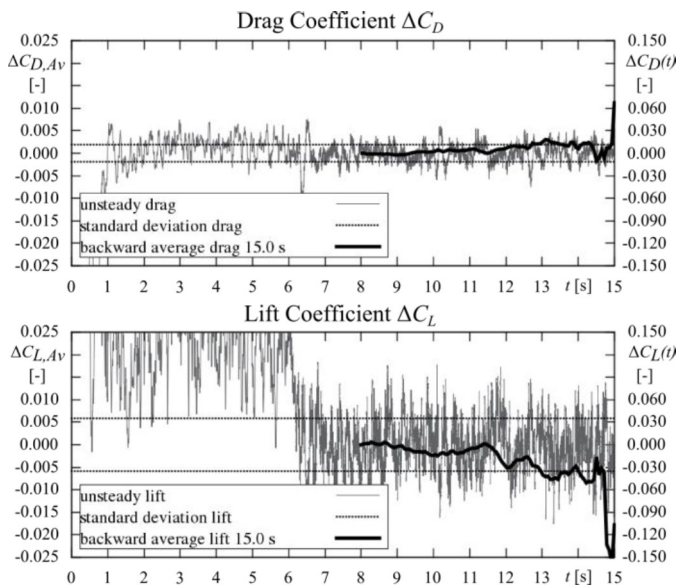


Figure 10. Unsteady and averaged aerodynamic drag (top) and lift (bottom) referenced by the average value from $t = 8$ s to $t = 15$ s. Unsteady forces and standard deviation are plotted on minor scale $\Delta C_i(t)$, backward averaged values on major scale $\Delta C_{i,Av}$.

To limit the overall simulation time, the end time was reduced from 20.0 s to 15.0 s without losing accuracy in the averaging of unsteady aerodynamic forces. As an example Figure 10 depicts the unsteady value for drag and lift coefficient (estate back, $A_N = 11.0$ m², $x_c = 3.984$ m) including raw data as well as backward averaged data from $t = 15.0$ s and the standard deviation. In terms of visualization, the plots are drawn on different scales - $\Delta C_{i,Av}$ for backward averaged

Test Cases and Validation

¼ Open Jet Test Case

In order to eliminate any static pressure gradients and to isolate the interference between model, nozzle and open jet, a constant static pressure inside the test section was desired. Therefore, the empty test section, as shown in Figure 7, served well as a test case. The slip wall condition on the ground as well as the plenum size³, which was considered infinite, turned out to be adequate conditions for a preferably free jet propagation. Due to the growth of boundary layers along the nozzle walls, there was a noticeable acceleration and a drop in static pressure inside the nozzle channel. However, the static pressure along the test section $c_p(x)$ turned out to be rather constant if the mesh refinement inside the free shear layer allowed for the solution of essential turbulent structures on grid scale. The reason for this is the fact that the sub-grid scale turbulence model Spalart-Allmaras works poorly in the free turbulent shear layers since it isotropically models turbulence. In this region the open jet propagation displayed a very sensitive behavior depending on the meshing resolution. Figure 11 shows an example for the effect of grid size inside the free turbulent shear layer on the velocity field for grid length scales of $\Delta x = 23$ mm and $\Delta x = 3$ mm. For a better visualization, the curves were fitted in a way to achieve $\Delta c_p(x=0) = 0$. For the grid with $\Delta x = 23$ mm considerable fluctuations at the jet boundary began further downstream at $x \approx 2$ m, whereas the same effect appeared further upstream at $x \approx 0.3$ m for the finer grid. It is obvious that turbulent structures had to increase in length scale to be resolved by the grid. The sub-grid scale model itself did not properly

³ Since the regions of reverse flow showed very low velocities $u/u_{ref} \ll 0.05$, the assumption of an infinite plenum was derived from the flow field measurement.