Comparison of Time-Resolved Experimental and Numerical Data in the Wake of a Full-Scale Passenger Car

Jonas Sebald 1) Jan Reiß 1) Marco Kiewat 2) Thomas Indinger 1)

1) Technical University of Munich, Chair of Aerodynamics and Fluid Mechanics
85748 Garching, Germany, (E-mail: jonas.sebald@tum.de)
2) AUDI AG
85045 Ingolstadt, Germany

Received on April 30, 2020

ABSTRACT: To further reduce the aerodynamic drag of passenger vehicles, a deeper understanding of the flow field is required. Analysis methods like the dynamic mode decomposition (DMD) are useful to investigate unsteady flow phenomena around the vehicle. DMD results are only relevant for the aerodynamic development, if the used numerical data is able to predict the unsteady physical flow field. Therefore, in this study unsteady hot-wire data in the wake of a passenger vehicle is compared to a numerical dataset of an unsteady Spalart-Allmaras Delayed Detached Eddy simulation. Differences and wind tunnel effects are found and explained.

KEY WORDS: heat · fluid, computational fluid dynamics, aerodynamic performance, Unsteady Aerodynamics [D1]

1. INTRODUCTION

The aim of the automotive industry to further reduce the aerodynamic drag of passenger vehicles, requests a deeper understanding of the flow field around the vehicle. A useful tool to analyze the flow field are numerical simulations. For automotive aerodynamics simulations Islam et al. (1) showed that a Detached Eddy Simulation (DES) is able to predict important characteristics of the time-averaged flow with a higher accuracy than a Reynolds-Averaged Navier Stokes (RANS) simulation. For an even deeper understanding of unsteady flow phenomena the use of transient numerical simulations gains higher importance to investigate not only the time averaged flow field but also temporally resolved unsteady phenomena. For the evaluation of the phenomena of the time-averaged flow field pressure- and velocity-distributions, streamlines in the flow field and statistical mean quantities are helpful tools. The analysis of time-resolved unsteady structures demands additional evaluation methods because of the complexity of the transient flow field at high Reynolds numbers. One helpful method is the dynamic mode decomposition (DMD), firstly introduced by Schmid (2), which helps to extract dominant flow characteristics out of various temporal overlapping flow features. Multiple works like Peichl et al. (3) and Kiewat et al. (4) improved the DMD algorithm for the use in the development process for vehicle aerodynamics and showed different possibilities to use the results for a deeper understanding of the flow field and how this is used to optimize the aerodynamics of a vehicle. The DMD is based on the time-resolved numerical data. Therefore, the unsteady numerical data has to feature the same time-resolved flow phenomena as the unsteady flow around a passenger vehicle in the wind tunnel. A comparison of the numerical and experimental data is necessary to check if the datasets, for this work especially the unsteady data, contain the same phenomena.

A similar comparison of time-averaged quantities only was made by Collin (5). He validated time-averaged flow features of the full-scale Aero-Acoustics-Wind tunnel of the AUDI AG (AAWK). He shows that the time-averaged pressure gradient of the experimental measurements fits the gradient of the numerical simulation using the geometry of the wind tunnel as boundary for the simulations, but left out a validation of the time-resolved flow phenomena. The same wind tunnel geometry will be used in this work.

A lot of research was conducted on characterizing the unsteady wake of simplified vehicles like the Ahmed Body (6). Therefore, different types of time-resolved experimental data was used. Duell et al. (7), Ishihara et al. (8), Vino et al. (9) and others used pressure probes to evaluate frequencies of unsteady flow structures. Thacker et al. (10) and Wang et al. (11) and many other studies used a Particle Image Velocimetry System (PIV) and a hot-wire anemometer in addition to pressure probes to investigate dominant flow structures of the Ahmed Body. Also Sims-Williams et al. (12) and Volpe et al. (13) used PIV to compare experimental measurements of unsteady flow phenomena of Ahmed Body geometries with numerical simulations. Lienhart et al. (14) used a Laser Doppler Anemometer (LDA) to analyze velocity fluctuations in the wake of the Ahmed Body.

Kawakami et al. (15) compared time-resolved hot-wire measurements of different configurations of a 28%-scale
They were able to capture typical unsteady flow structures, but also found differences of the Strouhal numbers compared to other studies.

Also Sims-Williams et al. (17) found a discrepancy of the Strouhal numbers in their wind tunnel tests to other investigations done with the same car model in different wind tunnels. They assumed that the unsteady flow phenomena of the car wake are dependant on the unsteady flow structures from the wind tunnel itself. However, they didn’t further investigate the finding nor did they compare the results to numerical data.

This work will provide novel results for a comparison between time-resolved numerical and experimental data in the wake of a full-scale passenger vehicle in a wind tunnel test. In addition, the dependency of unsteady wake structures and unsteady wind tunnel structures will be issued for the AAWK.

2. METHODS

2.1. Experimental Setup

The wake measurements are carried out in the AAWK at AUDI AG. Detailed information about the AAWK can be found in Wickern et al. (18). The wind tunnel nozzle has a cross section of 11.00 m². The selected vehicle has a frontal of 2.33 m². This leads to a blockage ratio of 21.2%. The investigated vehicle is a production 2019 Audi A6 with a notchback rearend which is depicted in Fig. 1.

Fig. 1 Wind tunnel test vehicle (a) front view, (b) side view

In Fig. 1 (a) the fully closed frontend of the car, to avoid air from flowing through the enginebay, can be seen. This helps to reduce errors made by potential differences in the modelling of parts in the engine compartment and possible discrepancies in the prediction of pressure losses due to the airflow through the cooling package. To further increase the comparability of the numerical and experimental data, the rolling road system of the wind tunnel as well as the boundary layer suction are deactivated and covered. This prevents potential errors that could occur owing to different boundary conditions in the wind tunnel and the numerical setup, like mismatching suction rates. All gaps of the body are closed to match the geometry used for the numerical simulations – see 2.3 Simulation Setup. Furthermore, the side view mirrors are removed with the intention to only measure unsteady wake structures of the main body.

The investigations are conducted with a wind speed of 39 m/s which leads to a Reynolds number of \( \text{Re}_L = 8.2 \cdot 10^6 \) with the wheelbase of 2.928 m as the characteristic length scale.

2.2. Measurement Equipment

The main measurement technique for this study is a Constant Temperature Anemometer (CTA) type hot-wire. Due to the fact that the used hot-wire has a single wire probe the specific velocity components of the flow cannot be separated. Therefore, the temporal evolution of unsteady structures in the flow field can be measured, but not their spatial evolution. Also, there is a correlation between the mounting position and orientation of the probe and the measured intensity of the velocity components. This correlation was examined by Becker (19). They provide two graphs with the relative amount of measured flow speed, depending on the orientation of the probe to the main flow. The used mounting position of the probe in the wind tunnel test leads to the following factors for the three x-, y- and z-velocity components \( u, v \) and \( w \) given in Table 1 and will be used later for the analysis.

Table 1 Hot-wire factors for velocity components

<table>
<thead>
<tr>
<th>Velocity components</th>
<th>( u )</th>
<th>( v )</th>
<th>( w )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Hot-wire factor</td>
<td>1.0</td>
<td>1.1</td>
<td>0.2</td>
</tr>
</tbody>
</table>

Fig. 2 shows the traversing arm and the hot-wire probe.
Spatial discretized points are measured in a y-z-plane 0.75 m behind the vehicle. The sampling-rate for the hot-wire measurements is set to 4096 Hz with a sampling-time of 9 seconds. In addition to the hot-wire probe, a three-component acceleration sensor is mounted at the hot-wire probe support to measure the frequencies of the oscillating motion of the probe support. The acceleration sensor is depicted in Fig. 2.

2.3. Simulation Setup

The computational fluid dynamics (CFD) simulations are done with OpenFOAM®. The chosen Delayed DES (DDES) simulation setup uses different turbulence models for different regions in the domain. For regions distant from the walls a LES turbulence model is used together with a Smagorinsky-like one-equation-model for the sub-grid scale viscosity. In near wall regions a Spalart-Allmaras RANS turbulence model is used for the modelling of the turbulent viscosity and the turbulent length scale. Additional, hybrid wall functions are employed in accordance to Spalding’s Law to solve the problem of turbulence modelling in a large range of y+ values. Islam et al. showed the accuracy of the used method for vehicle aerodynamics.

Collin introduced and validated a virtual model of the AAWK with full-scale dimensions which is used as boundary for the simulations. This domain contains the settling chamber, the nozzle of the wind tunnel, the test-section combined with the plenum and the collector. This geometric boundary box allows a comparison between the experimental data of the wind tunnel test and the numerical data. In the simulations as well as in the experiments the boundary layer suction and the rolling-road system are inactive. The virtual vehicle in the simulations was modelled to match the experimental vehicle as good as possible. A physical simulation time of 9 seconds is computed with a distributed random-access memory system on 960 cores and took a total cpu-time of 71k hours.

The mesh of the domain is created with snappyHexMesh, an automated mesh generator by OpenFOAM®. Refinement regions at specific areas of the car and the wake are applied to resolve small wake structures. This leads to a hex-dominant mesh with a total amount of approx. 241 million cells and a minimal edge length of 1.5 \times 10^{-3} m for the smallest cells, depicted in Fig. 3 (c) and (d). To enable an analysis of the time-resolved numerical data of the large simulation domain, only the velocity components of the specific points of the hot-wire measurement are saved with a time-step width of 0.0025 seconds. To avoid a dependency of the comparison between numerical data and experimental data on the temporal resolution the time-step width of 0.0025 is applied to the experimental data as well. Also, a comparison of two various edge lengths – with the size of the hot-wire and a larger one – showed just a negligible small influence on the velocity fields. Furthermore, the hot-wire factors of Table 1 are applied to the numerical data to reach a higher comparability for the characteristics of the spatial evolving structures. The measurement plane (red) in relation to the virtual vehicle can be seen in Fig. 3.

![Fig. 3 Vehicle model for numerical simulations with red measurement plane (a) rear view, (b) side view and mesh layout at rear-pillar (c), trunk lid (d)](image_url)

There are two major differences between the numerical and the experimental setup which could cause a deviation of the two datasets. On the one hand the AAWK is just partially modeled in the simulations and characteristics of the remaining wind tunnel will not be present in the numerical data. On the other hand the traversing arm is not modeled in the numerical setup.

3. RESULTS

3.1. Time-averaged flow field

To verify the comparability of the numerical and experimental data the two time-averaged velocity fields of the velocity magnitude (Umag) normalized to the maximum value of Umag...
will be given in Fig. 4 (a) and (b). For both Fig. 4 (a) and (b) the maximum Umag-value of the hot-wire measurement (38.02 m/s) is used.

The interpolated velocity fields of the specific data points for the experimental and numerical data with the same averaging time of 9 seconds are depicted. The data points for the interpolation are marked by the black crosses in Fig. 4 (a) and (b) and correspond to the hot-wire measurement points.

There is an overall match of the two velocity fields. Nevertheless, two discrepancies in the structure of the wake can be observed. One area which differs between the numerical and the experimental data is the upper part of the wake near \( y = 0 \text{ mm} \). A much higher shape of the wake is present in the experimental data compared to the numerical data.

A possible cause for the differences can be a wrong prediction of the flow detachment from the trunk lid and the underbody. In the numerical simulations the upper eddy caused by the trunk lid detachment is spatially greater than the eddy from the underbody detachment. Because of the small lower eddy, the upper eddy is able to extend more downwards. This distribution of the eddies, given in Fig. 5, leads to a lower overall wake. Given the the greater extension of the wake at \( y = 0 \text{ mm} \) in Fig. 4 (a) in positive \( z \)-direction of the hot-wire measurements, a different distribution of both eddies in the wind tunnel test can be assumed. Therefore, a spatially greater lower eddy could cause the wake to extend higher in \( z \)-direction. To clarify the distribution of the eddies in the wind tunnel test, further investigations are necessary. The deviation between the numerical and the experimental dataset has to be further investigated to fully explain the cause of the different shapes of the wake.

The second discrepancy between the numerical and the experimental time-averaged wake structures can be found at around \( y = -600 \text{ mm} \) in the upper part of the wake. The two tips of the wake in this area are caused by the vortex shedding at the rear pillar of the vehicle body. The wake of this vortex of the simulation is higher in \( z \)-direction than in the wake of the wind tunnel test. The difference can be caused by not ideal prediction of the flow detachment at the rear pillar. To fully clarify the cause for this problem further investigations are necessary. Apart from
the discrepancies mentioned above, there is a highly accurate overall match of the time-averaged numerical and experimental data. Thus, a fairly accurate prediction of the time-averaged wake structure can be made by the chosen simulation setup and also the used measurement technique is able to determine the correct shape of the wake structure in the measured plane.

3.2. Time-resolved flow field

In order to obtain knowledge about unsteady structures in the wake of the vehicle the Fast-Fourier-Transformation (FFT) is often used. This method is applied with the same parameters to the numerical and the experimental data. The governed solutions are given with respect to the dimensionless Strouhal number (St). For the vortex shedding phenomena at bluff bodies like a vehicle Vino et al. (15) suggest to use the square root of the frontal area. This characteristic length scale is used for the following investigations.

On each measurement point of the experiment respectively of each data point of the simulation a FFT is applied. The Power Spectral Density (PSD) amplitudes of each Strouhal number are averaged over all points to get a averaged FFT solution for all measurement points – a MeanFFT. This MeanFFT summarizes dominant frequencies of all single data points and gives an overview of the present unsteady flow structures in the whole measurement plane.

In Fig. 6 the MeanFFT of the numerical data with different peaks is depicted. The main peaks are at St = 0.07, St = 0.15, St = 0.25, St = 0.34, St = 0.43 and St = 0.55. Vino et al. (15) found a dominant peak in the wake of the Ahmed body at St = 0.39. They assumed that this unsteady structure is caused by the vortex shedding between the trunk lid and the underbody of the vehicle. Two nearby peaks at St = 0.34 and St = 0.43 can be found in Fig. 6 as well, but not as dominant as in Vino et al. (15). The peak at St = 0.25 could be referring to a vortex shedding between the two sides of the vehicle like the vortex shedding behind a cylinder. The exact cause for the dominant peaks of the MeanFFT can not be examined because the FFT analysis contains no spatial information about the unsteady structures. Therefore, there is also no thorough explanation for the other peaks.

The same analysis method is now applied to the experimental dataset. The results are given in Fig. 7. Different peaks compared to the MeanFFT of the numerical data in Fig. 6 can be seen there. The analysis of the acceleration sensor shows, that neither of the main peaks of the hot-wire MeanFFT are caused by structural oscillations of the traversing arm. During the measurements in the vehicle’s wake the sensor recorded the vibration of the hot-wire probe with main frequencies at St = 0.28 and St = 0.63. Therefore, the hot-wire MeanFFT peaks represent unsteady flow structures.

The MeanFFT peaks of the experimental data seem to be similar to the peaks of the numerical data. The peak of St = 0.26 in the experimental data could correspond to the St = 0.25 peak of the numerical data and could be caused by the sideways vortex shedding behind the car. Also the St = 0.38 peak of the experimental data could be caused by the z-directional vortex shedding, similar to the phenomenon described by Vino et al. (15) at St = 0.39, and could be related to the peaks of St = 0.34 and St = 0.43 of the numerical MeanFFT. Due to the lack of detailed spatial information, because of the single wire hot-wire probe, this phenomenon cannot be examined further. The peak at St = 0.17 could relate to the St = 0.15 peak of the numerical data. Discrepancies between the two MeanFFTs are the amplitudes for the different Strouhal numbers as well as the lack of further dominant peaks in the experimental data at St = 0.07 and St = 0.55.
Nevertheless a shift in the Strouhal numbers can be observed for the MeanFFT of the wind tunnel test. Sims-Williams et al. (17) supposed a lock-in phenomenon between the unsteady car wake structures and global wind tunnel structures. They observed shifts of the Strouhal numbers between different wind tunnels with the same vehicle. They describe a sensitivity of the unsteady flow structures of the vehicle’s wake to external influences, like wind tunnel pulsations. This leads to a Strouhal number shift caused by the lock-in of the wake structures to global unsteady wind tunnel structures with similar frequencies. To investigate this assumption the main peaks of the wind tunnel test are transformed to frequencies in Hertz given in Table 2.

<table>
<thead>
<tr>
<th>Strouhal number</th>
<th>0.17</th>
<th>0.22</th>
<th>0.26</th>
</tr>
</thead>
<tbody>
<tr>
<td>Frequency [Hz]</td>
<td>4.25</td>
<td>5.38</td>
<td>6.63</td>
</tr>
</tbody>
</table>

Wickern et al. (22) conducted research to the pulsations of the AAWK. They investigated main frequencies of the empty wind tunnel at different wind speeds. Neither of their chosen wind speeds correspond to the wind speed of this experiment and natural frequencies could hardly be transferred for this study. Even though they pointed out two main frequencies – 3.9 Hz at a wind speed of 33.3 m/s and 6.8 Hz at 55.6 m/s. Evert et al. (23) examined the behaviour of different active suppression methods for the natural frequencies of the AAWK. They conducted their research at a wind speed of 39 m/s combined with different vehicles. The frequencies they found are more important for this study. With a notchback vehicle placed in the test-section of the wind tunnel, two dominant peaks are present in the FFT of the wind tunnel flow. The first peak is found at 6.5 Hz and the second at 3.9 Hz. The FFT results of a measurements which is done like the one described by Evert et al. (23) are depicted in Fig. 8. These two peaks are very similar to the peaks of the MeanFFT of the vehicle’s wake, 6.63 Hz and 4.25 Hz, but also with a slight shift.

This shift in the frequency can be caused by the different vehicle geometry used for this investigation in comparison to Evert’s (23) study. Also the shift between the experimental and numerical MeanFFT peaks can be caused by the lock-in phenomenon. The natural frequencies of the AAWK are very close to the frequencies of vortex shedding flow structures of the vehicle’s wake. The fluctuations of the wind tunnel flow with their natural frequencies can cause a shift of the vortex shedding frequencies by triggering the flow detachment due to the fluctuating flow at its natural frequencies. The wind tunnel frequencies have an effect on the wake of the vehicle. At the moment those frequencies are not modeled in the numerical setup, differences in the wake of the vehicle between simulation and experiment can be caused by the absence of the wind tunnel pulsations in the numerical setup. Also compressible phenomena caused by the wind tunnel pulsations are not resolved by the CFD which leads to additional discrepancies between the numerical and experimental data. Nevertheless, the wind tunnel pulsation, possibly in combination with the vortex shedding structures of the vehicle, remain the dominant temporal structures in the wake of a vehicle in the wind tunnel. Further investigations are necessary to clarify the wake structure frequency shifts caused by wind tunnel pulsations and the presence of vortex shedding structures near those frequencies. A peak at or near 5.38 Hz, respectively St = 0.22, is not described by Evert. Therefore this frequency could relate to an unsteady structure caused by the vehicle, e.g. a sideways vortex shedding behind the car. The cause for this peak can not be stated due to the missing spatial information.

Besides the overall missing of initiated turbulence in the LES region of the CFD, the absence of the wind tunnel pulsations in the numerical simulations can be a part of the cause that the turbulent kinetic energy (TKE) in the vehicle’s wake of the numerical data differs from the experimental data. Fig. 9 shows the distribution of the TKE in the measurement plane. Both figures are normed to max(k) of the experimental data.

In both figures can be seen, that the region with the highest TKE is the shear layer between the vehicle’s wake and the freestream. The shape of the distribution is similar for both datasets. The major difference between the two graphs is the quantity of the TKE. The experimental data has regions with up to twice the amount of TKE compared to the numerical data. As the velocity fluctuations of the flow are the basis for the determination of the TKE, the experimental and numerical datasets are equally sampled. Both datasets are averaged with a moving average window with a width of 0.0025 seconds to ensure a matching temporal discretization. However, there are two discrepancies in the evaluation of the numerical TKE distribution given in Fig. 9.
(b) compared to the hot-wire data. On the one hand the spatial discretization of the CFD compared to the experiment is different due to a cell length which is double the size of the hot-wire probe. This leads to a varying spatial averaging of the velocity fluctuations. A simulation with a edge length of the mesh matching the hot-wire probe, showed a negligible effect due to the spatial averaging of the mesh with double the edge length. The coarser mesh is used for the TKE analysis because of the lower requirement of disk space especially for the long averaging time of 9 seconds. On the other hand the absence of unresolved turbulence of the sub-grid scale of the LES region in the analysis of Fig. 9 (b). The sub-grid scale turbulence of the wake’s shear layer has values in specific regions from $k/\text{max}(k) = 1 \cdot 10^{-5}$ to $k/\text{max}(k) = 2.5 \cdot 10^{-4}$. These values are order of magnitude lower and therefore can be neglected for the analysis. A large discrepancy of the numerical and experimental TKE values remains. The lack of TKE in the CFD compared to a wind tunnel test can be a reason for different flow detachment behaviour in numerical simulations.

4. CONCLUSION

The presented study provides an improved understanding of differences between an experimental wind tunnel test and a numerical simulation of a full-scale notchback passenger vehicle. At first a comparison of time-averaged flow field data shows the overall validity of the numerical, experimental and measurement setup. Except for minor deviations in the numerical averaged velocity field, which are mainly caused by the used turbulence model and its detachment prediction, the overall match with the hot-wire measurements has a high accuracy.

Larger differences are found in the comparison between the time-resolved datasets. The FFT analysis of the CFD data shows dominant peaks which could partly be related to unsteady vortex shedding structures. The Strouhal numbers of these structures are similar to other investigations, which were conducted at simplified vehicles. Applying the same FFT analysis to the experimental data presented similar but slightly shifted Strouhal numbers. The shift of the Strouhal numbers can be related to global wind tunnel pulsations, which are dominant at these frequencies. This shift leads to different unsteady wake structures in the wind tunnel test compared to the numerical simulation.

Furthermore, a discrepancy between the numerical and experimental TKE is stated. The different level of turbulence in the flow potentially leads to varying detachment processes which may cause Strouhal number shifts for unsteady structures.

Additional investigations are necessary to clarify the dependency of the unsteady vehicle wake structures on the pulsations of the wind tunnel. In addition, improvements to the numerical setup should be made to close the gap between the time-resolved numerical and experimental data. An implementation of the wind tunnel pulsations to the numerical setup is useful to achieve an improved correlation of the time-resolved datasets.

ACKNOWLEDGEMENTS

We would like to express our gratitude to our industry partner AUDI AG for their financial support and thank Dr. M. Islam for the opportunity to carry out research. This publication is supported by TUM Graduate School’s Faculty Graduate Centre of Mechanical Engineering. This paper is written based on a proceeding presented at JSAE 2019 Annual Congress.

REFERENCES