Experimental Investigation on the Aerodynamics of Isolated Rotating Wheels and Evaluation of Wheel Rotation Models Using Unsteady CFD

Lukas Haag 1) Thomas Blacha 2) Thomas Indinger 3)

1), 3) Technische Universität München, Boltzmannstraße 15, D-85748 Garching bei München, Germany (E-mail. lukas.haag@tum.de)
2) AUDI AG, Windkanalczentrum, D-85045 Ingolstadt, Germany

Received on June 24, 2016
Presented at the JSAE Annual Congress on May 25, 2016

ABSTRACT: Wheel aerodynamics of passenger cars has recently been under close investigation due to its influence on the aerodynamic forces on the vehicle. In modern wind-tunnels the wheel rotation can be resembled close to reality, while, in their CFD development process, car manufactures rely on simplified models for considering rotating wheels. In this paper, model-scale generic and production wheels are investigated experimentally, in respect of the influence of geometry, camber and a brake disc. The data is used to assess the accuracy of CFD simulations using open-source software and their applicability in a manufacturer’s CFD process.

KEY WORDS: heat_fluid, aerodynamic performance, computational fluid dynamics, wind-tunnel test, wheel aerodynamics, PIV

1. Introduction

In seeking to improve the aerodynamic performance of passenger vehicles, engineers have made the move from general optimisation of the vehicle body to optimising specific details and the interference between those details [1]. Studies have shown that particularly the vehicle wheels and wheel arches have a considerable influence on the aerodynamic properties and can actually contribute up to 25% of the total aerodynamic drag [1, 2]. Consequently, over the past two decades, wheel aerodynamics has been the focus of many studies and optimisation efforts in academia and in industry [2, 3, 4].

Despite the known potential in the optimisation of the flow around the rotating wheels, the understanding of the flow phenomena and their interaction with surrounding parts still offers considerable room for improvement. This can also be attributed to the uncertainty introduced by modelling approaches to resemble wheel rotation in CFD. Good quality CFD results, which resemble the important physical effects, can lead to a thorough understanding of the flow phenomena; however, this is impeded when modelling approaches and less detail are used. In terms of wheel rotation, the methods often used in industry, such as the Rotating Wall Boundary Condition (RWBC) or Multiple Reference Frames (MRF) incorporate a higher degree of modelling, while Sliding Mesh (SM) approaches effectively cover wheel rotation and can offer further insights into the aerodynamic characteristics at the cost of higher turn-around times.

In order to validate and facilitate SM simulations and to gain more insight into the effect of certain aspects of wheel aerodynamics, this paper presents experimental results on the flow around an isolated wheel with different closed and open wheel and rim designs, as well as the influence of the presence of a brake disc and different camber angles.

Furthermore, the same setup is investigated by unsteady CFD to assess the capability of the wheel rotation modelling approaches in order to predict the flow field around the wheel. The CFD method is then assessed as to whether it is fit for application in an OEM’s aerodynamics development process.

2. Experimental and Numerical Test Set-Ups

2.1. Investigated Wheel Models

To be able to assess different geometrical influences on the flow around wheels, multiple wheel geometries are investigated: two generic rim geometries and one production rim geometry. All the wheels are 40% scale models.

The two simplified generic wheel models exhibit few specific shape details. The rim is basically a flat plate. However, rim flanges are included. The closed generic rim, shown in Fig. 1 (left), can serve as a baseline case since it incorporates the lowest level of detail. The open generic wheel, which can be seen in Fig. 1 (middle), is basically the same geometry, but with ten circular holes. It represents a simplification of the standard steel rim without a design cover, with which many production cars are equipped. Both wheel models have a diameter of 258.2mm and an aspect ratio (AR, width/diameter) of approximately 35.5%. The tyre aspect ratio for both generic geometries correspond to a full-scale tyre dimension of 225/45 R17.

Furthermore, a realistic rim geometry is investigated. It is a production rim of AUDI AG’s compact luxury crossover SUV model, the Audi Q5. The rim, as shown in Fig. 1 (right), incorporates a six-arm design and is specifically optimised to produce low aerodynamic drag when mounted on the car [5].

Whilst for the production vehicle the tyre size is 235/65 R17, in the experiments, a tyre size corresponding to a real life size of 225/45 R17 is investigated for compliance with the generic wheel geometries. All the model tyres are made of CFRP and are therefore undeformable. The realistic wheel model has a slightly smaller diameter of 254.8mm. Furthermore, it can be equipped with a brake disc in order to investigate the influence on the flow field. In addition, the wheel support structure allows the...
application of camber of up to 0.5° to the wheels. The camber influence was also investigated in steps of 0°, 0.2°, 0.4° and 0.5°.

2.2. Wind-Tunnel

The experiments were conducted in the large model scale wind-tunnel A (WTA) of the Technical University of Munich (TUM). WTA is a Göttingen type wind-tunnel with a 3/4 open test section. It is equipped with a single-belt rolling road system (RRS), which can operate at speeds of up to 50 m/s. However, for operational safety reasons, the test speed is set to 45 m/s. With the wheel diameter as reference length this gives a Reynolds number of Re = 7.40e5 for the investigated cases.

The test section is 4.8m long and the distance between the rollers of the moving belt is 4.53m. The boundary layer exiting the nozzle is trimmed off with a boundary layer scoop which reduces the effective nozzle area to 1.746x2.4m (4.1904m²). The blockage ratio of the wheel and wheel strut is 1.5% which is very low as regards the recommended 10%\(^\text{(1)}\). Furthermore, the wheel strut incorporates an aerodynamically optimised aerofoil profile to influence the free stream as little as possible. The measurement position of the wheel axis is 2m behind the nozzle exit, where the free stream pressure gradient is less than ∆C\(_{\text{p,Max}}\) < ±1.5\(\text{e} - 3\)\(^\text{(9)}\) consequently no correction regarding force measurements is applied. The position of the wheel axis marks the origin of the x-axis of the used coordinate system (x=0mm).

2.3. Measurement Techniques

During the wind-tunnel experiments integral force measurements as well as flow field measurements were performed.

2.3.1. Force Measurements

The wheel drag force was measured using a one component force balance within the wheel axle. The specification of the wheel balance can be found in Mack et al.’s publication\(^\text{(7)}\). The sampling frequency was 1000Hz and 2000 samples were taken for each measurement. An averaging over 30 measurements has been carried out for each configuration giving 60s of averaged time for the force measurements.

2.3.2. Particle Image Velocimetry

The method which is used for the flow field measurements is stereoscopic PIV. In previous work by Hövelmann et al., a PIV system by LaVision was installed in the WTA\(^\text{(8)}\). The setup proposed in their publication was also used for the presented investigations. With the used model configuration, the setup results in a spatial resolution of approximately 4.35x4.35mm. 400 double-image pairs were recorded for each measurement with a sampling rate of 12.5Hz to 15Hz. This gives an averaging time of 26s to 32s for each measurement.

The PIV devices are mounted on a traversing system in order to capture multiple plane cuts perpendicular to the approaching flow at x=300mm, 375mm, 500mm, 625mm and 750mm downstream of the wheel axis location. In the proximity of the wheel, the measurement window is limited due to reflections of the laser at the wheel and the polymer belt of the RRS, as well as the line-of-sight of the upstream camera. Consequently, no experimental data of the wake in the close proximity (<1DWheel) or near the contact patch between tyre and ground can be recorded. However, the dominating ground vortex structures of the wake\(^\text{(9)}\) are expected to be covered with the presented setup.

2.4. Numerical Simulations

For the CFD-Simulations presented here the open-source program OpenFOAM\(^\text{®}\) was used. Axerio-Cilies has already shown that, for a similar single wheel setup, LES turbulence models give considerably better results compared to experiments, than simulations with RANS models accounting for the turbulent effects\(^\text{(10)}\). Yet, LES is very demanding in spatial and temporal resolution which is also pointed out by Axerio-Cilies\(^\text{(10)}\). This disadvantage becomes even more striking when thinking of the simulation of not only a single wheel configuration but a full vehicle.

The CFD simulations were therefore conducted using the hybrid approach Delayed Detached Eddy Simulation (DDES)\(^\text{(11)}\), to account for the effects of turbulence utilising the one equation Spalart-Allmaras turbulence model. Islam et al. have previously validated the application of the DDES model to automotive aerodynamics\(^\text{(12)}\). At the walls, hybrid wall functions according to Spalding’s Law\(^\text{(13)}\) are applied due to their universal applicability in a large range of y+ values. The time derivative is discretised by a second order implicit backward scheme. For spatial discretisation, second order central differencing is used as a convective scheme. For the velocity field, it is blended with a second order upwind scheme based on the local CFL number for stabilisation. Each computational run is parallelised on 256 cores.

For the mesh generation an automated meshing algorithm was applied. The size of the computational domain used for the simulations is 3.6x6x12m (HxWxD). Multiple refinement regions around the wheel are applied to the hexa-dominant mesh. The smallest hexahedral cells of the resulting mesh have an edge length of 1.172e-3m. The surfaces are refined with prism layers. The wheel is intersected with the floor by 1e-3m leading to a wedge shaped cells at the contact patch. This results in a total cell count of approximately ten million cells for the different wheel geometries. The inlet conditions of the simulation setup are set to meet the experimental conditions in terms of velocity, as well as turbulent quantities.

To account for the wheel rotation in the CFD simulations, two approaches were used. For the closed generic rim the RWBC is applied, meaning the appropriate velocity vector in every face
on the wheel surface is fixed according to the wheel rotation rate and the face location ($\omega_{\text{wheel}} \times r_{\text{face}}$). For the closed generic rim geometry, the use of this boundary condition is valid because it is rotationally fully symmetric, thus, the velocity vectors produced by the RWBC are tangential to the wall everywhere on the wheel surface. Furthermore, there are no geometrical details of the wheel that change their location compared to the surrounding geometries, e.g. the wheel strut.

For the open generic rim, as well as the realistic rim geometry, the RWBC cannot be used to resemble wheel rotation, since they are rotationally not symmetric. This would be physically incorrect for two reasons. Firstly, the application of the RWBC would lead to wall-normal velocity components at the openings of the rim geometries, e.g. the holes of the open generic rim or the spokes of the realistic rim. Wall-normal velocity components at boundary faces result in a mass flux through those faces which is evidently incorrect compared to real life conditions. Secondly, an RWBC is not able to cover the relative motion between geometrical features of the rim, like the holes or the spokes, and the surrounding objects, like the wheel strut. Consequently, for those geometries, the SM approach, which actually moves parts of the computational mesh, is used to cover the wheel rotation.

It is worth mentioning, that the first of the previously mentioned issues could be resolved by the application of the MRF model\textsuperscript{(14)} within the holes, or around the spokes of the two geometries in question. Nevertheless, amongst others, Hobeika et al. have shown that the model produces erroneous results for a simple bulk flow with an MRF region and that the application to rotating wheels lacks accuracy compared to reference experiments\textsuperscript{(15)}. In their publication, they propose an improved version, the MRFG model, which gives improved results compared to experiments \textsuperscript{(15)}. However, it does not resolve the second of the above-mentioned issues. The relative motion between the geometrical details of the rim and the surroundings remains uncovered. Thus, the MRF(G) models are not applied within this investigation.

The computation is run for 6s of physical time, averaging the flow field and force quantities over the last 4s. This time span is expected to be sufficient for two reasons. On the one hand, for cylindrical bodies of the diameter $d$ and the approaching flow velocity $u_\infty$, the dominant separation frequency $f_{\text{sep}}$ should give a dimensionless Strouhal Number $Sr$\textsuperscript{(16)}:

$$Sr = \frac{f_{\text{sep}} d}{u_\infty} \approx 0.2$$ (1)

For the wheels investigated here, this results in a dominant separation frequency of $f_{\text{sep}} \approx 35\text{Hz}$. This frequency is lower than the rotational frequency of $f_{\text{rot}} \approx 55.5\text{Hz}$ and is therefore the relevant phenomenon which needs to be considered for time averaging. With the chosen averaging time of 4s, this phenomenon is covered an adequate amount of 140 times. Furthermore, the 4s allow the flow to pass the domain length 15 times which is expected to suffice for convergence.

### 3. Experimental Results

In order to extract the main characteristics of the flow, the flow structures of the baseline case, the generic wheel with closed rim, is discussed thoroughly. This is done for the non-rotating (static), as well as the rotating case. Thereafter, geometrical influences such as opening the generic rim geometry, changing the position of the wheel support and using the realistic rim geometry without and with brake disc and camber will be presented. For better visibility, only every second inplane velocity vector is displayed in the presented PIV results.

![Fig. 2 Contours of mean velocity magnitude and inplane velocity vectors for the closed generic wheel configuration without rotation and with static ground at a) x=300mm, b) x=375mm, c) x=500mm](image-url)
3.1. Baseline Case – Generic Closed Wheel

3.1.1. Flow Field Investigations

Fig. 2 illustrates the first three investigated planes at: a) x=300mm (x/D = 1.16), b) x=375mm (x/D = 1.45), c) x=500mm (x/D = 1.94) for the setup without wheel rotation.

The vector plot of the inplane velocity exhibits a counter-rotating vortex pair (CVP) near the ground. This is an expected behaviour and consistent with previous publications\textsuperscript{(4),(9),(10)}. However, the wake is very asymmetric and the cores of the two vortices are strongly shifted to the outward side of the wheel. This can be attributed to the very distinguished hub vortex at the outward (left) side of the wheel at the height of the wheel support. Fig. 3 shows a schematic of the identified dominant vortices.

This hub vortex has previously not been observed that far downstream of the wheel axis by other researchers, especially not by Axerio-Cilies, who investigated a similar setup of a stationary tyre at x/D=1.1 by PIV\textsuperscript{(10)}. The shoulder vortices, which should emerge from the top aft wheel shoulders, as previously found by Wäschle, are not expected to be present this far downstream from the wheel either\textsuperscript{(4)}. Consequently, this hub vortex is unexpected. In addition to the outward deflection of the CVP near the ground, and the strong asymmetry of the wake, the fast dissipation of the outward ground vortex seems to be caused by this hub vortex. From results at x=625mm and x=750mm (not displayed), it is evident that this vortex structure is stronger and more persistent than the CVP near the ground.

An explanation for the presence of this hub vortex can be a peculiarity of the used test setup for the generic wheels. The wheel is fixed to the axle with a clamp in the size of 31x19x5mm which is again secured by an end cap. Despite not being visible in the front view, it laterally protrudes the rim plate by 15mm (see Fig. 1, left, black clamp). Therefore, this feature could be exposed to the reattaching flow around the tyre and cause this unprecedented phenomenon.

When rotation is applied, the hub vortex is less pronounced (Fig. 4). Furthermore, its location is more upward which can be attributed to the upward dragged flow at the rear part of the wheel. The CVP is located closer to the ground and, due to the greater distance to the hub vortex, is not as deflected to the outward side of the wheel as in the static case. Furthermore, the outward ground vortex is more pronounced compared to the static case. This can once again be attributed to the weaker influence of the hub vortex which interfered strongly with the outward ground vortex in the static case.

For the rotating closed generic wheel, it can be concluded that, due to the reduction of the influence of the hub vortex, the wake now resembles the upside-down T-structure (\textsuperscript{2}) previously observed by multiple researchers\textsuperscript{(4),(9),(10)}. However, the hub vortex still introduces a visible asymmetry and causes the CVP at the ground to be deflected outward further downstream, as it is still the most persistent vertical structure in the wake field. The CVP also moves slightly upward until it is dissipated in the downstream flow (evident from slices not shown here). Furthermore, it is apparent that, between the boundary layer scoop in the nozzle and the origin of the moving belt, a boundary layer develops which is not entirely washed-out up to the wheels’ location and is therefore visible in the investigated plane. This corresponds to the published boundary layer profiles by Mack et al. for the WKA with RRS\textsuperscript{(6)}.

3.1.2. Force Measurements

The results for the force measurements are shown in Table 1. It can be seen that the wheel rotation leads to drag reduction. The mechanism leading to this drag reduction has been explained previously by Wäschle\textsuperscript{(4)}. The rotation creates a ring vortex in the top wake of the wheel which transports high-energy air towards the wheel and increases the base pressure. Consequently, the drag is decreased. Unfortunately, due to limitations of the line-of-sight of the PIV system’s cameras, the measurement plane could not be moved closer to the wheel and this ring vortex cannot be seen in Fig. 4. However, the order of magnitude and relative drag reduction due to wheel rotation are in good agreement with the ones observed by Wäschle\textsuperscript{(4)}.

It is worth mentioning that the drag value for the rotating configuration is corrected to not include rolling resistance post-measurement. This is done according to a method applied to a similar setup by Mack et al\textsuperscript{(7)}, which is based on the decomposition of the drag forces acting on wheels into their sources. The approach was originally proposed by Wickern et
In this particular case, the rolling resistance accounts for 11 drag counts which is only 2% of the measured drag but in total values it is considerably more than the two drag counts experienced for a full car setup\(^{(7)}\).

Table 1 Force coefficients of the closed generic wheel (*The value for the rotating configuration is corrected by the calculated rolling resistance)

| Configuration                  | \(C_D\) | \(\Delta C_D\) |
|-------------------------------|---------|----------------|
| Closed generic rim static     | 0.589   | -              |
| Closed generic rim rotating   | 0.524*  | -0.065         |

### 3.2. Open Rim Geometries

To investigate the influence of a possible cross-flow through the rim, the open generic rim and the realistic rim geometry are investigated in the same way as the closed generic rim had been previously. For brevity, the analysis will be limited to the rotating configurations.

#### 3.2.1. Open Generic Rim

The in-plane velocity vectors for the open generic rim shown in Fig. 5 indicate that the relevant vortex structures of the closed generic rim are also found when flow through the rim is possible. The CVP near the ground and the hub vortex can be easily identified. However, their intensity and relative size has changed. The hub vortex is less pronounced, while the ground vortex on the outer side of the wheel is stronger. The ground vortex on the inner side of the wheel is very similar to the setup with the closed rim, both in location and intensity.

Once again, due to the weaker hub vortex, the wake profile is more symmetric and it can be observed that the momentum loss in the wake near the ground is increased. This results in a higher drag value compared to the closed wheel configuration as shown in Table 2.

An explanation for this behaviour could be found in the cross-flow through the rim openings which might weaken the hub vortex shortly after its creation by allowing mass flow against the clockwise rotating momentum of the hub vortex. A weaker hub vortex would lead to a smaller influence on the CVP at the ground which would, in turn, be more strongly developed. These stronger vortices can cause a greater energy consumption and increased drag force.

Table 2 Force coefficients of the different rim geometries (*All values are corrected by the calculated rolling resistance)

| Configuration                  | \(C_D\) | \(\Delta C_D\) |
|-------------------------------|---------|----------------|
| Closed generic rim rotating   | 0.524*  | -              |
| Open generic rim rotating     | 0.563*  | +0.039         |
| Realistic rim rotating        | 0.571*  | +0.047         |

#### 3.2.2. Realistic Rim

The wake flow field induced by the realistic rim looks considerably different to the results for the two generic rims. Fig. shows that the hub vortex has disappeared completely, while the outward ground vortex has the strongest intensity observed so far. The inward ground vortex is not visible in the displayed result; however, the in-plane velocity vectors at the inner side of the wheel suggest that it is present, but the vortex core is located very close to the ground. Consequently it is out of reach for detection with the PIV setup. Altogether a CVP appears to be present with the realistic rim with an increased focus on the outward ground vortex.

The absence of the hub vortex in this wheel configuration is likely to be explained by the different fixing mechanism of the wheel to the strut. It does not incorporate a clamp, as is the case with the generic wheels, but is only fixed with the end cap. Consequently, the source of the hub vortex appears to be absent in the realistic rim configuration. As previously observed, the hub vortex has interfered with the outward ground vortex. Hence, since the hub vortex is not apparent in this configuration, the outward ground vortex is stronger than in the cases investigated previously. Furthermore, in this result, the wake is the most symmetric and now fully resembles the upside-down T-structure \((\perp)\) that has been seen by other authors\(^{(4),(9),(10)}\). Furthermore, the

![Fig. 5 Contours of mean velocity magnitude and inplane velocity vectors for the open generic wheel configuration with rotation and moving ground at x=300mm](image1)

![Fig. 6 Contours of mean velocity magnitude and inplane velocity vectors for the realistic rim configuration with rotation and moving ground at x=300mm](image2)
wake area, and therefore the momentum loss, is slightly increased compared to the open generic wheel configuration. This leads to the slightly higher drag value shown in Table 2.

3.3. Effects of Geometrical Changes to the Setup

3.3.1. Application of a Brake Disc

The application of a brake disc inside the realistic rim geometry will resemble an even more realistic assembly. Since the brake disc impedes mass flow through the wheel, differences like the ones observed between the open and closed generic rim should be expected. Fig. 7 illustrates the inplane velocity vectors for the configuration with a brake disc, as well as the contours of mean velocity magnitude difference between with and without brake disc calculated as follows:

\[ \Delta U_{\text{mag}} = |U_{w/BD} - U_{w/oBD}| \]  

(2)

Apparently, the changes to the flow field due to the brake disc are almost insignificantly small. The inplane velocity vectors indicate an almost identical flow field and the velocity differences are also practically negligible except for small areas.

The brake disc causes a higher momentum loss in the area of the outward ground vortex, while in the upper wake of the wheel higher velocities occur. This behaviour is counter-intuitive to what was expected from the open vs. closed rim geometry comparison. Here, closing the rim leads to a lower momentum loss at the outward ground vortex. However, this contradiction could once again be attributed to the hub vortex which corrupts the results for the generic rim geometries.

The presented results reveal that the influence of the brake disc on the flow is not as high as expected. The brake disc is shadowed from the approaching flow parallel to the wheel. In real-life full car conditions, however, the approaching flow is yawed due to the deflecting effect of the vehicle’s front and the exiting engine bay flow into the wheelhouses. Here, the effect of a brake disc is known to be more significant.

3.3.2. Different Camber Angles

For the investigations on the influence of the applied camber, no clear trend in the influence of the applied angles of 0.2°, 0.4° and 0.5° could be identified. While the flow fields in the investigated planes show no difference from the baseline realistic rim case without camber, aside reproduction accuracy, an influence on the drag values can be identified. However, the relative influence is at most 2.5% as shown in Table 3. The absolute differences are in the order of magnitude of the rolling resistance; furthermore, no unambiguous effect of the camber is measured. While the smallest applied camber leads to the largest drag reduction, the largest camber gives almost no reduction, compared to the baseline case. Consequently, no ascertained conclusions on the influence of camber can be drawn here.

On first sight, this observation contradicts the one made by Knowles et al. When using an undeformable 40% scale wheel, a considerable change in flow field and drag (+12%) was observed\(^{[17]}\). In a later publication, the same authors attribute the changes mainly to the change in the contact patch size\(^{[19]}\). In the case presented here, however, the undeformable tyre is coated at the tread with a slightly deformable foam to minimise wear. This foam is squashable and can easily compensate for the 0.6mm of height difference between the two shoulders at a camber of 0.5°. Consequently, the contact patch remains identical.

Additionally, the maximum applicable camber of 0.5° is considerably lower than the 4° applied by Knowles et al.\(^{[17]}\). It is therefore probable that the mechanisms causing a change in the flow field around the wheel, and the larger changes in drag due to camber, cannot be triggered with the used facilities.

Table 3 Force coefficients of the realistic rim geometry at different camber angles (*All values are corrected by the calculated rolling resistance)

| Configuration         | \(C_D\) | \(\Delta C_D\) |
|-----------------------|---------|---------------|
| Baseline no camber    | 0.571*  | -             |
| 0.2° camber           | 0.557*  | -0.014        |
| 0.4° camber           | 0.558*  | -0.013        |
| 0.5° camber           | 0.568*  | -0.003        |

4. CFD Results

In order to evaluate the accuracy of the applied CFD setup, the baseline case, namely the closed generic rim, was investigated using the RWBC for wheel rotation. For the assessment of the validity of the SM approach, the open generic rim was selected for simulation. Both setups included the wheel strut from the experimental setup. For brevity, only the results of the rotating cases with moving ground are shown.

From Fig. 8, it can be seen that, for the closed generic rim, the wake structure is predicted to be more symmetric than it is evident from the PIV results shown in Fig. 4. This can be mainly attributed to the missing hub vortex which is hardly covered in the CFD results. Consequently, the location of the CVP near the ground is not properly resembled in the CFD results either. Both vortices have drifted too far away from the sides of the wheel and...
the inner vortex's location is too elevated. However, at least the CVP is predicted by CFD and, furthermore, the elevation of the area of momentum loss in the wake is covered well.

For the open generic rim, the prediction accuracy is slightly improved. From Fig. 9, it is evident that the hub vortex is covered better, also the asymmetry of the wake compared to the PIV results in Fig. 5 is slightly clearer than for the closed rim. Nevertheless, the location of the inner vortex of the CVP is predicted far too high and the momentum loss in the wake is too big. However, for the open rim, the dominant vortical structures are covered and, once again, the vertical extent of the wake region is covered well.

The impression of the improved prediction accuracy for the generic open rim is confirmed when comparing the measured and calculated drag values in Table 4. While the relative error for the closed rim is as high as 14.9%, the relative error for the open rim is only 3.7%.

Table 4 Measured force coefficients of the generic rim geometries compared to CFD results (*Measured values are corrected by the calculated rolling resistance)

| Configuration                  | $C_D$ | $\Delta C_D$ |
|-------------------------------|-------|--------------|
| Closed generic rim rotating Exp. | 0.524 | -            |
| Closed generic rim rotating CFD     | 0.602* | +0.078      |
| Open generic rim rotating Exp.    | 0.563* | -            |
| Open generic rim rotating CFD     | 0.542 | -0.021       |

In conclusion, from the presented CFD results it can be stated that the used numerical basis setup still offers room for improvements as regards the reproduction quality of the baseline case. Coverage of the hub vortex seems essential. However, it appears that the used Sliding Mesh rotation modelling approach does not interfere with the accuracy of the numerical simulations. Quite the contrary, the application of a rotating mesh region around the openings in the rim actually improves the CFD results. It should be noted that the application of the SM approach to the single wheel configuration leads to an increase in computational cost to a factor of 4.2. For multiple rotating regions, like the four rotating wheels of a car, this additional effort is expected to be even larger. This overhead seems questionable in respect of the application in serial development of a car manufacturer, but for investigations around the rotating wheels, this additional effort seems to be inevitable to cover the governing flow physics.

5. Conclusions

From the presented experimental and computational studies, a set of conclusions can be drawn for the presented single wheel setup. From the experimental results, it can be said that it is possible to identify a counter-rotating vortex pair (CVP) near the ground for all the investigated wheel geometries, but a third hub vortex is identified at the outward side of the rim for the generic geometries. It is very likely that this vortex is caused by the fixing clamp of this assembly. The hub vortex is not present for the realistic rim without fixing clamp. It leads to a strong asymmetry in the wake of the generic wheel geometries. The absence of the hub vortex for the realistic rim geometry leads to a more symmetric wake with the CVP, as has been observed by other researchers e.g. (4),(9),(10).

Contrary to expectations, the application of a model brake disc has a minor effect on the flow field. Presumably, this is due to the parallel approaching flow. The effect of a brake disc should be tested using either a model car front around the wheel, or at least a brake cooling duct. A distinguishable camber effect cannot be identified with the investigated setup. The main changes emerge from changes in the contact patch and tread deformation at higher camber angles (9).

For the computational results, the computed flow field and drag results for the baseline case predict the PIV results only rudimentary. Consequently, the baseline CFD setup still requires revision before applying it in an OEM’s aerodynamic development process. The application of the SM approach to the open rim geometry, however, does improve prediction accuracy compared to the baseline case and should be used for open rim geometries if high-quality simulation results are required.
6. Summary and Outlook

In the presented work, multiple influences on the flow around a model scale wheel have been investigated. The created data can serve as a validation base for CFD simulations. The first CFD setup still offers considerable room for improvement compared to the PIV results. Especially in the case of the closed generic rim, where the hub vortex is completely missed out. However, the application of the SM approach for wheel rotation seems promising. In future work, the CFD setup will be revised for better accuracy in order to cover all relevant vortical structures and to be able to figure out, why the hub vortex is missed out for the closed generic rim with the presented CFD setup. This will be done by, e.g. evaluation of a different turbulence model (kω-SST) for the RANS region of the DDES; as soon as the desired accuracy is reached, further insights on the setup are expected.

Acknowledgements

The authors would like to thank AUDI AG, especially Dr. M. Islam, for providing the opportunity to carry out this project. We would also like to thank the workshop staff at the TUM (W. Lützenburg, H.-G. Frimberger, M. Banzer) who helped with the WKA facilities and wherever a hand was needed. We also would like to thank our students. Parts of the computational resources for this project were provided by Leibniz Supercomputing Centre under grant: PR48GE. This publication is supported by TUM Graduate School’s Faculty Graduate Centre of Mechanical Engineering.

References

(1) Schütz, T. (Ed.). Hucho-Aerodynamik des Automobils. Springer-Verlag. (2012)
(2) Schnepf, B. et al. Further Investigations on the Flow Around a Rotating, Isolated Wheel with Detailed Tread Pattern. SAE Int. Journal of Passenger Cars, Vol. 8 No. 1, p. 261-274. (2015)
(3) Wickern, G. et al. Rotating wheels - their impact on wind tunnel test techniques and on vehicle drag results. SAE Technical Paper No. 970133. (1997)
(4) Wäschle, A. The Influence of Rotating Wheels on Vehicle Aerodynamics - Numerical and Experimental Investigations. SAE Technical Paper No. 2007-01-0107. (2007)
(5) Schütz, T. Aerodynamics of Modern Sport Utility Vehicles. Proceedings of the 8th MIRA Int. Vehicle Aerodynamics Conference – Low Carbon Vehicles, Grove, UK 13-14th October ISBN: 978-1-62276-977-3. (2010)
(6) Mack, S. et al. The Ground Simulation Upgrade of the Large Wind Tunnel at the Technische Universität München. SAE Technical Paper No. 2012-01-0299. (2012)
(7) Mack, S. et al. The interior design of a 40% scaled DrivAer body and first experimental results. In ASME 2012 Fluids Engineering Division Summer Meeting. American Society of Mechanical Engineers. (2012)
(8) Hövelmann, A. et al. AVT-183 diamond wing flow field characteristics Part 2: Experimental analysis of leading-edge vortex formation and progression. Aerospace Science and Technology, Vol. 57, p. 31-42. (2015)
(9) Saddington, A. J. et al. Laser Doppler anemometry measurements in the near-wake of an isolated Formula One wheel. Experiments in fluids, Vol. 42 No. 5, p.671-681. (2007)
(10) Axerio-Cilies, J. A. Predicting Formula 1 tire aerodynamics: sensitivities, uncertainties and optimization. PhD Thesis, Stanford University. (2012)
(11) Spalart, P. R. et al. A new version of detached-eddy simulation, resistant to ambiguous grid densities. Theoretical and comp. fluid dynamics, Vol. 20 No. 3, p. 181-195. (2006)
(12) Islam, M. et al. Application of detached-eddy simulation for automotive aerodynamics development. SAE Technical Paper, No. 2009-01-0333. (2009)
(13) Shih, T. H. et al. A generalized wall function for complex turbulent flows. Proceedings of the 8th Aerospace Sciences Meeting and Exhibit sponsored by the AIAA. (1999)
(14) Luo, J. Y. et al. Prediction of impeller-induced flow in mixing vessels using multiple frames of reference. IChemE Symposium Series. Vol. 136, p. 449-456. (1994)
(15) Hobeika, T et al. Study of different tyre simulation methods and effects on passenger car aerodynamics. Proceedings of the IMechE International Vehicle aerodynamics Conference. p. 187-195. ISBN: 978-0-08-100199-8. (2014)
(16) Hucho, W. H. Aerodynamik der stumpfen Körper. Vieweg & Sohn, Wiesbaden. (2002)
(17) Knowles, R. et al. On the near wake of rotating, 40%-scale Champ Car wheels. SAE Technical Paper No. 2002-01-3293. (2002)