On the Influence of Ride Height Changes on the Aerodynamic Performance of Wheel Designs

Bastian Schnepf 1)  Gregor Tesch 2)  Thomas Indinger 3)

1),3) Technical University of Munich, Institute of Aerodynamics and Fluid Mechanics, Boltzmannstr. 15, 85748 Garching bei Muenchen, Germany (E-mail: bastian.schnepf@tum.de)
2) BMW Group, Aerodynamics, 80788 Muenchen

Presented at the JSAE Annual Congress on May 21, 2014

Received on June 7, 2014

ABSTRACT: On the road a passenger car’s ride height is elevated both by the radial expansion of the tires due to centrifugal forces and aerodynamic lift. Wheel size and design influence these forces and therefore may affect aerodynamic drag more than predicted using fixed-ride-height tools. In this study, on-road ride height and surface pressure measurements for different wheel designs on a BMW 3 Series sedan are compared to wind tunnel tests and numerical simulations. Compared to still conditions, the vehicle is elevated by 5 to 7 mm when driving at 140 kph. This ride height change increases drag by 4 counts in the wind tunnel. However, the drag differences between the specific wheel designs are only altered marginally. Using CFD, areas sensitive to wheel designs are identified and analyzed. Furthermore, lift differences between the wheels are explained by the vehicle’s pressure distribution.

KEY WORDS: heat • fluid, aerodynamic performance, computational fluid dynamics, wind tunnel test, wheel, ride height [D1]

1. Introduction

Wheels and wheel houses are responsible for approximately 25 percent of a passenger car’s total aerodynamic drag. (1) Given upcoming regulations on CO2 emissions and the need for long-range electric vehicle concepts, this area has to be optimized aerodynamically in more detail than in the past. Furthermore, aerodynamic front and rear axle lift, which influence the driving stability and maneuverability, also depend on the wheel design. Wäschle(2), Modlinger(3), Landström(4) and others have pursued research in this area over the last years but still have not covered all aspects of this topic.

Ride height is a crucial parameter for the aerodynamic performance of a vehicle. An increase in ride height leads to higher drag. On the road, ride height is, however, not only influenced by the static load but also by velocity dependent forces. In fact, a vehicle is elevated both by the radial expansion of the tires due to centrifugal forces and by aerodynamic lift forces. Wheel size and design influence these forces. If the wheels consequently affect the dynamic elevation of the vehicle, they may have a larger effect on the aerodynamic performance than standard analysis tools would predict. Neither numerical simulations nor wind tunnel (WT) setups in which the model’s ride height is fixed throughout the measurement will reproduce the interaction of ride height variations with the flow around a vehicle and the resulting aerodynamic drag.

The purpose of this study is to clarify the influence of wheel-induced ride height changes on aerodynamic force deltas between wheel designs. Consequently, the authors investigated on-road ride height changes for three wheel designs on the proving ground and fed them back into both numerical simulations and WT tests. Furthermore, surface pressure distributions recorded on the test track at the identical vehicle are used for validating WT and CFD results.

2. Experimental Setup

2.1. Test vehicle

A 2012 BMW 3 Series sedan was equipped with three laser sensors measuring the distances to the road from a) front left wheel axle, b) front left underbody (body-fixed), and c) rear left underbody (body-fixed). In this way, ride height and pitch of the vehicle body, as well as the elevation of the wheel center from the ground could be captured. Moreover, out of this data the relative motion between wheel axle and body could be calculated. This data then was used to position the vehicle body and the wheels correctly in the WT and in the CFD setup.

In order to record the differences in surface pressure distribution between the wheel designs, holes of 1 mm diameter were drilled in the vehicle’s skin and then tubed to a 64-port pressure scanner. The holes are placed in regions sensitive to changes in wheel geometry that were identified through preliminary CFD investigations. All of them are located on the left side of the vehicle in order to achieve the best local resolution out of 64 available channels. Mounted in the middle of the license plate, a Pitot tube is able to measure the total pressure of the free-stream for the calculation of dimensionless pressure coefficients (Cp). In Figure 1, the distribution of measurement locations is displayed. Most of them are positioned in the vicinity of the left front wheel, but also at the rear wheel house, on the underbody and on the rear base.

In Figure 2 the three wheel designs investigated in this study are shown. They cover a wide range of drag and lift values, thus, allowing for representative statements regarding a typical wheel portfolio. All of them were tested on 225/45 R18 Pirelli P7 Cinturato run-flat tires in order to eliminate tire influences in this part of the study. In general, however, tire influence is not negligible since details like the design of a rim protection edge...
can affect the differences in aerodynamic performance between two wheels. Wittmeier\(^5\) has studied the aerodynamic properties of tires and their interaction with wheels extensively.

It is important to note that the vertical position of the vehicle body is fixed during the WT measurements. Hence, aerodynamic lift forces or tire expansion cannot elevate the body. The wheels are, however, mounted on the wheel axle in the common way, therefore being able to move vertically. As a result the distance between the wheel house shells and the tire decreases with increasing velocity.

2.3. On-road tests

On the fast course of the BMW testing site Aschheim (Fig. 4), the different wheel designs were evaluated at a velocity of 140 kph according to GPS measurements. There are two straights in opposite directions on which laser and pressure signals could be recorded and averaged over a period of 30 s. The average of three laps (six straights) was used as the reference for the comparison with WT and CFD results. Only when windsocks at the test track signaled calm conditions, measurements were accepted.

The vehicle’s elevation at 140 kph compared to still condition is of major interest in this study. Therefore, before and after each lap the car was driven at a low speed of 10 kph in order to record a “zero” level for the ride height sensors. At a complete stop imperfections on the road’s surface like small stones would have distorted the laser measurements leading to unrealistic values. Rolling at 10 kph, however, an average ride height could be determined without significantly altering the values in comparison to still conditions. On the straights the unsteady laser signals showed a standard deviation of 0.2 mm at the front wheel, 0.5 mm at the front body and 1.5 mm at the rear body when driving at 140 kph.
Regarding the loading of the vehicle, deviations from the loading standard (DIN 70020 / 1) had to be accepted as listed in Table 1. The resulting ride height in operating conditions at 140 kph differed from the theoretical CAD position of the vehicle. The deviations were, however, smaller than 3 mm and furthermore depended on the wheel geometry. In this study the focus is put on force differences that are caused by the ride height deviations. Therefore, small ride height differences would produce only small force differences. The exact values are given in section 4.1. Ride height.

The numerical simulations of this study were conducted using the commercial CFD solver Exa PowerFLOW 5.0. Its computations are based on the Lattice-Boltzmann method which derives the macroscopic flow variables like density, momentum and energy from microscopic particle distributions following the Boltzmann kinetic theory. This method is inherently transient and discretizes the particles’ motion both in velocity and direction on an equidistant, cubic lattice.\(^{(7)}\)

Since the spectrum of turbulent time and length scales is very large in the case of automotive aerodynamics a direct numerical simulation (i.e. resolving all turbulent scales) is not possible with today’s computational resources. Instead, the smallest turbulent scales are modeled using an enhanced two-equation RNG k-\(\varepsilon\) turbulence model. The boundary layer is modeled according to the logarithmic law of the wall, additionally taking into account the effect of pressure gradients on flow separation.\(^{(7)}\)

For this study, the detailed vehicle geometry consists of 10 million surface elements and is situated in a computational domain being 208 m long, 175 m wide and 138 m high. This lead to a blockage of 0.01 percent which is three orders smaller compared to the WT experiment and is supposed to be equivalent to idealized on-road conditions. At the inlet and on the moving floor a velocity of 140 kph was prescribed leading to a Reynolds number of 7 million, calculated using the wheelbase length. The finest voxel size is 1.5 mm around the wheels, brake discs, wheel spoilers and grills and 3 mm around the rest of the model including all entire wheel house volumes. Using 10 levels of resolution the grid was coarsened with growing distance from the car. In total, the discretization of the domain resulted in 150 million voxels and 30 million surfels (i.e. discretized volume and surface elements, respectively). Since only the finest scale participates in every time step’s computation, a so-called fine equivalent (fe) of voxels and surfels is determined for resources considerations by weighting voxels and surfels corresponding to their levels of resolution. Consequently, a total of 90 million fe voxels and 20 million fe surfels is attained for the case. The time step which is determined by the maximum expected fluid velocity and the smallest voxel size is \(5.3 \times 10^6\) s resulting in 376,000 time steps to be calculated for 2.0 s of physical time. Averaging the transient results over the last 1.6 s an accuracy of +/- 0.001 for the drag coefficient \(c_d\) and +/- 0.003 for the axle lift coefficients \(c_{l,1}\) (front) and \(c_{l,2}\) (rear) is achieved. This was determined out of the force signal of a 20 sec simulation at reduced resolution for a confidence level of 90\%. In total, 20,000 core hours were consumed for one simulation run.

3. Numerical Method

The numerical simulations of this study were conducted using the commercial CFD solver Exa PowerFLOW 5.0. Its computations are based on the Lattice-Boltzmann method which derives the macroscopic flow variables like density, momentum and energy from microscopic particle distributions following the Boltzmann kinetic theory. This method is inherently transient and discretizes the particles’ motion both in velocity and direction on an equidistant, cubic lattice.\(^{(7)}\)

Since the spectrum of turbulent time and length scales is very large in the case of automotive aerodynamics a direct numerical simulation (i.e. resolving all turbulent scales) is not possible with today’s computational resources. Instead, the smallest turbulent scales are modeled using an enhanced two-equation RNG k-\(\varepsilon\) turbulence model. The boundary layer is modeled according to the logarithmic law of the wall, additionally taking into account the effect of pressure gradients on flow separation.\(^{(7)}\)

For this study, the detailed vehicle geometry consists of 10 million surface elements and is situated in a computational domain being 208 m long, 175 m wide and 138 m high. This lead to a blockage of 0.01 percent which is three orders smaller compared to the WT experiment and is supposed to be equivalent to idealized on-road conditions. At the inlet and on the moving floor a velocity of 140 kph was prescribed leading to a Reynolds number of 7 million, calculated using the wheelbase length. The finest voxel size is 1.5 mm around the wheels, brake discs, wheel spoilers and grills and 3 mm around the rest of the model including all entire wheel house volumes. Using 10 levels of resolution the grid was coarsened with growing distance from the car. In total, the discretization of the domain resulted in 150 million voxels and 30 million surfels (i.e. discretized volume and surface elements, respectively). Since only the finest scale participates in every time step’s computation, a so-called fine equivalent (fe) of voxels and surfels is determined for resources considerations by weighting voxels and surfels corresponding to their levels of resolution. Consequently, a total of 90 million fe voxels and 20 million fe surfels is attained for the case. The time step which is determined by the maximum expected fluid velocity and the smallest voxel size is \(5.3 \times 10^6\) s resulting in 376,000 time steps to be calculated for 2.0 s of physical time. Averaging the transient results over the last 1.6 s an accuracy of +/- 0.001 for the drag coefficient \(c_d\) and +/- 0.003 for the axle lift coefficients \(c_{l,1}\) (front) and \(c_{l,2}\) (rear) is achieved. This was determined out of the force signal of a 20 sec simulation at reduced resolution for a confidence level of 90\%. In total, 20,000 core hours were consumed for one simulation run.

3. Numerical Method

The numerical simulations of this study were conducted using the commercial CFD solver Exa PowerFLOW 5.0. Its computations are based on the Lattice-Boltzmann method which derives the macroscopic flow variables like density, momentum and energy from microscopic particle distributions following the Boltzmann kinetic theory. This method is inherently transient and discretizes the particles’ motion both in velocity and direction on an equidistant, cubic lattice.\(^{(7)}\)

Since the spectrum of turbulent time and length scales is very large in the case of automotive aerodynamics a direct numerical simulation (i.e. resolving all turbulent scales) is not possible with today’s computational resources. Instead, the smallest turbulent scales are modeled using an enhanced two-equation RNG k-\(\varepsilon\) turbulence model. The boundary layer is modeled according to the logarithmic law of the wall, additionally taking into account the effect of pressure gradients on flow separation.\(^{(7)}\)

For this study, the detailed vehicle geometry consists of 10 million surface elements and is situated in a computational domain being 208 m long, 175 m wide and 138 m high. This lead to a blockage of 0.01 percent which is three orders smaller compared to the WT experiment and is supposed to be equivalent to idealized on-road conditions. At the inlet and on the moving floor a velocity of 140 kph was prescribed leading to a Reynolds number of 7 million, calculated using the wheelbase length. The finest voxel size is 1.5 mm around the wheels, brake discs, wheel spoilers and grills and 3 mm around the rest of the model including all entire wheel house volumes. Using 10 levels of resolution the grid was coarsened with growing distance from the car. In total, the discretization of the domain resulted in 150 million voxels and 30 million surfels (i.e. discretized volume and surface elements, respectively). Since only the finest scale participates in every time step’s computation, a so-called fine equivalent (fe) of voxels and surfels is determined for resources considerations by weighting voxels and surfels corresponding to their levels of resolution. Consequently, a total of 90 million fe voxels and 20 million fe surfels is attained for the case. The time step which is determined by the maximum expected fluid velocity and the smallest voxel size is \(5.3 \times 10^6\) s resulting in 376,000 time steps to be calculated for 2.0 s of physical time. Averaging the transient results over the last 1.6 s an accuracy of +/- 0.001 for the drag coefficient \(c_d\) and +/- 0.003 for the axle lift coefficients \(c_{l,1}\) (front) and \(c_{l,2}\) (rear) is achieved. This was determined out of the force signal of a 20 sec simulation at reduced resolution for a confidence level of 90\%. In total, 20,000 core hours were consumed for one simulation run.

The numerical simulations of this study were conducted using the commercial CFD solver Exa PowerFLOW 5.0. Its computations are based on the Lattice-Boltzmann method which derives the macroscopic flow variables like density, momentum and energy from microscopic particle distributions following the Boltzmann kinetic theory. This method is inherently transient and discretizes the particles’ motion both in velocity and direction on an equidistant, cubic lattice.\(^{(7)}\)

Since the spectrum of turbulent time and length scales is very large in the case of automotive aerodynamics a direct numerical simulation (i.e. resolving all turbulent scales) is not possible with today’s computational resources. Instead, the smallest turbulent scales are modeled using an enhanced two-equation RNG k-\(\varepsilon\) turbulence model. The boundary layer is modeled according to the logarithmic law of the wall, additionally taking into account the effect of pressure gradients on flow separation.\(^{(7)}\)

For this study, the detailed vehicle geometry consists of 10 million surface elements and is situated in a computational domain being 208 m long, 175 m wide and 138 m high. This lead to a blockage of 0.01 percent which is three orders smaller compared to the WT experiment and is supposed to be equivalent to idealized on-road conditions. At the inlet and on the moving floor a velocity of 140 kph was prescribed leading to a Reynolds number of 7 million, calculated using the wheelbase length. The finest voxel size is 1.5 mm around the wheels, brake discs, wheel spoilers and grills and 3 mm around the rest of the model including all entire wheel house volumes. Using 10 levels of resolution the grid was coarsened with growing distance from the car. In total, the discretization of the domain resulted in 150 million voxels and 30 million surfels (i.e. discretized volume and surface elements, respectively). Since only the finest scale participates in every time step’s computation, a so-called fine equivalent (fe) of voxels and surfels is determined for resources considerations by weighting voxels and surfels corresponding to their levels of resolution. Consequently, a total of 90 million fe voxels and 20 million fe surfels is attained for the case. The time step which is determined by the maximum expected fluid velocity and the smallest voxel size is \(5.3 \times 10^6\) s resulting in 376,000 time steps to be calculated for 2.0 s of physical time. Averaging the transient results over the last 1.6 s an accuracy of +/- 0.001 for the drag coefficient \(c_d\) and +/- 0.003 for the axle lift coefficients \(c_{l,1}\) (front) and \(c_{l,2}\) (rear) is achieved. This was determined out of the force signal of a 20 sec simulation at reduced resolution for a confidence level of 90\%. In total, 20,000 core hours were consumed for one simulation run.

Wheel spokes and vented brake discs are included inside sliding mesh regions, thus effectively rotating during the simulation. The setup moreover includes laser-scanned tire geometries. They were captured isolatedly from tires, mounted on a wheel arm\(^{(8)}\), at a load of 4,200 N and 140 kph in the WT. The resulting tire geometry, therefore, is deformed both by static and...
centrifugal forces. Being scanned during rotation the tire sections are rotationally averaged thus only including the longitudinal main grooves. Since the tires’ contact patches are deformed they cannot be included inside sliding mesh regions. Instead, they are modeled as rotating walls, i.e. the tangential velocity at the tire surface is prescribed. Differently from the experiments camber and toe-in of wheels and brake discs are set to zero to fit the tire models that had to be scanned at zero camber.

4. Results

4.1. Ride height

Comparing ride heights between still conditions (10 kph, rolling) and driving at 140 kph, Table 2 shows differences in distances for the investigated wheel designs. The front wheel center is lifted by 2.5 mm, independently of the wheel geometry. In the WT the same value is attained. Since the vehicle body is fixed in the WT aerodynamic lift forces attacking at the body are not responsible. Instead, centrifugal forces that expand the tire radially cause the elevation of the wheel. It could be noticed that the warming of the tires during a measurement increased the inflation pressure and thus additionally elevated the vehicle by 1 mm. This was also observed by Wittmeier et al.\(^5\). For the front part of the vehicle body an elevation between 5 mm (wheel design 416) and 7 mm (wheel design 416 Cover) compared to still conditions could be observed. The rear was not lifted as high as the front, namely between 1 mm and 2 mm. However, the accuracy of the rear laser is compromised as mentioned in section 2.3. On-road tests.

Table 2  Ride height changes at 140 kph compared to still conditions.

| Wheel      | Wheel center [mm] | RH front [mm] | RH rear [mm] |
|------------|-------------------|---------------|--------------|
| 416        | +2.5              | +5            | +1           |
| 416 Cover  | +2.5              | +7            | +2           |
| 419        | +2.5              | +6            | +2           |

The high absolute value for rear ride height on wheel 419 could not be explained by lift forces (see section 4.2) in this study and needs to be investigated further. A sensitivity analysis in the WT, however, showed no measurable impact of a rear ride height variation of 2 mm. For these reasons the rear ride height is not used for further interpretations in this report.

An overview of the configurations considering wheel geometry and ride height variations is listed in Table 3. On-road ride heights (4 - 6) agree well with the theoretical CAD positions (1 - 3) and the differences to be corrected in WT and CFD setups are smaller than 3 mm. Furthermore, an artificial ride height at still conditions using the experimental vehicle loading was added to the configurations. In that way the influence of driving, and therefore velocity induced elevation on wheel aerodynamics, could be assessed.

### Table 3  Overview of different ride height configurations.

| Conf. | Wheel      | RH front [mm] | RH rear [mm] |
|-------|------------|---------------|--------------|
| CAD   |            |               |              |
| 1     | 416        | Ref.          | Ref.         |
| 2     | 416 Cover  | +0            | -1           |
| 3     | 419        | +2            | +1           |
|       |            | +1            | +2           |
|       |            | -5            | -1           |
| On-road |            |               |              |
|       | 416        |               |              |
|       | 419        |               |              |
| 7     | 419        |               |              |
|       | 416 Cover  |               |              |

4.2. Integral forces

Starting with the drag differences in Table 4, the covered wheel (2) attains significantly lower C\(_D\) values than the uncovered wheel 416 (1). In the WT the drag difference \(\Delta C_D = -0.020\) (20 counts) for the CAD position while the CFD results show a smaller \(\Delta C_D = -0.013\). A similar observation can be made for wheel 419 (3) which yields \(\Delta C_D = -0.015\) in the tunnel and \(\Delta C_D = -0.009\) in CFD. Applying on-road ride height in the WT the drag differences of the covered wheel (5) and wheel 419 (6) to the reference wheel 416 (4) are decreased by 1 count. This can be explained by the front ride height which is lower for the reference wheel (2 and 1 mm respectively). For the ride height in still conditions, the drag differences are identical to the ones obtained in CAD position. Compared to the ride height at 140 kph, the absolute drag value is, however, reduced by 3 to 4 counts.

Analyzing the aerodynamic front axle lift C\(_L1\) the trend behaves in the opposite direction as the C\(_D\) values: the wheels with lower C\(_D\) attain higher C\(_L1\) values. In the WT using CAD ride height C\(_L3\) of the covered wheel and wheel 419 increases by 28 counts and 8 counts respectively. The CFD results confirm this order qualitatively but yield deviating magnitudes.

The lift differences at the vehicle’s rear are smaller and show opposite trends in the WT and in CFD. Whereas for the covered wheel a 10 count increase of C\(_L3\) can be determined in the WT a decrease of 8 counts is observed in CFD. Wheel 419 gives approximately the same C\(_L3\) value as wheel 416 in all configurations.

Combining lift results with measured ride height values a good correlation between C\(_L3\) and front elevation is obtained. In this specific case, an increase of 28 counts C\(_L3\) by covering the wheels causes an additional elevation of the vehicle’s front of 2 mm. Due to the small rear axle lift differences and laser measurement uncertainties this kind of observation cannot be confirmed for the vehicle’s rear.
### Table 4  Integral force coefficients.

| Conf. | $C_D$ | $C_{L1}$ | $C_{L2}$ |
|-------|-------|----------|----------|
|       | WT    | CFD      | WT       | CFD      |
| 1     | -0.020| -0.013   | +0.028   | +0.020   |
| 2     | -0.015| -0.009   | +0.008   | +0.014   |
| 3     |       |          | -0.001   | +0.003   |
| 4     |       |          |          |          |
| 5     | -0.019| -0.013   | +0.030   | +0.017   |
| 6     | -0.014| -0.009   | +0.005   | +0.009   |
| 7     |       |          | +0.000   | +0.000   |
| 8     | -0.020| -0.013   | +0.027   | +0.026   |
| 9     | -0.015| -0.009   | +0.008   | +0.014   |

4.3. Pressure distribution

Being a bluff body, the rear base of the vehicle is the region that has the largest influence on drag. For the investigated wheel designs the measured base pressure differences are, however, in the range of $\Delta C_P = +/-0.005$ which is within measurement accuracy. Expressed as base drag contribution this would result in +/- 2.5 counts $C_D$. Even for covered and open wheel 416 the overall drag difference of 20 counts cannot be correlated with the base pressure distribution. This is in agreement with Landström\(^{(4)}\) who investigated different wheel designs and found that base drag was not a good indicator for their overall drag. Analyzing data from other measurement locations no strong connections to the drag trend are found. Local changes in the pressure distribution can be proven, e.g. at the front shell of the front wheel house (Fig. 6). The gradients in the region above the wheel spoiler are strong and the pressure peaks move on the shell. The integral drag value, however, is only slightly affected. As will be shown in a more detailed CFD analysis in the next section, most of the drag difference is, in fact, generated directly at the wheels.

Regarding impact on the front lift, the static pressure differences compared to the baseline (4) for three regions are presented in Figure 7 and 8 for wheel 416 Cover (5) and 419 (6) respectively. The less ventilation is permitted through the wheel openings the higher the pressure is in and between the front wheel houses. This behavior is proven by the measurements taken in the wheel house and on the underbody (UB) engine bay cover. Furthermore, the reduction of the contact patch separation leads to increased pressure in the tire’s wake that can be measured on the UB covers behind the tire.

Overall, WT and CFD results agree well with the on-road deltas. For the covered wheel, however, the pressure difference on the UB is slightly underpredicted by approx. $\Delta C_P = 0.01$ in the WT and in CFD. This would imply that the lift increase caused by the cover could be stronger on the road than in the WT. Considering the projected areas of the UB shells and their respective levers to the wheelbase center, the following sensitivities for $C_{L1}$ can be approximated for a static pressure increase of $\Delta C_P = 0.01$: wheel house top +0.002, UB engine cover +0.004, UB front wheel wake +0.001. In that way, 7 counts could be added to the $C_{L1}$ value, measured in the WT.
4.3. CFD analysis

As mentioned in the previous section, the discrete pressure measurements on the vehicle’s surface did not reveal strong connections to the overall drag. In CFD, however, not only discrete points but also the whole surface can be examined to understand where drag is generated. In Figure 9, the development of the drag difference is plotted over the vehicle length. At the front, increased static pressure in the engine bay and the wheel houses decreases drag locally at the rearward facing shells, upstream of the front wheels. Downstream of the wheels, however, drag is increased again at the forward facing shells leaving only a slight net reduction of drag. An analysis of the specific drag contributions of the single vehicle parts shows that in the cooling package, modeled as porous media, and around the wheel spoilers drag is additionally reduced by 1 count each. However, the largest share of drag reduction comes from the wheels themselves. The front wheels decrease drag by 6 counts in the case of the covered wheel and 2 counts for wheel 419 while the rear wheels cause a larger drag reduction of 11 and 6 counts respectively. The rear base does not contribute significantly to the drag difference as was already concluded from the experimental data.

At the front shell of the rear wheel house, drag is increased by 3 counts compared to the open wheel case. This is caused by a pressure reduction that is \( \Delta C_P = 0.03 \) stronger than in the WT and on the road. Correcting the pressure level on the wheel house shell using the experimental value would increase the drag difference between open and covered wheel by 2 counts to a total of 15 counts. For the case of wheel 419 (3) this correction would lead to an increase of the drag difference by 1 to 2 counts. Although this correction would bring the CFD results closer to the WT values, for both cases 416 Cover (2) and 419 (3) the drag reduction in CFD would still remain 4 counts lower. As mentioned at the end of section 3, the camber of the wheels was neglected in the CFD setup which could cause a different flow topology, especially at the rear wheels. Therefore, two additional simulations including the correct camber angles by tilting the wheels in the setup were conducted. The results confirmed the simulations run without camber and did not change the force differences.

In his studies Landström\(^{(4)}\) showed that wheel designs interact strongly with the separation originating from the outer side of the front tire’s contact patch. In Figure 10, the flow field around the left front wheel (design 419) is illustrated by three cutting planes (250, 500 and 750 mm downstream of the front wheel axle) and streamlines, all colored by the total pressure distribution. The separation at the contact patch is enclosed by the streamlines and shall be examined for the three designs. Low levels of total pressure indicate irreversible energy losses in the flow field that increase drag. Comparing the slices 500 mm downstream of the front wheel axle in Figure 11, the trend of larger contact patch flow separations corresponding to higher \( C_D \) values can be observed. Closing the wheel, the separation is diminished and consequently drag is reduced which is in accordance with Landström’s findings \(^{(4)}\).

5. Conclusion

Three different wheel designs were investigated by numerical simulation, in the WT and on the proving ground. Compared to still conditions, the vehicle was elevated between 5 and 7 mm at the front and 1 and 3 mm at the rear when driving at...
140 kph. Firstly, this can be explained by centrifugal forces expanding the tire radially, and thus lifting the wheel center by 2.5 mm. Secondly, aerodynamic lift forces, depending on the wheel design, elevate the vehicle in addition. More specifically, on the sedan tested in this study, 30 counts difference in front lift between open and covered wheel lead to a ride height difference of 2 mm between the configurations. Applying the measured on-road ride heights in the WT setup, the drag of the covered wheel configuration increases slightly. Thus, the drag difference is reduced from 20 counts to 19 counts. Regarding both integral forces and surface pressure distributions, the deltas between the wheels show only light dependencies on the ride height although the absolute drag values are changed by up to 4 counts. Nevertheless, there are wheel designs that show larger differences in front axle lift of 50 counts and more. Extrapolating the trend, the vehicle’s front can be elevated by up to 5 mm leading to a drag difference of 2 to 3 counts. Depending on the level of accuracy demanded of the aerodynamic wheel assessment process, differences of that order would have to be expected.

6. Future Actions

In the context of the research project, that this study has been part of, the authors will investigate the flow topology around wheels in more detail, also evaluating different ground simulation techniques in the WT. Moreover, unresolved issues like the measurement of the rear ride height on the road will be revisited. Also following this study, the authors will investigate the influence of tires on wheel aerodynamics and their accurate modeling in the numerical simulation. Eventually, a strategy to assess wheel and tire aerodynamics numerically by CFD during the vehicle development process will be developed.

Acknowledgements

This publication was created within the TUM Graduate School.

The authors gratefully acknowledge the Gauss Centre for Supercomputing e.V. (www.gauss-centre.eu) for funding this project by providing computing time on the GCS Supercomputer SuperMUC at Leibniz Supercomputing Centre (LRZ, www.lrz.de).

References

(1) Wickern, G.; Zwicker, K.; Pfadenhauer, M. Rotating Wheels – Their Impact on Wind Tunnel Test Techniques and on Drag Results. SAE Technical Paper 970133. (1997).
(2) Wäschle, A. Numerische und experimentelle Untersuchung des Einflusses von drehenden Rädern auf die Fahrzeugaerodynamik. Dissertation. Universität Stuttgart. (2006).
(3) Modlinger, F.; Dernuth, R.; Adams, N. New Directions in the Optimization of the Flow Around Wheels and Wheel Arches. 7th MIRA International Vehicle Aerodynamics Conference, 15-16 October. Warwickshire, UK. (2008).
(4) Landström, C. Passenger Car Wheel Aerodynamics. PhD thesis. Chalmers University. (2012).
(5) Wittmeier, F.; Widdecke, N.; Wiedemann, J. Reifenentwicklung unter aerodynamischen Aspekten. FAT Schriftenreihe Nr. 252, Forschungsvereinigung Automobiltechnik, VDA. (2013).
(6) Duell, E.; Kharazi, A.; Muller, S. et al. The BMW AVZ Wind Tunnel Center. SAE Technical Paper 2010-10-0118. (2010).
(7) Exa Corporation. PowerFLOW® User’s Guide Release 5.0. (2013).
(8) Schnepf, B.; Tesch, G.; Indinger, T. Investigations on the Flow Around Wheels Using Different Road Simulation Tools. In: Wiedemann, J. (ed.): Progress in Vehicle Aerodynamics and Thermal Management, Proceedings of the 9th FKFS Conference (Stuttgart, Germany, October 01-02, 2013). Expert Verlag. (2013).