Moving Ground Simulation for High Performance Race Cars in an Automotive Wind Tunnel

- CFD Approach on Moving Belt Dimensions -

Christopher Collin 1) Thomas Indinger 2) Jörg Müller 3)

1-2) Technical University of Munich, Institute of Aerodynamics and Fluid Mechanics
Boltzmannstr. 15, 85748, Garching b. München, Germany (E-mail: christopher.collin@aer.mw.tum.de)
3) AUDI AG
85045, Ingolstadt, Germany

Received on July 1, 2016

Presented at the JSAE Annual Congress on May 27, 2016

ABSTRACT: For the wind tunnel testing of race car aerodynamics, it is essential to cover the real underbody flow field characteristics, as they affect the aerodynamic balance of race cars and, hence, the transferability of wind tunnel measurements to on-road conditions. In this study, numerical simulations have been performed to determine the influence of moving belt dimensions on the aerodynamic forces for a particular race car. The results are compared with an infinitely large moving ground in open road conditions. Thus, the requirements for the dimensioning of a single belt moving ground simulation in wind tunnels for race car aerodynamics are derived.

KEY WORDS: Heat - fluid, Wind Tunnel, Moving Ground Simulation, Motorsport Aerodynamics [D1]

1. Introduction

Aerodynamic performance is one of the crucial factors of many modern race cars since it often acts as a key driver for the specific car design. Today, computational fluid dynamics (CFD) and wind tunnel testing are the main tools in the development of race car aerodynamics. However, the scenario of CFD entirely replacing the use of wind tunnels is not yet expected in the foreseeable future. Aerodynamic development using the wind tunnel often remains the last step before testing a specific configuration on the race track. Thus, transferable results from wind tunnel to track testing are essential for a cost-effective development. Additionally, wind tunnel hours are restricted in many racing series today.

One of the main objectives of wind tunnel testing is to provide reliable data regarding the overall aerodynamic forces acting on the model. In particular, the aerodynamic efficiency, also known as the lift-to-drag ratio \( C_{L,F} / C_{D} \), as well as the aerodynamic balance of front lift \( C_{L,F} \) and rear lift \( C_{L,R} \) predominate the definition of the final setup of the car for a specific race track. For ground vehicles, the underfloor design turned out to be an excellent parameter in order to produce considerable down force with high aerodynamic efficiency. Due to the transformation of relative motion between fluid, ground and vehicle in an automotive wind tunnel, this entails the requirement of a moving ground simulation for the exact representation of real underfloor flow conditions.

Compared to passenger cars, high performance race cars are a lot more sensitive with respect to the realistic representation of the underfloor flow conditions by moving ground simulation techniques. Consequently, modern motorsport wind tunnels are typically equipped with a single-belt rolling road system, whereas 5-belt systems are expected to provide sufficient accuracy in flow conditions for a wide range of typical passenger cars. It is worth mentioning that 5-belt and single-belt systems are merely two possible methods to apply moving ground simulation in an automotive wind tunnel. Additional practical solutions are a 3-belt or a T-belt system, as investigated by Hennig et al. \(^{(1)}\), e. g. they show the results of a numerical approach in which significant differences in aerodynamic coefficients were recognized for both different car types and techniques of a moving ground simulation. However, in the present study, only the single-belt configuration was investigated in detail. In particular, the moving belt dimensions, such as length and width, were varied in order to obtain the necessary belt dimensions for the selected car geometry. Therefore, a detailed version of an Audi RS5 DTM was chosen, as it is expected to be an appropriate representation of a race car with a considerable ground effect. A more thorough description of the model is provided in section 2.4.

It is well known that the integration of a moving ground simulation system in an automotive wind tunnel is always connected to a careful treatment of the boundary layer profile in order to ideally achieve a block profile of the oncoming flow. However, this issue was eliminated for the present study, as the wind tunnel flow was set to be a slip wall until the beginning of the moving belt.

2. CFD Simulations

2.1. Geometry

The focus of this work is on the evaluation of interference effects for the dimensions of a single-belt moving ground system. However, a comparison of results for both a 5-belt and single-belt system was performed as well. Regarding the representation of
the “real” flow conditions in the underfloor region, the single-belt
is expected to be the best solution for a modern motorsports wind
tunnel. Thus, the following studies on moving belt dimensions are
limited to those of single-belt systems. To isolate the effects of the
moving ground simulation, the complexity of the CFD domain
was reduced. According to that, no specific wind tunnel geometry
was included. The simulations run in a rectangular shaped box for
which the blockage ratio was smaller than 0.5% and, hence,
negligible. Furthermore, the velocity profile of the oncoming flow
was idealized since the floor boundary was set as a slip wall and,
therefore, no boundary layer profile was produced until the beginning
of the moving belt. Further downstream, the wind tunnel floor
was modeled as a friction floor and, depending on the region, it
was either static or a moving wall (belt).

Therefore, the rear end belt length \( L_{B,r} \) as well as the belt
width \( W_B \) were examined. The approach to determine the effects
of \( L_{B,r} \) is shown in Fig. 1. For all those simulations, the front end
belt length \( L_{B,f} \) was fixed at 0.9 as well as the belt width \( W_B/W_M = 1.6 \)
were held to be constant. The belt dimensions refer to the length
\( L_M \) and width \( W_M \) of the race car (model). Based on these
conditions, the rear end length \( L_{B,r} \) was varied in the range of
0.3% \( L_{B,r} \leq L_M \leq 2.0 \). Therefore, the restriction for the shortest
belt \( L_{B,r}/L_M = 0.3 \) was given by the rear tyres.

The second part of the study focused on the belt width \( W_B \).
Based on the investigations on the belt length, a reference belt
length was defined with \( L_{B,f}/L_M = L_{B,r}/L_M = 0.9 \). This was
essential since \( L_{B,f} \) defines the beginning of boundary layer
growth (static floor) laterally along the moving belt. Again, the
floor boundaries were selected as stated above. Fig. 2 depicts four
selected widths \( W_B \) of the moving belt. Overall, the width was
varied in the range \( 0.98 \leq W_B/W_M \leq 5.00 \), whereas the minimum
was once again limited by the dimensions of the car.

The CFD simulations were performed as Delayed Detached
Eddy Simulations (DDES) \(^{(2)}\) by using an enhanced version of the
OpenFOAM® package 2.3.1 \(^{(3)}\). For turbulence modeling, the one
equation turbulence model introduced by Spalart and Allmaras in
1992 was used \(^{(4)}\). A validation of the DDES model applied to
automotive aerodynamics was given by Islam et al. \(^{(5)}\). A DDES
simulation, a hybrid RANS-LES approach, was used employing
the Spalding wall function for the turbulent viscosity \(^{(6)}\). For the
discretization in time, a second order implicit backward scheme
was applied. The velocity field was discretized as a convective
scheme using a second order central differencing, which was
blended with a second order upwind scheme. This blending
mechanism is based on the local CFL number to achieve a stable
simulation. However, due to the selection of a time step
\( \Delta t = 5 \times 10^{-5} \) s the maximum occurring CFL number was smaller
than 8 and, hence, the regions of blended cells were moderate. For
each simulation, a physical time of 10 s was covered, whereas the
unsteady fields were averaged in the last 7 s.

The aerodynamic forces of the race car were compared to
experimental data, whereas the differences in both drag and lift
coefficients were smaller than 5% compared to wind tunnel tests
for a Reynolds number of \( Re = 12.4 \) million.

2.3. Mesh Generation

The basic idea of the CFD setup was to create one single
volume mesh on which every simulation was run. The
OpenFOAM integrated automatic mesh generator, snapHexMesh, was used to create the unstructured hexahedral
grid. The variation in belt dimensions was achieved by changing
the boundary conditions on the wind tunnel floor. The main
advantage of this approach was to avoid potential uncertainties
produced by differences of the local grid especially close to solid
boundaries. The smallest cell within this grid had an edge length
of 3.125 mm.
The change in boundary conditions from moving belt to static wind tunnel floor resulted in an increase of average $y_{avg}$ since, at the same time, the near wall mesh was not changed. The final mesh was defined with the target to achieve a $30 \leq y_{avg} \leq 300$. Effectively, the average value was in the range of $y_{avg} \approx 40$ for the moving belt and $y_{avg} \approx 120$ for the static friction floor regions. Therefore, a pre-study was conducted in order to evaluate the sensitivity of the wall model for different values of $y^+$. See Fig. 3, which depicts that the Spalding wall function works reliably in the investigated range of ($40 \leq y^+ \leq 640$). No significant differences in the near wall velocity profiles were also investigated for a larger range of local Reynolds numbers $Re_x$.

As mentioned above, all simulations were performed on the same volume mesh with approximately 90 million cells. The simulations were parallelized on 512 cores, whereas the total run time for each simulation was approx. 130 h.

### 2.4. Boundary Conditions

As stated above, the simulations were performed inside a rectangular-shaped domain for the representation of open road conditions. Thus, no specific wind tunnel geometry was considered. A schematic illustration is shown in Fig. 4 whereas an overview of the boundary conditions is provided by Table 1 including the OpenFOAM® fields $u$, $p$, $\nu_{SGS}$ and $\bar{v}$.

### 2.5. Race Car Configuration

As a representation of a high performance race car that produces a considerably high amount of aerodynamic down force, a detailed model of an Audi RS5 DTM was selected for the present study. This specific type of race car is expected to be an appropriate model to evaluate the effects of a moving ground simulation since the predominant proportion of aerodynamic down force is created by a controlled underfloor flow, especially on the front axle. To enhance the quality of the CFD simulations, several details of the real race car were included. In particular, the CFD model contains all relevant underfloor components such as front splitter, skid block as well as front and rear diffusers. Furthermore, essential body parts, such as the rear wing, flicks, mirrors and wheel house openings, were added. Another important parameter, which is of high importance for the flow field, is the consideration of mass flows passing through the race car. For example, those are cooling flows for brakes and engine, and the process air for internal combustion (Fig. 5). For the combustion of air, both an outlet (air box) and an inlet (exhaust pipe) are included on which the volumetric flow rate is defined. For engine cooling, the heat exchangers are modeled by porous media and defined by pressure loss coefficients.
Fig. 5 Essential details of the race car (1: process air flow for combustion; 2: air flow for brake cooling; 3: air flow for engine cooling; 4: rotating wheels).

The rotating wheels are implemented using rotating wall boundary conditions for $\omega W, f$ and $\omega W, r$ (mathematical: $\omega \times r$) on both front and rear axle. The rotation wall boundary conditions was equivalent to the standard wall boundary condition but a tangential velocity $u_{\text{tan}} = \omega \times r$ was assigned to each face in the rotating region with respect to the radial distance $r$ to the rotational centre.

Extensive studies have been performed to improve the modeling of open rotating wheels in CFD, e. g. by Wäschle (7), Hobeika et al. (8), Landström et al. (9), Schnepf et al. (10), or Lewis et al. (11). The general conclusion can be drawn that the sliding mesh simulation is expected to be the most promising technique. However, the results published in the past sometimes show large errors for the sliding mesh too. Due to the enormous effort – e. g. $\approx 46\%$ stated by Lewis – it was decided not to use the sliding mesh approach for the present study.

The simulations were run for zero yaw conditions by using a ratio of front and rear ride height of $h_{M,f} / h_{M,r} = 0.75$. The entire input data applied on the present setup configuration was transferred from measurements obtained on the real race car. Thus, a sufficient overall representation of average track conditions is expected.

3. Results of CFD-Studies

3.1. Single-Belt vs. 5-Belt Moving Ground Simulation

To consolidate the requirement of a detailed moving ground simulation for aerodynamic development in motorsports, CFD was used to compare the aerodynamics forces acting on the Audi RS5 DTM for both a 5-belt and single-belt system. Therefore, Fig. 6 depicts both methods including the floor boundary conditions used. As stated above, the wind tunnel floor is treated as a slip wall until the beginning of the moving belt region. By this approach, an effective boundary layer profile was avoided in order to isolate the influences of the moving ground simulation. However, it is essential to mention that lateral regions of both the center belt and the single belt were set as static friction walls to consider the boundary layer developing from the beginning of the moving belts.

For both the 5-belt and the single-belt system, the belt length was $L_{B,f}/L_M = L_{B,r}/L_M = 0.9$. The belt width was $W_B/W_M = 0.55$ for the 5-belt and $W_B/W_M = 1.60$ for the single-belt, respectively. Each one of the wheel drive units for the 5 belt system had a width of $W_{WDU}/W_M = 0.2$ and a length $L_{WDU}/L_M = 0.06$.

To compare the results of both techniques for moving ground simulations, Fig. 7 depicts the differences in aerodynamic force coefficients referenced to the single-belt simulation. The largest difference occurs in the front lift $\Delta C_{L,f} = 0.529$, which can be explained by the boundary layer thickness that was present for the 5-belt system. The difference in rear lift $\Delta C_{L,r} = 0.063$ was only 12% of that in front lift. The lowest effect was investigated on the drag coefficient $\Delta C_D = -0.036$. The negative sign indicates that, due to the lower aerodynamic down force – the absolute lift values were negative – the induced drag increase was lower for the 5-belt system.

Compared to the results of Hennig et al. (13) the results for the Audi RS5 DTM race car are somewhat surprising since Hennig et al. recognize bigger influences on the rear lift rather than on front lift for both the generic F3 and the LMP race car. An explanation for the different behavior of the DTM car can be the ride height ratio between front and rear end $h_{M,f}/h_{M,r} = 0.75$. For the selected configuration, the front ride height was significantly smaller than the rear one. A more sensitive behavior at the front splitter and the front diffuser was then plausible.

Fig. 6 Schematic view of a 5-Belt 1) and a Single-Belt system 2) including boundary conditions on the wind tunnel floor.
However, from Fig. 7, the general conclusion can be drawn that a 5-belt system does not seem to be appropriate in order to cover the underfloor flow conditions sufficiently.

3.2. Effect of Belt Length on Aerodynamic Forces

Based on the knowledge from Fig. 7, the present study focused on the single-belt dimensions. Following the approach of Fig. 1, the rear end length of the moving belt was varied in the range of $0.3 \leq L_{B,r} / L_M \leq 2.0$ by keeping a constant belt width of $W_B / W_M = 1.6$. The results are shown in Fig. 8. As expected, the effect of the belt length was magnitudes smaller than for the comparison of 5-belt and single-belt. The maximum occurring difference was recognized for the rear lift with $\Delta C_{L,r} = 0.013$ with $L_{B,r} / L_M = 0.3$. By estimating the average accuracy of approximately 0.005, no consistent trend can be reliably drawn since the effects were too small. Furthermore, an effect of $\Delta C_{L,r} = \pm 0.013$ is expected to be negligible for the DTM race car since $\Delta C_{L,r} / C_{L,r,ref} < 1\%$.

The fact that the rear end length of the moving belt turned out to be of minor importance was somewhat surprising. Even with a moving belt immediately ending behind the rear wheel, no significant influence was observed. However, this result is well confirmed by the surface pressure distribution along the underfloor within the symmetry plane $y = 0$. Therefore, Fig. 9 shows $\Delta c_p$ whereas $c_p$ refers to the configuration with $L_{B,r} / L_M = 2.0$. Additionally, the 5-belt simulation is added. The plot provides an explanation for the high differences in front lift, if the wind tunnel operates with the 5-belt system. Especially in the region of the front splitter, a positive shift in surface pressure was produced even though the plot was extracted from the center line covered by the center belt.
front and rear end of the moving belt. Those effects can be observed in static pressure gradients for many wind tunnels, for instance Mack et al.\textsuperscript{12)}

Furthermore, the flow profile in a real wind tunnel needs to be cleaned for the boundary layer thickness exiting from the wind tunnel nozzle. Possible solutions are a boundary layer suction, scoop, tangential blowing or combinations of those. As shown in the latest EADE correlation test\textsuperscript{(13)} in many wind tunnels, a boundary layer treatment can affect the static pressure gradient within the test section significantly. In order to minimize a gradient in static pressure, a careful dimensioning of the single-belt length is recommended. Ideally, both the front and the rear end of the moving belt are in sufficient distance to the car model for which effects of possible flow disturbances do not interfere with the race car aerodynamics.

### 3.3. Effect of Belt Width on Aerodynamic Forces

As evaluated in section 3.2, the rear end length of the moving belt turned out to be of minor importance. However, as stated above, for other reasons, it is recommended to increase the belt length in a reasonable way. This is also somewhat relevant for the study regarding the effects of the belt width $W_B$. The reason is that the belt length defines the length of lateral static floor regions in which boundary layers develop. Therefore, a belt length of $L_{B,fr}/L_M = 0.9$ was selected, as it seems to be a reasonable size for a wind tunnel facility.

To determine the effects of the moving belt width, the approach of Fig. 2 was applied to achieve a variation in effective width $W_B$. The results, with respect to aerodynamic forces, are given by Fig. 10. Again, the plots show $\Delta C_D$, $\Delta C_{L,f}$ and $\Delta C_{L,r}$ for the drag and lift coefficients and referred to the configuration of $W_B/W_M = 5.0$. As for the comparison of 5-belt and single-belt in Fig. 7 as well as for the width of the single-belt, the largest effects were observed on the front lift $\Delta C_{L,f,max} = 0.051$, whereas the maximum effect on the rear lift was $\Delta C_{L,r,max} = 0.026$. Generally, the narrow single belts produced a positive shift in lift and, hence, a loss in aerodynamic down force. Consequently, the effect on the drag coefficient $\Delta C_D$ was negative, but of significant lower magnitude than for the lift coefficients. Furthermore, only for the belt width of $W_B/W_M = 0.975$ a considerable effect on both $\Delta C_D$ and $\Delta C_{L,r}$ was recognized, as, for $W_B/W_M > 1.0$, the changes were in the range of averaging accuracy of the unsteady aerodynamic forces ($\pm 0.005$). For the front lift $\Delta C_{L,f}$, the interferences died out for belts of larger width. Accordingly, the change in $\Delta C_{L,f}$ was negligible for $W_B/W_M \geq 1.5$.

To provide a better understanding regarding the local surface pressure distribution, Fig. 11 depicts $\Delta C_p$ along the underfloor on the center line $y = 0$. As expected from the results for the force coefficients, the effects were low compared to the 5-belt configuration. In contrary to the belt length study, the largest influence on $\Delta C_p$ was observed within the front and middle region of the underbody, with $\Delta C_{p,max} \approx 0.035$ for $W_B/W_M = 0.975$. The region of the rear diffuser remained unaffected in terms of surface pressure. Finally, the effect of the belt width was evaluated to be larger rather than the length of the moving belt. A moving belt width of $W_B/W_M \geq 1.5$ seems a reasonable size for the race car configuration under investigation.

![Fig. 10 Effect of the moving belt width on aerodynamic force](image)

![Fig. 11 Effect of the moving belt width on the underfloor surface pressure distribution; comparison of 5-Belt and Single-Belt with $W_B/W_M = 0.975$ and $W_B/W_M = 2.250$; the static pressure coefficient $\Delta C_p$ is referenced to the width ratio $W_B/W_M = 5.000$.](image)

### 4. Conclusions

In summary, it can be stated that the evaluated interferences of belt dimensions and race car aerodynamics die out quickly with increasing length scales. Accordingly, no considerable effect of the rear end belt length was recognized or at least it was in the range of the averaging accuracy of the CFD. For the moving belt width, a ratio of $W_B/W_M \geq 1.5$ was evaluated to be sufficient to minimize moving belt interferences. In contrast, a moving ground simulation using a 5-belt system turned out to be not acceptable for the selected configuration, as enormous losses in aerodynamic down force ($\Delta C_{L,f} = 0.529, \Delta C_{L,r} = 0.063$) were observed. It is worth mentioning that those results have been obtained for a
single race car geometry in one specific configuration and, hence, the transferability for a wide range of race cars is not given.

For example, the change in both front and rear ride height is experienced to highly affect the aerodynamic balance of the race car. A shift towards a higher sensibility in rear lift rather than front lift – for example at lower rear ride heights – cannot be ruled out and is even quite likely. Furthermore, uncertainties regarding the accuracy of the CFD model remain. For instance, modeling rotating wheels is a well known issue in CFD. The application of a rotating wall boundary condition to an open rim geometry is certainly not a perfect approach to properly cover the effects of wheel house aerodynamics. A comparison with advanced models, such as sliding mesh, is, therefore, recommended.

Another important point to consider is that the present study was performed under rather ideal conditions. Thus, practical issues in real life wind tunnel facilities, such as flow gradients, leakage flow rates, construction tolerances, blockage effects and others, were not covered. On the one hand, this was connected to the objective to isolate the effects of the moving belt; on the other hand, the transferability of the results is surely limited. However, this work provides a better understanding of the aerodynamic sensitivity of high performance race cars with respect to the dimensions of a single-belt system, even though a full transferability cannot be guaranteed.

Acknowledgements

The authors gratefully acknowledge AUDI AG, especially J. Monchaux and B. Jacquet from Audi Sport for providing the geometry of the Audi RS5 DTM race car as well as the essential input from wind tunnel tests. Furthermore, we would like to give special thanks to Dr. S. Blume and Dr. P. Unterlechner for their excellent collaboration on the development of CFD methods for the setup of race car simulations.

References

(1) A. Hennig, N. Widdecke, T. Kuthada, J. Wiedemann : Numerical Comparison of Rolling Road Systems, SAE Technical Paper, 2011-37-0017 (2011).
(2) P. R. Spalart, S. Deck, M. L. Shur, K. D. Squires, M. K. Strelets, A. Travin : 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).
(3) OpenFOAM® : The open source cfd toolbox, www.openfoam.com, Access Date April, 2, 2016.
(4) P. Spalart, S. Allmaras : One-Equation Turbulence Model for Aerodynamic Flows, Proceedings of the 30th aerospace sciences meeting and exhibit, AIAA, 92-0439, p.439 (1992).
(5) M. Islam, F. Decker, E. de Villiers, A. Jackson, J. Gines, T. Grahs, A. Gitt-Gehrke, J. Comas i Font : Application of Detached-Eddy Simulation for Automotive Aerodynamics Development, SAE Technical Paper, 2009-01-0333 (2009).
(6) T. H. Shih, L. A. Povinelli, N. Liu, M. G. Potapczuk, J. L. Lumley : A generalized wall function for complex turbulent flows, 8th Aerospace Sciences Meeting and Exhibit sponsored by the AIAA, 1999.
(7) A. Wäschle : The influence of Rotating Wheels on Vehicle Aerodynamics – Numerical and Experimental Investigations, SAE Technical Paper, 2007-01-0107 (2007).
(8) T. Hobeika, L. Löfdahl, S. Sebben : Study of different tyre simulation methods and effects on passenger car aerodynamics, Proceedings of the IMECE International Vehicle Aerodynamics Conference, p. 187-195, ISBN: 978-0-08-100199-8 (2014).
(9) C. Landström, S. Sebben, L. Löfdahl : Effects of wheel orientation on predicted flow field and forces when modelling rotating wheels using CFD, Proceedings of the 8th MIRA International Vehicle Aerodynamics Conference (2010).
(10) B. Schnepf, T. Schütz, T. Indinger : Further Investigations on the Flow Around a Rotating, Isolated Wheel with Detailed Tread Pattern, SAE Int. J. Passeng. Cars - Mech. Syst., Vol. 8, No. 1, p. 261-274 (2015).
(11) R. Lewis, M. Cross, D. Ladlow : The influence of rotating wheels on the external aerodynamic performance of a vehicle, The International Vehicle Aerodynamics Conference (2014).
(12) S. Mack, T. Indinger, N. A. Adams, P. Unterlechner : The Ground Simulation Upgrade of the Large Wind Tunnel at the Technische Universitaet Muenchen, SAE Technical Paper, 2012-01-0299 (2012).
(13) A. Hennig, E. Mercker : EADE Correlation Test 2010, Final Report, EADE Meeting at FIAT, Turin (2011).