

A Computational Investigation of Ground Simulation for a Saloon Car

Sofie Koitrand
Jaguar Land Rover

Lennart Lofdahl and Sven Rehnberg
Chalmers University of Technology

Adrian Gaylard
Jaguar Land Rover

ABSTRACT

Automotive aerodynamics measurements and simulations now routinely use a moving ground and rotating wheels (MVG&RW), which is more representative of on-road conditions than the fixed ground-fixed wheel (FG&FW) alternative. This can be understood as a combination of three elements: (a) moving ground (MVG), (b) rotating front wheels (RWF) and (c) rotating rear wheels (RWR). The interaction of these elements with the flow field has been explored to date by mainly experimental means.

This paper presents a mainly computational (CFD) investigation of the effect of RWF and RWR, in combination with MVG, on the flow field around a saloon vehicle. The influence of MVG&RW is presented both in terms of a combined change from a FG&FW baseline and the incremental effects seen by the addition of each element separately.

For this vehicle, noticeable decrease in both drag and rear lift is shown when adding MVG&RW, whereas front lift shows little change. The same trends are seen in both CFD and experimental data.

The addition of MVG alone increases both drag and front lift, whereas rear lift decreases significantly. The addition of RWF alone has little effect on the global results (aside from lift), whereas the addition of RWR alone decreases both drag and rear lift significantly. Combining the incremental changes produces values that align well to the MVG&RW case, with the exception of front lift.

This shows similar trends to previously published work, both the noticeable drag decrease due to the addition of MVG&RW, and the contributions of the individual components.

CITATION: Koitrand, S., Lofdahl, L., Rehnberg, S., and Gaylard, A., "A Computational Investigation of Ground Simulation for a Saloon Car," *SAE Int. J. Commer. Veh.* 7(1):2014, doi:10.4271/2014-01-0615.

INTRODUCTION

To evaluate the aerodynamic design of vehicles, OEMs (Original Equipment Manufacturers) and technical universities have utilised wind tunnels since early in the 20th century for measurements and simulations of on-road conditions [1]. Wind tunnels are a well established tool for the investigation of vehicle aerodynamic behaviour, but are expensive and time consuming to use. Relatively recently, Computational Fluid Dynamics (CFD) has become a viable complementary tool, due to improvements in mathematical techniques, physics models and computational power [2]. To improve the simulations, both physically in wind tunnels and computationally, it has become increasingly popular to add

moving ground and rotating wheels conditions, as opposed to fixed ground and fixed wheels. This has been shown, in previously published research, to yield more on-road like conditions, which not only guarantees a better estimation of integral forces (total drag and lift), but also better simulation of local flow fields in the underbody region [3]. There are different ways to implement MVG&RW, but in general it can be broken down into three different elements; front wheel rotation, rear wheel rotation and a moving ground plane.

This paper discusses the effects that MVG&RW have on the external aerodynamics of a saloon type passenger car (the Jaguar XF), and focuses mainly on CFD simulations. Particular

focus is given to the effect of rear wheel rotation and the interaction between rear wheel rotation and the base wake of the vehicle. The CFD data will be compared to experimental data for the same vehicle, and also to previously published research, both computational and experimental.

LITERATURE REVIEW

This topic has been researched and commented on previously, mainly in terms of physical testing but also to some degree by the use of CFD.

Wickern, Zwicker and Pfadenhauer [4] found that as much as 25 % of the total drag during MVG&RW conditions can be attributed to the underbody region, indicating the significance of correctly simulated wheel rotation and ground plane movement.

During experimental testing Wickern and Lindener [5] found that a moving ground together with a boundary layer removal system yields an increase in total drag and a decrease in rear lift for an Audi sports car. Wheel rotation alone lowered the overall drag and increased lift, especially front lift. Rear wheel rotation is reported to decrease the overall lift, whereas front wheel rotation leads to a lift increase.

By using generic versions of a Volvo sedan and a Volvo squareback, Elofsson and Bannister [3] performed wind tunnel testing, after which it was concluded that the most significant cause of the drag reduction caused by MVG&RW is the interference effect of the rear wheel and wheel housing wakes with the base wake. Adding only a MVG was found to increase drag and front lift slightly, while decreasing rear lift. Front wheel rotation resulted in a small increase in drag, whereas rear wheel rotation led to a large drag reduction.

A decrease in global drag with MVG&RW compared to stationary conditions has also been shown by Landström, Löfdahl and Walker [2]. Experimental wind tunnel data was compared with time averaged CFD data, using the MRF (Multiple Reference Frame) method to simulate rotating wheels. The CFD simulations broadly confirmed the experimental results, although there were some discrepancies.

Wäschle [6] found that the interference effects between the underbody flow and the rear wheel wakes contributed the most to the changes in drag and lift with MVG&RW and, in accordance with others, a decrease in total drag with MVG&RW compared to stationary conditions.

All conclude that MVG&RW ought to be incorporated in aerodynamic testing and simulation of road vehicles, due to its large influence on the flow structure and the overall drag.

Several published papers have investigated the accuracy of Computational Fluid Dynamics, including research performed with different methods (time-averaged and transient) and different algorithms (the finite volume method and the Lattice

Boltzmann method). In general, previous research shows good correlation between CFD and physical testing. This includes the previously discussed work of Landström, Löfdahl and Walker [2]. The correlation between CFD analysis and experimental results considering rotating wheels has also been investigated by Wäschle, Cyr, Kuthada and Wiedemann, [7] with good results. The paper compares one steady state solver (STAR-CD) and one transient solver (PowerFLOW) with experimental wind tunnel data.

Further work by others has evaluated the prediction of the Lattice Boltzmann based CFD software Exa PowerFLOW, with promising results for both drag and lift, [8], [9], [10], [11].

Duncan, Fischer and Kandasamy [8] performed wind tunnel and CFD investigations on four different vehicle models and compared the results. The results were found to correlate well and trends for moving ground effects were very well predicted. Front and rear lift values from the CFD simulations showed a trend toward higher lift values than the experimental values, although still close to perfect correlation.

A good correlation between CFD results and experimental results has also been shown by Kandasamy, Duncan, Gau, Maroy et al. [9] with a maximum of 2 % difference in the drag coefficient, and a maximum of 15 counts difference in the lift coefficients with a sliding mesh setup to simulate the rotating wheels. When using MRF or a rotating wall boundary condition to simulate rotating wheels a 5 % maximum error of the drag coefficient is instead noted. The lift values of the rotating wall method show an error of up to 27 counts, whereas those of the MRF method correlate well with the sliding mesh ones with values differing up to 13 counts.

By investigating a notchback and a SUV (sports utility vehicle) model in an open jet wind tunnel and comparing with CFD results, Fischer, Kuthada, Mercker, Wiedemann, et al. [10] found that for the notchback the drag coefficient differs by a few counts, whereas the total lift coefficient is about 20 counts higher for the CFD simulation for each setup. The front lift coefficient is between 6 and 7 counts higher for each setup whereas the rear lift coefficient is consistently about 15 counts higher for the CFD results with a CAD (computer aided design) model compared to the wind tunnel results.

Finally, an indication of an absolute difference in the lift coefficient C_L of ± 10 counts when comparing CFD results to experimental results has been shown by Boujo, Nakasato, Shiozawa, Miyamoto, et al. [11].

Ground simulation and the effect of belt width have also been commented on before. Dimitriou and Garry [12] performed research on the use of a narrow belt in wind tunnel testing, and its effects on the generated aerodynamic forces on a 50 % scale Formula-1 model. It was discovered that a so-called "edge effect" was apparent when using a narrow belt which results in a boundary layer development both above and at the sides of the belt. The size of the boundary layer was found to

be significant, and grew larger at the trailing edge. Generally, it was found that the drag coefficient was reduced with a narrower belt, the front lift was reduced, and the rear lift increased.

This paper is based on the Master's thesis work researched and written by Koitrاند and Rehnberg [13], during the spring of 2013 at JLR in Gaydon, UK.

METHODOLOGY

The simulations performed during the investigation have been prepared using ANSA by BETA CAE Systems and the simulations have been run using PowerFLOW, a CFD solver that uses the Lattice Boltzmann method.

In order to ensure realistic simulations, and to improve comparison with physical tests, the CAD models that the simulations are based on are very detailed in all the relevant areas exposed to airflow, including the engine bay and the suspension geometry. The aim is to have a computational model that to a large degree resembles the actual production vehicle.

The simulations have been performed using a large rectangular domain where 0.1 % of the virtual wind tunnel cross-sectional area is blocked by the vehicle. The vehicle has a frontal area of 2.33 m² and a length of 4.96 m. A suction point has been introduced in the virtual wind tunnel, 13.5 m in front of the car (measured from the centre of the wheelbase) to simulate the honeycomb screen at MIRA. The suction point also specifies where the floor friction will be turned on, and is hence turned off when simulating a moving ground.

To guarantee that this focus on detail improves the simulation results, the volume mesh cell size is reduced in areas of special interest. The volume mesh is divided into eleven different zones of refinement, where the cell size doubles with each zone. The smallest cell size is 1.25 mm and is used in critical regions on the vehicle, whereas the largest cell size of 1250 mm is used far away from the vehicle. All simulations are performed using a zero yaw onset flow, at a nominal velocity of 100 km/h (28.1 m/s).

The ambition to maintain a high level of detail also extends to the wheels, which for the cases with stationary wheels are scanned from physical tyres. This provides a realistic profile and tread pattern, as well as a realistically deformed shape where the tyre meets the ground. For the cases with rotating wheels the lateral profile of the scanned tyres is revolved around an axis through the wheel centre so that an axisymmetric tyre that resembles the scanned tyre is produced. This tyre is allowed to intersect the ground plane by a few millimetres to simulate the contact patch. No flow is permitted between the tyre and the ground. For some cases the stationary scanned tyres were exchanged for stationary axisymmetric tyres to investigate the influence of the tyre. The majority of the cases with rotating wheels are simulated using

the MRF method for the rotating rims while the rest of the wheels are assigned as rotating walls. The sliding mesh method was also evaluated as an alternative to the MRF method. When simulating the moving ground conditions, a section of the ground plane, 50 metres by 50 metres, was assigned a sliding wall boundary condition. This method was in some cases replaced with a narrower "moving belt" for comparison. In order to capture and investigate the influence of rotating wheels, both front and rear, and the moving ground plane, eight different cases were constructed based on previous work done by Elofsson and Bannister [3]. The different cases are explained in Table 1.

Table 1. Cases simulated

Case	Abbreviation	Description
1	FG&FW	Fixed ground and fixed wheels
2	MVG&RW	Moving ground and rotating wheels
3	MVG&RWF	Moving ground and rotating front wheels, fixed rear wheels
4	MVG&RWR	Moving ground and rotating rear wheels, fixed front wheels
5	RWF	Rotating front wheels, fixed rear wheels and fixed ground
6	RWR	Rotating rear wheels, fixed front wheels and fixed ground
7	RW	Rotating wheels, fixed ground
8	MVG	Moving ground, fixed wheels

CORRELATION STUDY

To provide confidence in the CFD results, comparisons with both MIRA full scale wind tunnel (FSWT) (refer to [14] for further information) and Volvo PVT (refer to [15] for further information) have been performed. MIRA results were compared to the baseline case with FG&FW whereas Volvo results were compared both to the FG&FW case and the MVG&RW case. However, this should not be viewed as an attempt at formal validation of Volvo PVT, as no particular effort

was made to match the boundary conditions of this tunnel. The Volvo PVT results are provided to give an indication of whether the CFD simulations were recovering a realistic trend [16].

As can be noted in Figure 1, CFD generally predicts drag quite well, especially compared to MIRA, with a difference of 4 counts (i.e. $\Delta C_D = 0.004$). With MVG&RW the drag is under predicted by 16 counts compared to Volvo PVT. CFD constantly yields lower front lift, but higher rear lift values, compared to the wind tunnel results. Interesting to note is the large over prediction of rear lift by CFD compared to FG&FW at Volvo PVT.

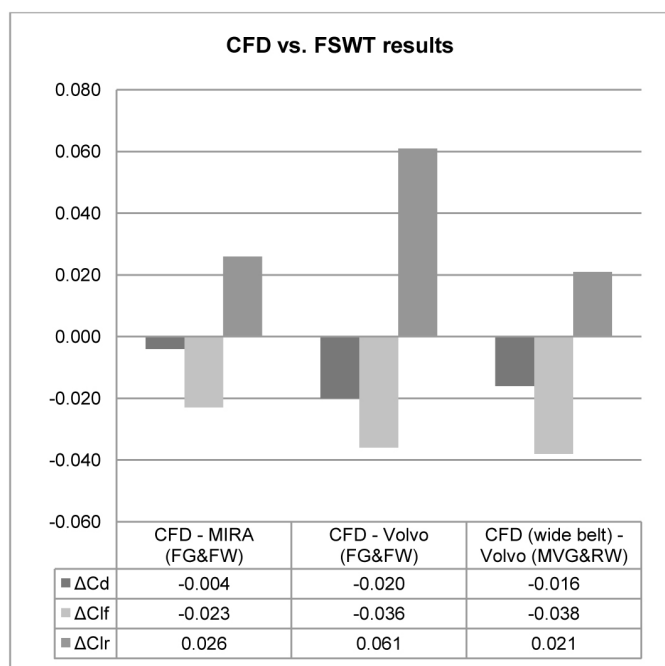


Figure 1. Difference between CFD data and FSWT data

ELEMENT STUDY

Three studies were performed to validate the use of certain elements; the use of MRF to simulate rotating wheels rather than sliding mesh, the simulation of a rolling road (wide belt) rather than a moving belt (narrow belt), and the use of scanned tyres versus the use of axisymmetric ones.

MRF vs. Sliding Mesh to Simulate Wheel Rotation

It has recently been shown that the use of sliding mesh instead of MRF to simulate wheel rotation gives a better correlation with wind tunnel testing results, especially in terms of drag [9]. It was also shown that the simulations with sliding mesh were consistently lower in terms of drag than the experimental values, whereas the simulations with MRF regions were consistently higher. Generally, lift predictions were good for both methods, although a slight lift sensitivity can be noted. Between the two methods differences in rear lift values of up to

13 counts are presented. It is stated by [9] that with MRF the local pressure distribution in the wheels is not as accurate as with the sliding mesh method.

Table 2 shows the difference in drag and lift values between the two simulation methods, representing the MVG&RW case. It is noticeable that there is little difference in the drag predictions made by the two methods, which differs from [9]. The changes in the flow field can be seen when investigating isosurfaces of $C_{p_{tot}}$ (coefficient of total pressure) less than zero, indicating areas of large losses and sources of drag. As can be seen, the pressure distribution close to the rear wheels changes considerably (Figure 2 and Figure 3), leading to changed interaction with the base wake. The front wheels on the other hand are less affected by the change of simulation method, which can also be noted when studying the vortices around the front and rear wheels. The vortices are visualised by identifying regions of the flow where the second invariant of the velocity tensor, Q , exceeds 3000. As can be seen in Figure 4 and Figure 5, the vortex structures around the front wheels are similar in the sliding mesh and the MRF simulations, whereas the ones around the rear wheels differ between the two methods. A noticeable vortex change can be seen on the leading lower edge of the rear wheel, where the sliding mesh simulation (Figure 5) shows several small vortices, whereas in the MRF simulation (Figure 4) they have merged together to form a large vortex.

Table 2. MRF vs. sliding mesh for simulating wheel rotation

	ΔC_D	ΔC_{LF}	ΔC_{LR}
MVG&RW			
Sliding mesh - MRF	-0.003	-0.001	+0.013

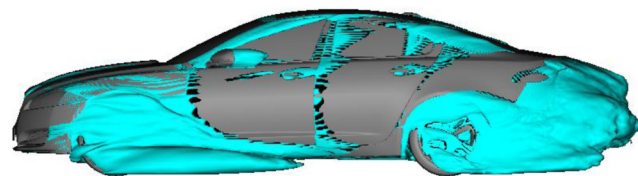


Figure 2. Isosurfaces representing $C_{p_{tot}} \leq 0$ - MRF

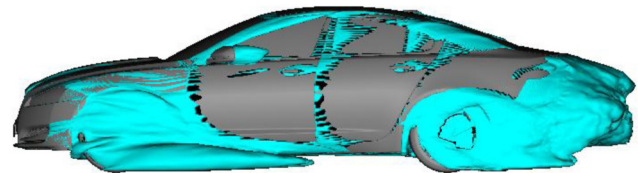


Figure 3. Isosurfaces representing $C_{p_{tot}} \leq 0$ - sliding mesh

By comparing the difference between the FG&FW case and the MVG&RW cases simulated with the MRF method and the sliding mesh method (Table 3) it can be noted that the large rear lift decrease experienced with the MRF method decreases with the sliding mesh method. As with [9] this difference is 13