

# Detailed Flow Studies in Close Proximity of Rotating Wheels on a Passenger Car

**Christoffer Landström and Lennart Löfdahl**

Chalmers University of Technology

**Tim Walker**

Volvo Car Corporation

Copyright © 2009 SAE International

## ABSTRACT

Moving ground systems with rotating wheels have been used in wind tunnel tests during the last decades. Several studies on the effects of rotating wheels and the importance of wheel aerodynamics have been published. It is well known that both the local flow field and the global aerodynamic forces are affected by rotation of the wheels. Different studies indicate that the most significant effect from rotating the wheels is interference effects between the rear wheels and the underbody and vehicle base [1], [2].

A detailed flow field investigation around the wheels in close proximity to the vehicle has been performed on a passenger car in the Volvo Aerodynamic Wind Tunnel. Two omnidirectional 12-hole pressure probes were traversed in a number of planes close to the wheels. Effects of changing different parameters such as ground simulation and rim geometry were investigated. The local flow field has been scrutinised and related to the global aerodynamic properties of the vehicle.

A clear dependency on ground simulation as well as a dependency on wheel geometry was found in the local flow field. All configurations showed lower global drag with moving ground activated. Generally a good correlation with numerical results was observed.

## INTRODUCTION

The importance of road vehicle aerodynamics on passenger car's fuel consumption is well known. As

allowable CO<sub>2</sub>-emissions decreases and the fuel prices increases, further reductions in fuel consumption are necessary. Besides improving engine efficiency and decreasing rolling resistance, reducing the aerodynamic driving resistance is the most effective way in achieving this.

However, considering that the upper body only represents half of the total aerodynamic drag on a typical passenger car [3], [4], [5], creating a nicely streamlined upper body will not be enough in order to be competitive in the future automotive industry. Optimising the flow on the underbody, in the engine bay and around the wheels and wheel houses can yield noticeable reductions in aerodynamic drag and are equally important as the upper body.

As the focus on the underbody and wheels has increased significantly, sophisticated ground simulation techniques have been developed and today several car manufacturers perform wind tunnel tests with moving ground and boundary layer treatment systems. Introducing such systems into wind tunnel testing affects both local and global aerodynamics on a vehicle as discussed in for example [1], [2], [6], [7] and [8].

Considering that these new advanced ground simulation systems have an effect on the global aerodynamic forces on the test object, it is also important to understand how the local flow field is affected. To understand what flow field properties are changed and how they interact with the general flow is essential in order to use these new systems with confidence.

Performing reliable flow field investigations around the wheels and underbody is not always straightforward. The flow is typically very complex and three dimensional in these areas and as a result advanced measurement systems are often required. In order to further increase understanding, numerical investigations should be performed in parallel to the measurements. Neither method will give the complete reality, however by using the strengths of the methods together a fairly reliable picture can often be achieved.

The main focus in this study was to increase the understanding of flow field properties governing the local aerodynamics around wheels and how the global drag is affected. A number of configurations were investigated in the Volvo Aerodynamic Wind Tunnel. Omni-directional pressures probes were used together with an automated traverse and data acquisition system.

The experiments focused on the local flow field around the front and rear right wheel on a modern passenger car. The influence of different rim geometries and the influence of different velocities as well as ground simulation were investigated.

For comparable purposes the experimental vehicle was also investigated numerically using CFD.

## EXPERIMENTAL SETUP

The Volvo Aerodynamic Wind Tunnel is a closed circuit tunnel with slotted walls in the test section. The general design of the tunnel is presented in [9] and a recent upgrade including a 5-belt MTS moving ground system with flat wheel drive units and improved boundary layer suction is described in [10].

A full scale Volvo C30 passenger car was used in this experiment. The vehicle was configured in the same way as the one used in the previous study presented in [11]. For illustrative purposes, a virtual model of a Volvo C30 with the investigated measurement planes and the numbering used is shown in Figure 1.

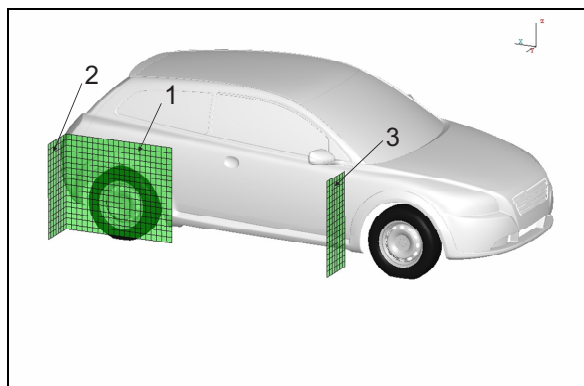


Figure 1 Measurement planes relative to the car

All three measurement planes start 25 mm from the ground and at the closest 30 mm from the vehicle. A distance of 50 mm between measurement points was used. The results in [11] showed that this grid density should be sufficient in order to capture the main flow features.

Plane 1 stretches  $\pm 500$  from the rear wheel centre and is 850 mm high. Plane 2 and 3 are 350 mm wide and are placed 500 mm behind respective wheel centre and have the same height as Plane 1.

## INVESTIGATED CONFIGURATIONS

The vehicle was equipped with two different wheel configurations. Both wheels were 205/55R16 but slightly different tyre treads were used. Rim 1 had a Pirelli P6000 tyre whereas Rim 2 was quipped with a Michelin Energy Saver tyre. The wheels are shown in Figure 2 and 3 respectively.



Figure 2 Wheel Rim 1 with a Pirelli P6000 tyre

Both wheels were investigated with and without the use of the moving ground system. When stationary wheels were used the position of the rims were kept as constant between the configurations as possible.



Figure 3 Wheel Rim 2 with a Michelin Energy Saver tyre

When comparing the results between the two wheels it should be noted that even though the tyres are fairly similar between the wheels there could still be some difficulties in separating the effect of tyre and rim.

Table 1 gives a summary of the configurations that was considered in this experiment. It should also be noted that the data shown for Configuration 5 and 6 are taken from the previous study in [11] and they are included for comparative purposes. The measurements in Configuration 5 and 6 were performed in a plane positioned 510 mm downstream of the front right wheel centre on an identical vehicle set up as the other Rim1 configurations in this study. Moving ground off implies that both the wheel drive units and centre belt was turned off.

| Configuration    | Rim | Plane | Moving ground | Velocity [kph] |
|------------------|-----|-------|---------------|----------------|
| 1                | 1   | 1     | OFF           | 100            |
| 2                | 1   | 1     | ON            | 100            |
| 3                | 1   | 2     | OFF           | 100            |
| 4                | 1   | 2     | ON            | 100            |
| 5 from ref. [11] | 1   | 3     | OFF           | 100            |
| 6 from ref. [11] | 1   | 3     | ON            | 100            |
| 7                | 2   | 1     | OFF           | 100            |
| 8                | 2   | 1     | ON            | 100            |
| 9                | 2   | 2     | OFF           | 100            |
| 10               | 2   | 2     | ON            | 100            |
| 11               | 2   | 2     | ON            | 110            |
| 12               | 2   | 2     | ON            | 120            |
| 13               | 2   | 3     | OFF           | 100            |
| 14               | 2   | 3     | ON            | 100            |
| 15               | 2   | 3     | ON            | 110            |
| 16               | 2   | 3     | ON            | 120            |

Table 1 Experimental configurations

## MEASUREMENT EQUIPMENT

Two 12-hole Omniprobes from Aeroprobe Corporation were used in a probe rake for the flow field measurements. The rake was connected to the wind tunnel main traversing unit at a 45 degree yaw angle to the wind tunnel free stream flow. This was done to move the traversing unit arm away from the vehicle and having the probe heads as closest to the vehicle. The angle was later accounted for in the post processing of the data. Figure 4 shows the two probes with support together with part of the traversing unit. The probes are positioned 50 mm apart from each other in the rake.

As visible in Figure 4 the size of the probe heads are slightly different. The lower probe has a probe head diameter of 6.35 mm whereas the upper probe has a head diameter of 9.52 mm. The reason for using two different probes was simply that these were the Omniprobes available for this test. A considerable time gain was considered to be superior to the fact that the

probes were slightly different in size. Considering the probe sizes and the wall distances described earlier, both probes satisfy the requirement of approximately two probe diameters solid wall distance in order to avoid wall effects [12].

The performances of the calibrated probes are stated by the manufacturer to be more or less equal between the probes. Using omnidirectional probes such as the 12-hole probes used here makes it possible to resolve yaw and pitch angles up to  $\pm 160^\circ$  [13], [14].

A PSI 8400 system was used in the acquisition of the port differential pressures. The probes were connected to a 64 ports PSI ESP Miniature Pressure Scanner which was calibrated before the experiment. A settling time of approximately 10 seconds was used after each traverse movement and 90 samples taken over 3 seconds were time averaged in each measurement point.

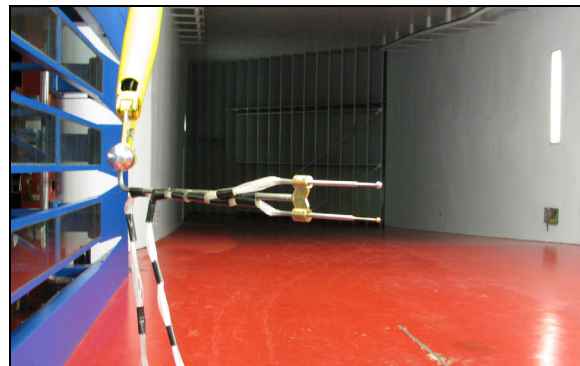


Figure 4 Omniprobes and probe support

Unfortunately it was discovered after the experiments that one of the probes produced non-realistic results in this experiment. An extensive investigation of the probes, which will not be covered in detail here, was performed in the Chalmers University of Technology probe calibration rig. It showed that both probes predicted the oncoming flow satisfactorily. By comparing with other published results with similar measurements and numerical results it was concluded that the smaller probe which is more sensitive to blockage was disturbed by the probe support giving a significant over prediction of the lateral velocity component. As a consequence, the data from this probe have been excluded from the results presented in this paper.

## NUMERICAL SET UP

A numerical investigation using Computational Fluid Dynamics (CFD) was also performed on some of the experimental configurations within the scope of a master thesis performed at Volvo Cars [15]. The vehicle model was configured as similar as the wind tunnel test vehicle as possible but was only modeled using a simplified

were some limitations in available CAD data for the specific test vehicle and it was necessary to make some simplifications in the diffuser area and around the engine. The CAD geometry was prepared in the software ANSA, meshed using Harpoon and solved using FLUENT. Post processing was performed in the software Enight.

The finished mesh consisted of approximately 44 million cells and the vehicle model is shown in detail in Figure 5 below. Realizable k-ε model was used with standard wall functions when solving in FLUENT. Moving ground and rotating wheels were modeled using a moving wall boundary condition on the ground and tangential velocity on the tyre geometry. The rims rotation was modeled using a MRF volume defined in a similar way as suggested in [16] and [17]. Figure 6 shows the MRF volume definition for Rim 1. Due to time reasons, only rotating wheels were considered in the CFD analysis.

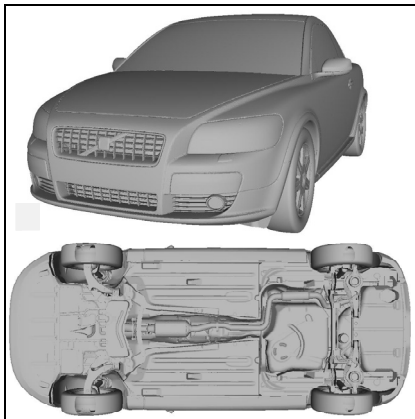


Figure 5 Surface representation of the vehicle model used in the CFD simulations [15]

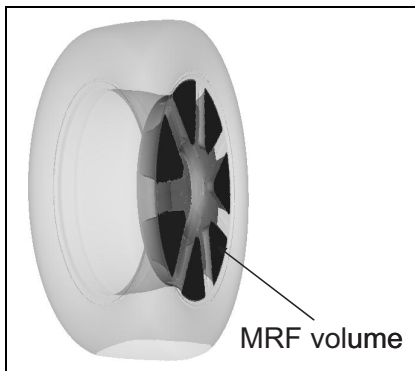


Figure 6 Definition of the MRF volumes used in order to simulate rim rotation [15]

## EXPERIMENTAL RESULTS

The experimental results will be presented in three parts. First the local flow field is presented and analysed for all configurations and thereafter the global aerodynamic properties are presented. Finally a comparison between the global aerodynamics and the local flow field is made.

## LOCAL FLOW FIELD

The local flow field is presented as contour plots of total pressure coefficient defined in Equation 1 as the local total pressure with respect to the reference pressure, normalized with free stream dynamic pressure. In-plane velocity vectors as well as a cut from the vehicle geometry are also visible in the figures.

$$C_{p\,tot} = \frac{P_t}{q_\infty} \quad (1)$$

### Plane 1, Parallel to rear wheel

Figures 7-10 shows the total pressure coefficient and in-plane velocity vectors in Plane 1 for Configuration 1-2 and 7-8 respectively. Comparing moving ground on and off show that the total pressure is generally higher towards the ground for both rims with moving ground activated. The flow structures between the two wheels appears overall very similar for moving ground on and off respectively. A drop in pressure can be noted at the upper left part of the wheel in all configurations. The moving ground on configurations show a somewhat larger area that is affected by this local decrease. This reduction in total pressure most likely corresponds to a small vortex structure shedding from interaction with the wheel and wheelhouse. With rotating wheel this phenomena is somewhat enhanced.

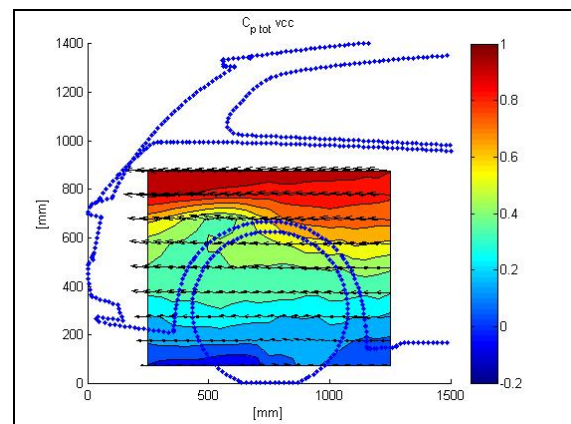


Figure 7 Total pressure coefficients and in-plane velocity vectors Rim 1 moving ground off

A local increase in total pressure is observed in the upper left corner towards the base of the vehicle. The area of high total pressure appears slightly larger for the moving ground condition indicating possibility for higher pressure recovery at the base. It can also be noted that total pressures in the upper right corner are noticeably higher with moving ground.

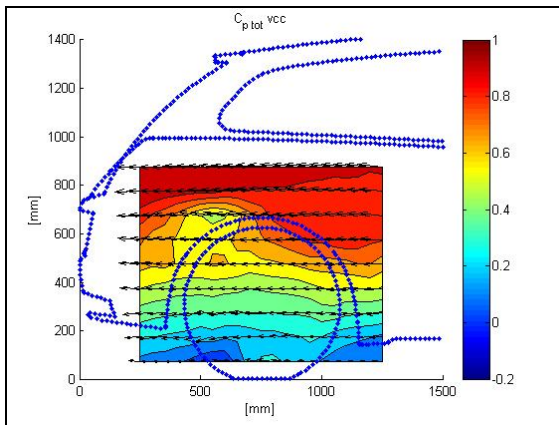


Figure 8 Total pressure coefficients and in-plane velocity vectors Rim 1 moving ground on

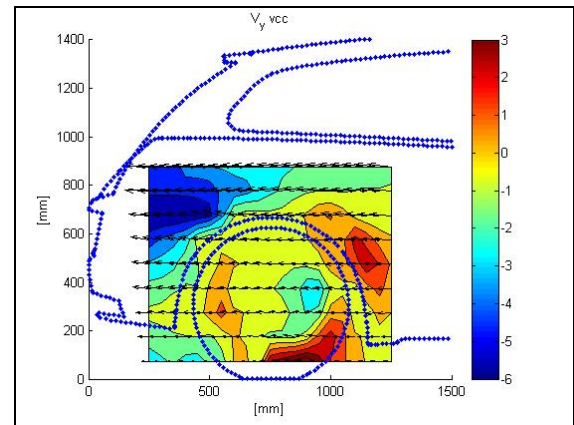


Figure 11 Lateral velocity component in m/s Rim 1 moving ground off

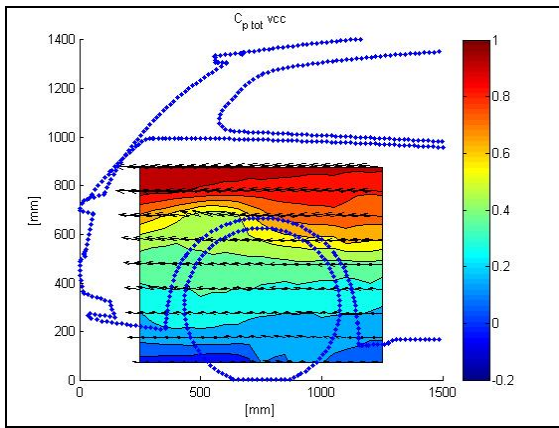


Figure 9 Total pressure coefficients and in-plane velocity vectors Rim 2 moving ground off

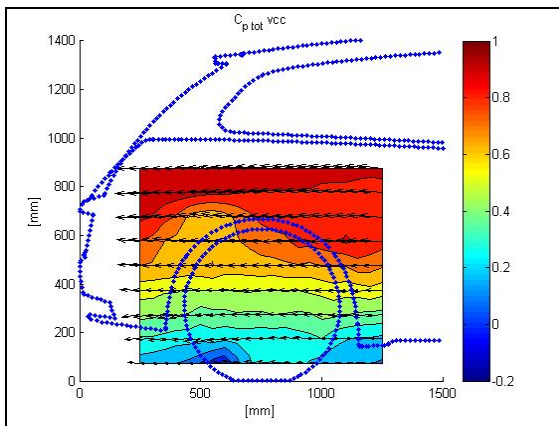


Figure 10 Total pressure coefficients and in-plane velocity vectors Rim 2 moving ground on

When comparing ground simulation for Rim 1 it is clearly seen that there are both similarities in the overall behavior as well as some noticeable differences. It is seen that the flow is angled slightly out from the vehicle at the front of the wheel house which makes sense when considering the shape of the wheel arch shoulders. There is also an outward facing motion close to the ground at the front of the contact patch. An interesting observation can be made for the moving ground condition with noticeably larger area of high velocity which supports the jetting phenomena discussed in [4], [18] and [19] among others. The high pressure created when the tyre approaches the ground creates an outward facing jet. In front of this high pressure area a horse shoe vortex is formed around the contact patch.

In the upper left corner there is a clear inflow towards the vehicle base for all four configurations, with slightly more inflow for the moving ground case, indicating pressure recovery in this area. In the lower left area there is a significant difference between stationary and moving ground. With stationary ground there velocity is relatively low whereas there is a much more significant inflow for the moving ground case. This agrees well with the total pressure iso-surfaces presented in [2] where a clear increase in inflow in the rear wheel wake was observed,

In order to further understand how the local flow in Plane 1 behaves; the lateral velocity component is shown in Figures 11-14. It should be noted that a positive value indicates that the velocity component is directed out from the vehicle.