Flow analysis around a rotating wheel

Emmanuelle Thivolle-Cazat, Patrick Gilliéron

1: Research Department, Renault, Guyancourt, France, Emmanuelle.thivolle-cazat@renault.com
2: Research Department, Renault, Guyancourt, France, Patrick.gillieron@renault.com

Abstract: Wheel’s contribution to vehicle aerodynamic drag is around 25%. Reducing this amount is of great interest to diminish fuel consumption and CO2 emissions. The knowledge of vortex structures around the wheel is therefore necessary to understand the physical mechanisms of vortex dynamics and to develop a strategy for flow control.

Particle Image Velocimetry (PIV) and aerodynamic torque measurements have been carried out in a wind tunnel in order to analyze the flow around a rotating wheel located inside a wheel arch. This study has been conducted in collaboration with RENAULT Research Department, Aero Concept Engineering (ACE) and ISAT (Institut Supérieur de l’Automobile et des Transports).

Detailed flow-field measurements of a 40 percent full-scale bluff body have been obtained using two-dimensional PIV in the wind tunnel facility of ACE in Magny-Cours, France. Drag coefficient data have also been acquired using balance measurements for several wheel arch volume parameters. A global increase in drag has been shown with wheel arch volume growth.

Longitudinal vortex structures such as “jetting” have also been visualized using PIV. Wake characterization for three wheel arch volumes has been obtained. Flow structure differences between stationary and rotating wheel have also been observed for one wheel arch volume configuration.

1. Introduction

During a car displacement, wheels’ rotation is responsible of about 25% of the aerodynamic drag of passenger cars (Wickern et al, 1997). Reducing this contribution is an interesting solution to diminish CO2 emissions and fuel consumption and to improve vehicle’s road behavior. To control the flow and then to achieve drag reduction, a better physical understanding of the flow mechanism is first needed.

The aerodynamic characteristics of rotating exposed wheels have received growing interest in recent years both experimentally and computationally (Axon et al. 1998, Mears et al. 2004, Wäschle et al. 2004). Differences between stationary and rotating wheel have been highly documented and since the work of Fackrell (1974), it is well-known that an isolated rotating wheel produces less drag than a stationary one. Cogotti (1983) and Mercker et al. (1992) showed that a rotating wheel, in presence of the ground, originates a system of three pairs of counter rotating longitudinal vortices in the wake (Fig. 1). A pair of them sheds from the top, a second from the wheel axis and the third from the bottom, attached to the ground. This last pair of higher intensity is called “jetting” vortices (Morelli, 2000). When the wheel is partially enclosed by the wheel arch, three out of the six vortices disappear: only the jetting vortices and the external vortex shedding from the wheel axis remain (labeled 1, 2, 5 in Fig. 1).
However, less work has been published concerning wheel located inside a wheel arch (Axon et al. 1999, Skea et al. 2000). Skea et al. compared the flow features obtained by CFD and experiments for two wheel width, rotating and non-rotating. Axon et al. found that a wheel located within a wheel arch produces more drag when it is rotating on a moving belt compared to when it is fixed on a stationary one. In that configuration, it is the opposite result as for an isolated wheel (Fackrell, 1974). To the authors’ knowledge, no detailed PIV measurements results have been reported concerning shrouded wheels.

Fig. 1: Visualization of the longitudinal structures around an isolated rotating wheel (Mercker & Bernerburg, 1992).

In this paper, wind tunnel results of PIV and balance measurements are presented in order to analyze the flow characteristics around a rotating wheel located inside a wheel arch, responsible to the drag. This study has been conducted in collaboration with RENAULT Research Department, ACE and ISAT.

2. Experimental setting and parameters

2.1 Wind tunnel configuration

Measurements have been conducted in the wind tunnel of Aero Concept Engineering in Magny-Cours, France. The wind tunnel has a moving ground with a boundary layer suction to approach road conditions. The wind tunnel dimensions allow a bluff body at scale 0.4. Air velocity and moving belt velocity can be adjusted between 0m.s⁻¹ and 40m.s⁻¹. Tests measurements have been conducted on a bluff body model representing a front left ¼ of a car-like shape at scale 2/5 (Fig. 2). With a relatively large working cross-sectional area compared to the cross-sectional area of the bluff body, the blockage effects are minimal and less than 4%. A symmetry plane is set along the right edge to reduce 3D effects with its thickness arbitrary fixed to 2 mm. The wheel has a diameter of 210mm and a width of 73mm (Fig. 3).

Fig. 2: Picture of the bluff body with the rotating wheel placed in the wind tunnel. Scale is 2/5. Fig. 3: Picture of the isolated wheel on the moving belt.

2.2 Aerodynamic torque measurements

SCx values have been acquired by an aerodynamic balance located respectively inside the bluff body (S being the reference wetted area). The bluff body is linked to the ceiling by an appropriate strut, which contains the aerodynamic balance. The wheel is linked to the floor by a sting.
2.3 Particle Image Velocimetry

The PIV apparatus was set up to analyze the flow-field around the wheel and thus give an insight into the rotating wheel aerodynamics, both qualitatively and quantitatively. The flow is seeded with particles (mixing of water + Safex) and lighted by a mini Yag 50mJ laser. Images are acquired using a Flowsense camera (1600 x 1200 pixels) and treated with Flowmanager 4.5, a DANTEC acquisition system. The flow condition inside the wind tunnel is at constant wind velocity \( V_0 = 30 \text{m.s}^{-1} \) and the belt can be fixed or similarly moving at 30m.s\(^{-1}\). PIV measurements are conducted in four horizontal and vertical planes in longitudinal and transversal directions. Velocity fields are averaged with a statistic of 200 instantaneous vector fields taken at a sampling frequency of 16Hz. The time period between a pair of images is set to 50.10\(^{-6}\)s for longitudinal planes and to 10.10\(^{-6}\)s for perpendicular planes. Velocity fields have a maximal size of 200mm x 200mm. The relative complexity of the flow and especially parasite reflections from the tire do not allow enough resolution of the flow field to resolve quantitative vorticity fields. The following analysis concentrates essentially on qualitative results based on velocity contours and streamlines.

2.4 Configurations

This study deals with flow analysis around the wheel and inside the wheel arch. The influence parameters are the non-dimensional radial \( a \) and transversal \( b \) spaces between the wheel and the wheel arch (Fig. 4). The radial space \( a \) and the transversal space \( b \) are respectively divided by the wheel diameter (210mm) and thickness (73mm). Four values of each non-dimensional parameter (\( a = 0.036, 0.095, 0.155, 0.214; \ b = 0.096, 0.233, 0.507, 1.055 \)) have been tested which leads to 16 configurations. Only three configurations of increasing wheel arch volume (\( V_1 = 1.15 \times 10^{-3} \text{m}^3, V_2 = 3.8 \times 10^{-3} \text{m}^3 \) and \( V_3 = 5.25 \times 10^{-3} \text{m}^3 \), see Table 1) have been explored by PIV in order to limit the amount of data. Influence of the moving belt is also investigated. Obviously the stationary wheel investigation does not represent any practical scenario but its inclusion helps for comparison between the configurations in the understanding of the flow physics.

![Fig. 4: Definition of parameters \( a \) and \( b \), view from top.](image)

<table>
<thead>
<tr>
<th>( V_1 )</th>
<th>( V_2 )</th>
<th>( V_3 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>( a )</td>
<td>0.036</td>
<td>0.214</td>
</tr>
<tr>
<td>( b )</td>
<td>0.233</td>
<td>0.233</td>
</tr>
</tbody>
</table>

Table 1: Table of different PIV configurations

3. Results and discussion

3.1 Aerodynamic torque measurements

Fig. 5 shows the results of global (wheel + bluff body) SCx balance measurements for 16 configurations, where S represents the bluff body reference wetted area. These results show a global increase of the drag coefficient SCx with the wheel arch volume (depending on \( a \) and \( b \)). For a given value of \( a \), the SCx value is almost stable, increasing very slowly with increasing \( b \). At the opposite side, for a given value of \( b \), the SCx value significantly increases with \( a \). It appears that the radial parameter \( a \) has therefore more impact on drag coefficients than the transversal parameter \( b \). Fig. 6 also represents global SCx results but only for the three volume configurations chosen for PIV. It points out a drag increase between volume V1 and V2 but not between V2 and V3. The drag increase between V1 and V2 can easily be explained by the increase of parameter \( a \) value. Nevertheless, for configurations V2 and V3, \( a \) is constant and \( b \) increases but a drag decrease is observed, which doesn’t follow the general trend. PIV measurements of the next section should
bring additional elements to understand this local phenomenon.

Fig. 5: SCx values for different values of \(a\) and \(b\) at velocity 30m.s\(^{-1}\) over a moving belt.

Fig. 6: SCx values versus wheel arch volume V1, V2 & V3.

3.2 PIV measurements

In the following analysis, the x-axis is oriented in the flow direction, the y-axis is horizontal, perpendicular to the flow direction and oriented form outside to inside the bluff body and the z-axis is vertical oriented to the top. The origin is located at the rear part of the wheel, at the floor level and contained in the vertical plane tangent to the wheel (Fig. 7). Velocity fields have been obtained in four different planes: two horizontal ones A & B at \(z = 125\)mm & \(z = 10\)mm, a longitudinal one C at \(y = -20\)mm and a vertical transversal one E at \(x = 20\)mm. Velocity results have been represented with streamlines superimposed to velocity contours for three different wheel arch volumes V1, V2 and V3. Results for volume V2 are presented with and without the moving belt.

**Plane A**

Fig. 7: Axis origin and referential around the wheel.

Fig. 8: Plane A located at \(z = 125\)mm.

Plane A is a horizontal measurement plane, located just above the wheel rotation axis, at \(z = 125\)mm (Fig. 8). It is intersected by a transversal plane E (located at \(x = 20\)mm). This configuration shows a large effect of the wheel sting on the flow along the y-direction (Fig. 9). In the part between \(y = -110\)mm and \(y = -200\)mm, the flow is not disturbed and the velocity values equals the wind velocity, around 30 m.s\(^{-1}\). This corresponds to the profiled part of the sting axis. In the non-profiled part between \(y = 0\)mm and \(y = -110\)mm, the flow is highly disturbed with an area of larger velocity along the wheel and a wake along the bluff body. This phenomenon is much more visible in the non-rotating configuration. This disturbance due to the wheel sting, and highly sensitive at this height (\(z = 125\)mm) does not allow a correct characterization of the wake along the wheel and the wheel arch. Drag and lift interferences due to struts and stings are discussed in more details by Hetherington & Sims-Williams (2004).
Fig. 9: Velocity contours (in m.s\(^{-1}\)) and streamlines in plane A in each configuration of volume V1, V2 and V3 with rotating wheel and for configuration V2 with a stationary wheel, view from above.

**Plane B**

Plane B is a horizontal measurement plane, located just above the floor, under the wheel sting, at \(z = 10\)mm (Fig. 10). It is intersected by the transversal plane E (located at \(x = 20\)mm). Results for the three volume configurations are presented in the rotating case with the moving belt on only. First results show a wake along the wheel and downstream the wheel beside the body (Fig. 11). Influence of wheel arch volume on wake can be characterized by the wake width \(D\) (measured in \(x = 80\)mm, Fig. 11), along the \(y\)-direction. Regarding the evolution of the wake width \(D\) for the three volumes, Fig. 12 shows an increase of wake transversal dimensions with wheel arch volume. These results are in agreement with aerodynamic torque measurements (shown in Fig. 5), which confirm the dependency between wake transversal dimensions and aerodynamic drag. Moreover, the increase of
D is less important for $a$ constant (between $V_2$ and $V_3$) than for $b$ constant (between $V_1$ and $V_2$).

**Fig. 11:** Velocity contours (in m.s$^{-1}$) and streamlines in plane A in each configuration of volume $V_1$, $V_2$ and $V_3$ with rotating wheel, view from top.

**Fig. 12:** Wake width D versus wheel arch volume.
Plane C

Plane C is a longitudinal measurement plane, parallel to the wheel plane at \( y = -20\text{mm} \) (Fig. 13). This plane is intersected by the two horizontal planes A (\( z = 125\text{mm} \)) and B (\( z = 10\text{mm} \)) and a transversal one E (\( x = 20\text{mm} \)), see Fig. 14. Different interesting flow mechanisms are revealed in this plane. First, at the intersection with plane B, results clearly show flow differences with and without the moving belt on for an equivalent wheel arch volume \( V_2 \). The boundary layer is much more developed above the floor when the moving belt is fixed, see thickness \( H \) (Fig. 14). Then, when considering the results obtained with the moving belt on, the boundary layer thickness \( H \) (measured in \( x = 0 \)) can be characterized as a function of the wheel arch volume. The thickness \( H \) increases between volume \( V_1 \) and \( V_2 \) and decreases between \( V_2 \) and \( V_3 \) (see Fig. 15), which can be directly correlated with SCx results obtained using aerodynamic torque measurements (Fig. 6).

![Velocity contours and streamlines in plane C for each configuration of volume V1, V2 and V3 with rotating wheel and for configuration V2 with a stationary wheel, side view from outside to inside the bluff body.](image-url)
Finally, a wake structure in the middle of the wheel (z = 105mm) appears at the intersection with plane E for all configurations studied. This phenomenon is discussed in the next section.

![Figure 15](image15.png)

**Fig. 15:** Boundary layer thickness H for configurations with the moving belt on versus wheel arch volume.

**Plane E**

Results obtained by PIV in the vertical plane E (Fig. 16) are useful to retrieve and visualize the longitudinal vortex structures reported in the literature (Mercker & Bernerburg, 1992) and designed by 1 and 5 in Fig. 17. This plane is localized in x = 20mm and cut in two parts (top and bottom) to enhance the spatial resolution of PIV. This plane is intersected by three other planes A (z = 125mm), B (z = 10mm) and C (y = -20mm) to approach three-dimensional phenomena.

![Figure 16](image16.png)

**Fig. 16:** Plane E located at x = 20mm

Velocity contours for each configuration (Fig. 17) show the vortex structures formed behind the wheel. The first one is located near the floor and the wheel (x = -20mm and z = 15mm). This structure number 1 corresponds to the jetting vortex identified on Fig. 1. The “viscous pumping” phenomenon that appears when the moving belt is on, yields a stronger jetting vortex which is closer to the wheel and then leads to a drag decrease. A second structure, designed by number 5 on Fig. 17, is located just below the rotation axis (x = -20mm and z = 90mm). These structures 1 and 5 correspond to the ones reported in the bibliography: they have the same clockwise or anticlockwise rotation and their location is similar. These two longitudinal vortices are located at the intersection with plane C (Fig. 17). They correspond to the wake zones identified in the plane C, where velocity is lower. These structures can be described as longitudinal vortex of a wake. A last vortex structure appears in y = -80/-100mm and z = 95mm. This vortex corresponds to the intersection between the profiled part and the non-profiled part of the wheel sting along the y direction and is probably due to the wheel sting (Fig. 3).
Fig. 17: Velocity contours (in m.s$^{-1}$) and streamlines in plane E for each configuration of volume V1, V2 and V3 with rotating wheel and for configuration V2 with stationary wheel. The image is cut in two parts to enhance the spatial resolution of PIV, view from behind at x = 20mm.
4. Conclusion & Future work
PIV and aerodynamic torque measurements have been carried out in a wind tunnel in order to analyze the flow around a rotating wheel located inside a wheel arch. The results obtained from PIV measurements have shown that:

- Wake zones are larger with a stationary floor compared to a moving belt,
- Wake thickness increases with the radial parameter and decreases with the transversal parameter of the wheel arch,
- Wake width increases with wheel arch volume, especially with the radial parameter increase,
- Flow structure between stationary and moving floor conditions is different: the boundary layer is thicker in the stationary case and jetting vortices are stronger in the moving floor case.

The results obtained from aerodynamic torque measurements are confirmed by PIV: drag globally increases with wheel arch volume but the effect of radial space on drag is the most important. For a constant radial parameter value, the increase of the transversal parameter induces a small increase of wake width and a decrease of wake thickness, which could explain a light SCx decrease.

This measurement campaign has been very useful to obtain flow visualization through PIV, and especially the three-dimensional structures of the flow around a rotating wheel, allowing a qualitative characterization of the wake vortex structures.

The wheel sting disturbance induces creation of additional vortex structures, locally disturbing the measurements. In this way, future PIV measurements are subject to a new experimental set up of the wheel.

Although the PIV measurements bring some first consistent results, they are limited by the small number of PIV planes realized and low spatial resolution not allowing vorticity fields to be resolved. New experiments will permit to characterize three-dimensional effects more precisely and in particular the unsteady flow by realizing 3D PIV measurements at a higher spatial resolution. Fluorescent paint could also be useful to diminish reflection parasite effects on the walls.

5. Acknowledgments
The authors acknowledge the Aero Concept Engineering society for their help and the loan of their wind tunnel and ISAT, Nevers University for the preparation of the scaled model.

6. References
Mercker, E. & Bernerburg, H. On the simulation of road driving of a passenger car in a wind tunnel.
using a moving belt and rotating wheels. 3rd Int. Conf. Innovation and Reliability, Florence, April 8-10, 1992.


