Aerodynamic analysis of an isolated vehicle wheel

To cite this article: P Leniewicz et al 2014 J. Phys.: Conf. Ser. 530 012064

View the article online for updates and enhancements.

Related content

- Aerodynamic analysis of potential use of flow control devices on helicopter rotor blades
  F Tejero, P Doerffer and O Szulc

- Numerical analysis of the flow around the Bach-type Savonius wind turbine
  K Kacprzak and K Sobczak

- Investigation of the tip-leakage losses in turbine axial stages
  A Szymaski, S Dykas, W Wroblewski et al.
Aerodynamic analysis of an isolated vehicle wheel

P Leśniewicz, M Kulak and M Karczewski
Institute of Turbomachinery, Lodz University of Technology, Poland

E-mail: lesniewicz.pawel@gmail.com

Abstract. Increasing fuel prices force the manufacturers to look into all aspects of car aerodynamics including wheels, tyres and rims in order to minimize their drag. By diminishing the aerodynamic drag of vehicle the fuel consumption will decrease, while driving safety and comfort will improve. In order to properly illustrate the impact of a rotating wheel aerodynamics on the car body, precise analysis of an isolated wheel should be performed beforehand. In order to represent wheel rotation in contact with the ground, presented CFD simulations included Moving Wall boundary as well as Multiple Reference Frame should be performed. Sliding mesh approach is favoured but too costly at the moment. Global and local flow quantities obtained during simulations were compared to an experiment in order to assess the validity of the numerical model. Results of investigation illustrates dependency between type of simulation and coefficients (drag and lift). MRF approach proved to be a better solution giving result closer to experiment. Investigation of the model with contact area between the wheel and the ground helps to illustrate the impact of rotating wheel aerodynamics on the car body.

1. Introduction

Nowadays, when car companies are trying to reduce fuel consumption and lower CO₂ emission, aerodynamic design of cars and its components plays important role. According to different sources even up to 25% of the car aerodynamic drag comes from wheels [1,2]. Car manufacturing companies and tyre manufacturers are facing problems connected with reduction of friction coefficient as well as improving air flow around the tyre and whole car body. Wheels are also an important factor taking into consideration noise generated by cars [3]. It is clearly visible that this branch of car design can give benefits connected not only with reduction of fuel consumption and CO₂ emission, but also with safety and comfort of modern vehicles. There are three pairs of major vortices induced by flow around wheel (see figure 1). Wake-horseshoe vortex denoted as numbers 1 and 2 (as a pair), another flow feature is a so called C-shoulder vortex (numbers 3 and 4) caused by separation of air passing over the top tyre shoulders. Third one, squash vortex is also presented in figure 1 (as 5 and 6).

1 To whom any correspondence should be addressed.
Despite wide investigations conducted in recent years, there is still a lack of information about flow structures inside tyre grooves, mainly caused by difficulties of accessing the tested region with measurement equipment. One of the first works devoted to this topic was performed in middle 70’s by [5]. Differences in circumferential pressure distribution between grooved and slick tyre were reported. In grooved tyre due to the Venturi effect, a sudden pressure drop was measured at place of wheel-ground contact, while the pressure reached the maximum in this place in case of the slick tyre. This was explained in the following manner: a sudden pressure drop inside grooves promotes the flow through the tread and diminishes the size of trailing vortices. Another work connected with automobile wheels was made by [6] whose investigation focused on identification of vortices in case of rotating wheel, while jetting effect was widely described by [7]. The importance of vortices located near the area of contact between the wheel and the ground was highlighted by [4]. As a result, the flow phenomena described by e.g. [4,5,6,7], among others, shed light at the importance of analysing vehicle aerodynamics with inclusion of rotating wheels, both experimentally and numerically, a practice that before 1990s was quite seldom. However, still a number of publications strictly connected with investigation of tyre geometry using Computational Fluid Dynamics (CFD) in synergy with experiment is limited. For example, very recently [8, 9] observed that there is a difference in aerodynamic drag depending on whether a vehicle is equipped with slick or grooved tyres. Namely, the aerodynamic drag is lower for an automobile with grooved tyre (a production sedan and a model of compact car were tested). This claim shifts analysis of tread pattern influence on vehicle aerodynamics back to isolated wheel study in order to search for explanation of this phenomenon. An alternative experiment, to Fackrell’s 1974 work, was performed by [10]. During this investigation a consistent negative peak in pressure distribution followed the pressure rise in place of contact for the slick tyre. This observation was in contrast to Fackrell’s work. No experiment was made for grooved tyre to make an adequate comparison. Yet an experimentally-based analysis of flow inside grooves for exposed wheel was performed by [11]. It was concluded that main difference between tyre with and without grooves in terms of pressure distribution is the jetting effect. No comments were made about potential drag difference between both tyre patterns. This paper aims at searching for explanation of origins of drag difference described above. Numerical model was prepared to simulate the conditions of the experiment in [5]. Explanation of phenomena responsible for negative peak of pressure for slick and grooved tyre will be discussed.

2. CFD Model

Description of wheel geometry, models used for wheel rotation simulation as well as numerical settings are discussed in this section.

2.1. Geometry

Two wheel geometries were analysed, one with three longitudinal grooves, and the other without grooves (slick). Tyre geometry was created basing on wheel investigated by [5] in order to compare results obtained using CFD with experiment. The wheel has a diameter equal to 416 mm and a breadth of 191 mm. Detailed geometry of the wheel is shown in figure 2. In order to investigate influence of tread, geometry was modified by adding three longitudinal grooves (rain grooves) spaced evenly with
respect to the wheel centreline. Each groove is 6.35 mm wide and 3.18 mm deep. Flow is considered as external 3D type, therefore boundary conditions must be set at such distances away from the object to ensure the least amount of influence. Based on domain size investigation it was stated that width should be equal 10D, height 10D, upstream 5D, downstream 15D where D is the wheel diameter. Due to the fact that in reality the contact between the wheel and the ground is not at one point, contact patch between the wheel and the ground was defined. Tyre deformation model study provided in [8] showed that uninterrupted grooves in contact patch area more realistically predict flows around tyres. The study presented herein adheres to this standard. Visualization of both, slick and groove tyre with a contact area are presented in the figure 3.

![Figure 2. Section view of wheel with dimensions (in mm) [3].](image)

![Figure 3. Area of contact between the wheel and the ground.](image)

2.2. Numerical model
Numerical investigation of flow was performed using ANSYS CFX v.14.0 software. Based on previous works performed in Institute of Turbomachinery related to the analysis of pumps, fans, turbines, and geometry optimization, which show the best compromise between accuracy of the results and time needed for simulation [12,13,14], Shear Stress Transport (SST) turbulence model was applied. Additionally, [15] made a study where non-linear viscosity model (Explicit Algebraic Reynolds Stress Model with Curvature Correction EARSM-CC) has been used in rotational frame of reference. Comparison to linear viscosity model reported by [15] reveals insignificant differences in pressure distribution modelling. Steady state analysis were performed with turbulence intensity at the inlet set to 0.002, while high resolution accuracy for advection terms of the Navier-Stokes equations was selected. Free stream flow speed was equal to 18.6 m/s and angular velocity of the wheel was 16.67 rev/s what matched the real travel conditions represented by stream flow speed. Rotation of the wheel was modelled using two approaches: Moving Wall boundary (MW), and Multiple Reference Frame (MRF). In the first method, translation of the wheel is created by applying linear velocity to the peripheral nodes located at the perimeter of the wheel, except the nodes situated in contact area where combination of boundary conditions is used (see figure 4). MRF is considered as an alternative way to model rotation of the wheel. It is partially based on MW method plus is extended by introducing a separate computational domain adjacent to the wheel rim. This results in creation of pair of interfaces presented in the left picture of figure 5 (marked in green). Grid A is attached to the stationary mesh B. Due to the angular velocity of the rim all nodes of mesh A will rotate, while mesh B will have translation definition according to MW method. Numerous tests for grid A were performed to determine its size as based on accuracy to represent the physical aspects of flow.
3. Results
Following section contains analysis of the results obtained from CFD model and experiment performed by [5,10,11].

3.1. Drag and lift coefficients
For the analysis that follows, the wheel with grooves is denoted as G, the slick one as S, slick tyre for which moving wall was applied is named S/MW, while grooved tyre with MRF model is denoted as G/MRF respectively. Global quantities such as \( C_x \) and \( C_z \) were calculated to evaluate the accuracy of the numerical model in comparison to the experiment presented by [5]. Comparison of obtained results of drag and lift coefficients are presented in the table 1.

<table>
<thead>
<tr>
<th>Tyre</th>
<th>( C_x )</th>
<th>( C_z )</th>
<th>Experiment ( C_x )</th>
<th>Experiment ( C_z )</th>
</tr>
</thead>
<tbody>
<tr>
<td>S/MW</td>
<td>0.709</td>
<td>0.320</td>
<td>0.51</td>
<td>0.28</td>
</tr>
<tr>
<td>S/MRF</td>
<td>0.535</td>
<td>0.295</td>
<td>0.51</td>
<td>0.28</td>
</tr>
<tr>
<td>G/MW</td>
<td>0.699</td>
<td>0.220</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>G/MRF</td>
<td>0.520</td>
<td>0.261</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

As it can be observed values of lift and drag coefficients are significantly smaller (except \( C_z \) for grooved tyre) in situation where Multiple Reference Frame model was used. In MRF simulations of slick tyre, reduction of lift is observed, while for tyre with grooves \( C_z \) values increase when compared to computations with the use of MW model. It can be concluded that use of Multiple Reference Frame decisively helps to obtain numerical model with higher accuracy with respect to the experiment not only for drag, but also for lift coefficient. Furthermore, a clear general trend of drag reduction for grooved tyre is observed and will be explained in next sections.

3.2. Tyre circumferential pressure distribution
It was decided to compare pressure distribution for geometries investigated in this research with experimental data elaborated by other researchers. Figure 6 presents measurements of centreline circumferential pressure distributions for rotating slick tyres experimentally tested by [5,10]. Results of those experiments are compared with centreline pressure distribution obtained by numerical investigation discussed in this paper.
Pressure increases from stagnation point to the contact area which is observed for all investigations. Pressure peak observed by Mears is shifted by few degrees. Immediately after, a sudden pressure drop is visible in place of contact patch (around 90° angle). This pressure drop is not visible for CFD simulation - contact patch area was simulated in such way as deformation of the tyre in this region blocked the flow. Furthermore, this is where the difference between Fackrell and Mears investigations is visible. For the first experiment a negative pressure peak is not observed and pressure stabilises.

![Pressure distribution](image)

**Figure 6.** Pressure distribution for slick tyre.

In case of Mears' investigation the negative pressure peak was observed. This change in pressure distribution is connected with a “suction cup” effect as it was named by [11]. The “suction cup” can be explained by creation of a vacuum between tyre and the ground. The vacuum area is suddenly expanded when tyre lifts up recovering the pressure right after contact area. This tendency is the same for CFD model where, despite lack of data points, clear pressure recovery is observed and is not visible in measurements made by [5] (area 1 in figure 6). After the contact patch distribution of pressure lines for numerical simulation and Fackrell's experiment seems quite similar. Pressure line for Mears' investigation after negative peak looks different. Behind contact area oscillations are observed. This phenomenon is caused by the fluctuations of the moving belt below the tyre. Second difference is located near place of flow separation (area 2 in figure 6). Values of pressure for Fackrell's experiment are higher than those recorded by Mears. Numerical investigation in this region shows pressure values between both experimental investigations. To sum up, the CFD model agrees quite well with data in Fackrell's experiment. Moreover, a part of negative pressure peak at contact patch, which was not observed by [5], but was presented by Mears is likewise predicted by the CFD simulation. Centreline circumferential pressure distributions inside a middle groove were analysed for the second tyre type. This time results of numerical investigation were compared with experiment performed by [5,11]. As it can be observed in figure 7 for all cases a sudden negative pressure peak appears near the area of contact, where slick portions of the tyre blocks the flow while the grooves allow it in. Hence observed pressure drop in this region is caused by sudden acceleration of the air rushing into the groove [11]. However, pressure drop for CFD simulation is significantly more intense. This can be connected with greater amount of “measurement points” (mesh nodes) in this area. Beyond this, pressure distribution is nearly the same for all cases. Near the flow separation area differences in
pressure distribution are observed but are smaller compared to the case of slick tyre. Overall, both experiments as well as numerical simulation show similar pressure representation.

![Pressure distribution](image)

**Figure 7.** Pressure distribution for tyre with grooves.

3.3. Velocity flow fields

2D velocity vector plot was used to visualize flow characteristics. Y-Z plane was created at the distance equal to 2.5 \(D\) behind the wheel (\(D\) as the wheel diameter). Comparing tangential projections of flow vectors for slick tyre (figure 8) and tyre with grooves (figure 9) following differences are observed. Velocity field for tyre with grooves is more even than for slick one. The small region of lower velocity behind slick tyre centreline is visible (figure 8, area 1). It can be treated as a result of the energy loss used for work which need to be done by medium to travel around the obstacle. One can also notice a vortex aside of right wheel edge (only on one side of wheel due to asymmetrical geometry of rim). It is present in both cases, however for slick tyre (figure 8, area 2) it is located higher and more to the outside with respect to the grooved wheel. The reason behind this shift may come from the fact, that bigger amount of fluid has to encircle the wheel – for the grooved tyre the vortex core remains at the distance equal to half of wheel width.

![Velocity vector plot - slick tyre](image)

**Figure 8.** Velocity vector plot - slick tyre.

![Velocity vector plot - grooved tyre](image)

**Figure 9.** Velocity vector plot - grooved tyre.
Behind the left wheel edge the flow is biased outwards in the grooved tyre case (figure 9, area 3). This phenomenon is an effect of influence of fluid which have propagated along the grooves, not present in the slick tyre case. Velocity flow field observations confirm previous conclusions related to differences in drag resistance for both tyres.

3.4. Drag differences between tread types
As it was presented, the negative pressure peak near 90° is presented for both, slick and grooved tyres, however its location is slightly different. In case of tyre with grooves it starts before place of contact, while for slick tyre initiates after 90° angle. What is more, the reason of its presence is different depending on the type of the tyre. In case of slick tyre, the negative pressure peak happens due to the suction cup. On the other side, the acceleration of the flow inside the grooves creates an under-pressure, which suppresses the jetting effect present in slick wheel what is manifested by a sudden pressure plunge at the contact patch. Table 2 presents the comparison of findings from current investigation to the study of enclosed wheels made by [9]. The drag coefficient reported in table 1 was multiplied by the frontal area of the wheel in order to make the comparison possible. This effectively created set of area drag coefficients for the three compared studies. Production size tyre was tested by [9], while a model of Formula 1 tyre was under investigation herein.

Table 2. Comparison of numerical results of isolated wheel to CFD simulation and experiment of enclosed wheel reported in [9].

<table>
<thead>
<tr>
<th>Tyre</th>
<th>SC</th>
<th>ΔSC</th>
<th>SC (Kulak et al.)</th>
<th>ΔSC (Kulak et al.)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Slick (MRF)</td>
<td>0.043</td>
<td>-</td>
<td>-</td>
<td>-</td>
</tr>
<tr>
<td>Grooved (MRF)</td>
<td>0.041</td>
<td>-0.002</td>
<td>-0.003</td>
<td>-0.002</td>
</tr>
</tbody>
</table>

It is evident that a drag difference presented by the current study is of the same order and, moreover, value when compared to results recorded for a model vehicle. Similar trend was observed by [8] where a difference of 4 drag counts in favour of grooved tyre was reported in tests of entire vehicle. This suggests that drag difference observed at vehicle scale can be purely related to the presence of grooves and, more importantly, to reduction of jetting effect provided by the grooved portions of the tyre. In the slick tyres, the work is being done on the air as it is squeezed between the wheel and moving ground giving rise to pressures that are greater than the total pressure in the working section. This is also the case for the groove tyre, except the places where a groove is present. This slight change in tread pattern has an important impact on the aerodynamic drag. It puts in question observations made by [5], where no change in drag was reported between tread patterns, but more studies are needed to confirm this hypothesis. Therefore, at Institute of Turbomachinery, in cooperation with PSA Peugeot Citroen, works are under way to develop an experimental stand for isolated wheel studies. At the same time, a reliable CFD model for wheel-ground contact is being developed.

4. Conclusions
Numerical model prepared for this investigation was able to predict major flow characteristics connected with aerodynamic analysis of an isolated wheel. The most important differences between flow around the tyre with grooves and the slick tyre were identified not only in terms of drag coefficient but also pressure distribution. Precise analysis of two approaches used for wheel simulation clearly showed that the use of Multiple Reference Frame model significantly improves the accuracy of numerical model. Difference in drag coefficient between numerical investigation and experiment for
slick tyre can be connected with pressure drop near the area of contact between the wheel and the ground. Same tendency can be observed for tyre with grooves, however jetting effect associated with slick tyre is responsible for bigger pressure drop and as a consequence for bigger profile resistance. Those phenomena were explained by Institute of Turbomachinery at Lodz University of Technology in cooperation with PSA Peugeot Citroen based on Peugeot 207 car body. Results of numerical investigations were verified in PSA wind tunnel.

Future attention should focus on CFD-based immersed body as a technique for rotating wheel simulation. Succeeding work will investigate tyres with various configuration of grooves. This is why in cooperation with PSA a new test stand for more precise analysis of wheel with use of Particle Image Velocimetry (PIV) is being created. What is more, turbulence model independence study, containing nonlinear eddy viscosity models [15], should be performed to verify its influence on the obtained results. Furthermore, based on analysis of pressure distribution for both tyres, it can be observed that the way how data are collected has a significant influence on results in most sensitive place – area of contact between the wheel and the ground. Numerical investigation can therefore help to supplement experimental results with expanded data especially in places hard to access.

5. References
[3] McManus J and Zhang X 2006 A computational study of the flow around an isolated wheel in contact with the ground J. of Fluids Eng. 128 520-530
[12] Karczewski M and Blaszczak J 2008 Performance of three turbulence models in 3D flow investigation for a 1.5-stage turbine TASK Quarterly 12 185-95
[14] Kabalyk K and Kryillowicz W 2011 Verification of the numerical model of the viscous gas flow within a centrifugal compressor stage Nauchno-Technicheskie Vedomosti SPbGPU 2 329-34