



Article Computational Investigation of the Aerodynamics of a Wheel Installed on a Race Car with a Multi-Element Front Wing

Carlo Cravero and Davide Marsano *

Dipartimento di Ingegneria Meccanica, Energetica, Gestionale e dei Trasporti (DIME), Università degli Studi di Genova, Via Montallegro 1, 16145 Genoa, Italy; cravero@unige.it

* Correspondence: davide.marsano@edu.unige.it

Abstract: The search for high aerodynamic performance of a race car is one of the main aspects of the design process. The flow around the basic body shape is complicated by the presence of the rotating wheels. This is especially true in race cars on which the wheels are not shrouded, where the effects on the flow field are considerable. Despite this, few works have focused on the flow around the rotating wheels. In this paper, CFD techniques were used to provide a detailed analysis of the flow structures generated by the interaction between a multielement inverted wing and the wheel of an open-wheel race car. In the first part, the CFD approach was validated for the isolated wheel case by comparing the results with experimental and numerical data from the literature. The wheel was analyzed both in stationary and unsteady flow conditions. Then, the CFD model was adopted to study the interaction of the flow structures between the wheel with the real grooves on the tire and the front wing of a Formula 1 car. Three different configurations were considered in order to differentiate the individual effects. The discussions were supported by the values of the aerodynamic performance coefficients and flow contours.

Keywords: race car; wheel; CFD

1. Introduction

The performance of a race car strongly depends on the car's aerodynamic efficiency. Several elements are used, such as inverted wings, flaps, endplates, diffusers, and barge boards, to control the airflow and generate downforce: this allows the vehicle to maintain its grip on the ground, even in extreme conditions. High vertical loads allow greater traction with the ground, and therefore a higher rating, but mostly guarantee more control when cornering and implicitly higher speeds, thus reducing the lap time. For all these reasons, any improvement in the aerodynamic design is considered strategic to increase the performance of a race car. In the past, basic aerodynamic concepts were developed [1-3]. In these studies, it was clear that the wheels were some of the most influential components to influence the aerodynamic behavior of the vehicle. In fact, if the front wing contributed to 30% of total downforce [4], the wheels of an open-wheel car covered 40% of the total drag [5,6]; this was because the wheel is a bluff body [7]. The interaction of the wheel with the front inverted wing substantially changes the performance of the two bodies compared to those of each single component. It is surprising, however, that in the literature, no data or information on this interaction can be found, apart from the investigations of single components that were individually analyzed. This was probably due to the difficulty in finding the actual geometries of race cars that could be published due to the extremely high confidentiality level of the data in motorsport.

Axon [8] studied an isolated wheel and compared a CFD analysis with experimental measurements. Mears [9] analyzed the pressure distribution around wheels experimentally using a particle image velocimetry (PIV) method to compare the CFD results from a RANS model. He also compared his results with the classical results of Fackrell [10].



Citation: Cravero, C.; Marsano, D. Computational Investigation of the Aerodynamics of a Wheel Installed on a Race Car with a Multi-Element Front Wing. *Fluids* **2022**, *7*, 182. https://doi.org/10.3390/ fluids7060182

Academic Editor: Ivette Rodríguez

Received: 10 March 2022 Accepted: 20 May 2022 Published: 25 May 2022

Publisher's Note: MDPI stays neutral with regard to jurisdictional claims in published maps and institutional affiliations.



Copyright: © 2022 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (https:// creativecommons.org/licenses/by/ 4.0/). McManus and Zhang [11] used an unsteady RANS approach to calculate the flow field around a wheel. More recently, Issakhanian et al. [12] carried out an experiment with PIV measurements to describe the flow field around a 60% scale model of an isolated Formula 1 wheel. They showed the reversed flow regions in the wake of the wheel with its swirling structures. Axerio et al. [13] investigated the flow structure of an isolated 60% scale Formula 1 wheel in stationary and rotating conditions [14]. Specific studies have been published on the reliability of RANS turbulence closures with a realizable k- ε model [15,16]. The influence of a rotating and a stationary wheel on a simplified model of the vehicle with a single airfoil and a smooth tire has been published [17,18]. Regert et al. [19] and Rajaratnam et al. [20] investigated the local flow field around the wheelhouse. They found that compared to stationary wheels, rotating wheels will induce a notable influence on the vortex structure and increase the total aerodynamic drag. Pavia et al. [21] studied the unsteady flow characteristics generated by rotating wheels and pointed out that wheel rotation could affect the wake bistability of the vehicle, as well as the aerodynamic forces. Bonitz et al. [22] found that the flow frequency downstream of the wheels could be altered by the wheel rotation. Wang et al. [23] conducted research on the effects of moving ground and rotating wheels on the aerodynamics of a square-back car model and found that the wheel and ground conditions mainly influenced the flow near the ground. The general wake structure and the total drag were not obviously altered. Wang et al. [24] proposed a wake status they called "wake balance" by comparing the flow field of a square-back model with rotating and stationary wheels. Yu et al. [25] investigated the aerodynamic influence of different ground and wheel conditions on the Notchback DrivAer using numerical simulations. Zhou et al. [26] experimentally and computationally investigated the aerodynamic characteristics of three tires of the 185/65 R14 type with different patterns under loading by comparing a simplified isolated tread tire with the real complex pattern. The geometrical details' influence on the flow structure (the effects of rim coverage area, fan spokes, spoke sharpness) and on the drag coefficient of a passenger vehicle were investigated by Bolzon et al. [27]. Hobeika and Sebben [28] evaluated the contribution of a rotating wheel to the aerodynamic drag of a passenger vehicle. The wheels also play a key role in the flow structure of a car during a braking-in-turn maneuver [29]. In a cornering maneuver, the modeling of moving wheels with respect to the steady case predicted a difference of 3% in the drag coefficient and 5% in the lift coefficient [30]. The numerical effects of three different wheel-rotation simulation methods (i.e., the steady moving wall, the MRF, and the unsteady sliding mesh) on the car aerodynamics were discussed in [31]. Yu et al. [32] investigated the influence of the wheel contact patch on the global car aerodynamic performance.

More recently several works have been published on the aerodynamics of race car front wings [33–35]. The Ansys CFX code, as in the present work, was used to investigate the ground effect in [36]. The CFD model setup was crucial for correctly comparing different racing scenarios [37] or for investigating the effect of the wake on the following car [38]. Moreover, car aerodynamics are subject to a number of random variables that introduce uncertainty into the downforce performance; the effects of the random variations in these parameters are important to accurately predict a car's performance during the race [39,40]. The authors carried out a fluid dynamics analysis of a multielement front wing with a Gurney flap on a Formula 1 car [41] and an extensive aerodynamics analysis on the profile of the ground effect with the Gurney flap to investigate the vortex-shedding phenomena that can occur in certain conditions [42].

Additional analyses have been performed on vortex-shedding generation to quantify the wake and the recirculation zone downstream of a bluff body [43], which can generate tonal noise in industrial applications [44]. The accuracy of the numerical prediction of unsteady flows is also essential for vibroacoustic analysis [45,46]. In this study, our attention was focused on the flow structure around the open wheel and its interaction with the front wing. In the first phase, the CFD approach was validated on a stationary and a rotating isolated wheel using reference data from the literature. In the second part, the aerodynamic interaction of a multielement airfoil installed on a Formula 1 car with a detailed tire with grooves is discussed using a set of CFD simulations.

The main scope of this work was to demonstrate the reliability of a RANS model to study the flow around the rotating unshrouded wheels of a race car, where the interaction with the multielement inverted wing is very notable, and to investigate the flow mechanisms of the above interaction.

2. CFD Analysis of the Flow Structures around an Isolated Wheel

. . .

The case studied in [3,8] has been considered as a reference for the validation of the CFD approach and for the discussion of the flow structures around the open wheel.

2.1. Governing Equations

The mathematical problem is set by the Reynolds-averaged Navier–Stokes equations. The conservation of mass and momentum take the Eulerian conservative divergence form:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \overrightarrow{u} \right) = 0 \tag{1}$$

$$\frac{\partial \left(\rho \vec{u}\right)}{\partial t} + \nabla \cdot \left(\rho \vec{u} \times \vec{u}\right) = -\nabla P + \nabla \cdot \tau + S_M \tag{2}$$

where τ is the tensor of the normal and tangential stress due to viscosity and S_M is the momentum source. The turbulence closure adopted to model the momentum source (the Reynolds stress tensor) is the k- ω SST. It is based on Boussinesq's hypothesis to model the Reynolds stress tensor. This model was developed to combine the accuracy of the k- ω model near the wall and the robustness of the k- ε in the free stream. It contains different terms with respect to the standard k- ω formulation. A blending function activates the models k- ω and k- ε depending on the local value of y+, i.e., close to or far from the wall [47]. A different formulation for the eddy viscosity and modified constants is introduced. The additional transport equations of the model are:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j} \left[\Gamma_k \frac{\partial k}{\partial x_j} \right] + G_k - Y_k + S_k \tag{3}$$

$$\frac{\partial}{\partial t}(\rho\omega) + \frac{\partial}{\partial x_j}(\rho\omega u_j) = \frac{\partial}{\partial x_j} \left[\Gamma_\omega \frac{\partial \omega}{\partial x_j} \right] + G_\omega - Y_\omega + D_\omega + S_\omega \tag{4}$$

where G_k represents the generation of turbulence kinetic energy due to mean velocity gradients, G_{ω} is the generation term of the specific dissipation rate, Y_k and Y_{ω} are the dissipation of k and ω , and S_k and S_{ω} are source terms.

The diffusivity is obtained by the following equations:

$$\Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \tag{5}$$

$$\Gamma_{\omega} = \mu + \frac{\mu_t}{\sigma_{\omega}} \tag{6}$$

The eddy viscosity is finally computed with:

$$\mu_t = \frac{\rho k}{\omega} \frac{1}{\max\left[\frac{1}{a^*}, \frac{SF_2}{a_1\omega}\right]} \tag{7}$$

The model constants are: $\sigma_{k,1} = 1.176$, $\sigma_{\omega,1} = 2.0$, $\sigma_{k,2} = 1.0$, $\sigma_{\omega,2} = 1.168$, $\alpha_1 = 0.31$, $\beta_{i,1} = 0.075$, and $\beta_{i,2} = 0.0828$.

2.2. CFD Model

The software ICEM CFD and Ansys CFX were used as simulation platforms. The operating conditions of the present model were obtained by scaling the data from the open literature cases. The flow domain consists of a rectangular hexahedron containing the wheel with a diameter d = 620 mm and width of the tread w = 320 mm. The domain has the following geometric characteristics (referred to as the tire diameter d): width in the y-direction y/d = 3.66, height z/d = 2.93. The inlet surface is placed at $x_1/d = 5.0$ and the outlet surface at $x_3/d = 15.0$. The domain is large enough to not influence the flow structure of the wheel, which is the common best practice in external aerodynamic simulations. The domain dimensions stem from previous works [41,42]. Figure 1 shows the sketch of the flow domain.



Figure 1. Flow domain of the isolated wheel.

The domain has been discretized with an unstructured grid with a prism layer of 10 layers near the wall of the vehicle and its components in order to solve the effects of the boundary layer. The size of the first cell has been set to have a y+ close to one with a dimension of 0.0054 mm. The size of the elements is increased from a minimum size of 0.5 mm of the tetrahedral elements at the walls of the grooves far from the body. The global mesh size consists of about 15 million cells. In Figure 2, the surface mesh detail of the wheel (with the tread grooves on the tire) and a cut plane of the volume mesh are shown.



Figure 2. Volume mesh cut plane (left) and surface mesh detail of the wheel (right).

The k- ω SST model has been designed to give accurate predictions of the onset and the amount of flow separation under adverse pressure gradients by the inclusion of transport effects in the formulation of the eddy-viscosity. This results in a major improvement in terms of flow separation predictions. The superior performance of this model has been demonstrated in many validation studies [48]. Moreover, it provides good results in

comparison to DES or LES models that require computational resources of a different order or higher magnitude. The following boundary conditions have been set: at the inlet of the domain, a uniform velocity of 12.85 m/s and a turbulence intensity of 0.2%. The incoming fixed flow has been calculated according to the diameter of the wheel to have the same Reynolds number as McManus and Zhang [11]: Re = 5.3×10^5 . At the outlet of the domain, a pressure ambient condition has been fixed. The lateral and the upper walls of the rectangular domain have been set as the inviscid wall, while all the remaining walls (including the ground and the wheel) have been treated as no-slip walls. Two cases have been considered for the isolated wheel model: stationary or rotating wheel. In this second case, the ground has been fixed at the same velocity of the incoming flow (in opposite direction) and an angular wall velocity of 41.42 rad/s for the wheel surfaces. The air has been modeled as an incompressible isothermal flow at 25 °C. All the equations have been solved with second-order numerical schemes carrying out steady simulation. The convergence of the simulation has been reached after about 1000 iterations when the continuity residues fall below 10^{-4} and all the other residues are abundantly under 10^{-6} . In the above conditions, the aerodynamic coefficients have an asymptotic fully converged trend.

2.3. Validation and Flow Structure Analysis

The use of a more realistic geometry that considers the wheel camber, grooves and footprint, involves an asymmetric flow structure, mainly evident in the stationary wheel, where the wake is simpler. The vorticity iso-surface and the streamlines around the wheel are shown in Figure 3 for the stationary wheel case. Here, the internal side vortex covers a larger area than the one generated with a wheel with zero camber [3] and develops from the contact area between the ground and wheel, unlike the outer one that is far away from the wheel with a lower intensity. The grooves affect the flow development on the tread with a less straight path in the back area if compared to a smooth tire as shown in Figure 4 with the surface streamlines of wall shear stress; flow recirculations are detected with curved and close lines. Due to the simplification of the rims with two flat circular plates, it is not possible to obtain the vortices originating on the sides as discussed in [3] on the edge of the upper rear tread between 10° and 30° from the horizontal plane.



Figure 3. Streamline and vorticity iso-surface on the stationary wheel.



Figure 4. Wheel wall shear stress surface streamlines.

As discussed by several researchers, there is a substantial difference in the flow structures between the stationary and rotating wheels. In the latter, there is the presence of a third vortex as shown in Figure 5. It develops from the top back of the wheel, forms a vortex-shaped arch, and causes the wake to develop more in the vertical direction and become more compact than in the stationary wheel case.



Figure 5. Streamline detail of upper vortex on the rotating wheel.

The obtained trends for drag and lift coefficients correspond to what is shown in the reference cases from the literature; they are reported in Figure 6. In both cases, there is a reduction in values if the wheel is rotating. The C_D is reduced by 7%; this is a higher value than that found by McManus and Zhang [11], but much smaller than the experimental evidence. The C_L decreases by 52% and is in line with what happens in the reference data. The same percentage reduction between the stationary and the rotating wheel (around 3%) has been detected in this work with respect to the numerical simulations of McManus and Zhang [11] for the integral pressure coefficient. No experimental measurements of the pressure coefficients are available for the stationary wheel from the literature reference. The differences between the numerical models can be attributed to different geometry details and different turbulence models (SST vs. $k - \varepsilon$) combined with a much coarser mesh (5:1) from the data in reference [3]. The difference in the experimental data can be due to the accuracy of pressure and force measurements in those conditions. The use of the fine mesh

combined with the turbulence model of the present results is considered adequate to give accurate flow details for the wheels to be used for the analysis of their interaction with the front wing.







Figure 6. Aerodynamic coefficients of isolated wheels.

3. Front Wing and Wheel CFD Model

The model used in this section consists of a multi-element wing with a nose cone. The wheel is included and the rear part of the car (with a length $l_v = 3.5$ m) is simplified up to a distance of 2.4 m from the front of the inverted wing with a tail shape in order to interfere as little as possible with the wake of the upstream components. The front wing is composed of a main wing, a flap and two endplates. The suspension arms are included and connected to the wheel rims that are simplified and closed with two circular surfaces.

The detail of the flow in the brake system is not included. The wheel is made of a tire with a grooved tread; the rotation axis is tilted from horizontal to simulate the wheel camber. In order to save a considerable amount of computational resources, only one-half of the front wing and the vehicle axial symmetry is considered. The main geometrical characteristic of this model has been reported in a previous work [41]. A similar CFD domain to the previous model for the isolated wheel (Section 2.1) is considered: the ground is a surface of $3l_v \times 8l_v$ and the entire domain has an height $2l_v$. In addition, the surface in front of the wing is placed at $2l_v$ and the surface at the outlet is $5l_v$ away. Figure 7 shows the scheme of the flow domain and the details of the vehicle geometry. The domain is discretized with an unstructured grid with a prism layer near the walls of the vehicle and the wheel, with the same characteristics as described in the previous section. Clearly, the presence of the wing changes the reference minimum (1.0 mm) and maximum (1.5 mm) mesh dimensions at the trailing edge with an increased number of overall mesh cells. Two different meshes have been generated: one for the model of the car without the wheel and one for the car with the real wheel. They have a global size of 34 and 45 million cells, respectively. This mesh strategy with the specific local refinement on the wings has been selected after a grid dependency analysis [41,42]. In a previous work [42], the CFD model has been validated by comparing the numerical results with detailed experimental data [49] at different distances from the ground and with different GF heights. In Figure 8, the detail of the surface mesh on the front wing is reported.



Figure 7. Computational domain and geometry details of the front wing and wheel.



Figure 8. Detail of the surface mesh on the endplate.

The k- ω SST turbulence model is set together with the following boundary conditions: at the inlet of the domain, a uniform velocity of 70 m/s and a turbulence intensity of 5%, and ambient pressure at the outlet of the domain. The symmetry condition at the symmetry surface and the no-slip wall condition with the same velocity as the car at the ground have been set. The external surfaces of the domain have been set as inviscid walls and the walls of the car and its components are treated with no-slip conditions. Only in the case with the rotating wheel, an angular wall velocity of 230 rad/s has been fixed at the wheel. The air is modeled as a perfect isothermal and incompressible fluid. The equations are solved with second-order numerical schemes and steady simulations. The convergence of the simulations is based on the values of the residues and forces acting on the car and on the wheel. The same convergence criteria as for the previous application are considered; the residues and the forces on the wheel reached asymptotic values after approximately 1500 iterations.

4. Flow Structure around an Open Wheel on Race Car with Front Wing

Three configurations have been simulated and compared, as reported in Table 1, to analyze the different flow structures and to highlight the different aerodynamic interactions between parts.

#	Case
Case 1	Geometry without suspension and wheel
Case 2	Complete model with stationary wheel
Case 3	Complete model with rotating wheel

Table 1. Configurations simulated.

All the cases are simulated with a steady flow assumption and with the model setup described in Section 3. To better understand the main differences in the flow structures between the three cases, some control planes have been added for the post-processing, as shown in Figure 9. The control planes are located immediately before (plane A: x = 0 m) and behind (plane B: x = 0.66 m) the car front, and at two further downstream sections (plane C: x = 1.5 m–plane D: x = 2.4 m). The wheel is positioned between plane B and plane C.



Figure 9. Control planes for the post-processing.

In Figure 10, the velocity magnitude contours with velocity vectors on the control planes are reported to compare the different flow structures of the three simulated cases. The contours plotted in the above figure span from 0 m/s (blue) to 100 m/s (red). The presence of the wheels behind the flap clearly affects the velocity values on the car upstream (plane A): in the area below the airfoil, the flow is always accelerated (as in case 1), but has lower values due to downstream obstruction.

The strong relevance of the wheel can be noticed also on plane B for case 2 and case 3. The flow is pushed outwards to the car lateral side and the small flow rate in the narrow area between the tire and the car is fully deflected upward. This is because the endplate generates a certain vorticity level with anticlockwise vortices, clearly visible in case 1. They are still visible but less intensive in the other cases.

A significant difference between the single front wing (case 1) and the other cases with the wheel is evident on control plane C. The presence of the wheel (stationary or rotating) induces a large area with a low speed and multiple vortex structures. With a stationary wheel, there are three distinct vortices on the ground with high intensity. The outer one is due entirely to the wheel (as in the isolated case previously described) while the middle one and the inner ones can be partially attributed to the wheel and partially to the Coanda effect from the wing endplate. In fact, the flow is diverted into the car body and when it finds the lower suspension arms it results in a swirling motion. With the rotating wheel, the vortices on the ground are only two, and are stronger and confined to a smaller area. The vortex that originates from the top back of the tread is present in both case 2 and case 3 as discussed in the single wheel analysis.

In the above cases, an additional vortex generates near the car body, not due to the wheel but due to the presence of the front suspension upper arm that, with its elliptical section, alters the local flow structures. In Figure 11, a 3D view of the above vortex formation is reported using streamlines and velocity magnitude contours on an orthogonal control plane. Here, the vorticity at the aileron endplate outlet hits the internal tread by reducing the stagnation zone. Due to the proximity of the elements with the tire, the flow resistance is lower than in the isolated wheel case.

An intermediate vortex is generated in case 3; it connects the standard upper vortex of the rotating wheel to the one that develops on the ground on the lateral side. This is due to the interaction of flows coming from the area between the wheel and the car and those passing outside the tire that tends to join the wake below: this interaction creates the clockwise vortex.

In both cases 2 and 3, there is a narrow area at a very slow velocity, mainly due to the flow that follows the upper profile of the endplate and flaps placed above, which impacts the upper arms linked to the wheel.



Figure 10. Comparison of the velocity contours on four transversal planes among the three analyzed cases.

In the last plane D, there is a fade of the vortices from case 1, while in the other two cases, the vortices are still present but they merge with reduced strength. If the wheel is stationary, the vortex core with lower velocity is close to the car body when it is placed on the ground; the outer eddy has regained strength thanks to its interaction with the incoming undisturbed flow.

The opposite occurs if the tire is rotating: the outer vortex is still very evident and very slow while the inner side has considerably reduced. The strong difference can be discussed by comparing planes C and D. In case 3, there are only two vortices behind the wheel rather than three, meaning the outer vortex also includes the middle one, increasing its intensity. Case 2 shows the opposite behavior. It is interesting to discuss the pressure distributions on the car components, paying specific attention to the wheels. The flow that impacts the tread is partly at a low velocity due to the inverted wing and endplate, thus reducing the area of stagnation pressure, as seen in Figure 12. In these pictures, the pressure difference between the top of the stationary and rotating wheel is highlighted. With the stationary wheel, the pressure has a minimum located on the top of the tread, while if the wheel is

rotating, the pressure reduction is less evident and more gradual due to the separation of the boundary layer of the fluid from the surface. Moreover, the stagnation zone in the rotating case is displaced on the portion of the tread which is more exposed to free flow, while the internal part has lower pressure (thanks to the vertical plate located in the front, which diverts the flow inwards). This avoids a strong flow impact with reduced velocities and high pressures.



Figure 11. Vortex formation from the upper suspension.



Figure 12. Pressure contours on 3D model with stationary wheel (upper) and rotating wheel (lower).

With the front wing installed, the stagnation zone on the wheel is less extended than in the isolated wheel case. The aerodynamic interaction between the wheel and front wing reduces drag and lift coefficients and slightly increases the pressure coefficient. In Figure 13, the aerodynamic coefficients for the wheel with the front wing are reported in both cases of stationary and rotating wheels. A reduction for all aerodynamic coefficients of the rotating wheel with respect to the stationary wheel is observed. This behavior matches the trends reported in the previous section for the isolated wheel, as well as with the literature reference for the entire car [8].



Figure 13. Performance comparison of the wheel with front wing -stationary and rotating wheel cases.

The aerodynamic interaction of the wheel with the front wing can be quantified by the pressure coefficient distributions for the wing sections for the main airfoil and the flap. In Figure 14, the above distributions are compared for a wing section near the endplates (positioned at y = -0.48 m) for the isolated front wing or with a stationary or rotating wheel installed. It is evident that the aerodynamic load of the isolated wing is higher than with the installed wheel. In fact, the negative peak near the lower leading edge of the main wing exceeds $C_p = -3$ but with the presence of the wheel, this peak is reduced by about a third, together with the depression on the suction side of the airfoils. This is because the wheel produces a flow blockage effect on the upstream wing that creates back pressure on the lower part (suction side) of the inverted wing by decreasing its velocity with a reduction of local depression. The effect of wheel rotation is modest on the above aerodynamic load. With a rotating wheel, a slightly deeper suction peak is observed. The different trends between the rotating and the stationary wheels are due to the flow energization given by rotation. The rotating wheel has a suction effect on the flow from the wing; the flow is dragged around the wheel with a beneficial effect on the suction peak, and it is possible to partially recover velocity and diminish the local pressure peak. The overall lift coefficient (downforce) of the isolated wing is $C_L = 1.25$ and it drops to $C_L = 0.819$ with a stationary wheel or $C_L = 0.957$ with a rotating wheel.



Figure 14. Pressure coefficient distributions on main airfoil and flap - section at y = -0.48 m.

5. Conclusions

This study focused on the detailed analysis of flow structures generated by the interaction between the multi-element airfoil and wheel assembly, including the entire front of the car, of an actual F1 model from the year 2000. In the first part, the isolated wheel flow structure was analyzed; the results with stationary or rotating wheels were compared with a numerical and experimental reference work to show the reliability of the CFD approach. A reduction in the aerodynamic coefficients was detected on the rotating wheel case with respect to the stationary case; the C_D was reduced by 7% and the C_L decreased by 52%. In the second section, the wheel's interaction with a front wing was analyzed and a more complicated flow structure was discussed. The study of vortex motions generated by the interaction of the flow with the bodies is the basis for the technical development of recent race cars. For instance, the peculiar layout of the endplate diverts part of the flow near the wheel between the two suspension arms thanks to the exploitation of the Coanda effect. It is not a coincidence that this space is chosen for the location of the air intake to cool the braking system. The swirling flow in that area has low pressure, $C_P = -0.11$, thus enhancing the air intake efficiency.

Similar considerations can be drawn for the control plane D where the air intake for the engine radiators is positioned. The air intakes on the side of the car body are positioned close to the inner vortex area, with local $C_p = -0.1$, which guarantees the fluid's suction to the radiators. Furthermore, the suspension profiles can strongly affect the fluid dynamics of the car according to the incoming flow structure delivered by the upstream components. The same applies to the endplates. They maintain high levels of downforce at the side end of the airfoils, but also drive the flow in dedicated areas to reduce drag and lift on the wheels. Since the wheels contribute to 40% of the total car drag, it is essential to try to reduce it as much as possible. The external flow's interaction with the braking system

inside the rotating wheel is another fundamental topic to enhance the braking efficiency by keeping low drag due to flow interaction effects [50].

Author Contributions: C.C. and D.M. have equally contributed to the concept of the research activity, the setup of the model, the discussion of the results, and the writing of the paper. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: Not applicable.

Conflicts of Interest: The authors declare no conflict of interest.

Nomenclature

- A Wheel frontal area: $d \times b$
- b Wheel breadth
- C Chord
- C_d Drag coefficient: L/(q $_\infty$ A)
- C_1 Lift coefficient: D/($q_{\infty}A$)
- $C_p \qquad \text{Pressure coefficient: } p p_\infty/q_\infty$
- d Wheel diameter
- D Drag force
- *k* Turbulent kinetic energy
- l_v Vehicle length
- L Lift force
- p_{∞} Freestream static pressure
- P Static pressure
- q_{∞} Freestream dynamic pressure: $(r_{\infty}U^2_{\infty})/2$
- *u* velocity
- U_{∞} Free stream velocity
- W Width
- y⁺ Non dimensional boundary layer distance from wall
- μ Dynamic viscosity
- ρ Density
- $\rho\infty$ Freestream density
- au Tensor of tangential and normal stress
- ω Specific rate of dissipation

References

- 1. Cogotti, A. Aerodynamic characteristics of car wheel. In Proceedings of the International Conference Held at the Café Royal, London, UK, 16–18 June 1982.
- Wickern, G.; Zwicker, K.; Pfadenhauer, M. Rotating Wheels–Their Impact on Wind Tunnel Test Techniques and on Vehicle Drag Results; SAE Technical Paper No. 970133; SAE: Warrendale, PA, USA, 1997.
- 3. Katz, J. Race Car Aerodynamics: Designing for Speed; Bentley Publishers: Philadelphia, PA, USA, 1995.
- 4. Katz, J. Aerodynamic model for wing-generated down force on open-wheel-racing-car configurations. SAE paper 860218. *SAE Trans.* **1986**, *95*, 129–137.
- 5. Dominy, R.G. Aerodynamics of Grand Prix cars. Proc. Inst. Mech. Eng. Part D 1992, 206, 267. [CrossRef]
- 6. Agathangelou, B.; Gascoyne, M. *Aerodynamic Design Considerations of a Formula 1 Racing Car*; SAE Technical Paper No. 980399; SAE: Warrendale, PA, USA, 1998.
- 7. Wright, P. Ferrari Formula 1: Under the Skin of the Championship Winning F1-2000; David Bull Publishing: Phoenix, AZ, USA, 2003.
- 8. Axon, L. The Aerodynamic Characteristics of Automobile Wheels–CFD Prediction and Wind Tunnel Experiment. Ph.D. Thesis, Cranfield University, Cranfield, UK, 1999.
- 9. Mears, A.P. The Aerodynamic Characteristics of an Exposed Racing Car Wheel. Ph.D. Thesis, Durham University, Durham, UK, 2004.
- 10. Fackrell, J.E. The Aerodynamic Characteristics of an Isolated Wheel Rotating in Contact with the Ground. Ph.D. Thesis, University of London, London, UK, 1974.

- 11. McManus, J.; Zhang, X. A computational study of the flow around an isolated wheel in contact with the ground. *ASME J. Fluids Eng.* **2006**, *128*, 520–530. [CrossRef]
- 12. Issakhanian, E.; Chris, J.E.; Kin, P.L.; John, K.E. An experimental study of the flow around a formula one racing car tire. *ASME J. Fluids Eng.* **2010**, *132*, 071103.1–071103.8. [CrossRef]
- 13. Axerio-Cilies, J.; Gianluca, I. An aerodynamic investigation of an isolated rotating formula 1 wheel assembly. *ASME J. Fluids Eng.* **2012**, 134, 121101.1–121101.16. [CrossRef]
- 14. Cilies, J.; Issakhanian, E.; Jimenez, J.; Gianluca, I. An aerodynamic investigation of an isolated stationary formula 1 wheel assembly. *ASME J. Fluids Eng.* **2012**, *134*, 021101.1–021101.17. [CrossRef]
- 15. Kang, S.O.; Jun, S.O.; Park, H.I.; Song, K.S.; Kee, J.D.; Kim, K.H.; Lee, D.H. Actively translating a rear diffuser device for the aerodynamic drag reduction of a passenger car. *Int. J. Automot. Technol.* **2012**, *13*, 583–592. [CrossRef]
- 16. Ting, L.; Qing, J.; Zhi, G.Y. The influence of rotating wheels on vehicle aerodynamics. Appl. Mech. Mater. 2012, 246–247, 543–547.
- Waschle, A.; Cyr, S.; Kuthada, T.; Wiedemann, J. Flow Around an Isolated Wheel–Experimental and Numerical Comparison of Two CFD Codes; SAE Technical Paper No. 2004-01-0445; SAE: Warrendale, PA, USA, 2004.
- Waschle, A. The Influence of Rotating Wheels on Vehicle Aerodynamics–Numerical and Experimental Investigations; SAE Technical Paper No. 2007-01-0107; SAE: Warrendale, PA, USA, 2007.
- 19. Regert, T.; Lajos, T. Description of flow field in the wheelhouses of cars. Int. J. Heat Fluid Flow 2007, 28, 616–629. [CrossRef]
- 20. Rajaratnam, E.; Walker, D. Investigation of wheelhouse flow interaction and the influence of lateral wheel displacement. *Energies* **2019**, *12*, 3340. [CrossRef]
- Pavia, G.; Passmore, M. Characterisation of wake bi-stability for a square-back geometry with rotating wheels. In Proceedings of the 11th FKFS Conference, Sttutgart, Germany, 26–27 September 2017; Springer: Cham, Switzerland, 2017; pp. 93–109.
- 22. Bonitz, S.; Larsson, L.; Sebben, S. Unsteady pressure analysis of the near wall flow downstream of the front wheel of a passenger car under yaw conditions. *Int. J. Heat Fluid Flow.* **2018**, *73*, 188–198. [CrossRef]
- 23. Wang, S.; Avadiar, T.; Thompson, M.C.; Burton, D. Effect of moving ground on the aerodynamics of a generic automotive model: The DrivAer-Estate. *J. Wind. Eng. Ind. Aerodyn.* **2019**, *195*, 104000. [CrossRef]
- 24. Wang, Y.; Sicot, C.; Borée, J.; Grandemange, M. Experimental study of wheel-vehicle aerodynamic interactions. J. Wind. Eng. Ind. Aerodyn. 2020, 198, 104062. [CrossRef]
- 25. Yu, X.; Jia, Q.; Yang, Z. Comprehensive study of the aerodynamic influence of ground and wheel states on the notchback DrivAer. *Energies* **2022**, *15*, 1124. [CrossRef]
- 26. Zhou, H.; Jiang, Z.; Wang, G.; Zhang, S. Aerodynamic Characteristics of isolated loaded tires with different tread patterns: Experiment and simulation. *Chin. J. Mech. Eng.* **2021**, *34*, 6. [CrossRef]
- Bolzon, M.D.P.; Sebben, S.; Broniewicz, A. Effects of wheel configuration on the flow field and the drag coefficient of a passaenger vehicle. *Int. J. Automot. Technol.* 2019, 20, 763–777. [CrossRef]
- 28. Hobeika, T.; Sebben, S. CFD investigation on wheel rotation modelling. J. Wind Eng. Ind. Aerodyn. 2018, 174, 241–251. [CrossRef]
- 29. Broniszewski, J.; Piechna, J.R. Fluid-structure interaction analysis of a competitive car during brake-in-turn manoeuvre. *Energies* **2022**, *15*, 2917. [CrossRef]
- 30. Piechna, J.R.; Kurec, K.; Broniszewski, J.; Remer, M.; Piechna, A.; Kamieniecki, K.; Bibik, P. Influence of the car movable aerodynamic elements on fast road car cornering. *Energies* 2022, *15*, 689. [CrossRef]
- Yu, X.; Jia, Q.; Bao, D.; Yang, Z. A Comparative Study of Different Wheel Rotating Simulation Methods in Automotive Aerodynamics; SAE Technical Paper 2018-01-0728; SAE: Warrendale, PA, USA, 2018.
- 32. Yu, X.; Jia, Q.; Rashidi, M.M.; Yang, Z. Comprehensive investigation on the aerodynamic influences of the wheel contact patch. *J. Appl. Comput. Mech.* **2020**, *6*, 934–955.
- 33. Huminic, A.; Chiru, A. On CFD Investigations of Vehicle Aerodynamics with Rotating Wheels' Simulation; Technical Paper 2006-01-08-04; SAE: Detroit, MI, USA, 2006.
- Wang, Y.N.; Tseng, C.Y.; Huang, Y.L.; Leong, J.C. Investigation of 2004 ferrari formula one race car wing effects. In Proceedings of the 2010 International Symposium on Computer, Communication, Control and Automation (3CA), Tainan, Taiwan, 5–7 May 2010; pp. 85–88.
- Kieffer, W.; Moujares, S.; Armbya, N. CFD study of section characteristics of Formula Mazda race car wings. *Math. Comput. Model.* 2006, 43, 1275–1287. [CrossRef]
- 36. Huminic, A.; Huminic, G. CFD investigations of an open-wheel race car. In Proceedings of the 4th European Automotive Simulation Conference, Munich, Germany, 6–7 July 2009.
- Soso, M.D.; Wilson, P.A. Aerodynamic of a wing in ground effect in generic racing car wake flows. *Proc. Inst. Mech. Eng. Part D* 2006, 220, 1–13. [CrossRef]
- Newbon, J.; Dominy, R.; Sims-Williams, D. CFD investigation of the effect of the salient flow features in the wake of a generic open-wheel race car. SAE Int. J. Passeng. Cars-Mech. Syst. 2015, 8, 217–232. [CrossRef]
- 39. Bradford, J.; Montomoli, F.; D'Ammaro, A. Uncertainty quantification and race car aerodynamics. *Proc. Inst. Mech. Eng. Part D J. Automob. Eng.* 2014, 228, 403–411. [CrossRef]
- 40. Ahlfeld, R.; Ciampoli, F.; Pietropaoli, M.; Pepper, N.; Montomoli, F. Data-driven uncertainty quantification for Formula 1: Diffuser, wing tip and front wing variations. *Proc. Inst. Mech. Eng. Part D J. Automob. Eng.* **2019**, 233, 1495–1506. [CrossRef]

- 41. Basso, M.; Cravero, C.; Marsano, D. Aerodynamic effect of the gurney flap on the front wing of a F1 car and flow interactions with car components. *Energies* **2021**, *14*, 2059. [CrossRef]
- 42. Cravero, C. Aerodynamic performance prediction of a profile in ground effect with and without a Gurney flap. *ASME J. Fluids Eng.* **2017**, *139*, 031105-1–031105-15. [CrossRef]
- Cravero, C.; Marogna, N.; Marsano, D. A Numerical study of correlation between recirculation length and shedding frequency in vortex shedding phenomena. WSEAS Trans. Fluid Mech. 2021, 16, 48–62. [CrossRef]
- 44. Cravero, C.; Marsano, D. Numerical prediction of tonal noise in centrifugal blowers. In Proceedings of the Turbo Expo 2018: Turbomachinery Technical Conference & Exposition, Oslo, Norway, 11–15 June 2018; ASME Paper GT2018-75243.
- Niccolini Marmont Du Haut Champ, C.A.; Silvestri, P. Experimental and numerical vibro-acoustic investigation on a trimmed car door to analyze slamming event. *Appl. Acoust.* 2020, 166, 107380. [CrossRef]
- 46. Guerrero, J.E.; Pacioselli, C.; Pralits, J.O.; Negrello, F.; Silvestri, P.; Lucifredi, A.; Bottaro, A. Preliminary design of a small-sized flapping UAV: I. Aerodynamic performance and static longitudinal stability. *Meccanica* **2016**, *51*, 1343–1367. [CrossRef]
- 47. ANSYS Inc. Ansys CFX Theory Guide v.17; ANSYS Inc.: Canonsburg, PA, USA.
- Bardina, J.E.; Huang, P.G.; Coakley, T.J. *Turbulence Modeling Validation Testing and Development*; NASA Technical Memorandum; Create Space Independent Publishing Platform: Scotts Valley, CA, USA, 1997; p. 110446.
- 49. Zerihan, J.; Zhang, X. Aerodynamics of a single element wing in ground effect. J. Aircr. 2000, 37, 1058. [CrossRef]
- 50. Cravero, C.; Marsano, D. Flow and thermal analysis of a racing car braking system. *Energies* 2022, 15, 2934. [CrossRef]