Numerical simulation and wind tunnel measurements on a tricycle wheel sub-system

47th APPLIED AERODYNAMICS SYMPOSIUM
PARIS, FRANCE, 26-28 MARCH 2012
Thomas Driant, Lakhdar Remaki, Stéphane Moreau, Alain Desrochers, Hachimi Fellouah*

Department of mechanical engineering, Université de Sherbrooke
2500, Boulevard de l’Université, Sherbrooke, QC, Canada
*Corresponding author: Hachimi.Fellouah@USherbrooke.ca

Abstract: The drag of an isolated rotating wheel sub-system including its accessories (tire, suspension, A-arms and fender) of a tricycle vehicle was studied numerically using the Star CCM+. The purpose of the present work is to optimize the wheel subsystem through drag analysis. The numerical model is validated by comparison to the wind tunnel experimental drag and pressure coefficient. Two aerodynamic shapes of A-arms are designed to perform drag reduction. Numerical simulations show a sensitive drag decrease up to 19.9% of the original wheel subsystem.

INTRODUCTION

The need to protect the environment and improve fuel efficiency has pushed aerodynamics toward the top of the automakers priority lists. Also as fuel prices are increasing, vehicle owners are more and more concerned by the gas mileage of their vehicles.

Reducing the energy consumption of a vehicle could be achieved by different ways for instance; reducing the vehicle mass, increasing the engine performance, reducing the rolling resistance drag, and optimizing the airflow around the vehicle to reduce aerodynamic drag. Some of these solutions are complex to implement on small vehicles. Based on an internal study, reducing 50 kg in the vehicle weight will provide 3% reduction of the consumption. However this solution requires a new study of the vehicle behavior and an increase in cost. Increasing the engine performance is also costly for a production vehicle. Reducing the aerodynamic drag has become an important parameter since it allows improving the fuel economy of the vehicle without adding any weight or without introducing any expensive technologies.

Moreover, the internal study shows a dissipation force distribution in a World Motorcycle Test Cycle (WMTC) of 32% for the rolling resistance drag and 68% for the aerodynamic vehicle total drag. The airflow past the wheel sub-system is complex, it is characterized by the formation of large vortices in both the vertical and lateral planes inducing drag. Understanding the dynamics of these vortices is thus very important in the design of an efficient aerodynamic vehicle.

Fig 1: The wheel sub-system. a) CAD design for simulation and tests, b) installed in the wind tunnel
According to Barlow’s review [1] the tricycle wheel subsystem (Fig. 1) contributes to a third of the vehicle total drag. The aerodynamic design of this subsystem has a major role in the overall design of the vehicle. This motivates the present work to be focused on the aerodynamic optimization of the A-Arms. To our knowledge, the proposed study is the first aerodynamic simulation and experiment on the whole wheel subsystem to study the effect of the A-Arms. The literature review shows, however, some detailed studies of an open rotating wheel of a formula 1 racing car [2-6].

In the present study, first, numerical simulations for different operating conditions on a standard wheel subsystem geometry are performed using the Star CCM+ commercial code. Results of these simulations are validated by comparing with experimental drag and pressure coefficient obtained during the wind tunnel experiments. Two aerodynamic shapes of the A-Arms are considered here following a strategy that is described below. A drag decreasing up to 19.9% of the original wheel sub-system installed on the present vehicle is then obtained.

1. EXPERIMENTAL SETUP

The experimental study is performed in the 1.82 m x 1.82 m section of the “S1 wind tunnel”, located at Sherbrooke University. The test section is 10 m long and the wind is generated by a 1.8 m diameter vane axial fan driven by a 200 hp electric motor. The rotational speed of the fan, and consequently the wind speed in the test section, can be varied using a controller variable frequency drive. The top air velocity in the empty 1.82 m x 1.82 m test section is 30 m/s. The velocity profile was measured uniform within ± 1% except in proximity of the wall where the boundary layer has a thickness of the order in 5 cm. The level of turbulence is less than 0.3%.

A locally designed wind tunnel aerodynamic balance (Fig 2) with three “S” load cell gauges were used to measure drag, lift and pitching moment. The balance is installed on the structure and isolated from the airflow during the measurements, as shown Fig 1b. The $C_d.A$ coefficient is then estimated by the equation (1):

$$ F = \frac{1}{2} \rho_{ah} C_d A V^2 $$

(1)

Where $F$ is the drag force, $\rho_{ah}$ is the air density (with temperature and humidity correction), $C_d$ is the drag coefficient, $A$ is the frontal area and $V$ is the main flow velocity.

Five piezoelectric differential pressure transducers of ±1PSI (Honeywell Sensing and control, SX01DD4, accuracy: 0.012% max) are installed on the wheel fender at different position to measure the wall pressure (the pressure taps have 0.5 mm hole diameter). Two instrumented fenders are used; one to measure the upper side pressure and the second to measure the lower side pressure.

For the positioning of the pressure taps, a cylindrical coordinate system (around the wheel axis) is used. The five positions of the taps are: 49, 134, 111, 74, and 14 degrees for the upper side (see Fig 3). The accuracy of the taps positioning angles is about ±1 degree.

Atmospheric conditions are recorded at the beginning and at the end of each test in order to measure the air density accurately (see [7] for more details). The output signal of the pressure transducers and load cells gauges was digitized using an E-Daq drive with an integrated run-time display for real-time tests. The time traces were recorded at a sampling frequency of 50 Hz during 90 seconds. The experiments were conducted for the rotated wheel factory fender at two incoming flow velocities 20 m/s and 27.7 m/s. A Fast Fourier Transform (FFT) is applied to the acquired time-
series of the instantaneous data from the pressure transducers and the cell gauges to control the resonance frequencies. Noise in the spectrum was reduced by employing a moving average filter. The experimental results obtained for the rotated wheel factory fender are shown in Table 1.

2. NUMERICAL MODEL

1.1. Governing equations

The governing three dimensional turbulent incompressible flow are expressed, relative to a Cartesian coordinate system \((x_1, x_2, x_3)\), over a fixed volume \(\mathcal{V}\) with a closed surface \(\mathcal{S}\), in the integral form

\[
\frac{d}{dt} \int_{\mathcal{V}} Q d\mathcal{V} = \int_{\mathcal{S}} F_{\alpha}(Q)n^\alpha d\mathcal{S} - \int_{\mathcal{V}} G_{\alpha}(Q)n^\alpha d\mathcal{V}
\]

where the summation convention is employed, with \(\alpha = 1, 2, 3\) and \(n = (n_1, n_2, n_3)\) denotes the unit outward normal vector to \(\mathcal{S}\). In this equation, the unknown \(Q\), the inviscid flux vectors \(F_{\alpha}\) and the viscous flux vector \(G_{\alpha}\) are defined by:

\[
Q = \begin{bmatrix} 1 \\ u_1 \\ u_2 \\ u_3 \end{bmatrix}, \quad F_{\alpha} = \begin{bmatrix} u_{\alpha} \\ u_1 u_{\alpha} + \frac{\rho}{\mu} \delta_{1\alpha} \\ u_2 u_{\alpha} + \frac{\rho}{\mu} \delta_{2\alpha} \\ u_3 u_{\alpha} + \frac{\rho}{\mu} \delta_{3\alpha} \end{bmatrix}, \quad G_{\alpha} = \begin{bmatrix} 0 \\ \tau_{\alpha 1} \\ \tau_{\alpha 2} \\ \tau_{\alpha 3} \end{bmatrix}
\]

Here, \(\rho\) denotes the average pressure of the fluid, \(u_{\alpha}\) is the average velocity of the fluid in the \(x_\alpha\) direction, \(t\) denotes time and \(\delta_{\alpha\beta}\) is the Kronecker delta. The averaged deviatoric stress tensor is defined by

\[
\tau_{\beta\alpha} = -\frac{2}{3} \frac{\mu}{\rho} \frac{\partial u_{\beta}}{\partial x_{\alpha}} \delta_{\beta\alpha} + \frac{\mu}{\rho} \left( \frac{\partial u_{\beta}}{\partial x_{\alpha}} + \frac{\partial u_{\alpha}}{\partial x_{\beta}} \right)
\]

Where \(\mu\) is the sum of the laminar and the turbulent viscosities. Steady state solutions of this equation set are sought in a spatial computational domain \(\Omega\).

1.2. Numerical Simulation

To build the numerical model, first the CATIA (V5R18) is used to create a CAD model and then STAR CCM+ is used to generate the mesh and solve the system of equations described above. For the mesh topology, polyhedral and hexahedral elements are tested; both give satisfactory solution in terms of accuracy with a better rate of convergence when using hexahedral elements with five prismatic layers to capture turbulent phenomena. Therefore the latest type, hexahedral elements, is selected and used in all simulations. To capture turbulence phenomena, five prismatic layers are generated ensuring that dimensionless distances to the wall, \(y^+\), remain within the range 30 and 50. Several turbulent models are tested as well, including \(K-\varepsilon\) realizable, \(K-\omega\) and \(K-\omega\) SST models along with the use of wall functions. Similar results are obtained. The discrepancy on drag is no more than 3% as shown in Fig 4.

A volume mesh refinement near the wheel has been added in the present simulations as shown in Fig 5. To estimate the necessary mesh resolution for a desired accuracy, the number of elements is plotted against the total drag force, in Fig 6. The latter figure shows that over roughly 9 million elements there is no noticeable change in drag. Consequently, this mesh size provides a good balance between accuracy and processing time. Finally, note that the rotating wheel is simulated by imposing a tangential velocity as a boundary condition. In fact a good accuracy is obtained between experimental and CFD results with frozen rotor and tangential velocity boundary conditions, as shown in Table 1. But the frozen rotor model approximately needs twice as long solving time as the tangential velocity boundary condition. Note, that all our simulations are stopped after the residual dropped by four orders of magnitude.

![Fig 4: Drag value for experimental results and different turbulent models](image)

![Fig 5: Wheel mesh section view](image)
3. VALIDATION

The numerical model described above was validated against experimental results before going to the aerodynamic optimization of the wheel subsystem with drag reduction as a target. The wheel subsystem experimental achieved in section 2, is simulated numerically at the same velocities considered experimentally 20m/s and 27.7m/s. Moreover the above wind tunnel test is simulated numerically. The $K_\varepsilon$ turbulent model is retained, because it produces a reasonable prediction with a good convergence rate. Table 1 shows a summary of the CFD results compared to experimental results. It can be seen that in all cases the error doesn’t exceed 3.5%, which demonstrates the reliability of the numerical model and then justifies its use for the wheel subsystem optimization.

<table>
<thead>
<tr>
<th>Flow speed (m/s)</th>
<th>Experimental results</th>
<th>CFD results</th>
<th>$C_dA$ Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$C_dA$ (m²)</td>
<td>Boundary condition</td>
<td>$C_dA$ (m²)</td>
</tr>
<tr>
<td>20</td>
<td>0.15</td>
<td>Tangential velocity</td>
<td>0.157</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Rotated mesh</td>
<td>0.153</td>
</tr>
<tr>
<td>27.7</td>
<td>0.16</td>
<td>Tangential velocity</td>
<td>0.156</td>
</tr>
</tbody>
</table>

Table 1: Numerical and wind tunnel results comparison.

The pressure coefficient distribution on the lower and upper surface of the stock fender measured using the 5 pressure transducers and calculated numerically with CCM+ is shown in Fig 7. As it can be seen, the experimental measurements have a good correlation with the numerical model for four points in both sides. Another important remark from Fig 7 is the two points (one in each side of the fender) located in the reticulating areas that are showing less accuracy.

4. DRAG OPTIMIZATION

The CFD model is applied to the wheel sub-system drag optimization by designing new A-Arms geometries and comparing the new drag force to the reference one given by the stock version. The pie-chart (Fig 8) provides the drag contribution of each part in the subsystem. It clearly points out that the fender is the most important contributor to the total drag followed by the A-Arms.

For the lower side of the fender, recirculations are located at the upper front and for the upper side; they are located at the lower front in back of wheel. The gap between experimental and numerical results is probably due to the turbulent model $K_\varepsilon$ used and the steady condition chosen; in fact the flow in these recirculation areas is often unsteady.
An efficient methodology to reduce drag on tricycle-vehicle through an aerodynamic optimization of the wheel fender is presented in [8]. Using he boat tailing optimisation technique, they proposed a new fender shape that reduces the wheel subsystem drag by 30.6% comparing to the stock version. Vortices and recirculation zone are reduced at the back of the wheel and between the subsystem parts. These improvements have an impact on reducing losses and consequently drag as demonstrated by the numerical simulations. Even with the optimized fender proposed in [8], the flow still shows strong vortices (Fig 9) due to the subsystem parts interaction.

**Fig 9:** Tangential velocity vector along longitudinal section x=-600mm. (a) Stock version, (b) Optimized fender, [8]

Flow analysis around the A-arms, see Fig. 10, shows a high pressure distribution on the deflectors, whose have been added to canalize and direct the flow towards the radiators.

**Figure 10:** Cp profile of the wheel subsystem, stock version

It has been showed in [9] that these deflectors are not efficient. This was observed in our wind-tunnel tests as well, for such a purpose wool wire is put down on the A-arms covers. Fig. 9 shows that the deflectors generate strong vertices. In order to minimize the contribution of the A-arms to the total drag, two types of A-arm shapes are designed.

**Fig 11:** A-Arms geometry a) Double-Wings, b) Single-wing
The first one is proposed by [9] and referred to by double-wing profile. It is obtained by covering the arms tubes to obtain an aerodynamic profile as shown in Fig 11. The second shape is proposed here and obtained by covering the whole arms with a single profile (Fig 11b); it is referred to by single-wing. A numerical simulation shows a significant drag reduction (Fig 12) with a better performance (19.8% of drag reduction) when the single-wing design is used. Moreover, vortices generated by the deflectors are completely eliminated (Fig 13).

Finally the effect of the optimized fender and A-Arms combined is simulated, the results shows an important drag reduction of 54.6% (Fig 12) and the elimination of almost vortices appearing in the stock version as shown in Fig 13.

5. CONCLUSION

This paper has proposed an efficient process to reduce drag on tricycle vehicle through an aerodynamic optimization of the wheel A-Arms. Numerical simulations of the flow past the system have been performed using the commercial Navier-Stokes solver Star CCM. Wind tunnel tests were achieved to validate the model. A new A-Arms shape is proposed in the present study that reduces the wheel subsystem drag by 19.8% compared to the subsystem stock version.

References
7. BIPM, 1981, "Formule Pour La Détermination De La Masse Volumique De l'Air Humide."