

Numerical and Experimental Analysis of Unsteady Separated Flow behind an Oscillating Car Model

Emmanuel Guilmineau

Ecole Centrale de Nantes

Francis Chometon

Conservatoire National des Arts et Métiers

Copyright © 2008 SAE International

ABSTRACT

The objective of this research is to improve the understanding of unsteady three-dimensional separated flows on bluff bodies. This research focuses on the analysis of the instability and safe maneuverability of passenger vehicles associated with unsteady atmospheric crosswind conditions. Experimental and numerical analysis were performed on the squareback Willy car model animated by an oscillating yaw angle in a steady wind. Experiments were carried out at a Reynolds number of $0.9 \cdot 10^6$, a frequency of 2 Hz, with an amplitude of $\Delta\beta = 10^\circ$ at the Conservatoire National des Arts et Métiers (CNAM). Unsteady wall pressures were obtained with sensors located inside the model. Computations were performed at the Ecole Centrale de Nantes (ECN). The ISIS-CFD flow solver, developed by the CFD Department of ECN, used the incompressible unsteady Reynolds-Averaged Navier-Stokes equations. Experimental and numerical comparisons are completed with numerical simulations at a larger yaw angle. The analysis of the physics of the unsteady flow around the model is done through the calculation of aerodynamic forces and moments, and cross flow velocities.

INTRODUCTION

Aerodynamics of road vehicles gather together a lot of specific problems due to the fact that an automotive is a bluff body of low aspect ratio with rotating wheels, open wheelhouses and which moreover is moving close to the ground. Except for fuel consumption or pollutant emission or non technical constraints like fashion or regulations, it remains that the main role of a road vehicle is to transport passengers and objects with the maximum of security. For example, optimization of the front part of the vehicle regarding to aerodynamics and aeroacoustics must not reduce the driver's visibility. On modern cars, separations on fore bodies are limited but base separations are inevitable and can be the cause of instability during passenger car overtaking.

On the road, the aerodynamic yaw angle increases with stronger wind conditions such as gusty cross winds caused by unsteady atmospheric conditions, tunnels, bridges or trucks passing. Separations appear on the leeward side of the body, interact with the base flow and cause unsteady vehicle loading. These aerodynamic elements, added to dynamic properties of the suspension, springing and tires are finally the factors in the vehicle's on-road aeroacoustics quality, of its dynamic stability and of the safe manoeuvrability appreciated by the driver, Hucho, [11].

One answer to this problem is to develop active control systems which integrate driver intents or faults, vehicle status measurement, and road bank detection, Tseng *et al*, [24]. Tires force distribution obtained by measuring in each direction forces between road surface and tires in real time can be integrated in order to enhance vehicle dynamics, Akazawa, [1]. In the same way, applications of vision-based vehicle detection where cameras are mounted on the vehicle seem a promising approach for driver assistance, Sun *et al*, [22]. At last, Rodic *et al*, [19], propose simulations based on a nonlinear model with 22 degrees of freedom ensuring simultaneous stability of the vehicle in longitudinal, lateral and vertical directions, roll, pitch, and yaw with perturbation of type of an uneven and slippery road, inadequately adapted ride velocity as well as wind gusts.

Nevertheless, the knowledge of the aerodynamic part of the dynamic stability of a full scale vehicle submitted to side gust of wind, passing or crossing is necessary. This task remains complex and many approaches use scale models. Macklin *et al*, [16], propose to propel the model on a rail crossing the test section of the wind tunnel. Ryan *et al* [20], used a technique where the side wind was produced by a cross jet. The approach retained here was proposed by Garry and Cooper [6], and consists of submitting the model to a periodic movement in a steady wind. This approach does not actually simulate the situation of passing but it does bring into evidence the phenomena of phase shifting or hysteresis and allows their analysis. More complicated problems like vehicle passing or crossing were correctly analyzed

on 1/5 scale models propelled on a rail by Noger *et al*, [18]. Experiments and analysis on models use commonly unsteady pressure and force measurements as well as unsteady PIV and LDV, Ferrand *et al*, [5], Gohlke *et al* [8], Sanquer *et al* [21], Tsuei and Savas, [24].

To date, accuracy of numerical simulation has been improved and simulation is now widely used in the automotive industry for exploration of flow physics or for effort prediction on full scale vehicles. Clarke and Filippone, [3], propose a full unsteady simulation of two vehicle overtaking each other at highways conditions. Structured meshes with sliding planes were used. As far as we know, many of the CFD tests are done at zero yaw angle for steady flows and use simplified bluff bodies, Khalighi *et al*, [14] or the reference Ahmed car model, Kapadia *et al*, [13], Krajnovic *et al*, [15]. Experimental and numerical data presented in this paper are concerned with the analysis of the unsteady flow around the Willy model, Guilmineau, [9], Guilmineau and Chometon [10]. The numerical tests are performed on the model animated by an oscillating yaw angle of amplitude $\Delta\beta = 10^\circ$ and $\Delta\beta = 30^\circ$, and a frequency $f = 2$ Hz in a steady wind at a Reynolds number of 0.9×10^6 . The experimental unsteady wall pressures obtained at $\Delta\beta = 10^\circ$ for a frequency $f = 2$ Hz are compared to the numerical results. The post-processing of other numerical data allows the understanding of the physics of phase shifting phenomena observed on wall pressures and aerodynamic forces.

THE WILLY TEST MODEL

Experimental and numerical tests are performed on the squareback Willy test model, which is realistic, compared to a van-type vehicle. A complete definition and the digital definition of the model through only 4 parameters are given in [2]. The overall length of the model is $L1 = 675$ mm, the width 240 mm, the maximum height 192 mm and its surface reference is the maximum cross section $S_{ref} = 41791$ mm². The ground clearance is $G = 29$ mm and the diameter of the four feet (f) which are used to secure the model to the floor of the wind tunnel is $\Phi1 = 20$ mm. All other dimensions are defined in the Table 1. The reference axis of the wind tunnel is the Eiffel axis where the axis Xo is parallel to the upstream velocity Vo . The origin of the Lilienthal axis X linked to the model lies at the point O located on the floor of the model, see Figures 1 and 2. This point O is the center of rotation of the model when submitted to a harmonic oscillation. The value of the yaw angle β is positive when the right side of the car model is windward.

L1	B	D	E	F	G	H	$\Phi1$	Sref
675	140	550	118	140	29	345	20	41791

Table 1 - Dimensions of the Willy test model (data in mm and mm²)

EXPERIMENTAL TECHNIQUE

Wall pressures measurements were performed in a Prandtl-type wind tunnel of the CNAM. The semi-open test section has a cross section of 1.45 m by 1.45 m and an effective length of 2.00 m. The wind tunnel is equipped with a six components balance. The ground of the wind tunnel is fixed.

The turbulence level at the center of the test section is 2%. For an upstream velocity $Vo = 20$ m/s, the thickness of the turbulent boundary layer at a point located on the floor of the wind tunnel, at $x = -670$ mm, i.e. upstream the body, and $y = 10$ mm is $\delta = 40$ mm with a shape factor $H = 1.29$. This data is obtained with the model at a yaw angle of $\beta = 30^\circ$.

The model is attached to a turntable of diameter $\Phi = 800$ mm inserted in a fixed plate of length 1.90 m and width 1.50 m, see Figure 3.

The experimental results given in the paper are obtained at a Reynolds number of $Re = 0.9 \times 10^6$ based on the velocity Vo and the length $L1$ of the model. For values of the Reynolds numbers from $Re = 0.4 \times 10^6$ to $Re = 1.4 \times 10^6$, steady conditions and $\beta = 0$, it was checked that the drag coefficient of the model does not depend of the Reynolds number. It was assumed that unsteady results only depend on the Strouhal number.

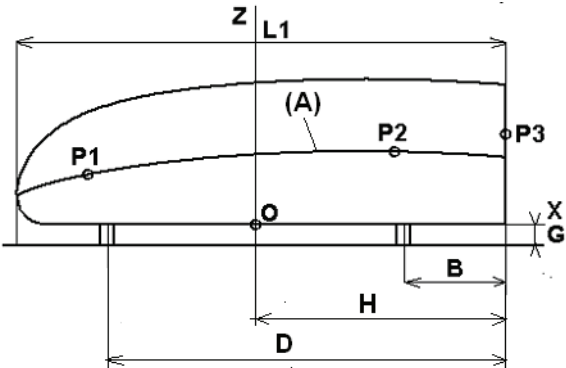


Figure 1 : Sideview of the model and dimensions

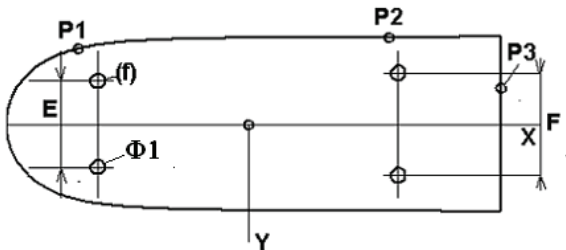


Figure 2 : Bottom view of the model and dimensions

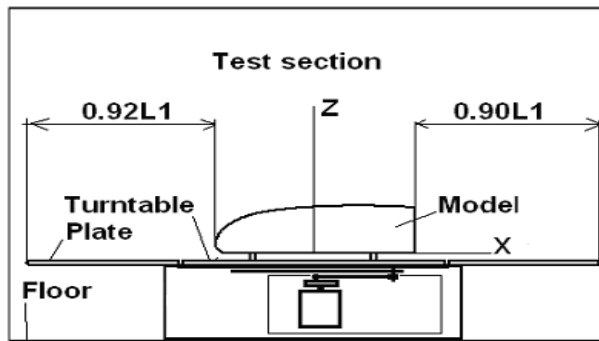


Figure 3 : Side view of the wind tunnel and model

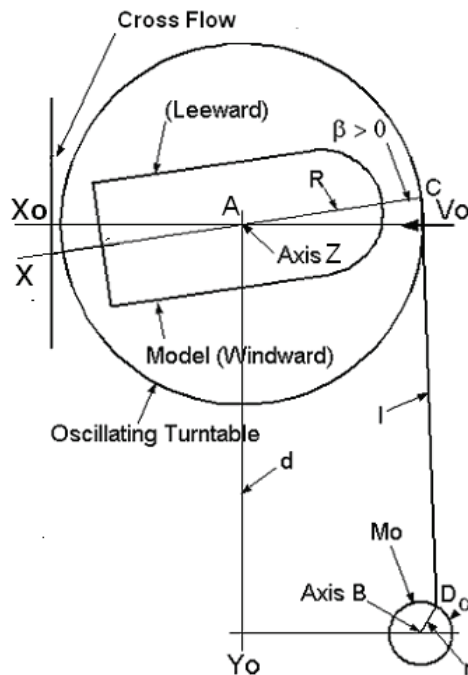


Figure 4 : Experimental set up and turnable

The set-up allowing the model to be put into movement at the frequency f is described in Figure 4. The turntable oscillates around its Z axis. A rod (CD) of length l is attached to the disc of a step motor (Mo). The set-up is conceived so that the rod (CD) is parallel to axis Y_o when $\beta = \Delta\beta$, where $\Delta\beta$ is the amplitude of the oscillation. In these conditions, $r = R \sin \Delta\beta$, and $l = d$. With $r = 0.0255$ m and $l = d = 0.3264$ m, the movement $\beta = f(\alpha)$ is close to the sine function. For an angle $\alpha = \omega t$, where ω is a constant angular velocity and t the time, the actual value of $\beta = f(\alpha)$ is the physical root of:

$$a \tan^2\left(\frac{\beta}{2}\right) + b \tan\left(\frac{\beta}{2}\right) + c = 0 \quad (1)$$

where,

$$\begin{aligned} a &= A + B \left(\cos \Delta \beta + \frac{\cos \alpha}{\sin \Delta \beta} \right) \\ b &= B \left(\frac{2 |\sin \Delta \beta|}{r} - 2 \sin \Delta \beta \sin \alpha \right) \\ c &= A - B (\cos \Delta \beta + \sin \Delta \beta \cos \alpha) \end{aligned}$$

$$A = B(1 + 0.5 \sin 2\Delta\beta \cos \alpha) - 2r_l \sin \alpha$$

Wall pressures are measured along the curve (A) of the model, see Figure 1. The internal diameter of the pressure taps is 1.5 mm, see Figure 5. Unsteady pressure coefficients $C_p(t)$ are defined by the ratio:

$$C_p(t) = \frac{p(t) - p_o}{q_o} \quad (2)$$

where p_0 and $p(t)$ are respectively the upstream static pressure and the local unsteady pressure, and q_0 the dynamic pressure $\frac{1}{2}\rho V_0^2$ where ρ is the fluid density.

Pressure measurements are performed with dynamic sensors Honeywell DC010BDC4 with pressure range ± 10 mb and combined linearity and hysteresis less than ± 0.25 %. Sensors are mounted inside the model and the electric wires pass through the rear-left foot (f). Port 1 of each differential pressure sensor is connected to each pressure tap, see Figure 5, and the ports 2 of all sensors are connected to the common reference pressure p_o .

The frequency response of this device was tested in a shock tube which was modified in order to provide weak shock waves suitable with the pressure range of the sensors. The natural frequency of the actual measurement device is 520 Hz. The bandwidth frequency at which the error is 5% is about 30 Hz.

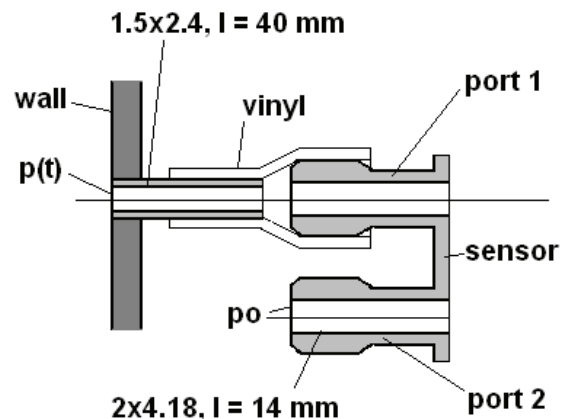


Figure 5 : Unsteady wall pressure measurement

For example, the time history of the rough experimental pressure coefficient measured on the wall of the model at the location P2, (see figure 1), is given in figure 6. These data which are obtained by dividing the unsteady local pressure by the reference pressure (see equation (2)) is then smoothed with the Savitzky-Golay filter based on a least-squares smoothing technique. The evolution of the pressure coefficient versus the yaw angle β is obtained by processing this result between the points A and B of the synchronizing signal (see figure 6). The phase shift introduced by the mechanisms

and by the control of the angle α of the motor (Mo) is calculated on quasi-steady reference results obtained at a the low frequency $f = 0.1$ Hz. It is admitted that this angle adjustment is independent of the frequency and applied to all tests.

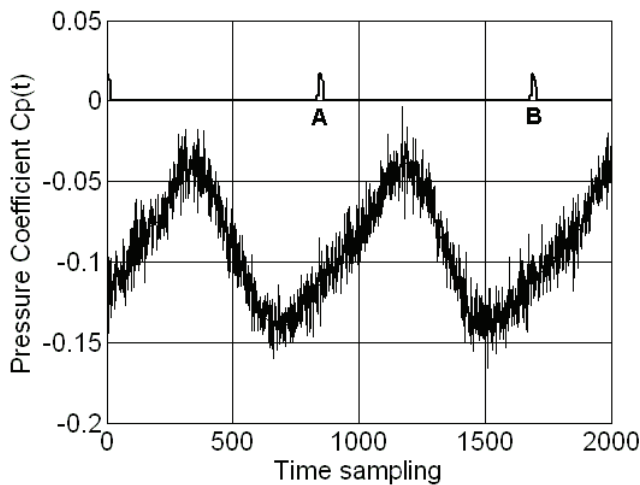


Figure 6 : Time history of the pressure coefficient at point P2

NUMERICAL APPROACH

FLOW SOLVER

The ISIS-CFD flow solver, developed by the EMN (Equipe Modélisation Numérique) of the Fluid Mechanics Laboratory of the Ecole Centrale of Nantes, uses the incompressible unsteady Reynolds-averaged Navier-Stokes equations (RANSE). The solver is based on the finite volume method to build a spatial discretization of the transport equations. The face-based method is generalized to two-dimensional or three dimensional unstructured meshes for which non-overlapping control volumes are bounded by an arbitrary number of constitutive faces. The velocity field is obtained from the momentum conservation equations and the pressure field is extracted from the mass conservation constraint, or continuity equation, transformed into a pressure-equation. A second-order accurate three-level fully implicit time discretization is used. Surface and volume integrals are evaluated using second-order accurate approximation, Jasak *et al.*, [12]. In the case of turbulent flows, additional transport equations for modeled variables are solved in a form similar to that of the momentum equations. They can be discretized and solved using the same principles. One of the turbulence models used is the well-know $k-\omega$ SST model of Menter [17]. It belongs to the category of linear eddy-viscosity model which is widely used in industrial applications. The second model selected in the present study is an Explicit Algebraic Stress Model (EASM). The EASM model is an explicit solution of the Algebraic Stress Model (ASM). We used the EASM model proposed by Gatski and Speziale [7] based on a three-term basis which is an approximation of the ASM model.

We solve the $k-\omega$ BSL model proposed by Menter [17] to determine k and ϵ . Contribution of the non-linear part of the Reynolds stress is not taken account in the production terms in the k and ω equations. Compared to the original BSL model, the $k-\omega$ model used for the EASM model differs only in the computation of the eddy-viscosity. All details are published by Deng *et al.* [4].

To take account of the oscillation of the model, all mesh points move with the same velocity than the one of the car model. In this case, the incompressible Unsteady Reynolds-Averaged Navier-Stokes (URANS) equations can be written:

$$\frac{\partial}{\partial t} \int_V \rho dV + \int_S \rho (\vec{U} - \vec{U}_d) \cdot \vec{n} dS = 0 \quad (3)$$

$$\frac{\partial}{\partial t} \int_V \rho U_i dV + \int_S \rho U_i (\vec{U} - \vec{U}_d) \cdot \vec{n} dS = \int_S (\tau_{ij} I_j - p I_i) \cdot \vec{n} dS$$

where V is the control volume bounded by a closed surface S moving at a velocity $\vec{U}_d$ with a unit outward normal vector $\vec{n}$. $\vec{U}$ and p represent respectively the velocity and pressure fields. τ_{ij} are the components of the viscous stress tensor whereas I_j is the identity vector whose components vanished, except for the component j which is equal to unity.

However, this approach does not represent the experimental set-up. In experiments, the model is attached to a turntable, and in numerics, the turntable is fixed. We suppose that this approach in numerics does not induce some secondary effects.

MESH

The computational domain starts $3.5 \times L_1$ in front of the model and extends $4 \times L_1$ behind the model. The width of the domain extends from $+1000$ mm ($1.48 \times L_1$) to -1000 mm and the height is 1050 mm ($1.55 \times L_1$). The

computational domain is presented in Figure 7. The mesh is generated by the unstructured mesh generator Hexpress. This software generates meshes only containing hexaedrals. The mesh is composed of about 6.3 millions of points with approximatively 93000 points on the model and 48100 points on the wind tunnel floor. The boundary conditions are shown in Figure 7. For the model, we use the near-wall low-Reynolds number turbulence model, the distance between the body and the first fluid points is fixed to 0.006 mm, i.e. $y^+ = 0.5$. For the wind tunnel floor, we use a wall function, and the distance between the floor and the first fluid points is fixed to 0.6 mm. At the outlet, the pressure is prescribed while for the other boundaries we use a Dirichlet condition. On these boundaries, we impose the