AERODYNAMIC DRAG AND WAYS TO REDUCE IT

T. Ivanić*, P. Gilliéron*
*Technocentre Renault, Direction de la Recherche, Groupe "Mécanique des Fluides & Aérodynamique", TCR AVA 058, 1 Avenue du Golf, France

Key Words : Aerodynamics / Automobile / Analytical approach

Introduction

Reduction of vehicle fuel consumption and aerodynamic noise associated with a wide variety of complex physical phenomena observed in the vehicle nearfield has recently become a strategic avenue of research. Directives from the European Union requiring a reduction in fuel consumption and CO_2 emissions (CO_2 *140g* per kilometer in *2008* against *164g* in *2003*) can only be achieved through significant technical and scientific progress. In this context, automobile aerodynamics plays a crucial role, considering that, for example, a drag reduction of *-20%* is equivalent to *a* CO_2 reduction of *-7* grams per kilometer.

Drag is the most important aerodynamic force exerted on passenger cars at normal highway speeds. Moreover, the factor which contributes most to aerodynamic drag is the separation zone at the rear of the vehicle. Fluid mechanics equations, applied to a moving vehicle, show that a reduction in aerodynamic drag requires a local flow modification, to eliminate or push the flow separation towards the base or reduce the development of the vortices in the vehicle wake. In the other words, the object is to control the flow locally in the vicinity of the separation points and thereby achieve an important drag reduction. This control can be ensured with (or without) energy input by means of active (or passive) devices. In this paper, the various parameters having a significant influence on aerodynamic drag will be identified and investigated according to analytical approaches based on the integral form of the momentum and energy equations.

1. Theoretical expression of the total drag

Although the velocity of an automobile is much lower than that of an aircraft, the fully turbulent flow around a car is just as complex – if not more so. In effect, the small relative length of the car (compared to that of a plane), the proximity of the ground layer and of other ancillary items all combine to induce the formation of interacting swirling structures, which renders the analysis of the flow structure around a motor vehicle a very complex process.

To obtain a better understanding of various physical phenomena (and their interaction), it is necessary to find simplified approaches conducive to identifying which parameters play a significant role in achieving a reduction in aerodynamic drag. The approach developed here is based on an integral form of the momentum equation, according to Onorato [1]. With such an analytical model, it is possible to simplify the analysis of the automobile aerodynamic drag and identify the phenomena which contribute most to its formation.

1.1. Theoretical Bases

At normal highway speeds, the Mach number of a vehicle is less than *M=0.3,* hence the compressible effects are negligible. Likewise, the air gravity forces exerted on the car are relatively weak compared to the viscous, turbulence and pressure forces. For these reasons, under normal driving conditions, the flow around a vehicle can be considered as incompressible and the gravity effect negligible.

Let us consider the closed surface S_C (representing a vehicle) and a fluid volume ϑ, representing a stream tube delimited by closed surface $\Sigma_e = S_1 + S_2 + S_L + S_S$ (Figure 1) where:

- S_1 is the inlet surface placed at a sufficient distance from the obstacle to consider flow as undisturbed and uniform, which permits the following assumptions: $\vec{V} = \vec{V}_O$ and $P=P_0$ at any point of this surface ($\vec{n}$ is the exterior normal).
- S_2 is the outlet surface and $\vec{n} = \vec{x}$ on S_2.
- S_L is the side surface composed of a stream lines. S_1 and S_2 are sufficiently large to consider that pressure acting on the lateral surfaces is constant, and $P=P_0$ and $\vec{V} = \vec{V}_O$ on S_L.
- S_S is the ground surface and $\vec{n} = -\vec{z}$ on S_S.

Let $O(\vec{x},\vec{y},\vec{z})$ denote the cartesian coordinate system in which the inlet velocity can be written as : $\vec{V}_o = V_o\vec{x}$.

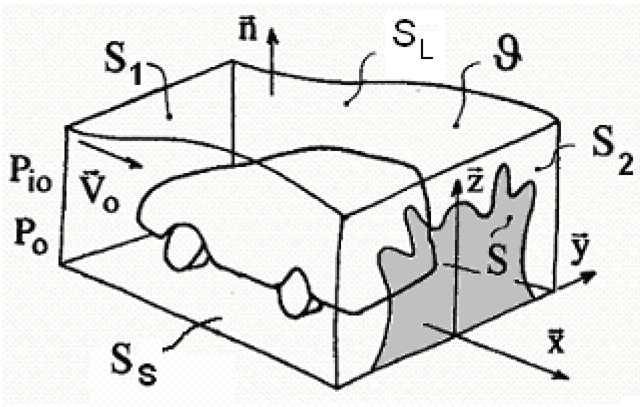

Figure 1 Stream tube of volume ϑ, Gilliéron [2]

Let us also consider the momentum equation applied to the air inside the stream tube delimited by the surface $\Sigma = \Sigma_e + S_C$ (S_C is the vehicle external surface):

$$\iiint_{\vartheta} \frac{\partial \vec{V}}{\partial t} d\vartheta + \iint_{\Sigma} \vec{V}(\vec{V}\vec{n})d\sigma = -\frac{1}{\rho}\iint_{\Sigma} P\vec{n}d\sigma + \frac{1}{\rho}\iint_{\Sigma} \bar{\bar{\tau}}_{\mu,t}\vec{n}d\sigma + \iiint_{\vartheta} \vec{f}d\vartheta \qquad (1)$$

where $\vec{V}$ is the flow velocity, ρ the fluid density, P the static pressure, $\vec{f}$ the body forces acting on the control volume, and $\bar{\bar{\tau}}_{\mu,t}$ the viscous and turbulence tensor stress.