J. Wind Eng. Ind. Aerodyn. 119 (2013) 146–157

Contents lists available at SciVerse ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

journal homepage: www.elsevier.com/locate/jweia

Interference effects of cooling airflows on a generic car body

Dirk Bäder [a,b,*], Thomas Indinger [a], Nikolaus A. Adams [a], Peter Unterlechner [b], Gerhard Wickern [b]

[a] Technische Universität München, Insitute of Aerodynamics and Fluid Mechanics, Boltzmannstr. 15, 85748 Garching, Germany
[b] Audi AG, Windkanalzentrum, 85045 Ingolstadt, Germany

ARTICLE INFO

Article history:
Received 19 August 2011
Received in revised form
11 April 2013
Accepted 1 May 2013
Available online 17 July 2013

Keywords:
Bluff body
Under-hood
Model radiator
Experiments
Numerical simulation

ABSTRACT

A significant contribution to the aerodynamics of road vehicles is caused by the cooling airflow. The cooling air flows through the engine compartment. Internal losses occur through the radiator and through the engine compartment. Additionally interference effects occur between the flow through the engine compartment and the flow field around the car. For most cars, all these effects result in additional aerodynamic drag, which is caused by the cooling airflow.

In this study, numerical and experimental investigations were performed to understand the effects of cooling-air flow on the vehicle's aerodynamics. A generic car-model was equipped with a simplified underhood section. The cooling air mass flow could be varied to understand the cooling air mass flow on the cooling drag. A variety of measurements and numerical simulations were performed to explain the phenomenona of cooling air flows. Total pressure and hot-wire measurements in the wake of the car visualise the flow field, which is altered by the cooling air. Force measurements, surface pressure measurements and oil-streak patterns were performed to understand the influence of the cooling air on the bluff body. With the measurements and with the CFD-simulations, this study will explain where the cooling drag originates from and how the cooling air interferes with the external aerodynamics.

1. Motivation and overview

Nowadays, the aerodynamics of vehicles have improved greatly. State-of-the-art automobiles exhibit a well-designed exterior with an intelligent flow interaction. Besides the inherently bluff-body shape of vehicles, complexity arises especially from wheel and wheel housing flows as well as from the under-hood flow with its interference effects on the inlets and the outlets of the cooling engine bay with the external aerodynamics. Aerodynamically relevant parameters of cars, especially the drag, but also the lift, can be improved further. Therefore it is of interest to explore the interaction of the flow field around a car to obtain an impression of how this flow is altered due to the under-hood flow for example.

Different authors have tried to understand and to explain the cooling-drag by the one-dimensional momentum equation. Soja and Wiedemann (1987) explain the influence of the cooling-air outlet-angle on the cooling-drag. Williams (2003) tries to quantify additionally the interference effects of the cooling air with the external flow at the inlet and outlet of the cooling duct. For the quantification, Williams (2003) uses semi-empirical formulas, where he adds terms for the interference at the inlet and at the outlet to the momentum equation. Barnard et al. (2004) state with the momentum theory that the variation of the cooling air exit is promising to reduce the cooling drag. Beside the theoretical quantifications, Barnard et al. (2004) also show the influence of the cooling air outlet-angle in their experiments.

Beside those one-dimensional examinations and explanations of the cooling drag, three-dimensional investigations were performed using experiments and numerical simulations. Jehle (2001) calculates the influence of different cooling air inlet positions and validates the CFD-simulations with experiments. He also tests empirically different cooling air outlet positions and states that an outlet position in front of the front wheels shows advantages regarding cooling drag. Herbig (2002) shows a relation between cooling drag and cooling air mass flow. Reducing the cooling air mass flow reduces the cooling drag and the efficiency of the engine. Finally he shows that an cooling outlet parallel to the street is optimal. Kuthada (2006) investigates state-of-the-art technical solutions of cars and characterizes the influence of a road simulation (i.e. rotating wheels) on the cooling drag. Tesch et al. (2010) investigate empirically different cooling air outlet positions on a road vehicle.

Most of the authors, who perform three-dimensional investigations, explain the physics empirically. Bäder (2012) and Bäder et al. (2012, 2011) explain the interference effects physically. The most interesting findings are presented in this study. A correlation between cooling air mass flow and cooling drag is presented, also a correlation between cooling air, mass-flow and surface pressure. With this investigation, a new design of cooling air is possible. This new approach is based on both, experiments and numerical

---

* Corresponding author.

E-mail addresses: dirk.baeder@aer.mw.tum.de (D. Bäder), thomas.indinger@tum.de (T. Indinger).

0167-6105/$ - see front matter © 2013 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.jweia.2013.05.009

Nomenclature

A	surface (m^2)
A_{ref}	car frontal surface (0.1198 m^2)
α	angle (deg)
b	vehicle width (m)
C^1	Darcy–Forchheimer tensor (Pa s^2/m kg)
C^2	Darcy–Forchheimer tensor (Pa s^2/kg)
$c_c = \dot{m}_c/(\rho \cdot U \cdot A_{ref})$	cooling mass-flow (–)
$c_d = F/(1/2 \cdot \rho \cdot U^2 \cdot A_{ref})$	drag coefficient (–)
$c_p = \Delta p/(1/2 \cdot \rho \cdot U^2 \cdot A_{ref})$	pressure coefficient (–)
$c_{p_{total}} = \Delta p_{total}/(1/2 \cdot \rho \cdot U^2 \cdot A_{ref})$	total-pressure coefficient (–)
k	turbulent kinetic energy(m^2/s^2)
l	characteristic vehicle length (1.05 m)
μ	dynamic viscosity (N s m^2)
$\dot{m}$	mass flow (kg/s)
$\dot{m}_c$	cooling-air mass-flow (kg/s)
ν	kinematic viscosity (m^2/s)
p	pressure (N/m^2)
ρ	density (kg/m^3)
Re	Reynolds-number of vehicle based on vehicle length (–)
Re_r	Reynolds-number of radiator based on hydraulic diameter (–)
U	absolute value of free-stream velocity (50 m/s)
u,v,w	velocity component (in the x-, y-, z-directions) (m/s)
x,y,z	cartesian coordinate (m)
$\zeta = \Delta p/(1/2 \cdot \rho \cdot U^2)$	dimensionless pressure drop (–)

Subscript

$(\ldots)_c$	cooling
$(\ldots)_r$	radiator
$(\ldots)_{total}$	total

Superscript

$(\ldots)^1$	coefficient 1
$(\ldots)^2$	coefficient 2

simulations. It provides a broad foundation and enables to understand the characteristics of cooling drag.

In this study, a systematic investigation of different vehicle parameters was performed regarding the influence of the cooling mass flow. Therefore a simplified generic bluff-body, the so-called SAE body, defined in the SAE-Report (SAE, 1997) and in Cogotti's study, was chosen to investigate the basic fluid dynamic effects. The SAE body was modified by adding a cooling duct of different outlet angles, different pressure losses in a model radiator and the possibility of adding wheels, see Figs. 1 and 10 and Chapter 2.1. The quarter-scale model was built similar to Kuthada's SAE body Type K (Kuthada, 2006; Kuthada et al., 2004) which allows comparisons. This study focuses on the flow field on, around and behind the body to obtain an impression of how the cooling drag is generated. Once it is understood where the drag originates from the development of an improved cooling system is eased. Last but not least the measurements were used and in future will be used as a validation basis for numerical simulations, for example in Bäder et al. (2011).

2. Wind-tunnel model

2.1. Modified geometry of SAE body

The quarter-scale SAE Body (SAE, 1997) was modified by a simplified under-hood flow including a well-designed model

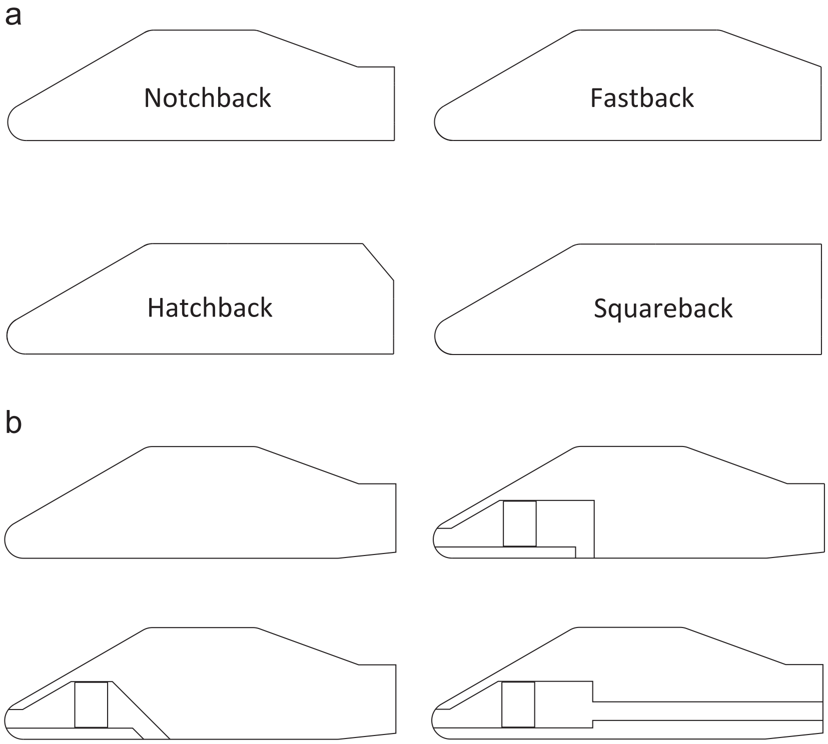

Fig. 1. Geometric modifications of the SAE Body. (a) Different rear end shapes of the SAE body, (b) Modified SAE Body with variable under-hood flow. The position of the model radiator is indicated. Top left: Mock Up, top right: 90°, bottom left: 45°, bottom right: 0°.