

Investigation of Aerodynamic Drag in Turbulent Flow Conditions

Daniel Stoll, Christoph Schoenleber, Felix Wittmeier, Timo Kuthada, and Jochen Wiedemann
FKFS

ABSTRACT

In this paper the influence of different turbulent flow conditions on the aerodynamic drag of a quarter scale model with notchback and estate back rear ends is investigated. FKFS *swing*[®] (Side Wind Generator) is used to generate a turbulent flow field in the test section of the IVK model scale wind tunnel.

In order to investigate the increase in drag with increasing yaw, a steady state yaw sweep is performed for both vehicle models. The shape of the drag curves vary for each vehicle model. The notchback model shows a more pronounced drag minimum at 0° yaw angle and experiences a more severe increase in drag at increasing yaw when compared to the estate back model.

Unsteady time averaged aerodynamic drag values are obtained at two flow situations with different turbulent length scales, turbulence intensities, and yaw angle amplitudes. While the first one is representing light wind, the second one is recreating the presence of strong gusty wind. The results are compared to a quasi-steady averaged aerodynamic drag value derived from steady state yaw sweep measurements in the low turbulence environment of the standard wind tunnel setup. It can be shown that aerodynamic drag at 0° yaw angle is not representative of the drag found under real onroad wind conditions. Furthermore, the results show that a quasi-steady analysis under predicts forces occurring under turbulent flow conditions. Additionally, aerodynamic modifications changing aerodynamic drag at 0° yaw angle are presented, and their potential under turbulent flow conditions is discussed. The method presented here does not only take the presence of yaw under different wind conditions into account, but also contains the unsteadiness inherent in the flow found on the road.

CITATION: Stoll, D., Schoenleber, C., Wittmeier, F., Kuthada, T. et al., "Investigation of Aerodynamic Drag in Turbulent Flow Conditions," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(2):2016, doi:10.4271/2016-01-1605.

INTRODUCTION

The determination of fuel consumption and CO₂ emissions under real-world driving conditions is of ongoing interest for car manufacturers. Improving the overall vehicle efficiency by reducing the overall resistance to motion is the primary goal in the development process of a vehicle. Aerodynamic drag is a main contributor to the overall vehicle resistance especially at higher driving speeds.

The aerodynamic properties of a vehicle are usually determined in a wind tunnel which simulates a steady state low turbulence environment. Realistic ground simulation techniques allow a close representation of the flow situation a vehicle encounters when travelling through still air. However, travelling through still air is a rather rare experience. The flow situation on the road differs considerably from that being simulated in the wind tunnel. Gusty wind, leading vehicles or road side obstacles can cause unsteady effects which are not present in the experiment described above. Thus, the drag coefficient determined in the commonly used low turbulence environment of today's wind tunnels is not representative for the resistance a vehicle experiences on the road.

In order to describe the wind conditions a vehicle is exposed to on the road, many investigations have been done in recent years. While there has been a lot of data collected from wind engineers and meteorologists focusing on the atmospheric boundary layer, little has been known about flow structures below a height of 2 m. An idealised mathematical model to predict the turbulence conditions experienced by a moving vehicle using atmospheric turbulence data from stationary sites was derived by Cooper [1]. However, locally existing conditions, like road side obstacles and the presence of other vehicles, cannot be described by extrapolated data collected from heights above 10 m. Also the presence of the ground will influence the flow structures. Vertical components will asymptotically decay at the ground. The turbulent energy from the vertical component is directed horizontally, resulting in the intensities increasing in the lateral and longitudinal directions. Due to the complex flow structures, only measurements with a vehicle moving through the variety of conditions can allow for a realistic determination of all properties. Different authors have conducted on-road measurements using vehicles equipped with measurement instrumentation in order to obtain the resultant approaching flow conditions (e.g. [2, 3, 4, 5, 6]). These investigations help to understand the size of turbulent length scales, the magnitude of velocity fluctuations and their distribution over frequency. Analysis of the data shows how both the turbulence

intensities and turbulent length scales are modified by terrain type, road side obstacles and the upstream wakes of other moving vehicles. The presence of natural stochastic crosswind contains turbulent length scales that are the size of the vehicle or larger and can thus cause unsteady loads that alter the unsteady response of the vehicle. These wind conditions have to be considered if a vehicle's performance under on-road wind conditions is to be assessed.

The effectiveness of different aerodynamic devices on the aerodynamic drag of trucks at different yaw angles was studied by Cooper [7] and Leuschen [8]. Windsor [9] investigated the drag rise with increasing yaw of different vehicle shapes. The results show that the drag rise with yaw varies considerably, even for cars of similar size and shape. There are cars with a low drag coefficient at 0° yaw angle that have a significant increase of drag under increasing yaw. Shape effects influencing the drag rise with yaw have been investigated by Carr [10, 11] and Howell [12]. It was shown that the use of aerodynamic devices, such as bug spoilers or airdams, cause an increase of drag at rising yaw angle. Similar effects are noted when the cooling air inlets are closed, and the same was found for a smooth underbody. The influence of more general shape effects, such as different rear end shapes, wind screen angles, rear slant angles and backlight angles, was investigated. It has been noted that for many shape configurations, there is a conflict between the drag reduction at 0° yaw angle and the drag rise with yaw.

In order to assess the drag rise with increasing yaw angle, and to allow for a quantification of the effectiveness of drag reducing devices, wind averaged drag coefficients have been introduced. The three commonly known methods are: MIRA [13], SAE J1252 [14] and TRRL Report 392 [15]. A detailed description of the three methods is given by Windsor [16]. These methods are very similar but use different approaches of weighting the results from steady state yaw sweep measurements in a low turbulence wind tunnel environment. Weighting factors for wind direction and velocity are derived from wind statistics data usually measured at heights of 10 m. However, there is no universal approach and each has its own restrictions. This is due to the large range of possible wind speeds and directions experienced on the road. Furthermore, these approaches are based on steady state measurements. Unsteady effects are neglected.

A theoretical method, taking into account the impact of unsteady aerodynamic loads on aerodynamic drag, has been introduced by Howell [16, 17]. The unsteady loads acting on the vehicle under the influence of natural wind are derived using a power spectral density approach. The wind energy spectra are represented in the von Kármán form, isotropic turbulence is considered. The investigations indicate that a quasi-steady analysis under predicts aerodynamic drag.

Coastdown tests carried out to measure aerodynamic drag in windy conditions on the road by Hamabe [18] and Buckley [19], suggest that the drag benefits from aerodynamic aids are less than those found using conventional wind tunnel tests. However, on-road measurements are not suitable in an early development stage since a prototype in running condition is needed.

The wind tunnel seems to be the more feasible development tool. Different methods for producing an unsteady flow field in the wind tunnel test section can be found in the literature. These methods include active and passive systems.

Passive systems, mostly in the form of grids or spires placed upstream of the vehicle, produce a homogenous turbulence distribution. The turbulent length scales generated are usually much smaller than the size of the vehicle [20, 21]. The effect of grid generated turbulence on drag reducing devices for trucks has been studied by Cooper [22] and Watkins [23]. It was shown that the performance of drag reducing devices is influenced by turbulence. On-road test results were compared to those measured with scale models in a wind tunnel under various levels of grid generated turbulence. It was found that the results from the smooth wind tunnel flow over predict the drag reductions of aerodynamic devices when compared to those in a turbulent flow with high yaw angles. A detailed summary of the effects of grid generated turbulence on the aerodynamic characteristics of car-like shapes has been given by Howell [20]. The findings indicate that aerodynamic drag for squareback shapes usually increases when compared to measurements in smooth flow. The benefits from aerodynamic devices are almost always reduced. The findings for hatchback shapes are more complex due to the interaction of the C-Pillar vortex and the wake of the vehicle. It was summarized that differences of drag in turbulent flow conditions are dependent on the vehicle's shape.

Active turbulence generation systems are required to achieve length scales that are the size of the vehicle or larger. Lift-based devices, usually in the form of airfoils placed upstream of the vehicle, allow the generation of the desired length scales [24, 25, 26, 27]. The influence of different turbulent flow conditions was investigated with the turbulence generation system (TGS) in the Pininfarina wind tunnel by Cogotti [28]. Large scale vortices were generated; results for a wagon type vehicle show an increase in drag due to the presence of turbulence. Kremheller [29] investigated a 30% van model in transient flow conditions using the turbulence generation system in the Durham University wind tunnel. Different transient yaw profiles were generated. It is shown that instantaneous peak values of aerodynamic drag can overshoot the steady state results by up to 45%. Time averaged results from the different transient yaw measurements show lower drag values when compared to the aerodynamic drag at 0° yaw angle in the smooth flow environment of the wind tunnel.

CFD Simulations using turbulent boundary conditions have been carried out to investigate aerodynamic drag under certain turbulence and yaw conditions. Simulations of a saloon car at 0° yaw angle with different turbulence intensities were carried out by Gaylard [30]. The results show that aerodynamic drag increases with higher turbulence in the onset flow. It was shown that the increased turbulence causes lower pressures in the wake of the vehicle and thus causes higher drag. The observed drag increase is also attributed to the changed velocity profiles and pressure distributions over different upper body design features. A similar simulation approach was utilized on a sedan model by D'Hooge [31, 32]. Transient yaw profiles and

different turbulence levels have been investigated. It was found that drag values are higher with upstream turbulence, while the vehicle's yaw sensitivity was decreased with increased turbulence intensity.

The variety of investigations found in the literature indicates that the presence of turbulence and yaw found on the road can significantly influence the aerodynamic performance of a vehicle and should thus be considered in the development process. In this paper an approach is presented that does not only consider the presence of yaw under different wind conditions, but also contains the unsteadiness inherent in the flow found on the road.

EXPERIMENTAL SETUP

All measurements were taken at the IVK model scale wind tunnel of Universität Stuttgart. This Göttingen type wind tunnel, i.e. wind tunnel with a closed air circuit and open jet test section, has a nozzle cross-sectional area of 1.65 m². The model can be rotated via the wind tunnel turntable to obtain a stationary angle of attack between vehicle and approach flow. The wind tunnel is equipped with a state of the art 5-belt system, displayed in Figure 1. The vehicle is mounted via the rocker panel struts and the wheels are placed on the balance mounted wheel rotation units. To obtain the unsteady aerodynamic forces and moments acting on the vehicle, a dynamic underfloor balance from A&D Japan, introduced by Oyama et. al [33], was installed in the IVK model scale wind tunnel. The plan view of the wind tunnel test section floor shown in Figure 2 provides an overview of the layout and positioning of the boundary layer control system. Boundary layer suction and tangential blowing are used to generate a block profile of the boundary layer before reaching the moving belt. The center belt runs at wind speed between the vehicle's wheels and simulates the relative motion between the vehicle underbody and the road. The wheels are driven by the wheel rotation units. The use of all described subsystems is referred to as moving ground simulation.

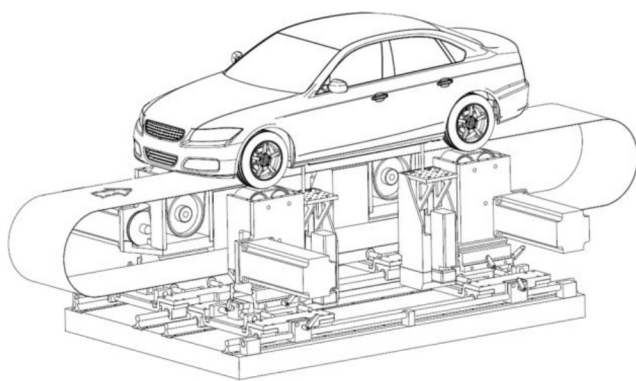


Figure 1. 5-Belt System with quarter scale vehicle model in the IVK model scale wind tunnel, based on [34]

To allow aerodynamic investigations on a realistic generic vehicle geometry, the so called DrivAer model was introduced by Heft et al. [35]. Compared to the often used simplified vehicle shapes, like the Ahmed Body or SAE Reference Body, this model offers a much more detailed representation of a realistic car. In order to investigate the influence of cooling air flow, an open grille geometry has been introduced by Wittmeier et al. [36]. At IVK/FKFS, a quarter scale model with detailed underbody and engine compartment geometry as

well as interchangeable notchback, fastback and estate back rear ends was built. For these investigations the DrivAer model with notchback and estate back was used as shown in Figure 3.

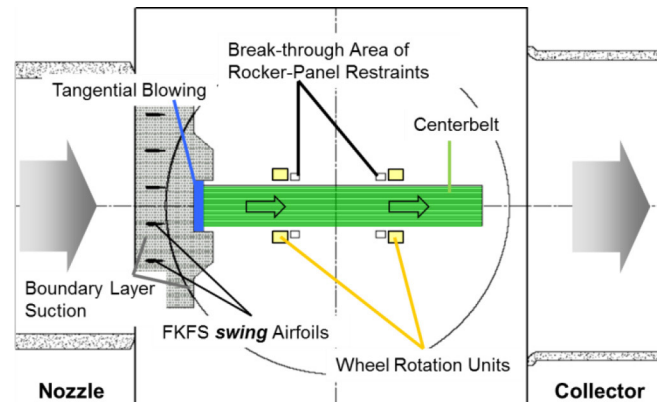


Figure 2. Test section floor in the IVK model scale wind tunnel with boundary layer control system (boundary layer suction, tangential blowing, centerbelt and wheel rotation units)

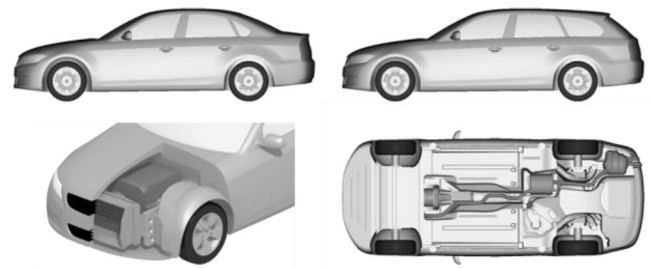


Figure 3. DrivAer model with notchback (top left) and estate back (top right) rear end, engine compartment (bottom left) and underfloor (bottom right)

In order to reproduce the spectral energy content of natural stochastic crosswind within the wind tunnel test section, FKFS *swing*[®] (side wind generator) can be installed at the nozzle exit of the model scale wind tunnel. The system is illustrated in Figure 4.

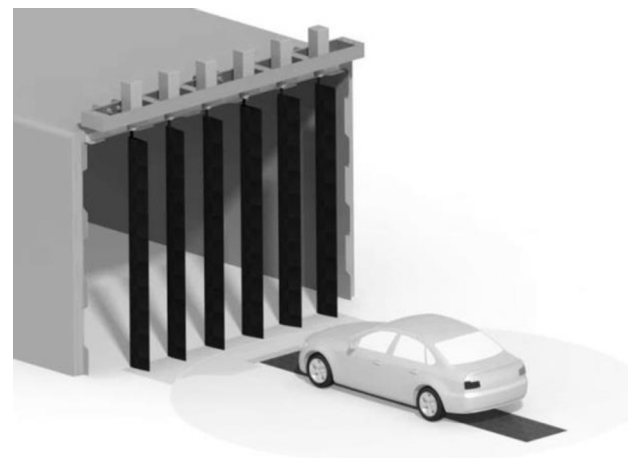


Figure 4. FKFS swing[®] in the model scale wind tunnel

The design is based on the previous system introduced by Schröck [26], but was improved to allow for the generation of higher gust amplitudes and frequencies while increasing the homogeneity of the flow field and minimizing the required drive torque due to aerodynamic and inertial