

The Effect of High Turbulence Intensities on Surface Pressure Fluctuations and Wake Structures of a Vehicle Model

David Schröck, Nils Widdecke and Jochen Wiedemann

IVK / FKFS Universität Stuttgart

Copyright © 2009 SAE International

ABSTRACT

The unsteady environment road vehicles are exposed to is subject of many investigations that are currently made. Yet, the approaching flow is only one aspect of unsteady forces acting on the vehicle. Unsteady wake structures also lead to time-varying surface pressures and consequently fluctuating forces even in steady and low turbulent flows. However, little is known about the influence of realistic flow conditions, i.e. as found on road, on the unsteady surface pressures and wake structures of a vehicle.

Therefore, to derive a deeper understanding of the unsteady aerodynamic properties of a vehicle this paper presents results of measurements conducted on a vehicle body both in smooth and turbulent flow conditions in the IVK model scale wind tunnel. Unsteady surface pressure measurements in the area where separation occurs and the base of the vehicle were made together with time accurate total pressure measurements in the wake. The analysis of periodic surface pressure fluctuations and flow structures reveals remarkable changes in time averaged and instantaneous properties of the flow field when the turbulence intensity is brought to levels typical for on road conditions.

These findings give insight into the interaction between realistic turbulence levels and changes in the flow around the vehicle body. From this the impact on unsteady forces is derived.

INTRODUCTION

The unsteady nature of the flow around passenger cars and the resulting surface pressure fluctuations have been heart of many investigations. But due to the complex nature of those unsteady structures, their dependency on Reynolds Number or turbulence level they partly remain misunderstood.

A large amount of work has been done especially on simplified car models (e.g.: Sims-Williams et al [1, 2, 3, 4], Duell et al [5], Vio et al [6]) and some production cars (e.g.: Sims-Williams et al [3], Gilhome [7]) investigating the time averaged topology of the wake as well as vortex shedding. As reviewed in Sims-Williams et al [3] and as outlined in Hucho [8] consensus exists that squareback and notchback vehicles exhibit diverse wake characteristics. The wake region of squareback models is dominated by vortices transverse to the mean flow direction with vortex shedding occurring at a Strouhal number of $Sr \approx 0.2$ based on free stream velocity and the square root of the frontal area. In contrast, the wake of notchback models is far more complex. The trailing vortices generated at the c-pillar constantly change their position in the wake leading to complex randomly occurring structures that are hard to identify.

Those findings are based on investigations done in the low turbulence environment of wind tunnels with conventional ground simulation. In recent years it was recognized that a better simulation of driving on road requires the installation of boundary layer conditioning

systems and moving belts. With such a system it is possible to condition the boundary layer to the desired "block profile" and to realize relative movement between the floor and the underbody of the vehicle as well as the rotation of the wheels. The environment simulated in such a wind tunnel perfectly describes the situation of a vehicle solely travelling on road on a windless day. But these conditions hardly ever exist. Most often the flow situation is somehow affected by other vehicles and the presence of natural winds, both leading to increased turbulence levels and time varying flow conditions. These wind conditions have been investigated and reviewed by Saunders et al [10], Cooper et al [11, 12], Watkins et al [14], and most recently by Lindener et al [13], and Schröck et al [15]. All report a flow field experienced by a vehicle that differs significantly from the one simulated in today's wind tunnels. Turbulence intensities of approximately 5 % and turbulent length scales up to the order of several vehicle lengths are reported. Depending on the scale these higher turbulence levels can have extensive effects on the flow around bluff bodies by either changing the local flow angle or by enhancing mixing between the free stream and boundary and shear layers. This later process promotes the transition from laminar to turbulent layers and raises the effective Reynolds number of the flow. This effect has been shown with grid generated turbulence by Wiedemann [16] and Howell et al [17].

However, there have been relatively few studies of the effect of free stream turbulence on unsteady surface pressure fluctuations and wake structures of bluff bodies. The time dependency of surface pressures leads to unsteady forces and moments which may have an impact on vehicle safety and comfort. Understanding the influence of free stream turbulence on these unsteady effects might help to optimize the vehicle in these premises. Therefore, in this paper a first attempt is made to investigate unsteady surface pressures and wake structures of a vehicle bluff body in both smooth and turbulent flow conditions. Further, experimental techniques suitable for time-accurate measurements are derived.

EXPERIMENTAL SETUP

THE IVK MODEL SCALE WIND TUNNEL

All measurements were done in the $1/4 - 1/5$ IVK model scale wind tunnel of Universität Stuttgart. It is a Göttingen type wind tunnel i.e. with a closed air circuit and an open jet test section. The exit nozzle cross-sectional area is $A = 1.65 \text{ m}^2$ with a maximum free stream velocity of 80 m/s. In 2001 the tunnel was equipped with a boundary layer conditioning system and a state of the art 5-belt road simulation system [9].

The quality of the flow leaving the nozzle exit is far better than the minimum requirements for automotive wind tunnels as stipulated by the SAE Report J2071 [18]. The value for overall turbulence intensity of the model scale wind tunnel is given with $Tu < 0.3 \%$. To achieve higher

turbulence levels and a velocity spectrum similar to that of the lowest part of the atmospheric boundary layer a turbulence generation system composed of a set of spires is placed in the settling chamber. The design of these spires is based on investigation by Counihan [19, 20, 21] and is widely used among civil engineers in boundary layer wind tunnels. Adapted by Wang [22] to meet the requirements of installation space, turbulence level and length scale the final design of the five spires used in this setup is given by:

- a height to width aspect ratio of $h / b = 10.8$
- lateral spacing of $d / b = 3$.

The spires are placed in the settling chamber of the model scale wind tunnel (Figure 1) to increase the distance to the model. When setting up such a system this length is a major parameter in order to obtain sufficient turbulent mixing of the flow.

To characterize the incident flow measurements were done 200 mm in front of the model using a fast response four-hole Cobra-Probe. Measuring time was 60 s at each position with a sampling rate of 1250 Hz, data was then low pass filtered with a 50 Hz Butterworth filter. In Figure 2 it can be seen that the boundary layer profile is not altered with the installed spires. In fact, a small boundary layer with a height of approximately 50 mm is present, because neither the boundary layer conditioning system nor the centerbelt have been used. In the center graph of Figure 2 an increase of the overall turbulence intensity due to the presence of the spires can be seen. With an average value of $Tu \approx 3.5 \%$ over the measured height from 20 to 300 mm above ground the turbulence level falls well among average values known from road measurements. The graph to the right of Figure 2 shows the distribution of turbulent length scales (TLS) of the three velocity components. Average values over the measured height are $TLS(u) = 0.28 \text{ m}$, $TLS(v) = 0.38 \text{ m}$, $TLS(w) = 0.52 \text{ m}$ for a fifth scale model. The turbulent flow structures generated by means of the passive spire system are of the size of length scales found under strong traffic conditions on motorways. The scales achievable with passive systems are limited to the size of the turbulence generation system and, of course, the size of the wind tunnel itself. In order to achieve larger structures of the size of several car lengths as found under windy conditions on road, active systems are necessary. For the purpose of the investigations done here the increase in turbulent length scales due to the passive system is sufficiently large to simulate a strong traffic condition and to result in a completely different flow condition compared to the standard wind tunnel configuration.

Values of mean velocities, turbulence levels and length scales are not sufficient to completely describe the properties of a turbulent flow because they are based on an analysis of average values. To further describe the flow, the energy distribution with frequency is necessary. Therefore, measured velocity spectra are compared to theoretical curves deduced from von Kármán's

interpolation formula [26]. These spectra are given by the following equations:

$$\Phi_u = \frac{2}{\pi} \cdot \left[1 + 70.78 \cdot \left(\frac{n \cdot L_u}{\bar{u}} \right)^2 \right]^{-\frac{5}{6}} \quad \text{eq. 1}$$

$$\Phi_{v,w} = \frac{2}{\pi} \cdot \left\{ \frac{\left[1 + 188.76 \cdot \left(\frac{n \cdot L_{v,w}}{\bar{u}} \right)^2 \right]}{\left[1 + 70.78 \cdot \left(\frac{n \cdot L_{v,w}}{\bar{u}} \right)^2 \right]^{\frac{11}{6}}} \right\} \quad \text{eq. 2}$$

In Figure 3 the spectra of the three velocity components are shown in comparison to the theoretical curves. A very good agreement is found indicating that the energy transported by eddies in the considered frequency range is distributed in accordance to the theory.

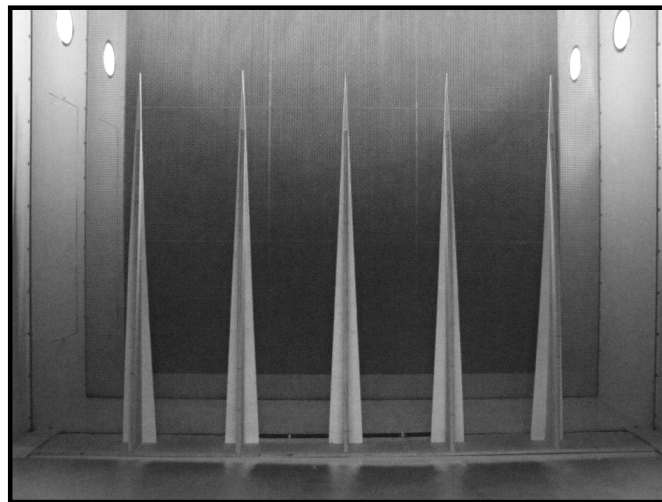


Figure 1: Turbulence generators in the settling chamber of the IVK model scale wind tunnel

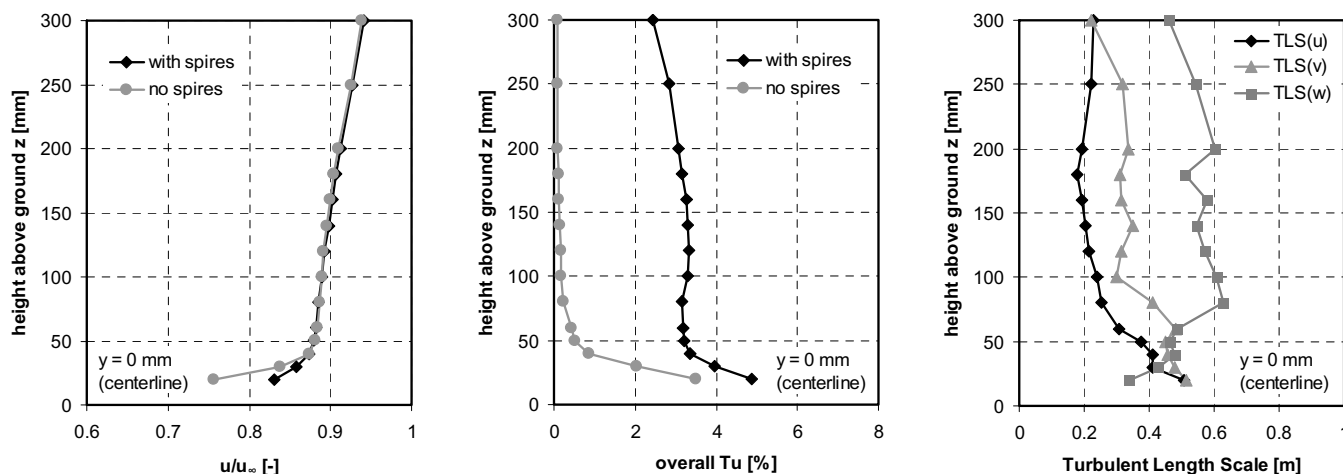


Figure 2: Characterization of incident flow 200 mm in front of model at centerline (with model present); left: boundary layer profile, middle: overall turbulence distribution; right: turbulent length scales

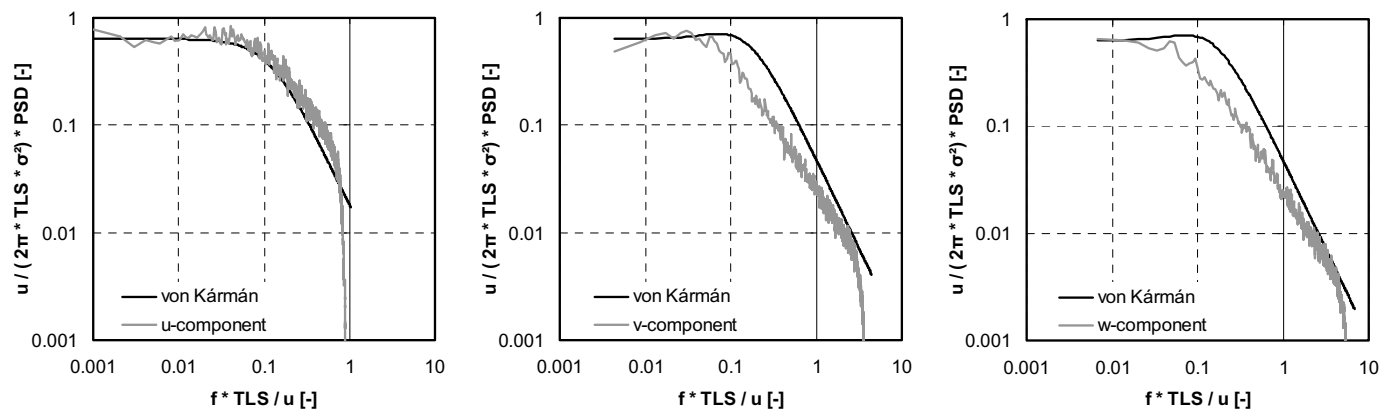


Figure 3: Dimensionless velocity spectra of u-, v-, and w-component of incident flow 200 mm in front of model at centerline, 140 mm above ground (with model present)

THE SAE MODEL

To investigate the influence of different wind tunnel parameters and to generate a simplified model for wind tunnel comparisons the SAE Reference Body was designed by the SAE Open Jet Interference Committee. The model is made of a simplified body held on struts with interchangeable rear end modules [18].

For the purpose of the measurements conducted here the shape of the squareback rear-end module was changed in order to achieve a high dependency on the turbulence level of the flow. As can be seen in Figure 4 a 120 mm radius was fitted to the top rear edge of the model resulting in a local Reynolds number for the velocity under investigation of $Re(35 \text{ m/s}) = 11.4 \cdot 10^5$.

The model is equipped with a set of 24 pressure taps located on the rounded rear end as indicated by the black marks in the rear view of the model in Figure 4. The pressure taps are spaced 20 mm along the circumference and 40 mm in y – direction.

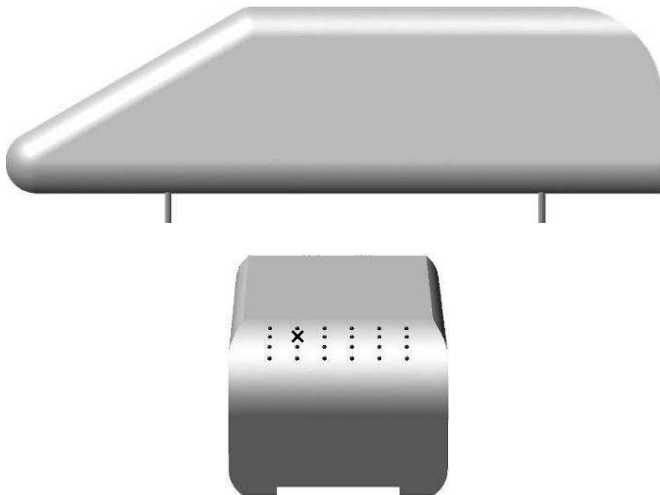


Figure 4: SAE Reference Model with rounded rear end edge, side and rear view

TIME ACCURATE PRESSURE MEASURING SYSTEM

In order to measure pressures on the surface of the model the pressure taps described above are connected to a series of differential pressure transducers mounted inside the model. The taps are connected with tubes to the transducers to transfer the pressure. In order to successfully conduct unsteady pressure measurements this system requires optimization because viscous damping and resonance in the tubes and the transducer housing alter amplitudes and lead to phase shifts. Those effects need to be corrected with the transfer function of the pressure tap – tubing – transducer housing – system.

The experimental setup used to determine the transfer function of the system can be seen in Figure 5. It includes a signal generator and an amplifier, a fluctuating pressure generator, a flush surface mountable reference pressure transducer, the pressure system to be

calibrated and a data acquisition system to determine the transfer function. The calibration method consists of incrementally increasing the frequency of the sinusoidal pressure fluctuations and measuring the analogue signals of the two pressure transducers. The transfer function is then derived by calculating the amplitude ratio of the two signals and the phase lag for each frequency under investigation. Knowing the systems transfer function, a distorted signal can be corrected and the correct pressure at the tap can be calculated.

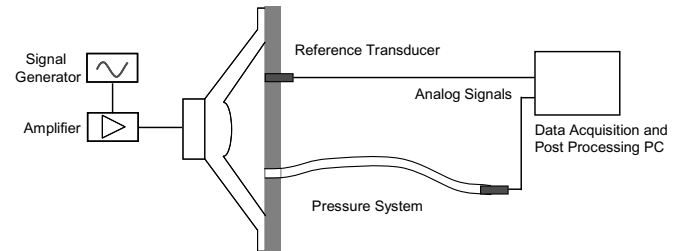


Figure 5: Experimental setup for the determination of tubing transfer functions

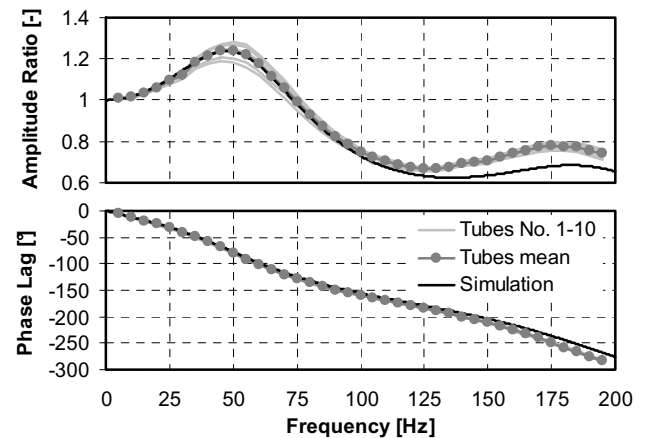


Figure 6: Transfer functions of pressure tube systems from experiment and simulation

In addition to the experimental determination of the transfer function it can also be derived theoretically using a theory from Bergh and Tijdemann [23]. Here the internal diameter of the tubing, its length and the volume of the transducer housing must be accurately known together with atmospheric conditions at which measurements are taken in order to calculate the systems transfer function. In this simulation errors can arise from a number of causes:

- the internal diameter of the tube is neither accurately known nor is it constant over the entire length due to inaccuracies during production. Therefore, a “theoretical radius” has to be determined which can be used in the simulation.
- the properties of the tubing material, its viscous and damping effects and temperature dependencies are not taken into account.

Therefore, considerable effort has been put into the optimization of the theoretical model by [24].