

Vehicle Wind Noise Measurements in a Wind Tunnel with a Contoured Top Profile

Vincent Rovedatti and Jacob Milhorn
Calvin College

Richard DeJong
Calvin Engineering

Gordon Ebbitt
Toyota Technical Center USA, Inc.

ABSTRACT

A 1/4 scale model vehicle profile has been tested in a wind tunnel with speeds up to 360 km/h. In order to simulate the free field flow over the vehicle, the top surface of the wind tunnel is contoured. A CFD simulation of the free field flow at various speeds is used to identify the desired top streamline. Then the boundary layer growth on the top surface is calculated and the top contour is adjusted accordingly. Since this contour changes very little with flow speeds of interest, an average contour is used for a fixed top surface of the wind tunnel. Pressure drop measurements are used to verify the flow similarity to the CFD model. Wind noise measurements using surface mounted pressure transducer arrays are used to determine the acoustic loads on the vehicle surfaces.

CITATION: Rovedatti, V., Milhorn, J., DeJong, R., and Ebbitt, G., "Vehicle Wind Noise Measurements in a Wind Tunnel with a Contoured Top Profile," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(1):2016, doi:10.4271/2016-01-1316.

INTRODUCTION

Wind tunnel tests have been used previously to study the changes in the frequency spectrum of turbulent flow over varying surface profiles and pressure gradients [1, 2]. In this study a 1/4 scale model center profile of a double cab pickup truck is used to measure the turbulent field behind the rear panels and to correlate this with wind noise models used to determine the acoustic loadings on the vehicle surfaces.

Exact scaling to a full scale vehicle is not possible with a 1/4 scale model because different features in the turbulent flow scale with velocity, dimensions or Reynolds number [3]. However, with an analytical model of the turbulent pressure spectrum, such as that developed by Chase [4], the physical scale model can be used to validate important features in the analytical model.

The focus of the work presented here is to develop a validated analytical model of the separated flow region behind the rear panels of a pickup truck which has not been accomplished in previous work [2].

CFD SIMULATION

The first step in designing the 1/4 scale model wind tunnel test section is to determine the top surface profile which will simulate free field conditions. This is done using a CFD simulation of the free field

flow over the vehicle (see Fig. 1) and identifying a streamline to be replicated in the wind tunnel. When this streamline is replaced with a solid upper surface, a boundary layer will grow and this must be accounted for. The following expression is used to determine the boundary layer thickness, δ , as a function of the distance, x , along the test section

$$\delta = 0.16 x \text{Re}_x^{-1/7} \quad (1)$$

where the Reynolds Number is $\text{Re}_x = Ux/\nu$, with U being the free stream velocity and ν being the kinematic viscosity. This thickness is added to the height of the wind tunnel. The resulting test section profile is shown in Fig. 2.

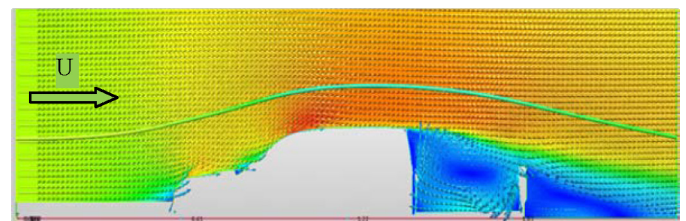


Figure 1. CFD Model of Pick Up Truck Profile Showing Top Streamline at 240 km/h. Full CFD Model Includes Air to 20 Times Vehicle Height.

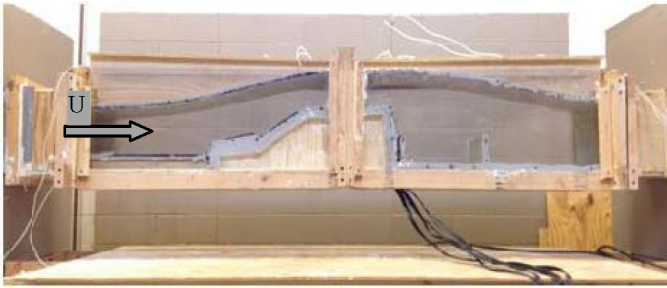


Figure 2. Wind Tunnel Test Section with Pick Up Truck Profile and Contoured Ceiling..

Since the boundary layer size changes slightly with flow speed, the contour for 240 km/h was chosen as a compromise for the range of test speeds (120 - 360 km/h).

To verify that the wind tunnel profile is accurately reproducing the free field flow conditions, the static pressure measured along the ceiling is compared to the CFD results (see Fig. 3). At 240 km/h the agreement is within 15% full scale, except near the exit of the test section where the wind tunnel contour had to be adjusted to fit the wind tunnel expansion section and the CFD simulations are not viewed to be very accurate. At the other flow speeds the agreement is within 25%..

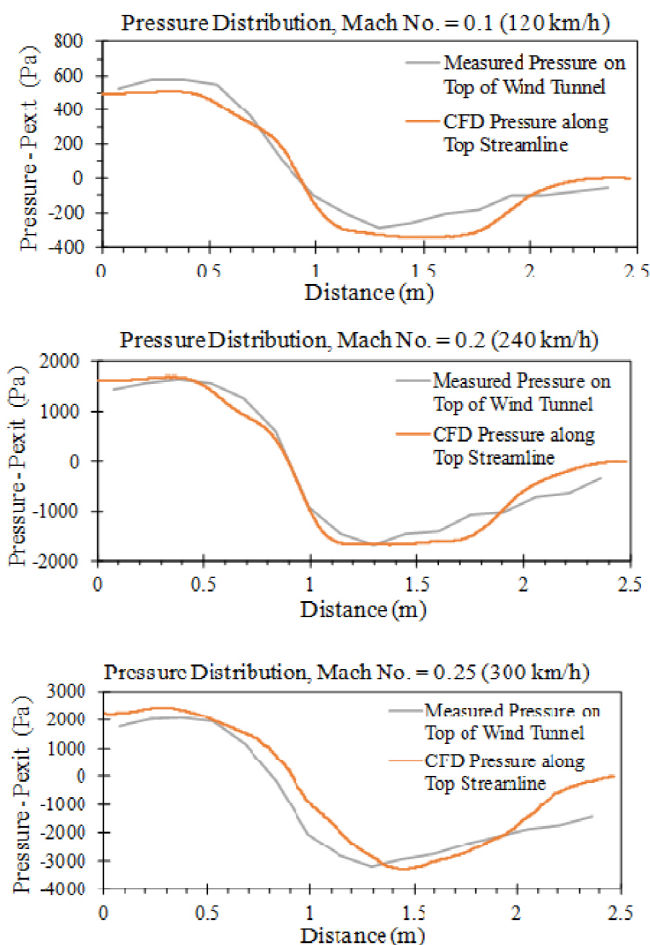


Figure 3. Pressure Drop Measured Along Wind Tunnel Top Surface Compared to CFD Simulation of Free Field Streamline..

WIND NOISE TRANSDUCER ARRAY

To measure the wind noise, surface mounted transducer arrays have been designed to detect both turbulent flow pressures and acoustic pressures. Using a pair of electret transducers (6 mm diameter, 2 mm thick) the phase delay between them can be used to determine the propagation speed of the pressure disturbance. Fig. 4 shows the pressure signals of two microphones spaced 6.35 mm apart placed on the surface of the wind shield of the vehicle model with a turbulent flow speed of 30 m/s.

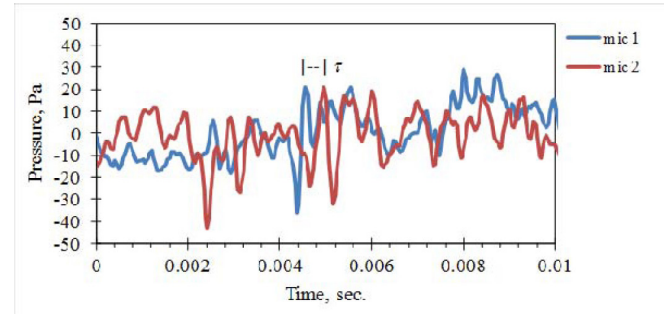


Figure 4. Pressure Signals with 6.35 mm Spacing and 30 m/s Flow Speed..

At a turbulent (convection) flow speed of $u = 30$ m/s and a spacing of $d = 6.35$ mm there is a delay of about $\tau = 0.2$ msec in the measured signal of successive transducers. The phase (ϕ in degrees) in a frequency analysis of these signals then grows in proportion to the frequency (f in Hz) by $\phi = 360^\circ f d / u$, as shown in Fig. 5. Up to about 7000 Hz, the measured phase agrees well with this model of the turbulent flow.

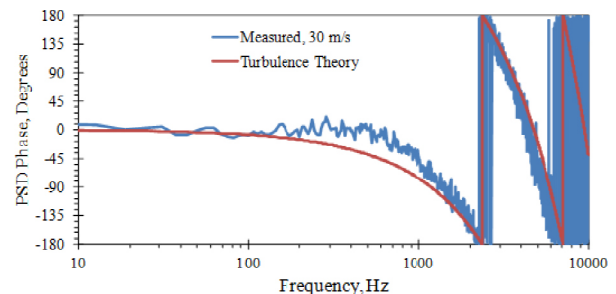


Figure 5. Phase of Cross Spectrum of Pressure Signals in Fig. 4.

The magnitude of the frequency spectrum of the measured surface pressure is shown in Fig. 6. The spectrum has the typical "hay stack" shape of a turbulent boundary layer. The "theory" line is from the basic Chase model discussed below.

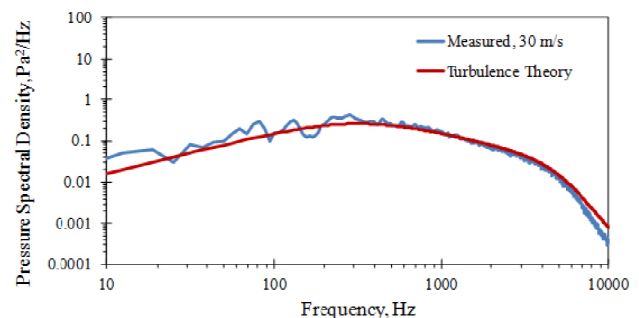


Figure 6. Magnitude of Cross Spectrum of Pressure Signals in Fig. 4..

To measure any acoustic pressure present, a wider spacing of the transducers is more useful (as in acoustic intensity probes). Fig. 7 shows the array design used here.