

Base Pressure and Flow-Field Measurements on a Generic SUV Model

Andrew Wood, Martin Passmore, David Forbes, and Daniel Wood
Loughborough Univ

Adrian Gaylard
Jaguar Land Rover

ABSTRACT

The pressure on the base of a vehicle is a major contributor to the aerodynamic drag of all practical vehicle geometries, and for some vehicles, such as an SUV, it is particularly important because it can account for up to 50% of the overall drag. Understanding the mechanisms that influence the base pressure and developing our simulation tools to ensure that base pressure is accurately predicted are essential requirements for the vehicle design and engineering process.

This paper reports an experimental study to investigate the base pressure on a specifically designed generic SUV model. The results from $\frac{1}{4}$ scale wind tunnel tests include force and moment data, surface pressures over the base region and particle image velocimetry (PIV) in the wake.

Results are presented for the vehicle in different ride height, underfloor roughness and wheel configurations and the paper includes some description of the experimental errors. Some initial CFD simulations are also reported.

CITATION: Wood, A., Passmore, M., Forbes, D., Wood, D. et al., "Base Pressure and Flow-Field Measurements on a Generic SUV Model," *SAE Int. J. Passeng. Cars - Mech. Syst.* 8(1):2015, doi:10.4271/2015-01-1546.

INTRODUCTION

The automotive industry faces a number of challenges in the design and optimization of vehicles for aerodynamic purposes, particularly with the pressing need to reduce energy consumption. However, a number of aspects of vehicle aerodynamics are not sufficiently well understood for further progress in drag reduction and accurate simulation during the design and engineering process. One such area is the numerical prediction of base pressure, where the contribution to the overall drag is very important but the accurate prediction is difficult because of an incomplete understanding of the main physics and flow-field mechanisms. In the case of SUV's the base pressure is a particularly large contribution to the total drag so an understanding of the driving mechanisms is critical. This paper serves a number of complimentary purposes. Firstly a simple generic SUV model is developed and described and made available for ongoing CFD development; secondly a description of the effect of wake structure upon base pressure and the resulting effect on drag is given for this generic geometry. Finally the experimental data will be available for download from the Loughborough University Institutional Repository [1]. The data includes details of the geometry and tunnel setup, full descriptions of data and its location in the model frame of reference.

METHODS AND MATERIALS

Model Design

The use of simplified vehicle models for experimental and computational research is well established in the aerodynamics community and has a proven record in enhancing our understanding of the underlying fundamentals and in the development of CFD methods. Well known examples of this are the Ahmed geometry [2], designed to capture the influence of a specific vehicle feature (the rear slant) or the SAE reference model [3], designed as a general purpose simplified vehicle. The focus of these models has been towards the conventional vehicle configurations, such as fastback, notchback and square-back but none exist to capture the underlying geometry and shape of an SUV. A new model was therefore developed based on an analysis of the large SUV market segment. Twenty-seven external dimensions were taken for a range of 39 vehicles from 12 manufacturers with model years spanning 1970 to 2011. The side profiles are shown overlaid in Figure 1. With normalized measurements and the data color coded with model year.

To define the wind tunnel model the characteristic data was summarized to identify trends and an initial geometry based on these trends generated. This was then further simplified to remove the influence of the large wheel arch eyebrows, often evident in these vehicles, and to remove the tumblehome and backlight rake from the

glass house. These simplifications reduce the Reynolds sensitivity of the model and allow a wider range of future parametric studies to be undertaken based on the generic SUV. Finally a radius was applied to all leading edges to prevent local separations that would not be representative of full scale. The size of the radius was based on the work of Newnham [4].

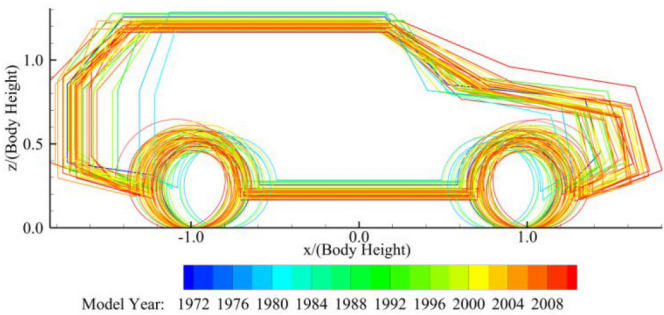


Figure 1. Outlines of benchmarked SUV models, coloured by model year

The model was designed to facilitate changes in ride height (tunnel floor to model underside), underfloor roughness and for testing with and without wheels. The configurations tested are listed in Table 1, with images of the changes in ride height, underfloor roughness and wheels shown in Figure 2, Figure 3 and Figure 4 respectively. The wheels and tires do not include any detail. The underside of the wheel is flattened to simulate the contact zone and there is a small gap (2mm) between the underside of the tire and the tunnel floor.

Table 1. Tested configurations

Configuration number	1	2	3	4	7
Ride Height	Nominal (65mm)	Low (50mm)	High (80mm)	Nominal	Nominal
Underfloor roughness	No	No	No	Yes	No
Wheels	Yes	Yes	Yes	Yes	No

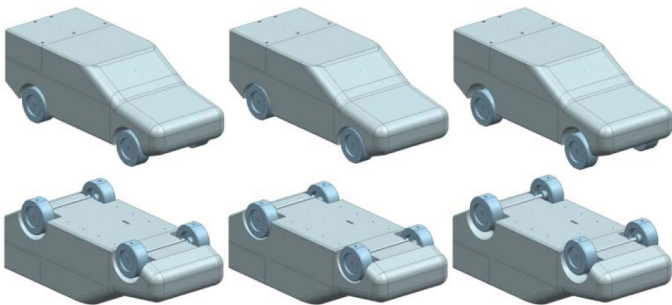


Figure 2. Ride height variations

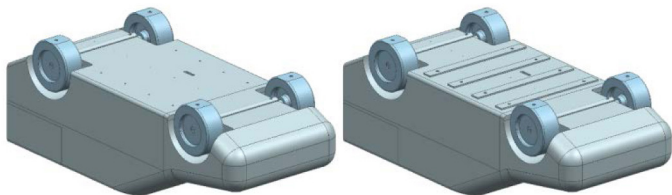


Figure 3. Underfloor roughness strips

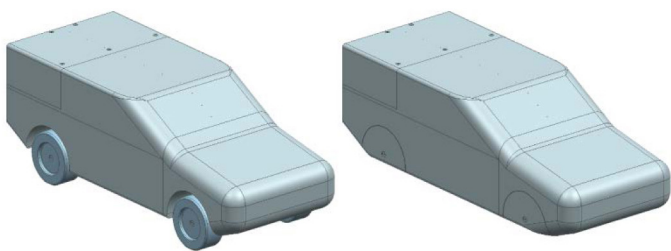


Figure 4. With and without wheels configurations

As the focus here was to consider the base pressure it was considered useful to make a comparison of the boundary layer development along the model compared to that at full scale. Velocity profiles were recorded using a hotwire system on the model and two full scale SUV models. Data were taken at a range of positions over the length and span of the model and vehicles. The results show a similar form factor validating continued use of the model.

Wind Tunnel

All testing was conducted in the Loughborough University model scale wind tunnel. The layout is shown in Figure 5. Automotive models up to approximately ¼ scale can be tested at approximately 5% blockage. A 140kW fan is capable of producing a flow velocity of up to 45m/s in the 1.92 by 1.32m working section. The contraction ratio is 7.3:1 and flow conditioning turbulence screens limit freestream turbulence intensity to 0.2%. In the work reported here all testing was conducted at 40m/s, giving a Reynolds's number of 2.85×10⁶ based on model length. The tunnel is operated in a fixed floor mode with a boundary layer displacement thickness at the model center of 9.4mm [5].

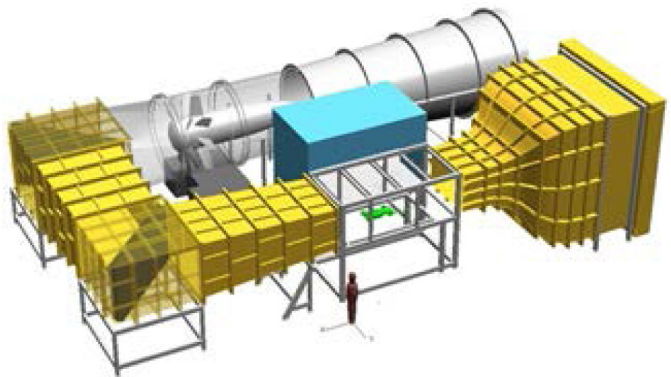


Figure 5. Loughborough University wind tunnel

Balance Measurements

Models are mounted onto the six component underfloor balance via four 8mm supporting pins that pass through 10mm holes in the tunnel floor into holes in the underside of the model wheels, leaving a small clearance under the wheels that have a flattened bottom to represent the contact patch. Adjustment of the supporting pins ensures a consistent setup. The balance and central disk in the tunnel floor are connected to a yaw drive to facilitate automated yaw sweeps. Balance data were sampled for thirty seconds at each yaw position after a ten

second settling time. Further details of the balance accuracy can be found in Johl [5], along with further information on the wind tunnel itself. Yaw sweeps were conducted from -20 to $+20^\circ$.

Surface Pressure Measurements

The base and rear diffuser of the model were populated with a grid of pressure tapings as shown in in Figure 6. The tapings are limited to one half of the model to allow for greater resolution. The pressure measurements were made using two 64 channel miniature pressure scanners located inside the model and accurate to $\pm 0.15\text{mm H}_2\text{O}$. Samples were triggered by an externally supplied 260Hz signal generator and samples were taken for 31 seconds for each model configuration. Data are recorded against tunnel static pressure.

Pressure coefficients for the model surface have been calculated using the free-stream dynamic pressure (recorded at the start of the working section). All results have been blockage corrected using the MIRA blockage correction (based on continuity), Equation 1.

$$C_{P_{\text{corr}}} = 1 - \left(\frac{1 - C_p}{\left(1 - \frac{TA}{A}\right)^{-2}} \right) \quad [1]$$

TA - Tunnel Area

A - Model Frontal Area

C_p - Recorded Surface Pressure Coefficient

$C_{P_{\text{corr}}}$ - Continuity Corrected Pressure Coefficient

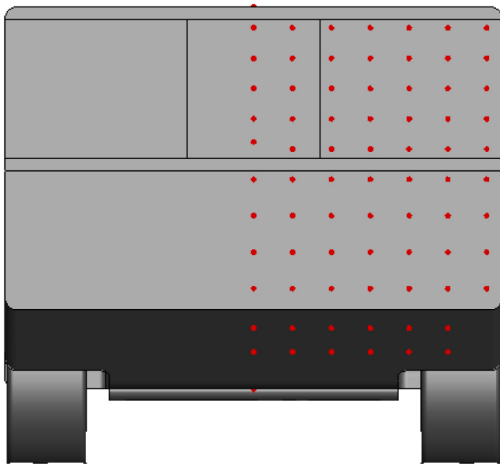


Figure 6. Surface pressure measurement locations

Mean pressure coefficients, $\overline{C_p}$, for the base and diffuser are presented separately and additionally the base + diffuser $\overline{C_p}$, which is based on the relative rearwards facing area of each constituent part, is calculated.

Particle Image Velocimetry (PIV)

Two-dimensional planar PIV measurements have been conducted in a number of planes in the vehicle wake region, as represented in Figure 7. The two horizontal planes are at half upper base height (blue) and

half lower base height (red). The vertical planes are the vehicle centerline (green), the wheel centerline (purple) and a location midway between these planes (yellow).

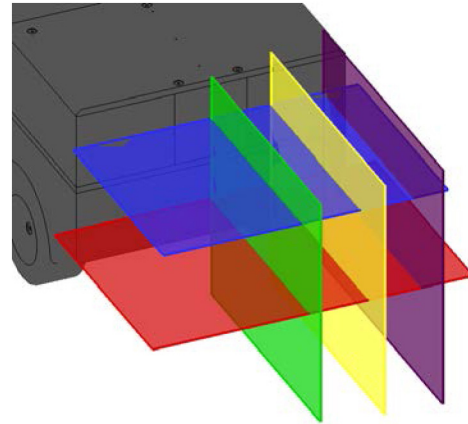


Figure 7. PIV measurement planes

Dual-frame images were acquired with a 4 mega pixel, 14bit camera equipped with a 50mm lens in combination with a frequency doubled Nd:YAG Litron LASER with 200mJ pulse energy, operating using the LaVision DaVis software. The image area was $\sim 400 \times 400\text{mm}$ giving a spatial resolution of 0.2mm/pixel . 1000 image pairs were captured for each plane at a recording frequency of 7.26Hz.

CFD Methodology

Computational validation of the wind tunnel tests was performed using Exa PowerFLOW, a commercial code that utilizes the Lattice Boltzmann Method (LBM). This method is a special discretization of the Boltzmann equation in space, time and velocity. It simulates the flow and collision processes of particles within a Newtonian fluid [6]. The turbulence model employed in the code is similar to Very-Large Eddy Simulation (VLES) with a $k-\epsilon$ RNG model acting as a sub-grid scale model. The LBM solves the transient, turbulent flow of air around the SUV and the solution is averaged in time to give the mean flow solution.

Spatial discretization in PowerFLOW generates what is known as a lattice, containing 'voxels' (cuboidal volume cells) and 'surfels' (surface cells generated as a voxel intersect a surface). The lattice used for this study was generated using a best practices template currently employed by a leading company in the automotive industry with automated voxel sizes (minimum 0.5mm) and refinement regions based upon model dimensions and geometric features. The smallest voxel size generated was 10^{-3} percent of the model length. A cross-section of the lattice shows the locations of refinement around the vehicle, figure 1. The total number of voxels in the fluid domain is approximately 70 million with y^+ values in the range of $15 \leq y^+ \leq 100$ over the surfaces of the SUV, hence wall functions are applied for the entire surface.

A velocity inlet and pressure outlet are used, with slip walls defined until the start of boundary layer growth, (4.5m upstream of the origin) at which point no-slip wall conditions are applied to all tunnel and SUV surfaces. To replicate the experimental conditions, values for free stream velocity and turbulence intensity are set to 40m/s and