

A Parametric Study of Automotive Rear End Geometries on Rear Soiling

Anton Kabanovs, Graham Hodgson, Andrew Garmory, and Martin Passmore
 Loughborough University

Adrian Gaylard
 Jaguar Land Rover

ABSTRACT

The motivation for this paper is to consider the effect of rear end geometry on rear soiling using a representative generic SUV body. In particular the effect of varying the top slant angle is considered using both experiment and Computational Fluid Dynamics (CFD). Previous work has shown that slant angle has a significant effect on wake shape and drag and the work here extends this to investigate the effect on rear soiling. It is hoped that this work can provide an insight into the likely effect of such geometry changes on the soiling of similarly shaped road vehicles. To increase the generality of results, and to allow comparison with previously obtained aerodynamic data, a 25% scale generic SUV model is used in the Loughborough University Large Wind Tunnel. UV doped water is sprayed from a position located at the bottom of the left rear tyre to simulate the creation of spray from this tyre. Having a single source of contamination simplifies the configuration of both experimental tests and simulations. It also improves analysis by allowing the soiling pattern from only one wheel to be seen in isolation. In order to provide further insight into the flowfield and its interaction with the spray CFD simulations are also performed at the same scale. A Detached Eddy Simulation approach is used, specifically the Spalart Allmaras formulation of the IDDES CFD model. Lagrangian particle tracking is used to model the dispersed phase. This CFD methodology has been found to give good agreement for soiling pattern with experiment for baseline cases.

CITATION: Kabanovs, A., Hodgson, G., Garmory, A., Passmore, M. et al., "A Parametric Study of Automotive Rear End Geometries on Rear Soiling," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(2):2017, doi:10.4271/2017-01-1511.

INTRODUCTION

The management of surface contamination, or soiling, of vehicles is important to all vehicle manufacturers. Three sources of contamination are generally described [1]. Primary sources are contamination directly resulting from rain; third-party contamination refers to contaminants resulting from other vehicles and self-soiling refers to contaminants resulting from spray generated by the vehicle itself. The work reported here is focused on the latter phenomenon that is particularly relevant for vehicles with blunt rear geometries, such as off-road vehicles, Sports Utility Vehicles (SUVs) and estates (Station wagons) [2]. Self soiling is particularly important for these vehicles because the blunt rear geometry causes the formation of strong large-scale recirculating vortices that draw the spray towards the base [3]. As a result, base contamination generates various issues for the driver and passengers including reduced visibility (for both the driver and any camera systems), a reduction in perceived quality and the potential for soiling to be transported to skin and clothing.

As the rear soiling is closely linked to the structure of the wake the issue of rear surface contamination is best addressed concurrently with aerodynamic drag. For example, for commercial vehicles a reduction in overall vehicle drag can also lead to a reduction in spray [4,5]. Therefore, the ability to predict the self-soiling phenomenon early in a vehicle development programme would provide a useful

insight and allow designers and aerodynamicists the opportunity to resolve soiling issues alongside the more usual areas of concern such as drag reduction, aero-acoustics and cross-wind stability.

Much of the published work has understandably focused on specific vehicles. For example, a very comprehensive study into the prediction of surface contamination for a detailed SUV has been reported by Gaylard and Duncan [3], as well as Gaylard et al. [6, 7]. Conversely, simplified bodies that lack details and styling of specific vehicles, are commonly used to study automotive aerodynamics. These bodies allow the investigation of the relevant flow features in well characterised and repeatable environments with geometries that can be employed by different research groups. The work reported here applies this same practice by considering surface contamination in controlled wind-tunnel conditions with a simplified vehicle body (Generic SUV) and conducting the same experiments numerically. This will improve the generality of any conclusions.

Whereas much published work [7,8,9,10] has focused on the final outcome with regard to the surface contamination, or on comparing CFD simulations with experimental work, the work presented here studies the variation of base contamination with changing roof slant angle. Roof slant angle variations were selected for the study because

they are commonly employed as a drag reduction method [11]. The work reported contains a thorough description of the experimental and numerical methods and analysis of the results.

EXPERIMENTAL METHODOLOGY

Facilities

Testing was performed in the Loughborough University Wind Tunnel, Figure 1, full details of which can be found in [12]. The operating velocity in the test section was 40 m.s^{-1} . The freestream turbulence intensity in the middle of an empty test section is approximately 0.2 % and flow uniformity ± 0.4 %. The pressure data presented in this work has been corrected to account for blockage effects using a continuity based correction method shown in Equation (1) [13].

$$C_{P\text{-corrected}} = \frac{C_P + 2 \frac{MA}{TA}}{1 + 2 \frac{MA}{TA}} \quad (1)$$

In addition, force coefficients have been calculated using the corrected value for velocity, estimated using a continuity correction seen in Equation (2) [14]. In the equations, TA and MA correspond to the wind tunnel cross section area and the model frontal area, respectively. C_p and $C_{p\text{-corrected}}$ are the recorded surface pressure coefficient and the corrected surface pressure coefficient, respectively. Finally, v and $v_{\text{corrected}}$ are the recorded reference velocity and the corrected reference velocity respectively. These pressure and force correction procedures have been also applied to CFD data.

$$v_{\text{corrected}} = \frac{v \cdot TA}{TA - MA} \quad (2)$$

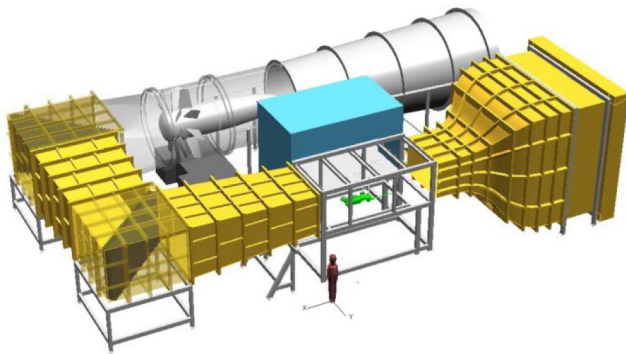


Figure 1. The Loughborough University Wind Tunnel

Model

A quarter scale generic SUV Model, Figure 2, was used for this work. The model is constructed of a machinable model board and gives a blockage ratio of 5% in the 2.5 m^2 working section. The model has large leading edge radii to prevent separation and sharp edges at the rear. The geometry is representative of a typical SUV with configurable elements including ride heights, roof tapers, diffusers and roughness strips. More information can be found in [15]. For this study the diffuser is fixed at 10 degrees, no roughness strips are

present and three taper angles are investigated; 0, 10 and 20 degrees. The ride height is fixed at 65 mm. SUVs have large bases and these are prone to contamination effects. As such, this model is highly suited to such a parametric study.

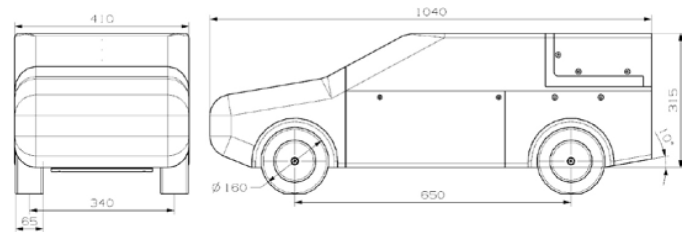


Figure 2. Dimensions (in mm) of the generic SUV model

Figure 3 shows the side profile of the model with the three tested roof tapers colour coded: grey 0 degree, blue 10 degree and pink 20 degree. The 10 degree diffuser is also shown in blue.

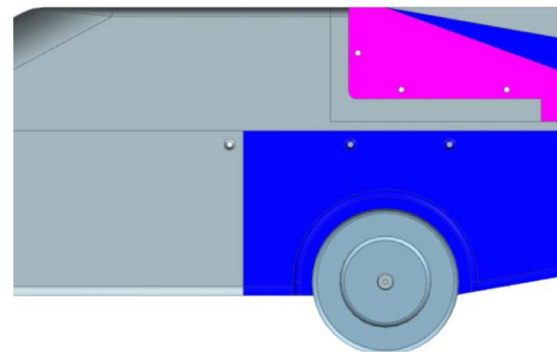


Figure 3. Roof taper geometries

Droplet Characterisation

To ensure simulations were matched to experiments the droplet characteristics have been measured in a separate Phase Doppler Anemometry (PDA) experiment. This allows the simulation to more accurately replicate the experiment than in previous work [9].

The PDA setup can be seen in Figure 4 with equipment as described in [16,17]. Traverses at 2, 3, 7, 10, 20 and 30 mm from the nozzle outlet face were conducted on the half-cone in a no-flow (air) arrangement. These tests identified droplet diameter (with a resolution of approximately 0.18 microns) and two velocity components for every droplet over a 5 s test at points typically 1-2 mm separation.

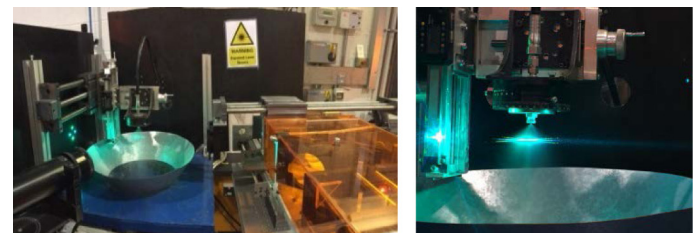


Figure 4. PDA experimental setup

Seeding Setup

A schematic of the seeding setup used in the wind tunnel can be seen in Figure 5. A single nozzle, installed 86 mm behind the rear near-side wheel center and directed 45 degrees downstream was used. This location was chosen to reasonably represent the typical source [7] of self-soiling and ensure that sufficient particles of fluid were entrained in the body and wheel wakes. Using a single source, rather than one located behind each rear wheel ensures that a clear analysis of the source and outcome can be achieved. A pair of solenoid valves controlled electronically allow for known, fixed duration of spray injection ensuring repeatable spray characteristics that can be provided as input to a numerical model. This setup also ensured a sharp start and end to injection, reducing experimental variation.

A mixture of de-ionised water and UV dye (Uvitex at 0.03 % concentration) was used as the contaminant with an UV lamp as the illumination source. Following every test the light source and a DSLR camera were placed in the tunnel and an image captured. A testing pressure of 110 bar resulted in a mass flow rate of approximately 3.5 g.s^{-1} .

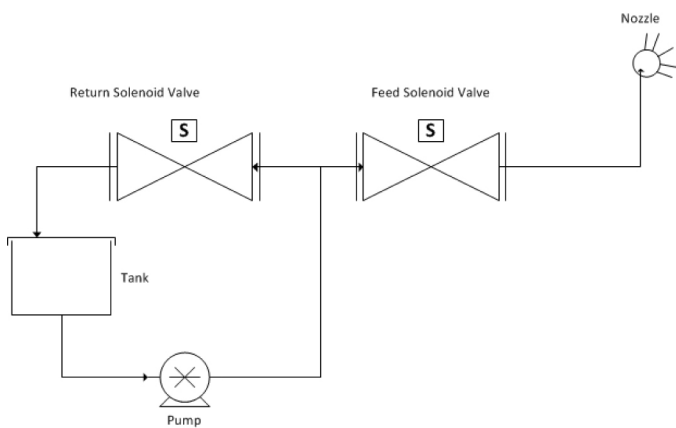


Figure 5. Seeding setup

Measurements

The model was run in the wind tunnel for sufficient time (~30 s) to settle prior to the spraying and every injection was 12 s in duration. This time was selected to be short enough to ensure no rivulet formation occurred in any configuration and long enough to account for any very low frequency wake structures that may impact soiling. Rivulet formation, while of interest in the broader Exterior Water Management (EWM) field, ensures the location of the droplet impact is lost and relocates the mass of fluid to the lower regions of the base. Captured images are stored in RAW format to ensure image processing is consistent across tests. Combined with an appropriate calibration the Beer-Lambert Law (Equation 3) can be used to calculate a depth of fluid from an image intensity.

$$g_t = [A(1 - e^{Bx}) + C]^D \quad (3)$$

The calibration is achieved by placing a calibration vessel similar to that used in [9,18] in the tunnel next to the model at the end of every test so that it is captured in the image. Figure 6 shows the variation of intensity with depth across the calibration vessel. It can be seen that with the illumination, exposure and dye concentration used that the relationship is reasonably linear, i.e. intensity is directly proportional to depth.

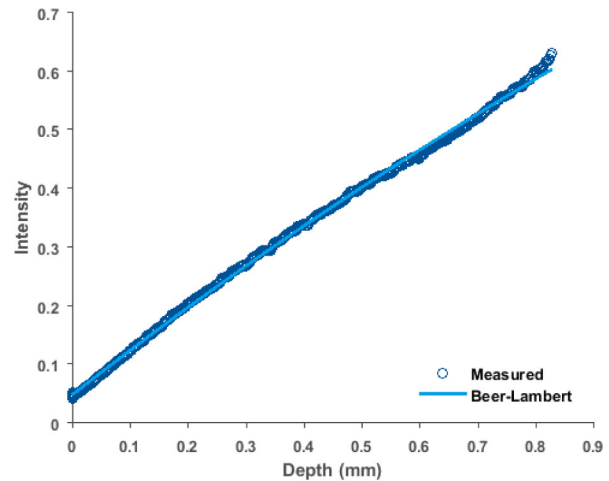


Figure 6. Typical Beer-Lambert curve fit

CFD CONFIGURATION

Mathematical Models

Computational simulations were performed using OpenFOAM®, an open source CFD tool. The continuous phase (air) was computed using the Improved Delayed DES (IDDES) formulation of the Spalart-Allmaras Detached Eddy Model. The DES formulation [19] and its recent modification, delayed DES (DDES) [20], use RANS to compute the whole or at least the major part of the attached boundary layer, while an LES approach is used in regions with separated flow. The recent modification of the DDES model, Improved DDES (IDDES), has an enhanced capability to identify which mode (RANS or LES) it should be operating in near the wall [21]. In addition, it allows some of the boundary layer to be resolved by LES, resulting in a quicker transition to resolved turbulence and hence increased resolved turbulence activity near the wall [21].

In this study the Navier-Stokes equations were solved using the PISO (Pressure Implicit with Splitting of Operators) algorithm. The maximum global Courant Friedrichs Lewy (CFL) number, defined as

$$CFL = \frac{U * \Delta t}{\Delta x} \quad (4)$$

was found to be of order 10, however the CFL number in the separated region was less than 1 to ensure that eddies are resolved in time and space. The dispersed phase (water particles) was computed using the Lagrangian approach [22]. The Lagrangian particle tracking approach allows simulations to be simplified by tracking groups of particles together as a numerical parcel. Although increasing the number of parcel streams (in the limit to match the number of