

Numerical Investigation of Features Affecting Rear and Side Body Soiling

Jonathan Jilesen
Exa Corporation

Adrian Gaylard
Jaguar Land Rover

Jose Escobar
Exa Corporation

ABSTRACT

Vehicle rear and side body soiling has been a concern since the earliest cars. Traditionally, soiling has been seen to be less importance than vehicle aerodynamics and acoustics. However, increased reliance on sensors and cameras to assist the driver means that there are more surfaces of the vehicle that must be kept clean. Failure to take this into consideration means risking low customer satisfaction with new features. This is because they are likely to fail under normal operating conditions and require constant cleaning.

This paper numerically investigates features known to have an influence on side and rear face soiling with a demonstration vehicle. These changes include rim design, diffuser strakes and diffuser sharpening. While an exhaustive investigation of these features is beyond the scope of this study, examples of each feature will be considered. This will include discussion on what flow changes were being attempted with a given feature and how this was expected to influence the soiling performance.

Numerical simulation will then be used to evaluate the impact of a given feature and discern if the desired impact was achieved.

CITATION: Jilesen, J., Gaylard, A., and Escobar, J., "Numerical Investigation of Features Affecting Rear and Side Body Soiling," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(1):2017, doi:10.4271/2017-01-1543.

INTRODUCTION

A fundamental objective of vehicle aerodynamics development is to ensure that system function, driver's vision, vehicle visibility and aesthetics are maintained as it is operated in the on-road environment. This means that the design needs to be robust to, amongst other things, the presence of rain, road-soil and spray; the latter generated by both its own and other vehicle's tires. Spray generated by the vehicles rear tires poses a particular challenge as it can be advected to the rear surfaces where it can degrade: rear screen vision, rear wiper life, rear lamp visibility, rear camera performance and vehicle appearance. *Square-backed* vehicle types such as hatchbacks, estates (station wagons) and SUVs are most prone to these effects [1] [2].

Historically, approaches to mitigating rear surface contamination have focussed on turning air from the roof boundary layer flow, which is free of road spray, parallel to the rear screen. This, in turn reduces the advection of road-spray present in the base wake to the rear screen. However; this can be costly, in terms of both additional aerodynamic drag and lift [3] [4]. Although some rear geometries enable pressure recovery from the additional flow down the rear screen that can mitigate the drag penalty [5], this is by no means a

universal solution. Hence, this approach is rarely considered acceptable as aerodynamic drag targets become increasingly stringent.

In addition, this issue has become more pressing as rear view camera systems have been deployed, initially as parking aids. Although these could, at least in principle, be covered or retracted when not in use, this approach has not been taken frequently as it introduces additional cost and complexity. As these systems become used as part of vehicle safety systems, it has become vital to maintain their function *and* impossible to retract or cover.

The current view appears to be that these devices "cannot be kept clean with aerodynamic measures" [6], which seems unnecessarily pessimistic to the authors. Hence, this work seeks to describe the beginnings of a research effort to understand and quantify sensitivities of rear surface contamination to vehicle aerodynamic design interventions, with the aim of at least being able to redistribute contaminants away from the most sensitive locations, such as camera positions.

Recent developments in multi-phase, unsteady aerodynamic simulation mean that it is now viable to explore this issue with numerical simulation, thereby opening up additional insights. Hence, this paper presents a numerical investigation of this topic with the aim of generating initial hypothesis which can be physically tested.

Previous Relevant Work

The first published application of simulation to rear surface contamination was published by Yoshida *et al.* [7]. This used both Lagrangian and Eulerian methods to estimate the deposition of soiling on the body sides and rear of an SUV. Due to resource limitations the geometry was highly simplified; although an unsteady flow solver was used, no turbulence model was included. Yoshida *et al.* [7] concluded that as the contaminants did not affect the flow field, it was not necessary to solve for the dispersed and continuous phases simultaneously. Also, they saw no differences between using an unsteady and a time-averaged flow field. Similarly, Roettger *et al.* [8] [9] used introduced Lagrangian particles into a time-averaged flow field to assess the likelihood of deposition on the rear surfaces of an estate car.

However, the work of Paschkewitz [10] [11] on particle dispersion through the wake of commercial vehicles demonstrated that the vertical distribution of the particles differed significantly between steady-state and unsteady flow fields.

This early work did not seek to model tyre spray in a way which related to experimental measurements. Kuthada & Cyr [12] provided the first calibrated tyre spray model, based on adjusting the diameters of Lagrangian particles released into a time-averaged flow field generated by a model of a single wheel in isolation until the experimental spray topology was recovered. It is noteworthy that the diameter which provided the best match to experiment (0.2 mm) matched the mean droplet diameter measured behind a car installed in a wind tunnel reported by Bouchet *et al.* [13]. More comprehensive experiments were later reported by Spruss *et al.* [14] which enable this calibrated tyre spray model to be further refined.

This calibrated spray model was applied to simulations of body side and rear soiling for a saloon car and SUV by Gaylard & Duncan [15]. An attempt was made here to account for flow unsteadiness by tracking the particles from source to “destination” for a series of short-time averaged flow fields and then summing the resultant distributions of particle impact locations over the vehicle surface. Whilst good results were obtained for body side soiling, the rear soiling pattern for the SUV was not convincingly reproduced. Taken with the observations of Paschkewitz [11], [10] this indicated that the initial conclusions of Yoshida *et al.* [7] were premature: properly coupling particle dynamics to flow unsteadiness is required to address this problem.

Jilesen *et al.* [16] reported body side and rear soiling simulations combining concurrent calculation of the unsteady flow field and particle paths, combined with a surface film solver to account for deposition. These were contrasted with advecting particles through a sequence of frozen flow fields, demonstrating the importance of capturing the effect of unsteady flow on the dispersed phase. Their work also included models for splash, break-up and re-entrainment. Gaylard *et al.* [17] took this approach further, demonstrating the importance of including the effect of the momentum of the dispersed phase on the continuous phase in this type of simulation (two-way momentum coupling).

It is now appreciated that the credible prediction of rear surface contamination patterns on the rear surfaces of square-backed vehicles requires the concurrent calculation of particle paths and the unsteady flow field. Further, that the flow field needs to be captured by the use of an unsteady eddy-resolving approach, as both RANS and URANS fail to recover the rear wake structure and base pressure distribution, which are key controlling mechanisms for this problem [18].

Physical Mechanisms

Deposition of surface contaminant on the rear starts with the source: rear tyre spray. Weir and co-workers [19] [20] identified the structure of tyre spray, during work on heavy commercial vehicles. They categorised water lifted from the road surface by the tyre as: (a) *bow wave*, (b) *side splash wave*, (c) *tread pickup* and (d) *capillary adhesion*. Bow and side splash waves are types of splash, characterised by the presence of larger droplets that follow a ballistic trajectory and tend to quickly fall back to the road surface. Hence, these are of little significance to the rear surface contamination problem. *Tread pickup* and *capillary adhesion* form the relevant fraction of the tyre spray. Tread pickup is the result of water passing through tread grooves that is thrown off close to the road. In contrast, the capillary adhesion spray fraction is generated by a retained (capillary) film on the tyre surface which is ultimately thrown off later in the tyre rotation; hence further above the road surface. Weir *et al.* [19] estimated a droplet diameter range for tread pickup from <1 mm to ~4 mm. The capillary film fraction of the spray generally comprises a very fine spray which accounts for estimated 1% of the water volume picked up by the tyre tread.

In detailed laboratory-based rig experiments [21] [22] relationships have been noted between droplet diameters, rotational Weber number (hence speed and tread depth) and tread volume.

As shown in Figure 1. A schematic of rear surface contamination mechanisms tyre spray is advected into the base wake as a consequence of the aerodynamic interaction between rear wheel wakes and vehicle base wake [23]. Square backed vehicle wakes can be considered, in the time-mean, to comprise a toroidal ring-vortex [24]. The return flow through the centre of this structure provides a strong mechanism for the advection of spray droplets back to the vehicle surface. This aligns with the observation made by Costelli [25] that deposits tend to be most intense where base pressure is highest.

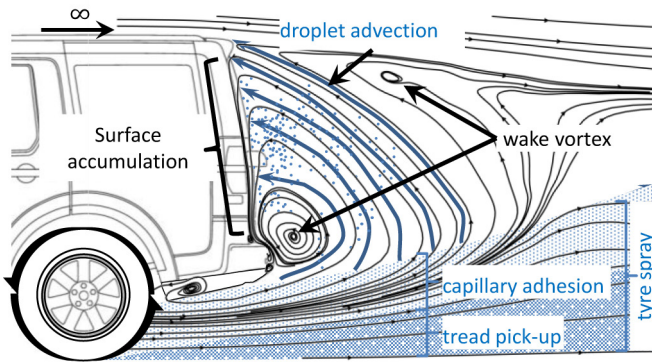


Figure 1. A schematic of rear surface contamination mechanisms [23]

METHODOLOGY

CFD simulations of turbulent flow were performed using a commercial numerical solver based on the Lattice-Boltzmann Method (LBM) [27], [28]. LBM differs from traditional finite volume methods (FVM) in the sense that it is formulated in terms of molecular motions instead of the macroscopic or continuous systems which are used in FVM. In general, a kinetic system can be expressed in terms of single particle motion at the molecular scale. In particular, LBM states the system behavior in terms of a distribution function which is the number density of molecules. This variable provides information about the instantaneous position and velocity of the molecules. Kotapati *et al.* [29] provides a detailed explanation of the LBM used here for the solution of turbulent fluid flow field. Proper discretization of the distribution function in space and in the velocity domain guarantees the recovery of the Navier-Stokes equations [29]. Integration of the distribution function yields macroscopic fluid quantities such as fluid density, velocity and pressure [30].

In this scheme turbulence is accounted for by what can be described as Very Large Eddy Simulation (VLES) [27]. This is a two-way approach where the large turbulent structures are solved directly on the computational grid, whereas small scales are modeled. Directly solving the large Eddies captures the anisotropy that emerges in the flow field, especially in regions where separation occurs. On the other hand, the more uniform (universal) scale turbulence was modeled using the RNG k-epsilon formulation. LBM is well suited for performing transient simulations because it is by definition unsteady and has very low numerical diffusion. By introducing a turbulence model in to the LBM, significantly more complex physics emerges from the RNG k-epsilon equations compared to their more usual application in FVM where it is introduced through eddy viscosity in the Navier-Stokes equations [30].

This approach has been shown to be able to address a wide range of flows associated with automotive development, including the aerodynamic pressures and forces exerted on reduced-scale models of cars [31], [32]; full scale vehicles [33], including the detail of wheel and tire effects [34] along with the additional physics associated with thermal and aeroacoustics problems [35], [36], [37].

A new capability to solve particle motion and surface film dynamics has been incorporated in this LBM-VLES software [12] [16], [38]. It has been applied to the simulation of airborne droplets comprising tire spray and the surface water film which results when they interact with vehicle surfaces. Lagrangian particles are used to represent the dispersed (droplet) phase and the solution of particle motion accounts for droplet breakup, splash and the re-entrainment of film in the surrounding air flow. The breakup model is based on the work of O'Rourke and Amsden [39] which used the Taylor Analogy Breakup (TAB) model. The splash model follows the work of Mondo *et al.* [40] along with O'Rourke and Amsden [41]. Finally, the re-entrainment model is based on a critical thickness specified by the user, in this case 0.3 mm. When the film thickness reached this critical value the excess is re-introduced in the surrounding flow as particles.

The particle motion was solved using the following assumptions:

- If the particle is in the fluid flow region, gravity and drag are the two forces acting on it. Also, if the conditions specified in the TAB model are met then the particle is subdivided into "child" particles.
- When a particle hits the surface a fraction of its mass is used to form a film, with a residual fraction passing back into the fluid flow due to splash. This surface film is subject to a number of forces, including those due to: gravity, friction between the liquid and the solid surface, the shear exerted on the surface by the surrounding flow field. When the film thickness reaches a critical film height, the excess is re-entrained into the surrounding flow.

The setup of the vehicle transient simulations followed a set of previously established best practices to ensure a grid size and distribution suitable for aerodynamics simulation, Samples *et al.* [42]. In general terms, the computational grid (lattice) follows a Cartesian coordinate system and is composed of cubic volumetric cells (voxels) with an aspect ratio of unity. Spatial resolution is controlled using hierarchical nested regions where voxel edge lengths are doubled as the lattice progresses to the next lowest resolution level. In this case, the smallest voxel edge length was 2.5 mm, which was used in regions with the largest spatial gradients of critical fluid mechanics quantities (e.g. a-pillar region). Outside of these zones the entire vehicle was immersed in a zone comprising voxels with a 5.0 mm edge length.

The numerical simulation was performed in two stages: first the unsteady aerodynamic flow field was established; next, the fluid flow field and the particle motion were solved simultaneously. In this case, two-way momentum transfer between the particles and fluid was also included.

In order to simulate the rear soiling problem, particles were emitted from the rear tires only. Following the work of Kuthada and Cyr [12], the tire spray model used emitters placed along the tire circumference, between 35° and 325°, which excludes the tire contact patch.