

Simulation of Rear Glass and Body Side Vehicle Soiling by Road Sprays

2011-01-0173

Published
04/12/2011Adrian Philip Gaylard
Jaguar Land RoverBradley Duncan
Exa Corporation

Copyright © 2011 SAE International

doi: [10.4271/2011-01-0173](https://doi.org/10.4271/2011-01-0173)

ABSTRACT

Numerical simulation of aerodynamics for vehicle development is used to meet a wide range of performance targets, including aerodynamic drag for fuel efficiency, cooling flow rates, and aerodynamic lift for vehicle handling. The aerodynamic flow field can also be used to compute the advection of small particles such as water droplets, dust, dirt, sand, etc., released into the flow domain, including the effects of mass, gravity, and the forces acting on the particles by the airflow. Previous efforts in this topic have considered the water sprays ejected by rotating wheels when driving on a wet road. The road spray carries dirt particles and can obscure the side and rear glazing.

In this study, road sprays are considered in which the effects of additional water droplets resulting from splashing and dripping of particles from the wheel house and rear under body are added to help understand the patterns of dirt film accumulation on the side glass and rear glass. Results are compared to experimental photographs for both a sedan and an SUV tested using luminescent dye in a wind-tunnel as well as in road testing. Results show that the addition of splashing and dripping particles increases the impingement of particles on the surface and provides excellent correlation to the experimental data. This allows use of simulation for identifying problems and potential solutions for upper body soiling issues.

INTRODUCTION

The contamination of vehicle surfaces, both glazed and unglazed, by water and solid contaminants, is becoming an increasingly important issue in vehicle development. In light rain, or when following other road vehicles on wet roads customers reasonably expect to have an unobstructed view through the front side glazing; particularly as it enables the use

of exterior rear-view mirrors. The need for excessive use of rear wash-wipe systems, due to the aggregation of solid contaminants, should also be avoided. Neither do customers appreciate having their hands or clothing soiled when they come into contact with exterior surfaces and handles. Demands of this kind will continue to grow, particularly with the increasing number of camera systems fitted to vehicles to facilitate reversing, emergence from junctions and the detection of collision threats.

These expectations give rise to the need for simulation methods, both physical and numerical, that can be deployed at suitable stages of a vehicle development programme to ensure that the product will meet the needs of the customer. Ideally, numerical simulation methods are required that enable soiling performance to be determined in the early phases of the development, when physical test properties are either not yet available or not sufficiently representative (i.e. lacking under body systems or the ability to have wheel rotation applied).

Surface contamination can be characterised as either *third-party* (or *foreign-soiling*), originating from the wakes of other road vehicles, for example, or *self-soiling*. The latter is typically caused by spray being driven into the flow around the vehicle by the rotation of the vehicle's wheels. The following work explores the simulation of this latter type of self-generated road spray, using Lagrangian particle tracking to simulate the spray droplets, combined with a transient flow field simulation. It should be noted that this type of simulation does not account for droplet break-up, coalescence, the formation of surface films or the stripping of droplets from surface films. Rather, the spray is modelled using particles of a fixed diameter and density.

PREVIOUS RELEVANT WORK

Road sprays have been a long-standing concern for the commercial vehicle industry. With large numbers of wheels and axles, tractor-trailer combinations can generate substantial road-spray, which then becomes a hazard for other road users. A useful summary of work in this area has been provided by Gotz and Mayr [1]. Nevertheless, a review of specific contributions of workers in this field that includes recent developments is apposite.

Clarke [2] explored the reduction of splash and spray from commercial vehicles; building on work done during the 1970s. This work looked at road-spray generation mechanisms, the conversion of splash to spray, wheel set interaction, fender and wheel well treatments. The investigation was undertaken using actual vehicles running through water troughs. Olson and Fry [3] reported an investigation of the effect of aerodynamic aids on splash and spray generation using water troughs, combined with video recording and a laser measurement system.

Lajos, Preszler and Finta [4] looked at mud and dust deposits on bus bodies via scale-model wind tunnel testing. They demonstrated the effect of a moving belt road simulation, finding that the, “rear wall deposits depend mainly on the structure of the wake and the characteristics of the under body flow.” Fujimoto *et al* [5] combined full scale trials and reduced-scale wind tunnel testing to elucidate the mechanisms of mud accretion on the rear surfaces of a “van type truck” and the potential for drag-reducing devices to mitigate both this and road spray.

Goetz and Schoch [6] reported on a laser-illumination method for measuring the spray generated by both commercial vehicles and cars. They also demonstrated a number of modifications to the wheel arch system that reduced road spray (wheel arch liner grooves and extended mud flaps).

Kim [7] investigated the ability of a truck-cab corner turning-vane to reduce cab side dirt contamination. The work used both Reynolds Averaged Navier Stokes (RANS) CFD simulation to study the *aerodynamic characteristics*, along with full-scale road tests.

The literature for cars follows a similar pattern, starting with full-scale track testing, moving into wind tunnel and ultimately computational simulations. Early work by Dawley [8] focussed on the use of a roof trailing edge spoiler to keep a station wagon backlight clear of contamination, using a test track based experiment. In a similar vein, Costelli [9] published on the aerodynamic development of the Fiat UNO; part of this work demonstrated the effectiveness of a variable-attitude spoiler for reducing soiling on the backlight. This work appears to have used wind tunnel based test methods.

Currently, the major theme in the published literature is the management of water and soiling on the front side glass and

door mirror glass. Karbon and Longman [10] demonstrated the viability of combining Lagrangian particle representation of airborne water droplets with a thin (surface) film model to account for water movement on vehicle surfaces, within a RANS simulation. An interesting feature of this work was the use of a periodic surface water source on the windscreen to model the effect of windscreen wipers driving surface water over the a-pillar.

Waki *et al* [11] investigated door mirror design, assessing mirror glass contamination. They used a full-scale track test with photosensitive paper attached to the mirror glass and spraying “developing liquid” into the airflow. In a second phase of the work, a water tunnel was used to perform flow visualisation experiments. A simple assessment of aeroacoustic impact was also made. Bannister [12] provided a seminal analysis of the relationship between aerodynamic drag, door mirror design, and side glass soiling. This work was based on track testing and the use of a rain grid in a full-scale aerodynamic wind tunnel. Borg and Vevang [13] investigated droplet size distribution within wakes (foreign-generated contamination), enabling the development of a wind tunnel based *foreign-soiling* simulation using a bespoke turbulence generator and rain grid. They demonstrated the efficacy of this system in assessing front side-glass contamination. The work also included an assessment of body side *self-soiling*, introducing water with a UV tracer onto a dynamometer roller. Bouchet *et al* [14] used specially designed rain collectors in the Jules Verne climatic wind tunnel to determine the intensity spatial gradient in the *wake* of a vehicle. Their measurements of the droplet size distribution generated by the rear wheels of their test vehicle showed more negative skew and positive kurtosis (leptokurtic) compared to natural raindrop size distributions. Both the negative skew and kurtosis tended to decrease (slightly) with increasing vehicle speed, the *mode* diameter decreasing and thus becoming more centred in the range. The mode droplet diameter for a vehicle speed of 80 km/h was 0.2 mm.

Foucart and Blain [15] reported a multi-stage computational method for simulating rain collection on the windscreen and subsequent movement of a water film over the a-post onto the side glass. The initial computation of the flow field used a Lattice Boltzmann solver. This was used to provide the boundary conditions for a local unsteady RANS model focussing on the windscreen and front side glass. Rain droplets were represented via Lagrangian particle tracking, with surface water simulated by an Eulerian model.

Experimental validation was presented, using wind tunnel experiments with a UV tracer in the water. Borg and Vevang [16] examined the use of Lagrangian particle tracking methods with and without a wall film model to examine front side-glass water contamination, within the context of a RANS simulation. Kruse and Chen [17] reported the development of an Euler-Lagrange method, to address the same issue. This work also

included experimental verification using a water spray with a UV tracer in the FKFS thermal wind tunnel. Although the paper concentrated on front side glass, it is clear that the experimental method was more widely applicable.

Road sprays, the topic of this paper, have been explored using a combination of unsteady Lattice Boltzmann flow simulation and Lagrangian particle tracking, starting with Roettger *et al* [18, 19]. Kuthada and Cyr [20] provided a comprehensive overview of vehicle soiling issues and explored the use of this combination of modelling techniques for road spray driven off the vehicle's tyres. In addition, this work provides an experimental validation for the road spray generated by a wheel in isolation. Finally, Borg [21] reported the use of RANS flow simulation plus Lagrangian particle tracking to analyze body side and chassis soiling from vehicle generated road sprays.

Other surface contamination issues reported in the literature include the nature of the rainwater film that forms over the bonnet of a road vehicle [22] (building on the earlier development of a bespoke test facility [23]) and the deposition of brake dust on the front wheels of a saloon car [24].

Experimental methods continue to be developed; Aguinaga and Bouchet [25] illustrate the 'state of the art' in test techniques for side-glass contamination measurement. Bouchet [26] has explored 'rain' intensities due to splash and spray from a vehicle running on a wet road. Future developments in the numerical simulation of soiling deposition may well include the use of unsteady Large Eddy Simulation (LES) [27] and the adoption of open-source CFD code as the simulation framework [28].

This work explores the extension of a combination of Lagrangian particle tracking and flow field simulation using a transient Lattice Boltzmann solver, where additional particle sources are added to account for splashing and dripping from the wheel house and rear under body.

VEHICLES

SUV

Two vehicles were selected for this work: a classic, two-box, SUV and a 'fast back' saloon. The 2.7L diesel variant of the 2005MY Land Rover Discovery 3 (LR3) was selected for the SUV geometry. This vehicle emphasises a command-driving position and exhibits classic Land Rover styling cues, such as verticality of surfaces. The result is a bluff body with a large near-vertical base area.

The LR3 is 4.838m long, with a 2.885m wheelbase. The fixed-roof variant height is 1.837m; vehicle width (mirrors deployed) is 2.176m with front and rear track of 1.605m and 1.612m, respectively. The projected frontal area is 3.15m² and has a drag coefficient of 0.40. Experimental work was carried out on

a pre-launch field evaluation unit (FEU); numerical simulation was based on a production-representative CAD model.

The typical concern with road sprays for this type of vehicle is soiling of the rear surfaces. However, body side soiling is still relevant, particularly with reference to door handle cleanliness.

SALOON

The 2.7L V6 diesel variant of the 2008MY Jaguar XF was chosen for the saloon car geometry. Its aerodynamics has been discussed in detail by Gaylard [29]. This full-sized saloon aims to fuse the style and performance of a sports car with the space and refinement of a luxury saloon. The sports car heritage is seen in the generally aggressive stance; epitomised by large 'haunches' over the rear wheels along with an upright and aggressive front grille. Alongside this it incorporates a number of (potentially) aerodynamically beneficial features: a well raked screen, cambered roofline, glass house tumblehome, reasonable front tyre-face coverage, squared-sills and a 'fastback' style rear with a sharp boot deck trailing edge condition.

The XF is 4.961m long, with a 2.909m wheelbase. Its height is 1.460m; vehicle width (mirrors deployed) is 1.877m with front and rear track of 1.559m and 1.571m, respectively. The projected frontal area is 2.33m². This is combined with a drag coefficient that is relatively representative of contemporary full-sized saloon vehicles, of 0.29.

The experimental work was completed using a full-scale aerodynamic test property ('aerobuck'). This married a new fibreglass upper body with an updated donor 'go-cart', providing a fully representative test property with complete under body and engine bay. The numerical simulations use an equivalent CAD model.

Generally, for a vehicle of this type, the principal road-spray concern is body side soiling as it relates to door handle cleanliness and the soiling of customers' clothes as they enter and egress from the vehicle. For this reason, the experimental investigation did not consider soiling of the rear surfaces.

SIMULATION METHOD

NUMERICAL

High-fidelity transient aerodynamics simulations were performed for the SUV and saloon using commercial Lattice Boltzmann Method (LBM) software. The simulations were performed using a grid size and distribution suitable for aerodynamics simulation (See [30], for example.). The simulations were setup with rotating wheels. To match the test environment of the wind tunnel, the floor was fixed rather than moving, and small sliding belts were added under the wheels to simulate the wind tunnel dynamometer. The transient flow data was recorded with a 100Hz sampling rate in order to provide a *sequence* of flow data frames for the particle advection calculations.

Particle advection was computed using the post-processing approach described in Kuthada and Cyr [20]. The advection is computed using Lagrangian integration of particle trajectories, considering the local aerodynamic drag force opposing the direction of motion, the pressure gradient force, as well as effects of mass and gravity. For efficiency of the calculation, the advection is computed in a post-processing procedure that assumes a frozen flow field and computes the motion of particles from the release location until they stop on the surface or exit the flow domain. The particle advection calculation is repeated for *each* frame of flow field data sampled from the transient flow. The deposition is accumulated over all frames. The advection of particles through a frozen flow field allows the effect of the relatively long time-scales (many minutes) required for deposition of material on the surface to be recovered using a relatively short-duration time simulation (about 1 second of real time). Summing the deposition over many frozen flow fields, sampled from the transient flow, captures the spreading of particles by random structures in the turbulent flow over the surface of the vehicle.

Particles were released from the tyre at 10-degree stations around the front or rear wheel as shown in Figure 1, and were ejected tangential to the direction of motion of the tyre. The particles were assumed to be water droplets with density 1000 kg/m^3 , and the distribution of particles sizes was determined from the experimental results of Kuthada and Cyr [20] (in preference to the measurements made in a *vehicle wake* by Bouchet *et al* [14]). A mean droplet size of 0.165 mm was used. Splashing of droplets was roughly approximated by allowing them to reflect from surfaces as if they were solid particles rather than liquid. Therefore, droplet break-up during splash was not considered. Particles that hit any surfaces of the vehicle were considered to reflect (splash) from the surface except when velocity was near zero, in which case the particle was considered to come to rest on the surface. For either reflected or stopped particles, the location on the surface is marked as a “hit point” where dirt, sediment, and water can be deposited on the surface of the vehicle. The hit points are marked with dots in the visualization to assess the surface deposition.

Dripping from wetted surfaces was also considered as an additional particle source. The dripping was simulated by releasing additional low-velocity small particles with a distribution of sizes up to 0.2 mm from the wheelhouse and under body - regions that could have a significant impact on the body side or rear face surfaces. Larger particles were also considered, but just impact on the ground so they can be neglected.

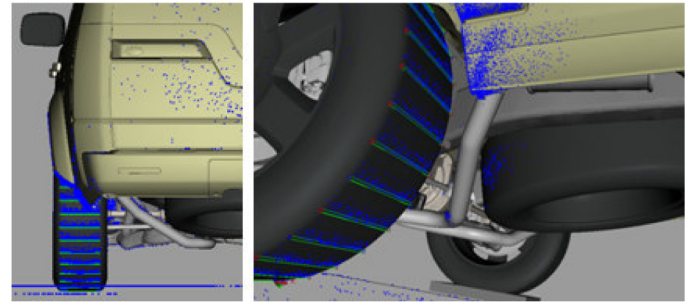


Figure 1. Rakes for emitting wheel spray particles are placed around the tyre in 10-degree increments.

The particles are released from the spinning wheel surface, parallel to the tyre/wheel at each station. Rakes are omitted for the front lower quarter of the tyre since these particles will spray directly into the ground.

In Figure 2(a), the advection of particles is shown with reflection turned off - particles that hit the wheel housing surface just stop on the surface, so only some regions of spray from the wheel will escape the wheel housing and contribute to the soiling of the body side. In 2(b) and 2(c), particles are reflected back into the flow, creating a more realistic spray from the wheel and wheel arch. Droplets may reflect multiple times on the wheelhouse as well as the body side. This results in more particles being emitted from the top of the wheel, forming a large wheel spray region downstream.

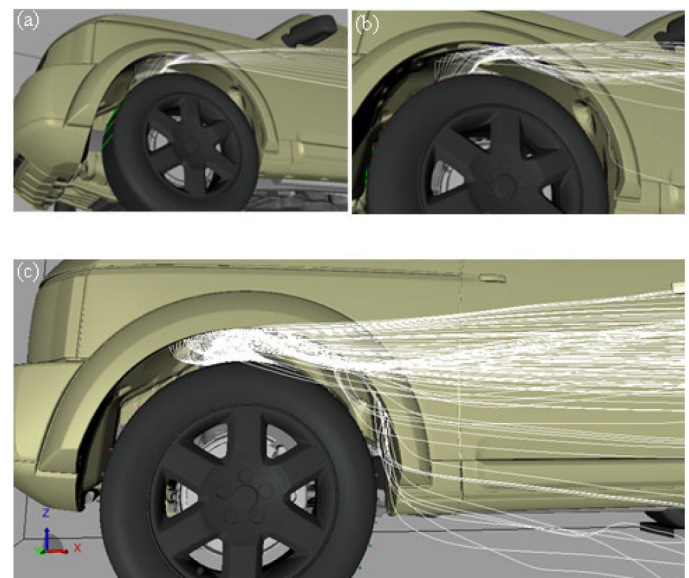


Figure 2. Splashing in the wheelhouse is simulated by allowing particles to reflect in the wheel housing. (a) No reflection in wheelhouse, (b) and (c) With reflection in wheelhouse