

Simulation of Rear and Body Side Vehicle Soiling by Road Sprays Using Transient Particle Tracking

Jonathan Jilesen
Exa Corporation

Adrian P. Gaylard
Jaguar Land Rover

Bradley Duncan, Alex Konstantinov and John Wanderer
Exa Corporation

ABSTRACT

Numerical simulations have proven to be effective tools for the aerodynamic design of vehicles, helping to reduce drag, improve cooling flows, and balance aerodynamic lift. Aeroacoustic simulations can also be performed; these can give guidance on how design changes may affect the noise level within the cabin. However, later in the development process it may be discovered that soiling management issues, for example, necessitate design changes. These may have adverse consequences for noise or require extra expense in the form of technological counter-measures (i.e. hydrophobic glass).

Performing soiling simulations can allow these potential issues to be addressed earlier in the design process. One of the areas where simulation can be particularly useful is in the prediction of soiling due to wheel spray. Our previous work on this topic examined the uses of pre-calculated flow fields and particle 'hit points' to predict dirt coverage. [1]

In this study, the advantages of using fully transient particle tracking along with a film model for the simulation of wheel spray are investigated. These include the elimination of *ad hoc* particle emitters (used to account for dripping), a more accurate splash model, and the ability to account for fluid particle interaction. Results are compared to both wind tunnel and track test data, along with those of a previous study which used a pre-calculated flow field methodology on the same vehicle. The results for the transient solver show improved side and rear surface coverage. These improvements are credited to more accurate transient tracking of particle trajectories, the inclusion of more realistic physics and the ability to simulate a far larger number of particles.

CITATION: Jilesen, J., Gaylard, A., Duncan, B., Konstantinov, A. et al., "Simulation of Rear and Body Side Vehicle Soiling by Road Sprays Using Transient Particle Tracking," *SAE Int. J. Passeng. Cars - Mech. Syst.* 6(1):2013, doi: 10.4271/2013-01-1256.

INTRODUCTION

The ability to predict vehicle soiling is becoming increasingly important. Although drag reduction and NVH performance are generally considered to be more important, issues such as handle cleanliness and the ability to get in and out of the vehicle without coming into contact with soiled surfaces are getting more attention, particularly for premium products. Additionally, as more camera systems are applied to vehicles, maintaining an acceptable rate of soiling is important if these systems are to meet customer expectations. Unfortunately, soiling issues usually surface late in the design process, as vehicle testing requires full scale prototypes. This is problematic because, while early in the design process

changes can be made that balance aesthetic, aerodynamic and wind noise requirements, as the process moves forward this becomes increasingly difficult; the scope for change becomes restricted and changes become more costly.

Key issues usually relate to reduced driver visibility through soiling of glazing (side and rear), door mirror glass and camera lenses. The source of the soiling can be *foreign-soiling* (*third-party*), originating from other vehicles, for example; or *self-soiling*. This study focuses on the second, with soiling in the form of wheel spray. Wheel spray *self-soiling* usually causes difficulty with rear window visibility, vehicle or camera lens cleanliness.

This work is a follow up to a previous study [1], where our first phase results were presented. This focused on a pre-calculated flow field methodology for predicting wheel spray soiling. The methodology was limited in that it could not account for droplet break-up, splash, or coalescence. Additionally no model for the formation of surface films or the stripping of droplets from the surface was included. In the current study all of these deficiencies, except for the modeling of particle coalescence, have been remedied in a methodology where particles are simulated concurrent to the flow simulation. In addition to providing maximum temporal resolution, having the particle and film simulations concurrent with the flow solver allows for bi-directional momentum coupling between the fluid and particle solvers. Results generated with the concurrent solver are compared both to results generated using the previous methodology [1], as well as experimental results presented in that study.

PREVIOUS RELEVANT WORK

While an extensive review of specific contributions in the field of road sprays can be found in our previous study [1], it is apposite to revisit the most relevant works.

Road sprays have been a long-standing concern for the commercial vehicle industry. During the 1980s Clarke [2] and Lajos *et al.* [3] were using experimental techniques to investigate soiling for commercial vehicles. Clarke [2], investigated road-spray generation mechanisms, the conversion of splash to spray, wheel set interaction, as well as fender and wheel well treatments using actual vehicles. Lajos *et al.* [3] instead used scale-models to look at mud and dust deposits on bus bodies.

Numerical techniques were not applied to wheel spray soiling until the 2000s when Kim [4] investigated the ability of a truck-cab corner turning-vane to reduce cab side soiling. This work comprised of Reynolds Averaged Navier Stokes (RANS) CFD simulations as well as full-scale road tests. The Lattice Boltzmann method combined with Lagrangian particle tracking was applied to road sprays a year later by Roettger *et al.* [5] [6]. A comprehensive overview of vehicle soiling issues and the use of Lagrangian particle tracking in combination with the Lattice Boltzmann method can be found in Kuthada and Cyr [7]. Work with RANS simulations has also continued with Borg [8] analyzing body side soiling from road sprays using a combination of RANS and Lagrangian particles.

Soiling simulations require information on particle size distributions for boundary conditions. Experimental work measuring the droplet size distribution within vehicle wakes such as Bouchet *et al.* [9] provided important measurements for calibrating numerical simulations. The particular UV dye experimental technique, used in this study is that developed by Borg and Vevang [10].

APPROACH

Experimental Setup

The results of two different experiments are used in this study. These experimental results were first reported by Gaylard and Duncan [1] where they were used to verify the pre-calculated flow field approach. In the previous work, both a sedan and a 2.7L diesel 2005MY Land Rover Discovery 3 (SUV) were studied while this paper focuses on the SUV alone. These comprise track and climatic wind tunnel testing.

During the track testing the SUV was driven at 80 km/h over a graded and wetted gravel track at Volvo's Hällered complex. The resulting soiling pattern was recorded photographically.

The wind tunnel testing was conducted using the technique described by Borg and Vevang [10]. This involved injecting water treated with UV dye into the two rear tyre contact patches, while the wheels are driven by a single axle dynamometer. During testing both the onset wind speed and roller speeds were set to 80 km/h. Video recordings were taken of the resulting soiling pattern and these images were post processed to provide information on the distribution of film thickness.

Simulation Approach

All of the numerical results shown in this paper are the result of high-fidelity transient aerodynamics simulations, which were performed using commercial Lattice Boltzmann Method (LBM) software. The simulation setups used were based on recommended best practices to ensure a grid size and distribution suitable for aerodynamics simulation, see Samples *et al.* [11], for example. The transient flow data was recorded with a sampling rate of 100Hz in order to provide a sequence of flow data frames for use with the pre-calculated flow field approach.

Particle Tracking and Film Tracking

Two different methods were used for predicting particle advection, these were a post-processing approach described in Kuthada and Cyr [7] and a concurrent approach where particle trajectories are calculated within the solver at each time step. These two different methodologies are contrasted in the results section.

In the post-processing approach the flow field is assumed to be frozen for the computation of the particle advection. This means that for each frame of flow field data particles are released and tracked until they either exit the flow domain or stop on a surface. This process is repeated for each frame of flow field data sampled from the transient flow, with the 'hit points' being accumulated over all the frames. The advantage of using a frozen momentum field is that relatively short-duration time simulations (around one second of physical time) can be used to predict the surface deposition resulting from much longer time-scales (in the order of minutes). The use of multiple time frames of data is a means of

economically accounting for the transient nature of the flow field.

The simulation approach used for the fully integrated method was similar, except that the trajectories of the particles were updated at each time step. Since the simulation of the particle trajectories is performed by the flow solver, post processing is simpler and only requires visualization of the scalar properties provided by the solver. Capturing the full transient behavior of the particles within the flow field has a significant impact on the distribution of particles within the flow field as well as the distribution of 'hit points' on the surface. This will be covered in detail by the results section. Solving the particle motion within the fluid solver has the benefit of allowing for two-way coupling with the surrounding fluid. The drag force acting on the particles within the surrounding air also has a reaction force which acts on the surrounding air. This force can have a significant impact on the motion of the air medium when there is a large density of small particles. One current disadvantage of this in-line particle simulation methodology is that the physical time scales for the particles are directly tied to the physical time covered by the fluid simulation.

In addition to the differences listed above the concurrent processing simulation model used in this study includes several improvements to physics modeling over the post-processing algorithm. These improvements include models for the splash and breakup of particles, as well as a surface film solver.

The film solver used with the concurrent method requires time-accurate mass fluxes and surface tractions as boundary conditions and therefore needs to be coupled to a time accurate simulation of the surrounding airflow.

The modeling of surface film dynamics is based on the assumption that droplets form a thin film of fluid on top of a solid, driven by the shear stresses on both the air-liquid interface and the liquid-solid interface. In addition to these two surface forces, gravity is treated as a body force acting on the volume of the film. The height of the film is determined during an advective step which enforces continuity, and then the velocity is computed based on the momentum balance including the surface and volumetric forces described above.

Overview of Splash model

When water particles hit a surface there is a high likelihood that splashing will result in multiple child-particles as illustrated in Figure 1. The correlations provided by Mondo *et al.* [12], and O'Rourke and Amsden [13] are implemented within the particle solver of the commercial LB solver.

Splash modeling was not available for use within the post-processing methodology. Splashing within the post-processing methodology was mimicked by enabling surface reflection; except for at very low particle velocity, effectively treating all splashes as events where no remnant is left on the surface. This has the disadvantage that droplet break-up during splash was not considered.



Figure 1. Splash Schematic.

Overview of Breakup Model

The shape of water droplets continually changes as the particles move through the surrounding air medium. At the extremes of these deformations there is the possibility for particle breakup, this breakup is one of the factors which determines the size of particles in the flow. Particle size has a significant impact on particle trajectory, which makes it important to include particle breakup in the modeling of particle advection. The fully transient particle advection solver used in this study includes a model based on the TAB model of O'Rourke and Amsden [14]. This model uses a spring-mass-damper model to calculate the vibration of the water droplets and predict when a deformation sufficient to result in breakup occurs.

Modeling particle breakup is particularly important in regions where the water particles undergo acceleration, such as when moving around the A-pillar of a vehicle or when drops are entrained into fast moving air. In both the pre-calculated flow field and concurrent simulation processes particles that drip from the surface have a larger volume (approximately 3 times larger) than those which are initially emitted into the simulation. In the concurrent simulation these particles are able to breakup into smaller particles, while in the post-processing methodology particles are unable to breakup.

Discussion of Re-Entrainment Model

A re-entrainment model was included in the concurrent solver that causes particles to be released from the surface film. This is important, as along with the inclusion of a surface film model, it allows water to move along the surface of the vehicle, pool and then re-enter the airflow as larger droplets. These dripped particles have a significant impact on the soiling pattern seen on the vehicle, as shown in Gaylard and Duncan [1]. Without the inclusion of a film and re-entrainment model within the post-processing methodology, engineering judgment had to be used to introduce secondary emission boxes, to account for these dripped particles.

Wheel Spray Configuration

Particles are released from the tyre at stations separated by 10 degree intervals around each wheel as shown in Figure 2. These emitters are as wide as the tyre and emit particles with a diameter of 0.165 mm. This is to match the size of particles observed in Kuthada and Cyr [7] (in preference to the measurements made in a vehicle wake by Bouchet *et al.* [9]). The particle density is set to 1000 kg/m³. The mean velocity of the particles is tangential to the surface of the tyre and equal to the surface speed. A standard deviation of 5 m/s

is added to this velocity magnitude and the direction was allowed to vary within a solid angle of 10 degrees.

In the concurrent simulation 20 million particles per second are emitted from each station, (some being masked by the floor). So for all four tyres, a total of 2.72 billion particles per second are being emitted into the flow. In the post-processing method far fewer particles (hundreds) are emitted at each station, due to software limitations. However, additional particles are released at very low velocity from the wheelhouse and under body to account for dripping, which is implicitly included within the concurrent method. These are regions that have a significant number of 'hit points' from the primary emitters. The dripped particles are given a distribution of sizes up to 0.2mm, as in Gaylard and Duncan [1].

Two different soiling conditions are simulated with the concurrent process; the first replicates wind-tunnel experiments where only the two rear tyres (one axle) emit particles, and a second replicates the on-track test condition with all four tyres (two axles) emitting particles.

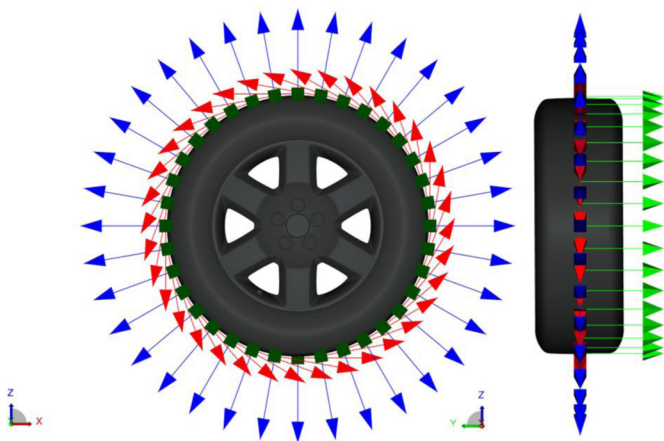


Figure 2. Wheel Spray Setup

RESULTS

Experimental Results

Images of the soiling patterns on the rear of the SUV from wind tunnel and on-track testing are shown [Figure 3](#). In [Figure 3\(a\)](#), which shows the wind tunnel results, the film thickness has been visualized through post processing video footage of a water film containing UV dye. The UV dye process cannot be used in on-track testing, so no film thickness information is available for the image seen in [Figure 3\(b\)](#). Without the film thickness information it is difficult to clearly see the extent of the coverage obtained during the on-track testing. The track test also introduces additional uncertainties, such as cross winds and steering angle. The gravel track at H  ller  d is oval so steering angle may bias soiling results for wheel spray testing. Comparing [Figure 3\(a\)](#) and [Figure 3\(b\)](#) though it does appear that there is

more soiling of the bottom rear corners of the SUV in the track test compared to the wind tunnel.

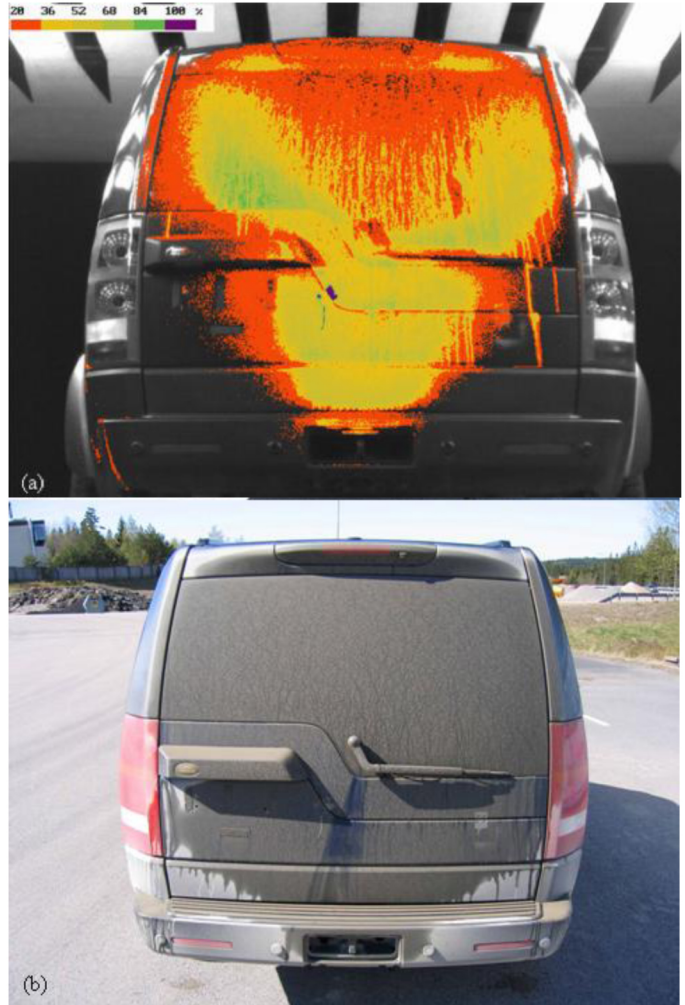


Figure 3. (top) Wind tunnel deposition intensity analysis and (bottom) on road test experimental pictures.

Aerodynamics Results

The introduction of a moving road boundary condition has a significant impact on the wake structure seen behind the SUV, reducing wake length and increasing 'up wash'. This can be seen in [Figure 4](#), which shows the centerline velocity profile for both boundary conditions. Examining the streamlines in these images it can be seen that the vertical position of the vortex center behind the rear bumper shifts upwards in the case with the moving road boundary condition. This increases the (up wash) angle of streamlines originating from underneath the vehicle. The particles which impact the rear of the vehicle are advected by the mean flow, thus enhanced up wash due to the upwards shift of the vortex center behind the rear bumper will introduce more particles into the near wake.