

Modelling A-Pillar Water Overflow: Developing CFD and Experimental Methods

Adrian P. Gaylard, Michael Fagg and Mark Bannister
Jaguar Land Rover

Bradley Duncan, Joaquin Ivan Gargoloff and Jonathan Jilesen
Exa Corporation

ABSTRACT

Water accumulating on a vehicle's wind screen, driven over the A-pillar by a combination of aerodynamic forces and the action of the windscreen wipers, can be a significant impediment to driver vision. Surface water film, or streams, persisting in key vision areas of the side glass can impair the drivers' ability to see clearly through to the door mirror, and laterally onto junctions. Common countermeasures include: water management channels and hydrophobic glass coatings. Water management channels have both design and wind noise implications. Hydrophobic coatings entail significant cost.

In order to manage this design optimisation issue a water film and wiper effect model has been developed in collaboration with Jaguar Land Rover, extending the capabilities of the PowerFLOW CFD software. This is complimented by a wind-tunnel based test method for development and validation. The paper presents the progress made to date. The test method, based on installing a rain grid in a climatic wind tunnel, is described. Boundary condition data is extracted to aid CFD model development. This includes: flow similarity assessment against a full scale aerodynamic wind tunnel, water spray flux measurements, screen water distribution, and A-pillar overflow measurements. The CFD method is described and its predictions are compared to experimental data. This includes a simple model to represent the effect of the wiper system. Finally, areas for further improvement in both experimental and computational methods are identified.

CITATION: Gaylard, A., Fagg, M., Bannister, M., Duncan, B. et al., "Modelling A-Pillar Water Overflow: Developing CFD and Experimental Methods," *SAE Int. J. Passeng. Cars - Mech. Syst.* 5(2):2012, doi:10.4271/2012-01-0588.

INTRODUCTION

When designing a vehicle it is important to consider the management of water on the surface of the vehicle in order to ensure that the driver's vision is not impaired. The A-pillar design is central to preventing the reduction of driver visibility due to the presence of water rivulets on the side glass. Currently, it is only possible to test the effectiveness of different A-pillar designs in suitable full scale wind tunnel facilities. This is problematic because it requires the project to have progressed to a point where a representative full scale test property can be constructed. Additionally, as the A-pillar design also has a large impact on wind noise production an iterative process between acoustic and water management wind tunnel testing is required before a final A-pillar design can be produced. This generally requires testing in two separate facilities, raising cost and availability issues. These factors make the development of a numerical alternative appealing.

In this study, Jaguar Land Rover has collaborated with Exa Corporation in the development and validation of a numerical approach for simulating thin films needed for water management testing. The Lattice Boltzmann Method (LBM) flow solver was extended to include time-accurate, transient particle tracking in the flow field, as well as advection of surface water film on the vehicle. In addition, the water film simulation was enhanced with a wiper *effect* model to simulate the forcing effect of windshield wipers on the surface film. In order to validate this new simulation tool a wind-tunnel based test method, which uses a rain grid within a climatic wind tunnel is being developed. The details of the experimental method are described in the method section of this paper along with information on the numerical methods used.

The development work carried out to this point has been based around an SUV, the Range Rover Sport (RRS).

In order to correctly simulate the water droplet content of the onset flow provided by the rain grid, many input

parameters are required, such as the diameter of the particles produced by the spray nozzles, the initial velocity of these droplets and the topology of the spray (hollow core, full cone, flat etc.). While nozzle catalogues can provide guidelines as to the ranges to use for these inputs many of them are affected by variables specific to the test environment. This meant that before performing simulations with the vehicle *in situ* initial tests with an empty tunnel were conducted to verify the test boundary conditions and their representation by the numerical simulation. These tests consisted of both experimental and numerical runs whereby the particle injection properties were tuned. These tests are detailed in the empty tunnel section of this paper.

In addition to the calibration of the particle injection properties, the degree to which the climatic wind tunnel needed to be simulated was determined. In order to ensure that all of the important features of the tunnel were included, experimental testing using pressure taps and pitot rakes was conducted for test vehicle both in a full scale aerodynamic wind tunnel and within the climatic wind tunnel. Equivalent CFD simulations were also performed. The details of the wind tunnel model verification are detailed in the section on Climatic Wind Tunnel Simulation Verification.

After validating the aerodynamics simulation of the wind tunnel and the rain grid, full simulations of the water spray interacting with the vehicle were conducted. These simulations were conducted for two different A-pillar configurations and two wiper speed settings and compared against experimental results in order to demonstrate the ability of the new code to simulate complex water management problems.

PREVIOUS RELEVANT WORK

The literature on surface water film management for vehicles is limited. To date, the main theme in the work reported is the management of water on the front side glass and door mirror glass. This has been approached using a range of experimental and computational methods.

In a ground-breaking paper Karbon and Longman [1] 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. A novel contribution 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 [2] assessed door mirror glass contamination, using 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. Bannister [3] provided an important 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 [4] measured droplet size distribution within wakes, enabling the development of a wind tunnel based water and soiling contamination simulation using a bespoke turbulence generator and rain grid. They demonstrated the efficacy of this system in assessing front side-glass contamination. This work uses their rain grid design as a starting point.

Bouchet et al [5] used specially designed rain collectors in the Jules Verne climatic wind tunnel to determine the intensity spatial gradient in the wake of a vehicle. The work reported in this paper uses one of the rain collector designs described by these workers.

Foucart and Blain [6] reported a multi-stage computational method for simulating rain collection on the windscreen and subsequent movement of a water film over the A-pillar 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 [7] 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 [8] 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.

Other vehicle surface contamination issues reported in the literature include bonnet water film [9], which built on the earlier development of a bespoke test facility [10]).

Progress in the development of experimental methods continues, as evidenced by Aguinaga and Bouchet [11] who illustrate the current ‘state of the art’ in test techniques for side-glass contamination measurement.

METHOD DEVELOPMENT

TEST VEHICLE

The vehicle selected for this study was the Land Rover, Range Rover Sport. This SUV is 4.783 m long, with a 2.745m wheelbase. The roof height is 1.789 m; vehicle width (mirrors deployed) is 2.158 m with front and rear track of 1.605m and 1.612m, respectively. The projected frontal area is 2.99 m² and has a drag coefficient of 0.37. Experimental work was carried out on a super-charged production vehicle; numerical simulation was based on a production-representative CAD model.

EXPERIMENTAL

Experiments were conducted in a climatic wind tunnel (MIRA CWT2). This simple and economic facility was locally available and able to cope with large quantities of water in the test section. However, lack of turning vanes on

the flow path corner directly behind the vehicle necessitated opening an access door behind the vehicle. A rain grid was mounted 2.3m ahead of the nozzle outlet with the centre of the rain grid 1.27 m above the floor. The centreline height of the rain grid was selected to correspond with the height of the centre of the external rear-view mirror glass. The rain grid was 0.684 m wide and 0.855 m high with 13 nozzles, based on a previous design by Borg and Vevang [4]. The arrangement of the nozzles can be seen in Figure 1. All of the nozzles used in this experiment were hydraulic atomizing spray nozzles from Spraying Systems Co. The top eight nozzles were 1/4M-SS2 nozzles while the bottom five nozzles were 1/4M-SS8 nozzles. The manufacturer's data (Spraying Systems Co., 2007) suggests a median volume diameter DV0.5 (i.e. diameter where 50% of the total volume (or mass) of the spray comprises drops with diameters larger than the median value and 50% smaller than the median value) of 1.6×10^{-4} m; though this does not account for the effect of the air flow.

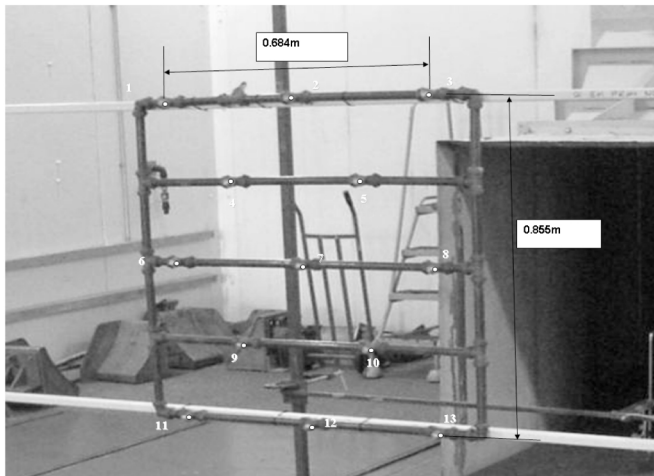


Figure 1. Rain Grid Geometry

During all experiments the test wind speed of 100 km/h (60 mi/h) was attained prior to starting the rain grid. The rain grid was connected to the water mains via a Flowquip FT3DS turbine flow meter, a Contrec Model 202D display, and a flow control valve. The water mains provided an approximate pressure of 2.07 bar and a flow rate of 3.5 litres/min. In order to eliminate difficulties with overshoot in the initial flow rate clamps were used to turn on and off the flow and the flow control valve was set prior to the experiment.

When tested, the vehicle was mounted 4.0m behind the rain grid with centreline of the steering wheel aligned with the centre of the rain grid with no yaw. This resulted in the vehicle being approximately 410 mm offset from the tunnel centreline. Prior to testing the A-pillar finisher and windscreen surfaces were checked for wettability ensuring that there was no contamination from hydrophobic substances such as wax-type materials. To further increase repeatability, these key surfaces were cleaned and rinsed before each test.

The A-pillar was wiped dry after each test to reduce the possibility of a pre-wetted A-pillar influencing water transport.

When wipers were used during testing they were set to maximum on the control stalk. Video analysis showed that the wiping rate was 1.18 wipes/sec.

NUMERICAL

Three dimensional aerodynamic CFD simulations using the Lattice Boltzmann Method (LBM) software were conducted for comparison against the experimental results. The grid used during the simulations containing a vehicle had a size and distribution appropriate for aerodynamics simulations (See [12] for example.). MIRA CWT2 does not have a moving floor, so the floor was fixed during all simulations. Simulations of the empty tunnel used a reduced mesh in which the high resolution regions needed to accurately resolve the vehicle were removed. To simulate the rain grid spray, time-accurate Lagrangian particle tracking of water droplets was used in the bulk flow. Droplets that hit the surface formed a water film. A transient thin film solver was used to compute the advection of the water film, along with the forcing effect of wiper blades. The transient flow data from the simulation of the air flow, bulk flow particle advection and water film surface advection was collected with an averaging interval of 10000 time steps for the bulk flow and 100 time steps for the centreline for the purposes of analysis and visualization.

Transient Lagrangian particle tracking was incorporated into the LBM solver using one-way coupling dynamics in which the particle momentum effect on the air flow is neglected. The thin film simulation calculated the mean velocity and thickness of a thin fluid film over arbitrary geometry. The film solver uses 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 modelling of the surface dynamics of the water droplets is based on the assumption of a thin film of fluid on top of 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 distribution of the film velocity field is computed based on the momentum balance using surface and volumetric forces described above. The thin film simulation approach used here is similar to that used in previous studies such as reference [13].

Simulation of the actual number of water particles leaving the nozzles into the fluid flow would be prohibitively costly, computationally; so the number of particles released was scaled down by a factor of approximately 45. This did not affect the surface film build up though because the volume of each particle hitting the surface was increased by the same scale factor.

The water droplet particles were injected into the flow through point sources located at the same location of the nozzles in the experiment. The particles were assigned diameters based on a Gamma distribution with a mean of 1.73×10^{-4} m and a variance of 7.94×10^{-9} m. The nozzles were given random initial velocities based on a uniform distribution within the ranges defined by the nozzles flow rates, cone angles and orifice diameters. Particle data was collected at a sampling frequency of 250 Hz both within the fluid domain and on the sampling surfaces.

EMPTY TUNNEL SIMULATIONS

EXPERIMENTAL

In order to ensure that the computational modelling of the rain grid was adequate, tests were conducted with an empty test section. During these empty tunnel tests rain gauges (See Bouchet et al [5]) were used to collect water at 36 discrete locations on a plane located 5.63 m downstream of the rain grid. These measurements were achieved by moving an array of four rain gauges to 9 different locations. [Figure 2](#) shows the rain gauge array at the centre location.



Figure 2. Rain Gauge Array at Centre Location

NUMERICAL

During the numerical simulations of the empty tunnel the geometry of the rain gauges was not included in the simulation. Hence, in this work, we have assumed that the rain gauges do not significantly perturb the flow field. Not including these devices in the simulations also enabled a course volumetric grid to be used. The smallest volumetric element (voxel) in these simulations had an edge length of 0.04 m. The region over which these were applied is shown in [Figure 3](#).

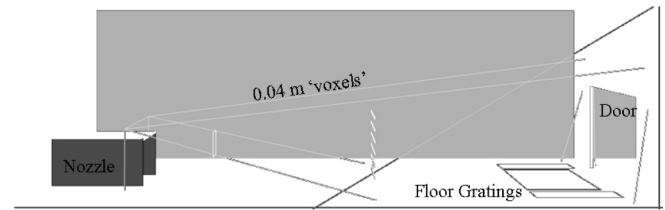


Figure 3. Schematic Showing Region of Highest Spatial Resolution

The wind tunnel nozzle was modelled, with the computational inlet contained within it, as shown in [Figures 3](#) and [4](#). Four different outlets were included in the simulation. Three of these outlets are shown in outline on the floor of the wind tunnel. These outlets model the vertical turn behind the vehicle position. The open access door in the test section is also modelled, with an outlet downstream. The mass flow through the three vents in the floor balances with the mass flow supplied by the inlet. This translated into a flow velocity of -5.47 m/s in the vertical (z) direction. The fourth outlet was located at the far end of the domain in the x-direction, 143 m downstream of the nozzle. This outlet was specified as a static pressure boundary condition and did not see any significant flow.

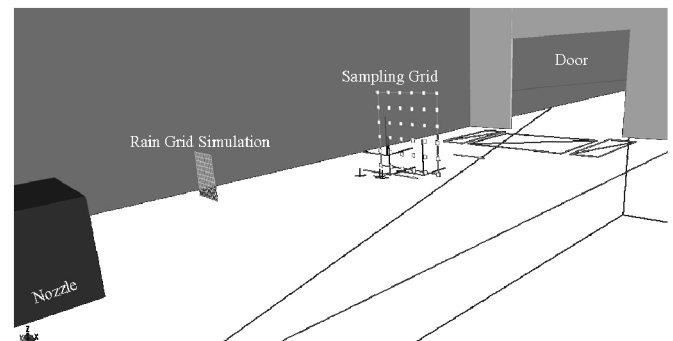


Figure 4. Schematic Showing Simulated Rain Grid and Sampling Grid

The location of the rain and sampling grids was the same as their experimental locations and can be seen in [Figure 4](#). The sampling grid was made up of 36 different sampling surfaces which were located at the same locations as the rain gauges in the experiments. The sampling surface area at each location was the same size as the rain gauge inlets. This numerical set-up allowed the same post processing to be applied to both the numerical and experimental data.

RESULTS

The data captured is presented in [Figure 5\(a\)](#). Here the grid nodes of the contour plot correspond to the 36 measurement locations. It can be seen that the largest water volume flux was just above the minimum rain grid height. This was expected, as the top of the rain grid was slightly higher than the exit of the air jet nozzle. This meant that