

Application of CFD to Predict Brake Disc Contamination in Wet Conditions

Lara Schembri Puglisevich, Adrian Gaylard, and Matthew Osborne
Jaguar Land Rover

Jonathan Jilesen and Adriano Gagliardi
Exa Corporation

ABSTRACT

Brake disc materials are being utilised that have low noise/dust properties, but are sensitive to contamination by surface water. This drives large dust shields, making brake cooling increasingly difficult. However, brake cooling must be delivered without compromising aerodynamic drag and hence CO₂ emissions targets.

Given that front brake discs sit in a region of geometric, packaging and flow complexity optimization of their performance requires the analysis of thermal, aerodynamic and multi-phase flows. Some of the difficulties inherent in this task would be alleviated if the complete analysis could be performed in the same CAE environment: utilizing common models and the same solver technology.

Hence the project described in this paper has sought to develop a CFD method that predicts the amount of contamination (water) that reaches the front brake discs, using a standard commercial code already exploited for both brake disc thermal and aerodynamics analysis.

This project builds on the capabilities of the Lattice Boltzman CFD code, Exa PowerFLOW, using its inherently unsteady simulation capability to capture the flow around the front brakes of an SUV (Range Rover Sport), sliding mesh to represent wheel rotation; along with Lagrangian Particle Tracking and Thin Film modelling to account for both airborne spray and surface water, respectively. Validation data is provided from experiments carried out in the FKFS Thermal Tunnel.

The numerical method is seen to reproduce the trends from the wind tunnel experiment.

CITATION: Schembri Puglisevich, L., Gaylard, A., Osborne, M., Jilesen, J. et al., "Application of CFD to Predict Brake Disc Contamination in Wet Conditions," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(2):2016, doi:10.4271/2016-01-1619.

INTRODUCTION

Vehicle front brakes are placed in a region of packing, geometric and flow complexity. They are co-located with the front wheel and tyre in the wheel arch cavity. Flow deflectors may sit upstream of them diverting direct air flow, whilst cooling flow may be introduced into the wheel arch after having passed through the cooling pack. Their flow environment is complex: the motion of the vehicle delivers a yawed onset flow, whilst the wheel imparts rotation to the flow.

Yet, to dissipate heat generated by the frictional contact of braking and maintain their function, they must receive significant cooling flow.

Also, to avoid having their performance degraded they must also be shielded from environmental water [1,2]. This is usually achieved using a dust shield, which can reduce the flow of cooling air to the discs.

Recent trends in brake disc design have favoured weight reduction, to aid compliance with CO₂ emissions targets, along with improving NVH and dust issues. These goals have been achieved through the

use of discs which both require a greater cooling air flow and are more sensitive to surface water contamination.

This can require new cooling strategies which need to be optimized to ensure that aerodynamic drag is not compromised in the process, eroding any CO₂ emissions benefits delivered through brake disc design.

Against this backdrop, the current study seeks to provide a methodology which allows for the analysis of brake cooling performance, the aerodynamic drag effect and accumulation of surface water on the brake discs in the same CFD solver environment.

The commercially available Lattice Boltzman (LBM) CFD code, Exa PowerFLOW, has been shown to be capable of predicting vehicle aerodynamics performance [3] and has also been used to design brake cooling strategies [4]. Further, recent developments have added the capability to simulate both airborne spray and surface water [5]. Hence, this is the CFD approach selected for this work.

PREVIOUS RELEVANT WORK

Tyre Spray

In this work the vehicle's own tyre is considered as the source of contamination; often referred to as self-soiling [6, 7]. Relatively little data has been published on the characteristics of the spray generated by passenger car tyres.

Bouchet *et al.* [8] made measurements 1 m behind the rear wheels of a test vehicle in the CSTB "Jules Verne" environmental wind tunnel. In this case average (mode) droplet diameter varied little between 80 and 140 μm , being around 0.2 mm; though higher speeds generated a greater proportion of smaller droplets.

In contrast Kuthada and Cyr [6] examined key parameters for the topology of a tyre spray generated by a wheel in isolation. Their approach mounted the wheel on a dynamometer roller in the test section of the FKFS Thermal Wind Tunnel. Water containing a UV fluorescing dye was injected onto the roller and laser light sheet was used to visualize the droplet path. A CFD simulation was carried out using the same commercial Lattice Boltzmann solver used in this work. Lagrangian Particle Tracking was used to model the tyre spray. The particle diameter was adjusted until the computational spray topology matched experiment. Consistent with Bouchet *et al.*, particle diameters of 0.2 mm provided the best match. This work was revisited in the more comprehensive experiments and simulations reported by Spruss *et al.* [9]

This literature provides the basis for the tyre spray model used. Given that the complex process of surface water pick-up by the tyre tread, and subsequent release to produce a spray with a distribution of droplet sizes, is to be represented by a simplified particle-release approach, this work can also be seen as a validation of the *local* spray characteristics provided by this model.

Surface Contamination Simulation

In the broadest sense of the term, this is not the first investigation of brake related surface contamination. Previously Gaylard *et al.* [10] used Lagrangian Particle Tracking and the same solver technology to assess soiling of wheel hubs caused by dust particles generated by braking. This issue was explored further using a combination of CFD and experiment (PIV) by Augsbury *et al.* [11].

However, this is the first study to examine the contamination of brake discs with water provided by the local tyre spray. As such it draws on the approach developed to simulate the soiling of the body sides and rear of passenger cars.

Attempts to simulate surface contamination on cars using particle modelling can be traced back to Yoshida *et al.* [12], who used a "Dirt particle path" method to calculate body side and rear soiling patterns on an SUV. However, the flow field was approximated by a RANS "steady state". Roettger *et al.* [13, 14] pioneered the use of an unsteady flow field (LBM) for this type of simulation. However, they

took the time-average of the unsteady flow field and advected particles through this as a post-processing step.

Building on this Gaylard and Duncan [14] advected particles through a set of short-time averaged flow fields; summing to determine the resulting particle distributions on an SUV body side and rear surfaces. Hence, some element of flow unsteadiness was accounted for. A study fully accounting for the unsteady flow field was reported by Gaylard *et al.* [16]. This incorporated the particle tracking simulation within the LBM flow solver, directly coupling the particle and flow simulation.

The first combination of particle and thin film modelling applied to vehicle surface contamination simulation was reported by Karbon and Longman [17]. Lagrangian particle tracking was used to simulate rain impacting on a vehicle wind screen a thin (surface) film model allowed particle mass transfer to the surface, modelling water accumulation and movement onto the front side glass. However, this was with a RANS (steady-state) CFD representation of the aerodynamic flow field.

Other workers have sought to use the combination of particle and thin film modelling to address surface water management issues at the front side glass, using a RANS "steady state" flow field [18,19,20]. In contrast, Foucart and Blain [21] used a LBM solver to provide unsteady aerodynamic boundary conditions for a sub-model of a car windscreen and front side glass. Airborne water droplets were modelled using Lagrangian particle tracking, and the surface water dynamics were represented using a thin film model in a specifically modified URANS CFD code.

A combination of unsteady flow field, and fully coupled particle and thin film modelling was used by Gaylard *et al.*, [16]. A model for "droplet stripping" from the surface film due to aerodynamic shear was also included. The approach used in this A-post overflow study is the direct precursor of the methodology used in this work.

It was further extended [5, 22] to include the break-up of airborne particles under aerodynamic shear, along with splashing on impact with surfaces. The current study takes this approach and adds sliding mesh representation of wheel hub rotation to provide a comprehensive simulation of the self-soiling of front brake discs on an SUV.

The following sections outline the experimental and computational methods, before the results are presented and discussed.

EXPERIMENTAL METHOD

Test Vehicle

A 14MY Land Rover, Range Rover Sport was used as the test vehicle in this study. This is a full size SUV with key dimensions of 4850 mm x 2073 mm x 1780 mm (L x W x H). Its wheelbase was 2923 mm, with front and rear track of 1690 mm and 1685 mm, respectively.

It has a projected frontal area (A) of 3.04 m². The principal aerodynamic coefficients (at zero yaw) for this type of vehicle are:

- $C_D = 0.36$
- $C_{L_F} = 0.09$
- $C_{L_R} = 0.02$

This particular example had a 3.0l V6 Gasoline engine, with an 8 speed automatic transmission. The wheels were a ten-spoke alloy design fitted with 255/55 R20 Michelin tyres. Front and rear trim heights were set to 815 mm.

Test Facility

The experimental approach used closely follows that described by Spruss *et al.* [1, 2]. Their approach used a combination of twin-axle dynamometer in the FKFS Thermal Wind Tunnel [6], a representation of spray generated by the tyre and advected in the onset flow, to provide the boundary conditions for the determination of brake disk soiling, braking force and adhesion coefficient. This work uses the same wind tunnel and dynamometer combination, but focusses on brake disc contamination due to tyre spray alone.

The wind tunnel is an open-jet Göttingen type, with both front and rear axle dynamometers. The working section has a jet nozzle with selectable areas of either 4 m² or 6 m² and the tunnel can attain a maximum velocity of 240 km/h. The tunnel layout is shown in Figure 1.

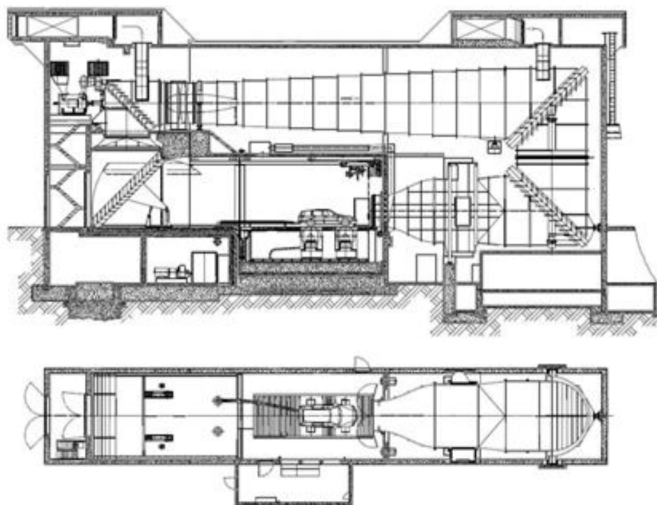


Figure 1. FKFS Thermal Wind Tunnel Layout [6]

The vehicle was tethered via two cables at the front tow eye (Figure 2) and three cables at the rear tow eye, with exhaust extraction tubes connected to the tail pipes. An accelerated self-soiling condition is created via water spray ejected at a constant flow rate onto the front of the dynamometer contact patch area as show in Figure 3. The water is mixed with a UV dye of known concentration that fluoresces, re-emitting radiation in the visible part of the electromagnetic spectrum once excited with a specific UV radiation frequency range [2].

To minimize reflections and provide good contrast for recording the data, the brake disc and assembly were painted matt black (Figure 4).

To prevent friction at the brake disc surface from affecting the measurements, and to facilitate the CFD modeling, the brake pads were offset from the brake surface by 3 mm.



Figure 2. Test Vehicle Setup on the Dynamometers at the FKFS Thermal Wind Tunnel

Each test run begins with cleaned brake discs installed at the front axle with the water spray initiated once the test velocity of 80 km/h is reached. Once initiated, a constant flow rate is maintained and a self-soiling condition is created, lasting for 8 minutes. Once the test run is complete and the wheels have come to rest, the front axle is jacked up to remove the front wheels. The brake discs are then extracted whilst still wet, and placed onto a measuring rig. The contamination pattern remains undisturbed because the adhesive forces acting on the fine beads of water are greater than the gravitational and inertial forces during transportation. Each side of the disc is photographed in a darkened room under UV illumination. Analysis of these images then permits the assessment of the extent and depth of residual water left on the discs.

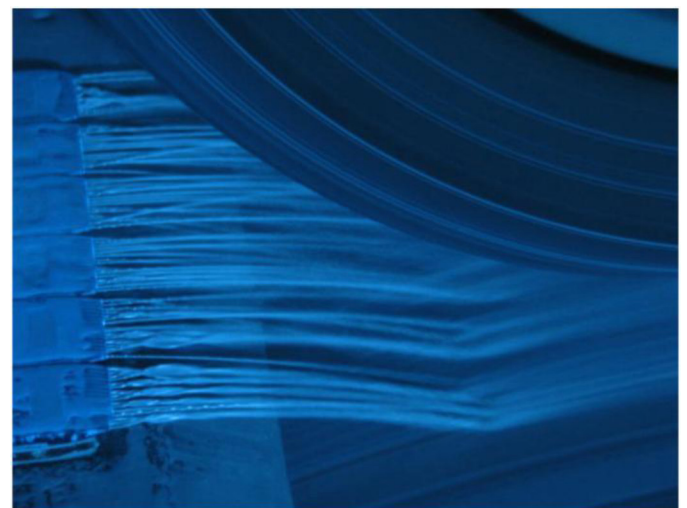


Figure 3. Spray Nozzles at the Front Wheel Dynamometer [22]