Investigation into the dynamics of wheel spray released from a rotating tyre of a simplified vehicle model

Anton Kabanovs\textsuperscript{a}, Andrew Garmory\textsuperscript{a,}\textsuperscript{*}, Martin Passmore \textsuperscript{a}, Adrian Gaylard \textsuperscript{b}

\textsuperscript{a} Department of Aeronautical and Automotive Engineering, Loughborough University, Epinal Way, Loughborough, LE11 3TU, United Kingdom
\textsuperscript{b} Jaguar Land Rover, Banbury Road, Gaydon, Warwick, CV35 ORR, United Kingdom

\textbf{A R T I C L E   I N F O}

Keywords:
Vehicle aerodynamics
Unsteady
Surface contamination
CFD
IDDES
Tyre spray
Generic SUV body

\textbf{A B S T R A C T}

Accurate prediction of vehicle soiling is an important step towards being able to understand and reduce this problem. Previous work has shown that eddy resolving CFD methods are capable of predicting the soiling pattern on the surface of automotive geometries when a known spray source is used. Here the influence of the wheel, ground and spray boundary conditions on a simulation of rear soiling of a generic SUV are investigated. The inclusion of rotating wheels led to a greater vertical distribution of the soiling pattern whereas the moving ground plane led to increased lateral distribution. The total soiling rate with moving wheels and ground, as well as with the offset between wheel and ground removed, was approximately 50\% higher than the experimental conditions. When spray was released from around the rotating wheel it was found that a large majority of the parcels which ended up on the base originated from close to the contact patch, indicating that this is the most important region for the tyre spray model. A model based on a measured distribution of droplet sizes from downstream of the wheel gave good agreement with previous experimental work for the spray topology around the contact patch.

1. Introduction

Management of road vehicle surface contamination from rain and spray is of increasing interest to automotive manufacturers. Apart from compromising aesthetic appearance, dirt can transfer to the hands and clothing of vehicle users. More importantly, deposition of contaminants can reduce vision through windows, the visibility of lights and the function of camera and sensor systems. The development of an accurate simulation methodology to predict contamination at an early design stage is important for vehicle manufacturers. A recent review of the topic can be found in Gaylard et al. (2017b). Due to the practical nature of this problem much of the published work has naturally concentrated on the on-road behaviour of real vehicle geometries (see, for example, Jilesen et al. (2017)). However to gain a more general understanding of the factors controlling the contamination of road vehicles the use of simplified vehicle geometries, representative of a particular class of vehicle, offer several advantages. Trends in aerodynamics and soiling can be attributed to key features of the general design rather than styling details that may not be relevant for other vehicles. Most importantly soiling patterns can be obtained in repeatable, well defined, conditions for which aerodynamic data for velocity, pressure and force coefficients are known.

This approach has recently been taken in Kabanovs et al. (2017a,b, 2018) in which experimental and computational fluid dynamics (CFD) simulation data were compared for the soiling of the rear surface, or ‘base,’ of a quarter scale generic SUV body in a wind tunnel test. This approach has shown the importance of correctly capturing the unsteady nature of the spray transport in the wake (Kabanovs et al., 2017a) and the role that evaporation and coalescence can have on spray dynamics (Kabanovs et al., 2018). The experimental setup in these studies differed from on-road conditions in two important respects; stationary wheels and ground plane were employed and the contaminant was introduced using a pressure atomising spray positioned behind the rear wheel. These conditions (in particular the spray) allow for well defined and repeatable boundary conditions to the problem but may cause important differences from both real world conditions and testing in climatic tunnels which commonly use rotating wheels but stationary ground planes (Gaylard et al., 2017b). In this paper we investigate the influence of the wheel and ground boundary conditions and the method of introducing spray into the simulation. This aims to provide insight into the correlation between test and real-world conditions as well as the physics of vehicle soiling and how this can be accurately modelled in an efficient manner.

There has been an extensive experimental and computational work carried out to analyse the influence of rotating wheels and a moving
ground plane on the aerodynamics of car-like bodies and production vehicles (Wiedemann, 1996; Eloffson and Bannister, 2002; Wäschle, 2007; Landström et al., 2009; Koitrand et al., 2014; Forbes et al., 2017). Most of the previous work has shown that the addition of rotating wheels and a moving ground reduces aerodynamic drag. According to Wäschle (2007), a significant part of drag reduction due to rotating wheels is attributed to the interference effects between the underbody flow and the wakes of the rear wheels. The wide wakes produced by stationary wheels can impede the flow expansion in the rear diffuser, increasing drag. This effect is reduced when the rear wheels rotate, as this makes the wheel wakes shorter and narrower, also adding to the flow up-wash (Koitrand et al., 2014). The rotation of the rear wheels also increases pressure on the base as the mass flow entering the wake is increased (Forbes et al., 2017). The influence of rotating wheels on spray dynamics has not been directly studied due to the common practice of releasing spray from tyre surfaces, hence making it obligatory to model rotation in the first place. This agrees with the typical experimental conditions used in soiling tests, in which the wheels are usually set in motion while the ground is kept stationary. Nevertheless, any change in the interaction between the base and the rear wheel wakes will inevitably affect the rear-surface contamination (Jilesen et al., 2017). For example, Lajos et al. (1986) estimated the mud concentration on the base of a bus to increase by 16–20%, caused entirely due to wheel rotation.

The influence of a moving ground has been studied by Landström et al. (2009), who noticed a significant increase in flow in-wash towards the centreline of the model behind the rear wheels. This also resulted in a reduced aerodynamic drag of a passenger hatchback vehicle. According to Jilesen et al. (2013), the introduction of a moving ground in simulations with a production SUV vehicle increased flow up-wash at the rear and reduced the length of the vehicle wake, letting spray maintain more of its momentum and more readily reach the base. This study also showed that the wake is laterally more unsteady when the moving ground is considered. Lajos et al. (1986), who carried out a study on mud deposition on bus geometries, reported that the effect of the moving ground is very much dependent on the body shape and the ground clearance. Ground motion increases the underbody flow rate, which in turn enlarges the quantity of mud entraining into the near wake. On the other hand, it can significantly change the flow pattern in the wake and decrease mud deposition. For example, they noted that the rear stagnation point can shift down due to ground motion. It is assumed that this would cause deposition either to shift or expand vertically. For the two bus geometries, Lajos et al. (1986) reported 34% increase and 18% reduction in the deposition when the ground motion was considered.

The four primary mechanisms for water ejection by a rotating tyre have been identified by Weir et al. (1978), who studied spray generated by heavy goods vehicles. The four mechanisms are bow wave, side splash wave, tread pickup and capillary adhesion. The first two categories are types of splash and the droplets that they generate are usually large in size. The bow wave and side splash wave generally do not contribute to vehicle surface contamination as the droplets follow a ballistic trajectory, impacting either the under-body surfaces or falling back to the road surface. The spray primarily responsible for self-soiling is generated by tread pickup and capillary adhesion. The tread pickup mechanism describes the process in which water is passed through the tread grooves and is ejected early in the tyre rotation, forming droplets that range from small (less than 1 mm) to reasonably large (3–5 mm) (Weir et al., 1978). According to Koesler et al. (1957), who studied spray fog of free and contact rotation of the spray, water was retained on an angle less than 30° from the ground. Some portion of water is retained on the tyre as a result of capillary adhesion and is stripped off due to an incoming air-stream later in tyre rotation, generating a very fine spray containing 1% of the water volume picked up by the tyre tread (Weir et al., 1978).

The droplet size distribution produced by rotating tyres mounted on a full-scale vehicle has been studied by a number of researchers. Experiments were carried out by Shearman et al. (1998) and Borg and Vevang (2006), aiming to quantify spray generated by moving lorries. The measurements were taken some distance from vehicle rear surfaces (up to 50 m), suggesting that the source of third-party contamination was of main interest. Shearman et al. (1998) used a high-resolution laser-based system (PDA) to measure the distribution of droplet diameter in a spray generated by a lorry moving on a wetted test track. Borg and Vevang (2006) used hydrophobic plates to identify droplet distribution in the wake at different distances behind a moving track. Unlike Shearman et al. (1998) and Borg and Vevang (2006), who focused on quantifying spray responsible for third-party soiling, spray measurements obtained by Bouchet et al. (2004) at a distance of only 1 m behind the rear wheels of a test vehicle provide important insights into the nature of spray responsible for self-soiling.

The most widely used numerical tyre spray model in recent work has been proposed by Kuthada and Cyr (2006). The model was validated against experiments, carried out with a free rotating isolated full-scale wheel mounted on a stub axle, with water supplied to the roller at the contact patch. The spray structure generated by the wheel rotating at 80 km/h was visualised using laser light sheet illumination. The diameter of particles injected from the tyre surface in the numerical model was iteratively varied until the appropriate size was found, such that the topology of the modelled spray matched the one determined in the experiments. Later work by Spruss et al. (2011) used a spray size that was allowed to vary around the circumference of the wheel although details of this size distribution were not included. According to Kuthada and Cyr (2006), the experimental spray pattern was matched when a particle diameter of 0.2 mm was used in the numerical model, thus matching the experimental findings in Bouchet et al. (2004), who used the same flow conditions. As a result, this study provided a modelling approach for numerical simulations of vehicle self-soiling that has been widely cited by many researchers and car manufacturers (Gaylard and Duncan, 2011; Jilesen, 2013; Jilesen et al., 2013, 2017; Gaylard et al., 2014; Hu et al., 2015; Schembri Pugliese et al., 2016). However, although the work of Kuthada and Cyr (2006) propose a particle diameter of 0.2 mm, these studies typically employ a diameter of 0.165 mm without providing an explanation for the reduction in size. It should be also emphasized that this approach of modelling tyre spray has limitations. For example, using a single particle to characterise both the tread pickup and capillary adhesion mechanisms in the tyre spray is a significant simplification.

In this paper a CFD simulation of an experimental test case for a quarter scale generic SUV is carried out to provide a validated baseline case before further simulations are carried out to investigate firstly the effect of wheel and ground boundary conditions, when a fixed spray is used, followed by simulations using a rotating wheel with three different spray injection models. The three models are:

- Particle size varying by circumferential release position to match spray topology of an isolated wheel based on the approach of Kuthada and Cyr (2006); Spruss et al. (2011).
- Releasing the measured distribution from Bouchet et al. (2004) from all circumferential positions.
- Using a single droplet size based on the peak diameter from the distribution of Bouchet et al. (2004).

This is done to observe the differences in predicted soiling pattern and rate and to identify where droplets depositing on the vehicle base originated from on the wheel. The paper is structured as follows: the test case and methodology is outlined in Section 2, the wheel spray models are introduced in Section 3, results for the SUV soiling simulations and discussion are presented in Section 4 and the conclusions are in Section 5.

2. Computational methodology

2.1. Geometry and test cases

The physical model used in this study and shown in Fig. 1 was a generic quarter scale SUV model with the roof taper and bottom diffuser
set to 0° and 10°, respectively. The model has smooth underbody and a ground clearance of 0.065 m in the baseline configuration. The model is 1.04 m long, 0.41 m wide and the rear surface is 0.295 m high. Further details about the model can be found in Wood et al. (2015). It has been developed at Loughborough University to study the aerodynamics, and later soiling characteristics, of SUV type geometries. Experimental data for pressure and velocity has been collected for various configurations using a fixed ground plane and stationary wheels.

Soiling test data has also been collected using a spray of UV doped water injected from behind one of the rear wheels. The UV light emitted from the contaminated surface can then be related to the level of soiling on that surface. Currently quantitative data about the mass deposited on the surface is unavailable. Instead the data is used to give a relative soiling ‘intensity’ distribution by normalising the UV level detected by the maximum on the surface. Further details of the method can be found in Kabanovs et al. (2017a, b, 2018).

The configuration used here was selected over model variations with different taper and diffuser angles due to its overall resemblance to the 2005MY Land Rover Discovery 3 (LR3), which has been previously used in soiling studies (Gaylard and Duncan, 2011; Jilesen et al., 2013) due to its susceptibility to rear soiling. In common with the chosen SUV configuration, LR3 has a relatively square rear surface with a moderate bottom diffuser.

As experimental data is available for a case with stationary wheels and fixed ground plane this is used as the baseline case to validate the modelling approach used. In the experiment the model is supported on pins passing through the bottom of the wheels leaving a small gap or stand-off between the floor and the flattened base of the wheel and this is also included in the baseline simulation. To investigate how aspects of the wind tunnel setup may affect the soiling behaviour compared to on-road conditions four numerical cases were considered. This allows for a systematic transition from the baseline case to a case more realistic of on-road conditions which consider wheel rotation and ground motion. When the effect of rotating wheels is investigated, the model is lowered to ensure contact between the wheel flats and ground plane, reducing the ground clearance by 0.004 m. All numerical cases are given in Table 1.

### Table 1

| Case | Abbreviation | Description |
|------|--------------|-------------|
| 1    | WSO          | Wheel stand-off (Baseline case). |
| 2    | FGFW         | Fixed ground and fixed wheels in contact with the ground. |
| 3    | FGWRW        | Fixed ground and rotating wheels. |
| 4    | MVGRW        | Moving ground and rotating wheels. |

- **Fig. 1.** Generic SUV model with 0° rear roof taper and 10° bottom diffuser. The surface referred to as the base is indicated in the figure.

- **Fig. 2.** Wheel and ground boundary conditions in all studied cases. The position of the spray injector when a point spray is used is also indicated.
2.2. Numerical set-up

The numerical approach is the same as used in Kabanovs et al. (2017a, b, 2018) which have shown good agreement with experiment for flow field and contamination patterns. Simulations were performed using the OpenFOAM CFD package. The turbulent flow of the gas phase was computed using the IDDES approach, with the RANS branch of this hybrid method making use of the Spalart-Allmaras turbulence model. The all-\( y^+ \) approach, suggested by Spalding (1961), was used to model the boundary layer. This uses the log-law to set boundary conditions for velocity and turbulence properties if the cell is found to have a ‘high’ \( y^+ \) but uses a smooth blending function between this and a low \( y^+ \) (viscous sub-layer) solution if the cell \( y^+ \) drops below the high \( y^+ \) regime. This is done as the computational requirements needed for high resolution of the turbulent wake and tracking a large number of parcels would make using a low \( y^+ \) approach expensive.

The computational domain and mesh topology were the same as those used in Kabanovs et al. (2017a). The wind tunnel working section, including divergence, is modelled and the total cell count is approximately 65M. The inlet boundary condition was set to 40 m/s to match the...
wind tunnel speed used in the experiment. Different combinations of boundary conditions were used on the wheels and floor to test the influence of the wind tunnel setup on the soiling behaviour. Each numerical case and its boundary conditions are shown in Fig. 2. In the fixed ground & rotating wheel and moving ground & rotating wheel cases, the rotational velocity boundary condition was applied to rotate
wheels at an angular velocity of 500 rad/sec. This is equivalent to a translational speed of 40 m/s, matching the inflow velocity. This is the most straightforward method of including wheel rotation in a CFD simulation which is generally used for solid, cylindrical wheels. For more realistic wheel shapes the Moving Reference Frame (MRF) (Wäschle, 2007; Landström et al., 2009) approach can be used, which attempts to model the added momentum generated as a result of unsteady mass flow through the rotating wheel. In order to simulate the moving ground plane in the moving ground & rotating wheel case, a velocity of 40 m/s was applied to the corresponding surface, matching the flow direction. The same approach to model wheel rotation and ground motion was also used by Forbes et al. (2017), who investigated the influence of these boundary conditions on the aerodynamics of the generic SUV geometry with a different diffuser angle.

Following Kabanovs et al. (2017a, 2018) the dispersed phase is simulated using Lagrangian parcel tracking in OpenFOAM, where each parcel carries the equivalent mass of a number of droplets of a particular size. The Lagrangian solver was modified to allow further post-processing including recording the starting position of the parcel when released from the surface of the wheel. Parcel trajectories are calculated...
concurrently with the unsteady flow solver and the modelled forces acting on the parcels were sphere drag, shear lift and gravity. One-way momentum coupling only (i.e. the dispersed phase does not affect the continuous phase) was considered as two-way coupling is likely to have only a very small effect, as seen in Kabanovs et al. (2017a). Neglecting this effect allows multiple spray populations to be included in one simulation which allows a direct comparison of different spray models in the same computed turbulent flow field. Parcels are assumed to stick and deposit their mass on the surface when they collide with the geometry. The influence of this modelling choice is investigated to some extent in Section 4.3.3 by investigating the results of allowing parcels to bounce in the wheel arch. For each case, an initial single-phase simulation was run to compute 1 s of simulated time and therefore establish the flow field. Then, soiling simulations were run to compute 1.5 s of the deposition process. This was also the time period over which the forces and volumetric fields were averaged. This period is equivalent to 60 flow through times, based on the freestream speed and vehicle length, and previous work (Gaylard et al., 2017a) has suggested that the soiling rate is reasonably linear and the spray pattern unchanged over this time. The experimental data used in this paper does not include quantitative information on the soiling rate so only the soiling pattern can be combined. In addition, the spray mass ejected by a rotating wheel is unknown in this work. As such the absolute mass injected into the simulations is less important (provided this is kept constant between different cases) than the number of parcels used which must be chosen to balance computational cost against statistical accuracy. A total of 25 million parcels, with the same total mass, were released per second for each case. This was found to be the maximum that allowed results to be produced with the computational resources available.

3. Spray modelling

In this paper the aims are to investigate the influence of wheel and ground motion on the vehicle soiling dynamics and also to investigate the influence of how the spray is released into the air flow. To do this the dispersed phase will be injected into the simulations in four ways. The first is to match the experimental setup by simulating the spray injected behind the wheel to provide validation results and then apply each of the boundary condition combinations described in the previous section. The other three methods introduce the spray from the surface of the wheel in an attempt to mimic spray production on the road, based on models available in the literature which have been used for soiling studies. The three wheel spray models are used in simulations with rotating wheels but fixed ground, as this combination is typically found in climatic wind tunnels used for surface contamination tests. These are the most likely source of comparative data.

3.1. Point spray injection

To match conditions in the experiment, a cone injector is placed on the tunnel floor just behind the contact patch between the left rear wheel and the ground. Previous work by Kabanovs et al. (2018) used a spray size distribution measured in a separate PDA test and investigated the role of evaporation, breakup and coalescence on the spray development. It was found that there was a significant amount of coalescence predicted close to the injector and that including this gave improved agreement with the experimental soiling pattern. Further, it was found that injecting the ‘developed’ size distribution after the high coalescence region gave the same soiling prediction without the very high cost of including the coalescence model. Therefore, the same approach has been used here; the ‘developed’ spray from Kabanovs et al. (2018) is used as the injection spray boundary condition. This distribution has a peak droplet diameter of approximately 25μm. The spray boundary condition is applied at the position shown in Fig. 2.
3.2. Tyre spray model development

The three tyre spray models have been developed using simulations of isolated full scale wheels as experimental data is available for these for the spray topology as well as downstream size distribution measurements.

3.2.1. Wheel geometry

The geometry used to develop spray models is the same Michelin Pilot Primacy tyre as used in Kuthada and Cyr (2006) and later in Spruss et al. (2011). This is shown in Fig. 3. The geometry has some differences to the experimental geometry (in particular the axle system and tyre tread are removed and the hub has some changes). These differences are expected to give minor changes to results and it should also be stressed that neither of the numerical set-ups given in the original studies ((Kuthada and Cyr, 2006) and (Spruss et al., 2011)) fully replicated the experimental set-up.

3.2.2. Computational set-up

The computational domain used in simulations with an isolated rotating wheel is shown in Fig. 4a. The dimensions of the domain (3.88D × 3.88D × 16.8D, where D is the diameter of the wheel) were smaller than those in the original simulations in Kuthada and Cyr (2006) and Spruss et al. (2011), which is why the sides of the domain seen in Fig. 4a were set to a slip wall condition. The ground was stationary and the pressure at the outlet was set to atmospheric pressure. The inlet velocity was set to 22.2 m/s (80 kph) as this speed was used in the tests of both Kuthada and Cyr (2006) and Bouchet et al. (2004). The wheel, shown previously in Fig. 3c, was placed in the computational domain 7.3D away from the inlet and 9.5D upstream of the outlet. It was sliced at a height of 0.008D to produce an approximate contact patch with the ground. The rotational velocity boundary condition was applied to simulate wheel rotation and the translational velocity was equal to the air velocity, similar to Kuthada and Cyr (2006). The topology of the volume mesh, generated using the blockMesh and snappyHexMesh utilities, is shown in Fig. 4b. There are three refinement boxes and prism layers around the wheel. The all-y + wall treatment was used in simulations to model the near-wall region. The total number of cells is around 5 million.

Modelling of the continuous phase was performed using the same IDDES approach as for the main study. In each numerical case, the mean flow field was obtained by time-averaging the instantaneous flow over a time period equivalent to 375 times of flight past the wheel. The dispersed phase was modelled using the Lagrangian particle tracking approach.

### Table 2

|        | WSO (Baseline) | FG&FW | FG&RW | MVG&RW |
|--------|----------------|--------|--------|---------|
| C_d    | 0.434          | 0.439  | 0.444  | 0.433   |
| C_l    | 0.127          | -0.015 | 0.077  | 0.022   |

Fig. 14. Wheel stand-off case: experimental and numerical flow field shown on the vertical ZX planes.

Fig. 15. Soiling intensities for the wheel stand-off case: (a) Experimental UV level; (b) Mass deposited in CFD results using ‘developed’ spray. Intensity is defined as the percentage of the maximum in the image.

3.2. Tyre spray model development

The three tyre spray models have been developed using simulations of isolated full scale wheels as experimental data is available for these for the spray topology as well as downstream size distribution measurements.
A total of 26 particle emitter boxes (see Fig. 3c), distributed evenly on $270^\circ$ of the wheel surface, were used to release spray into the mean flow field, similar to Kuthada and Cyr (2006). The particles were released tangentially to the surface with the same initial speed as that of the tyre outer surface. Particles were released such that the total mass was evenly distributed between the emitter boxes. While this is unlikely to be true in real world situations the aim is to observe the influence of size and starting position on the likelihood of being deposited on the vehicle rear surface. The choice of parcel size was made according to the following three models.

3.2.3. Spray model 1

Here the basic numerical approach discussed in Kuthada and Cyr (2006) and Spruss et al. (2011) was replicated whereby the particle sizes were iteratively varied until the spray topology seen in the experiments was achieved. Unlike Kuthada and Cyr (2006), who suggested uniform $200 \mu m$ particles to represent spray, Spruss et al. (2011) correlated the particle diameter with the wheel translational velocity and particle's circumferential release position, hence demonstrating better agreement between the numerical and experimental spray. Unfortunately, quantitative information on particle sizes was not disclosed by Spruss et al. (2011). The current work followed the same approach and a specific particle diameter was determined for all 26 release positions around the tyre surface, leading to the numerical spray shown in Fig. 5b. It can be seen that the particle paths in the numerical model match the paths of droplets, shown in Fig. 5a, reasonably well. The maximum and minimum particle diameters in the spatial distribution are $604 \mu m$ and $308 \mu m$ respectively. If Stokes number is defined as,

$$Stk = \frac{\rho d_p^2 U_c L_c}{18 \mu L_c}$$

Fig. 17. Evolution of base total contamination.
where the subscript ‘p’ refers to reference values of the particle and ‘c’ to reference values of the continuous (or carrier) phase, then this leads to a Stokes number range of \(10^{-38.5}\). It should be emphasized that particle paths are very sensitive to their sizes since drag is proportional to \(d^2\) and inertia to \(d^3\). Therefore, a relatively small change in particle diameter can lead to a very different particle path.

Although the paths of particles modelled numerically agree well with the visible paths of droplets obtained experimentally (especially in the upper region), it can be seen that the experimental spray topology in the region behind the contact patch is different. The high-intensity regions highlighted in the experimental results in Fig. 5a indicate a very dense spray in this region, that is likely to be composed of many small droplets generated due to the primary breakup of the liquid sheet leaving the tyre surface. It is evident that the current numerical model is unable to accurately reproduce spray in these regions. This can be seen in Fig. 6 which shows that while the top region of spray is very well captured by the model, the region close to the ground is not reproduced (seen most clearly in Fig. 6c and d). It will be shown later in this work that this is the major drawback of the current approach, as smaller, and therefore more responsive droplets, generated at the contact patch play a primary role in the rear-surface contamination phenomenon.

### 3.2.4. Spray model 2

To the authors’ knowledge, there is currently no quantitative experimental data of spray generated immediately behind the contact patch, making it difficult to model spray in this region. However, the experimental data obtained by Bouchet et al. (2004) provide a good insight into the ‘developed’ tyre spray size distribution, as it was measured around 1 m behind a full-scale vehicle. The vehicle was a small van, as shown in Fig. 7, which is similar in size to an SUV. The size distributions measured at three speeds are reproduced for reference in Fig. 7. Unfortunately, nothing is known about the origin of each specific droplet size. Therefore, a complete distribution of particle sizes, measured at a speed of 22.2 m/s, is used in Spray Model 2, with particles released randomly from all 26 emitter boxes. The distribution was found by digitizing the data from Fig. 7b. Fig. 8b shows the topology of the generated spray. The experimental spray (Fig. 8a) obtained for an isolated wheel in Kuthada and Cyr (2006) is shown for reference.

### 3.2.5. Spray model 3

The third spray model uses the peak size of the distribution employed in Spray Model 2 and represents a typical approach used by vehicle manufacturers and many researchers (Gaylard and Duncan, 2011; Jilesen, 2013; Jilesen et al., 2013, 2017; Gaylard et al., 2014; Hu et al., 2015; Schembri Puglisevich et al., 2016) to model tyre spray. The peak size of this distribution is 200 \(\mu m\), which matches the size used in Kuthada and Cyr (2006) to reproduce spray generated by an isolated wheel.

Fig. 9b shows the topology of the numerical spray, obtained in this work by releasing 200 \(\mu m\) particles from the tyre surface of a representative wheel. It can be seen that this numerical spray is significantly different from that collected experimentally (Fig. 9a) or numerically in Kuthada and Cyr (2006). Although there are some differences in the wheel geometry used, Fig. 9b shows a larger disagreement with the findings presented in Kuthada and Cyr (2006) than can be explained by this.

---

**Fig. 18.** Instantaneous spray (iso-surfaces show cell-based volume fraction, value \(25 \times 10^{-5}\)) and air velocity streamlines released behind the left rear wheel (fixed ground & rotating wheel case); (a) Early (distance-wise) and (b) late (distance-wise) entrainment of spray.

**Fig. 19.** Paths of 220 parcels released and deposited within specified time periods (fixed ground & rotating wheel case); Iso-surfaces (cell-based volume fraction, value \(25 \times 10^{-5}\)) show spray at (a) 1.328 s and (b) 1.516 s.
Due to the substantial difference between the numerical results obtained in this work and those reported by Kuthada and Cyr (2006), a second solver Star-CCM+ (CD-adapco, 2014) was employed to verify the particle tracking algorithm implemented in OpenFOAM. Fig. 10 shows very good agreement between the spray patterns for a generic wheel using the two solvers. The wheel was not in contact with the ground, explaining the reason for the relatively undisturbed particle paths seen behind the wheel. This numerical case gives confidence in the implementation of the particle tracking methodology used in OpenFOAM as applied in this work.

3.2.6. Comparison of wheel spray models

It has been shown that Spray Model 1 is able to reproduce the capillary adhesion mechanism and match spray topology in the upper region while it does not account for the primary breakup of spray in the tread pickup process close to the contact patch. Fig. 11, which presents coverage areas for each spray model, shows that Spray Model 2 is able to better capture the topology of spray behind the contact patch of the wheel, while also adequately reproducing spray in the top region. This is because Spray Model 2 uses a complete droplet size distribution, which contains particles of various sizes. As a result, it is able to capture the capillary adhesion and tread pickup mechanisms reasonably well. Spray Model 3, on the other hand, is able to reproduce the tread pickup mechanism but it lacks bigger particles to match the topology of spray in the upper region. This suggests that overall the most representative tyre spray model presented in this work is Spray Model 2.

3.2.7. Scaling of wheel spray models

The wheel spray models have been developed by considering full scale isolated wheels. In order to test the models using the generic SUV, and compare results with the experimental spray case, it is necessary to adapt the models to the scale of the wind tunnel SUV geometry. The generic SUV’s wheels have a diameter that is 25% of the wheel used in this section. We have scaled the problem by maintaining constant Reynolds number,

$$Re = \frac{\rho_c U_c L_c}{\mu_c}$$  \hspace{1cm} (2)

where subscript ‘c’ refers to the continuous phase, and ratio of particle diameter to the continuous phase reference length, \(d_p/L_c\). If these are constant and the fluid viscosity and densities are constant, then from the definition of Stokes number in Equation (1) it can be seen that the Stokes number will remain the same and the scaled particle behaviour will be

Fig. 20. Difference in the mean longitudinal and vertical velocity components on the vertical plane at \(Y = -0.09\ m\).
the same. This is discussed in Kabanovs (2018) where it is also shown that the same particle behaviour is indeed maintained if \( Re \) and \( dp/L_c \) are kept the same.

To maintain the same Reynolds number as for the isolated wheel would require a tunnel speed of 88 m/s. If this is done and the parcel diameters are also scaled by 25\% then the same spray topology is predicted but this tunnel speed is not achievable. However Bouchet et al. (2004) provides experimental data for the full scale wheel at 11.1 m/s; to match the Reynolds number for this case at model scale would require a tunnel speed of 44.4 m/s which is very close to that actually used. Therefore for SUV simulations using spray model 2 the 11.1 m/s distribution shown in Fig. 7b is used with diameters scaled to 25\%. The uniform size used for spray model 3 is the peak diameter from the 11.1 m/s again scaled by 25\%.

Unfortunately no data is available from Kuthada and Cyr (2006) at other speeds than 22 m/s. Therefore to scale spray model 1 an approximate method based on the trend seen from the data in Bouchet et al. (2004) is used. In the latter experiment halving the speed from 22 to 11 m/s increased the ensemble averaged diameter by 16\%. Therefore in the SUV simulations here the circumferential size distribution at full scale is first reduced by 25\% to match the model scale and then increased by 16\% to account for the reduced non-dimensional speed. The size distributions are shown in Fig. 12.

4. Results

4.1. Validation of the baseline case

In order to validate the numerical method used in this work, results are compared for aerodynamic data and rear soiling distribution between experiment and CFD for the wheel stand-off case (i.e. fixed ground and a gap between non-rotating wheels and floor) with the point spray injection. This is the only case for which experimental data is available. Fig. 13 shows the time-average base pressure distribution, obtained experimentally and numerically. The numerical data shows very good agreement with experiment. The numerical prediction of the wake structure is also very accurate and can be seen in Fig. 14. CFD appears to predict a slightly weaker upwash from the diffuser, possibly being the reason for a small discrepancy in the location of the bottom vortex in Fig. 14d. On the other hand, the corresponding plane is located centrally and even a small asymmetry in the wake can make the comparison difficult. The predicted and experimental force coefficients for the wheel stand-off case can be seen in Table 2, which shows that the drag has been accurately predicted while there is a disagreement in the coefficient of lift. The reason for the discrepancy in lift has been already discussed in previous work (Kabanovs et al., 2017a; Forbes et al., 2017). It is thought that the clearance gaps around the pins connecting the wheels of the model to the balance affect the flow under the wheels and change the lift. The clearance gap was not included in the CFD setup.

Fig. 15 presents experimental and numerical rear surface soiling patterns, obtained for the baseline case. The base surfaces are coloured by soiling intensities. These are found by normalising the UV level in the experimental image by the maximum level in the image and the local mass deposited in the simulation by the maximum on the surface. These are generated by averaging the data over 5 mm cells in post-processing to produce smoothed soiling patterns. Fig. 15a shows the experimental pattern. In this study, the soiling experiment was carried out over a 57-s long soiling test. This time was long enough to obtain a well-developed pattern but at the same time short enough to avoid generation of any rivulets on the surface. This gives a significant improvement compared to those shown in Kabanovs et al. (2017a) which used a shorter 12-s test time. This time was too short for enough liquid to deposit and form a clear soiling pattern, leading to very ‘noisy’ soiling intensity plots. Fig. 15 shows that good agreement has been achieved between experiment and CFD using the developed spray to account for coalescence close to the nozzle, especially considering the relatively short simulated time.

The work presented in this section shows that the chosen numerical approach is able to accurately predict the flow field and the pattern of spray deposition on the base of the baseline generic SUV. This suggests that the numerical investigation can now be extended beyond the experimental case to include rotating wheels and a moving ground plane. The analysis of aerodynamic and soiling trends associated with these boundary conditions is carried out in the next section.

4.2. Effect of wheel boundary conditions on aerodynamics and soiling

In this section the different aerodynamic and soiling behaviour seen in the four cases given in Table 1 are presented. Table 2 presents the time-averaged force coefficients obtained for all cases considered. It can be seen that the average drag increases and the downforce reduces significantly when the stand-off pins are removed and the model is lowered and placed on the ground. Although it is difficult to verify this trend due to the lack of experimental work, it is clear that the gap between the underside of the wheel and the floor in the baseline case generates a region of high-speed air. This results in a negative net pressure, thus increasing the downforce.

When the wheels are set to rotate, the coefficients of drag and lift are reduced, which agrees with the computational study carried out for a saloon car in Koitrand et al. (2014). By introducing the moving ground plane both the lift and drag are further reduced, although the drag is not affected as much as the lift. This trend from fixed ground & fixed wheel to moving ground & rotating wheel also agrees with the results presented in Koitrand et al. (2014) (saloon car) and Forbes et al. (2017) (generic SUV with 30° rear bottom diffuser).

4.2.1. Contamination patterns and soiling rates

Fig. 16 presents soiling intensities on the base of the model obtained in four simulations. To generate soiling intensity plots, the deposited mass in each numerical case was normalised against the maximum value in the fixed ground & rotating wheel case, as this was the largest value found across all cases. While the general shape of deposition in the fixed ground & fixed wheel case is very similar to the wheel stand-off case, Fig. 16 shows that the pattern becomes confined to a smaller region at the top centre of the base when the stand-off pins are removed and the model is lowered so that the wheels can form a contact patch with the ground. The vertical distribution of the contaminant across the base is increased when the wheels are set to rotate in the fixed ground & rotating wheel...
The addition of the moving ground plane (moving ground & rotating wheel case) further increases deposition in the left middle area and leads to a wider pattern.

Fig. 17 shows the evolution of the total deposition, found by integrating the mass off all parcels hitting the base surface at each time, for each case relative to the wheel stand-off case. The time axis starts at the point when spray injection starts and so there is a delay before any parcels reach the base. It can be seen that lowering the model such that there is no longer a gap between wheel and floor leads to an increase in total deposition by 21%. This is primarily due to the reduced vertical distance between the spray source and the wake leading to a greater amount of the spray being entrained into the wake from where it deposits.

Fig. 22. Standard deviation of vertical (left) and lateral (right) velocity components on the vertical centreline (Y = 0 m) and at Z = 0.14 m, respectively. Shown for two cases: fixed ground & rotating wheel (top) and moving ground & rotating wheel (bottom).

Fig. 23. Difference in cell-based volume fraction of spray: Red iso-surface (value − 25 × 10⁻⁶) - spray reduced; green iso-surface (value + 25 × 10⁻⁶) - spray increased. (For interpretation of the references to colour in this figure legend, the reader is referred to the Web version of this article.)
on the surface. The removal of the region of high speed air flowing through the gap under the wheels also has the effect of allowing more spray to be entrained rather than carried downstream. The addition of rotating wheels and moving ground plane leads to an increase in the total deposited mass as well as leading to a wider spatial distribution on the vehicle base.

The increased vertical distribution of spray on the base when the wheel rotates is linked to the mechanism of spray entrainment into the wake. Fig. 18 shows the ‘pumping’ nature of the wake whereby the length of the wake increases and decreases over time. This determines the longitudinal distance from the base at which spray is entrained. Fig. 19 shows paths of parcels that deposit on the base in two separate time windows. The particle size range in the plots is consistent with the peak of the injected spray size distribution taken from Kabanovs et al. (2018). It can be seen that early (distance-wise) entrainment leads to a lower deposition on the base and vice versa. In the fixed ground & rotating wheel and moving ground & rotating wheel cases, a larger fraction of spray is able to enter the wake closer to the base than in the wheel stand-off and fixed ground & fixed wheel cases, hence leading to increased lower deposition on the rear surface. The reason for this can be explained by looking at the trends in the flow field.

Fig. 20 shows the difference in the time-averaged horizontal U and vertical W velocity fields between the fixed ground & fixed wheel, fixed ground & rotating wheel, moving ground & rotating wheel cases and the baseline wheel stand-off case on the vertical plane at Y = −0.09 m. This plane is located approximately midway between the centrelines of the left rear wheel and the base surface. This figure shows that the mean vertical velocity W is increased when the stand-off pins are removed, possibly explaining the tendency of spray to concentrate in the top region on the base for this case. The addition of rotating wheels, alone or in combination with a moving ground plane, results in the reduction of the longitudinal U velocity in the diffuser and bottom near-wake region. This slows down spray as it is advected into the near-wake region, influencing it to entrain closer to the base and leading to an increased vertical distribution of deposition on the rear surface. Fig. 20 suggests that this effect...
is stronger for the moving ground & rotating wheel case. This case also shows a significant reduction in the vertical W component roughly 0.3 m away from the base, where the ‘late’ entrainment generally occurs (see Fig. 19b). This reduces ‘late’ entrainment, further explaining the increased deposition in the middle left region on the base in the moving ground & rotating wheel case.

Fig. 16d shows that the addition of a moving ground plane leads to a deposition that is evenly distributed vertically across a wider area on the base than in the fixed ground & rotating wheel case. Fig. 21 shows that there is an increased mean flow towards the centre of the vehicle when the moving ground plane is used which spreads the spray over a wider area laterally. Furthermore, Fig. 22 shows that there is also an increased standard deviation of the vertical and lateral components of velocity in the wake which also explains the wider contamination pattern. This effect of the moving ground was also reported by Jilesen et al. (2013).

The change of the mean spray volume fraction, averaged over the 1.5s simulation, with the changes to wheel and ground boundary conditions is visualised in Fig. 23. These figures highlight the change in mean spray location by showing regions of reduced volume fraction in red and increased volume fraction in green. Using this the increased entrainment of spray into the shear layer when the wheel stand-off is removed can be seen. This is enhanced when wheel rotation is included and the increased concentrations at the separate early and late entrainment points can be seen. Finally the addition of the moving ground plane can be seen to shift the spray plume inboard. These changes are potentially important when relating wind tunnel results to on-road soiling performance. As well as the difference from the stationary tunnel setup (wheel stand-off) the difference between fixed ground & rotating wheel and moving ground & rotating wheel is also significant as fixed ground & rotating wheel corresponds to the setup most often used in climatic wind tunnel tests.

4.3. Simulations with the generic SUV using wheel spray models

In this section the three wheel spray models described in Section 3.2 are applied to the simulation of the generic SUV. As spray will now be released into the wheel arch the interaction of the spray with inside of wheel arch could affect results. To test this, in Sections 4.3.1 and 4.3.2 parcels are allowed to bounce multiple times in the wheel arch region and in Section 4.3.3 this is compared with results where parcels are removed when they first hit the wheel arch.
4.3.1. Comparison of soiling patterns

Fig. 24 shows soiling intensity plots for the rear surface of the generic SUV model, obtained in the numerical cases which used the three tyre spray models presented in Section 3.2. It can be seen that all numerical cases show a similar location of the highest contamination, which is in the top middle region of the base. In addition, unlike the results presented in Section 4.2 for the experimental spray modelled in the same numerical setup (i.e. fixed ground & rotating wheel), the vertical distribution of deposited spray in all cases shown in Fig. 24 is significantly smaller, and all contamination patterns match the experimental pattern obtained for the baseline wheel stand-off case, shown previously in Fig. 15a. The results in Fig. 15a used particles of smaller size, and injection was carried out from a single point in space. Due to the reduced inertia of smaller particles, this spray was more sensitive to the unsteady structures in the flow field. This means that it was more susceptible to dispersion and entrainment into the near-wake region close to the base. This promotes deposition on the lower part of the rear surface. This shows that for the same flow field the size of particles and injection setup can influence the details and amount of deposition. As such an accurate model for the spray production would be of great value in surface geometries. Using a size distribution rather than a single size does not significantly increase the details and amount of deposition. As such an accurate model for the spray production would be of great value in surface contamination studies and is an important avenue for future research.

Fig. 24c also shows that deposition modelled with Spray Model 3 covers a wider area of the base than that modelled with Spray Model 2, shown in Fig. 24b. Although the difference between the two patterns is small in this case, we cannot rule out bigger differences for other vehicle geometries. Using a size distribution rather than a single size does not add significantly to the complexity of the Lagrangian simulation and hence this potential increase in accuracy does not come at a high cost.

The total base soiling rates, normalised by the total contamination obtained for Spray Model 1, are shown in Fig. 25. The results for ‘bounced p removed’ refer to cases where the particle boundary condition in the wheel arch is changed which are discussed below. The results confirm that, in this particular case, there is little difference in soiling rate between model 2, which uses a size distribution, and model 3 which uses only the peak size from that distribution. The most significant result here is that from model 1 which shows a significantly reduced soiling rate compared to the other models. The reasons for this are considered below.

4.3.2. Importance of parcel release position

An important consideration for surface contamination studies is the origin of those spray particles that deposit on the vehicle surface. This is important from a practical and a computational point of view. From the practical point of view it is useful to know where the contaminant originated so that appropriate mitigation can be taken. From the computational point of view it is important to know where particles should be released into the simulation as releasing particles which do not get transported to the surface adds to the computational cost without increasing the statistical accuracy of the simulation. Furthermore if modelling is to be improved by using more accurate size distributions then it is important to know where these distributions should be measured; i.e. from around the whole circumference or just at the contact patch for example. Fig. 26 shows the percentage of parcels deposited on the base which originated at a circumferential position as a function of that circumferential position. From all three models it can be seen that the majority of parcels contaminating the base originated from close to the contact patch.

As the majority of rear contamination originates from close to the contact patch, the ability of the chosen wheel spray model to capture the topology of the spray in this region is therefore more important than matching the topology above the wheel. For this reason wheel spray model 1 appears to be a less appropriate choice than models 2 or 3. Fig. 11 shows clearly that while the model matches the topology above the wheel (as it was tuned to do) it does not capture the behaviour around the contact patch. The smaller droplets included in the models based on the measurements in Bouchet et al. (2004) can capture this and can therefore be expected to provide a better soiling prediction. The difference caused in predicted rear contamination with the two approaches has already been shown in Fig. 24. The reason for this difference can be seen in Fig. 27, which shows paths of particles released from close to the contact patch for model 1 and 2. The particles from model 1 are significantly larger, and their high momentum carries them down-stream of the model thus avoiding deposition. The distributed, and crucially smaller, particle sizes in model 2 show much more dispersion and are small enough for the reversed flow in the wake to reverse their direction and deposit them on the base.

4.3.3. Spray boundary condition within the wheel-arch

The results in the previous section were produced by using a bounce boundary condition within the wheel-arch. This means that the parcels can bounce off the interior surface of the wheel-arch an infinite number of times, which is physically unrealistic. Fig. 28 shows the results for the paths of particles released from different sections of the wheel. The path lines are coloured according to the number of times the parcel bounced. It can be seen that for those parcels released further away from the contact patch the number of bounces increases. This is particularly the case for spray model 1 although even for model 2 most parcels released away from the contact patch have bounced at least once. It can also be seen that parcels released from the upper parts of the wheel tend to escape from the side of the wheel-arch from where they are unlikely to be entrained into the wake and are instead carried downstream.

To check the influence of the assumption that the particles can repeatedly bounce in the wheel-arch, all parcels which counted at least one bounce were removed from the results. This is analogous to assuming that any particle sticks to the inside of the wheel-arch. The soiling rate obtained by doing this is shown as the ‘bounced p removed’ lines in Fig. 25. For spray models 2 and 3 this causes around a 20% reduction in the total soiling rate while for model 1 it causes a small reduction. Fig. 29 repeats the analysis for the origin of the deposited parcels with those that have bounced removed. This shows that only a very small amount of spray (0.21%-1.67%, depending on the spray model) released from 0° to 180° of the tyre surface is able to reach the base without interacting with the walls. This means that spray released from a small region behind the contact patch contributes more than 98% to the total deposition.

![Fig. 27. A total of 400 particle paths generated by tracking spray from evenly distributed points on the tyre surface within 31° of the contact patch (from wheel spray model 1 and 2).](image-url)
As the largest amount of spray is generally thrown off in a small angle due to the tread pickup mechanism, the tyre spray modelling could be significantly simplified by only modelling spray behind the contact patch. Alternatively, modelling of these mechanisms in the wheel arch region may be required. The effect of liquid film stripping from the wheel arch, and its significance on rear surface contamination, still needs to be properly studied.

5. Conclusions

In this study, the influence of wheel, ground and spray boundary conditions on predictions of road vehicle rear surface contamination was investigated. The effect of rotating wheels in isolation and in combination with a moving ground has been assessed numerically for a generic SUV, configured to have 0° rear roof taper and 10° bottom diffuser. Due to the absence of experimental data for cases that consider wheel rotation and ground motion, the numerical approach was first validated for a baseline case, for which both the aerodynamics and soiling tests, based on a fixed spray source, had been carried out. Very good agreement was achieved with the experimental data for wake structure and surface deposition by using the IDDES turbulence modelling technique, combined with Lagrangian particle tracking. On the basis of these results, the investigation was then extended to include rotating wheels and a moving ground. This numerical method was also able to correctly predict the expected trends in the aerodynamics due to these boundary conditions, further confirming the suitability of the chosen approach.

It was shown that the patterns of rear-surface contamination, as well as the deposition rates, can vary significantly with the boundary conditions applied to the wheels and the ground. For example, the total deposition was increased when wheel rotation was considered. However the increase in total soiling rate due to wheel rotation was similar in magnitude to the increase seen when removing the offset between wheel and floor used in the experiment. This emphasizes the sensitivity of vehicle soiling to parameters such as ride height. The increased soiling with wheel rotation is associated with extra up-wash caused by the wakes of rotating wheels, which are able to deliver more spray for entrainment into the wake close to base where it then becomes available for transport onto the lower part of the base. As a result, the vertical dispersion of the soiling pattern is increased. The total deposition rate was only slightly increased when the rotating wheels were modelled in combination with a

---

**Fig. 28.** Paths of parcels released from the following circumferential positions (0° is the horizontal downstream direction): (top row) – 78° to – 47°; (middle row) – 47° to 25°; (bottom row) 25° to 180°. The inset figure also indicates the region of the wheel from which spray is emitted on each row.
moving ground, suggesting that the wheel rotation has a greater influence. The most heavily contaminated area had a wider extent at the vehicle mid-height when the moving ground was included. This is due to the increased mean lateral velocity component towards the centre-line as well as greater lateral unsteadiness of the wake.

Three tyre spray models were used based on the full-scale experimental work of Kuthada and Cyr (2006), Spruss et al. (2011) and Bouchet et al. (2004). The models were then scaled down and used in simulations with the generic SUV geometry to study the dynamics of tyre spray. The first model used a circumferential size distribution of particles and was developed by matching the pattern of experimental spray released from an isolated rotating wheel. It used particles of a relatively large size in order to capture the trajectories of the largest droplets, which could be seen in the experimental data. As a result, this model was able to reproduce the overall shape of the spray formed as a result of the capillary adhesion mechanism. The fine spray formed due to the break-up of the liquid sheet at the contact patch of the wheel (tread pickup mechanism), however, was not reproduced. The second model employed the data of a developed tyre spray measured 1 m downstream of the wheel and used this size distribution for all circumferential positions. This spray was able to reproduce both the capillary adhesion and tread pickup mechanisms reasonably well. The third model used particles of a single size based on the peak size from the distribution used in model 2. This model was able to reproduce the tread pickup mechanism, but the particle size was insufficient to capture the capillary adhesion mechanism properly. It was also shown to be impossible to replicate the complete spray pattern with any single particle size. As a result, the most representative tyre spray model implemented in this work used the full distribution of a ‘developed’ tyre spray.

An important conclusion of this work is that the largest fraction of spray that deposits on the base is released from a small region of the tyre close to the contact patch. If allowed to bounce in the wheel-arch, spray released from further away from the contact patch tended to escape from the side of the wheel-arch. From where it was not entrained into the wake. If any parcels that bounced within the wheelhouse were removed from the calculation then the percentage of parcels reaching the base which originated near the contact patch was even greater. This was common across all spray models used in this study. Further work needs to be carried out to identify whether this is general or is significantly affected by the design of the wheelhouse. If this is common, this would suggest that the numerical tyre spray model could be simplified to only account for spray released from behind the contact patch of wheel and ground. It also places more emphasis on capturing the tread pickup mechanism correctly than the
capillary adhesion mechanism in the spray injection model. As mentioned above the model developed by matching the spray topology above the wheel did not capture the spray pattern around the contact patch well and this led to a significantly smaller predicted deposition rate on the base. The smaller particles found in distribution used in model 2, as well as giving the correct spray topology around the contact patch, were much more likely to reverse direction and be deposited on the base of the vehicle. The importance of the size distribution created at the contact patch suggests that further research, both experimental and numerical, into this process would be of great value in improving understanding and prediction of road vehicle soiling.

Acknowledgements

This work was supported by Jaguar Land Rover and the UK Engineering and Physical Sciences Research Council grant EP/K014102/1 as part of the jointly funded Programme for Simulation Innovation. This research has made use of the HPC-Midlands-Plus Tier-2 computing system, supported by the research computing staff of Loughborough University, and funded through EPSRC e-infrastructure grant EP/P020232/1.

References

Borg, A., Vervang, R., 2006. On the prediction of exterior contamination with numerical simulations (simple Lagrangian particle tracking models with and without wall film model). In: 6th MIRA International Conference on Vehicle Aerodynamics, pp. 389–388.

Bouchet, J.-P., Delpech, P., Palier, P., 2004. Wind tunnel simulation of road vehicle in driving rain of variable intensity. In: 5th MIRA International Conference on Vehicle Aerodynamics-2004. CD-adapco, 2014. Star-CDM+ [CFD Software].

Elofsson, P., Bannister, M., mar 2002. Drag reduction mechanisms due to moving ground and wheel rotation in passenger cars. In: SAE Technical Paper 2002-01-0531. https://doi.org/10.4271/2002-01-0531.

Forbes, D., Page, G., Passmore, M., Gaylard, A., 2017. A study of computational methods for wake structure and base pressure prediction of a generic SUV model with fixed and rotating wheels. Proc. Inst. Mech. Eng. - Part D J. Automob. Eng. 231 (9), 1222–1238. https://doi.org/10.1177/0954407016685496, ISSN 0954-4070.

Gaylard, A., Kabanovs, A., Jilesen, J., Konstantinov, A., Wanderer, J., apr 2013. Insights into near surface contamination using simulation of road spray and aerodynamics, 2014–01–0610 SAE International Journal of Passenger Cars - Mechanical Systems 7 (2). http://doi.org/10.4271/2014-01-0610. ISSN 1946-4002. http://papers.sae.org/2014-01-0610/.

Gaylard, A.P., Duncan, B., apr 2011. Simulation of car spray and body side vehicle soiling by road sprays, 2011–01–0173 SAE International Journal of Passenger Cars - Mechanical Systems 4 (1). http://doi.org/10.4271/2011-01-0173. ISSN 1946-4002. http://papers.sae.org/2011-01-0173/.

Gaylard, A.P., Pittman, J., Jilesen, J., Gagliardi, A., Duncan, B., Wanderer, J., Konstantinov, A., apr 2014. Numerical simulation of spray injection model based on discrete phase model, Adv. Mech. Eng. 7 (7) https://doi.org/10.1177/1687814015597190, ISSN 1687-81401687814015597190 http://adc.sagepub.com/lookup/doi/10.1177/1687814015597190 http://journals.sagepub.com/doi/10.1177/1687814015597190, ISSN 1687814015597190.

Hu, X., Liao, L., Lei, Y., Yang, H., Fan, Q., Yang, B., Chang, J., Wang, J., jun 2015. A numerical simulation of wheel spray for simplified vehicle model based on discrete phase method, Adv. Mech. Eng. 7 (7) https://doi.org/10.1177/1687814015597190, ISSN 1687-81401687814015597190 https://adc.sagepub.com/lookup/doi/10.1177/1687814015597190 http://journals.sagepub.com/doi/10.1177/1687814015597190.

Jilesen, J., 2013. Multi-disciplinary design applications of unsteady aerodynamics and water sprays on road vehicles Multi-disciplinary design applications of unsteady aerodynamics and water sprays on road vehicles. In: 6th Graz Symposium Virtuelles Fahrzeug (GZVF), (May 2013),