Journal of Wind Engineering & Industrial Aerodynamics 165 (2017) 13–22

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

journal homepage: www.elsevier.com/locate/jweia

Simulation of rear surface contamination for a simple bluff body

A.P. Gaylard[a,b,*], A. Kabanovs[c], J. Jilesen[d], K. Kirwan[e], D.A. Lockerby[f]

[a] *Jaguar Land Rover, Banbury Road, Gaydon, Warwick CV35 ORR, UK*
[b] *WMG, International Manufacturing Centre, University of Warwick, Coventry CV4 7AL, UK*
[c] *Aeronautical and Automotive Engineering, Loughborough University, Leicestershire LE11 3TU, UK*
[d] *Exa Corporation, 55 Network Dr, Burlington, MA 01803, USA*
[e] *WMG, International Manufacturing Centre, University of Warwick, Coventry CV4 7AL, UK*
[f] *School of Engineering, University of Warwick, Coventry CV4 7AL, UK*

ARTICLE INFO

Keywords:
CFD
VLES
Aerodynamics
Surface contamination
Windsor body
Lattice Boltzmann

ABSTRACT

Predicting the accumulation of material on the rear surfaces of square-backed cars is important to vehicle manufacturers, as this progressively compromises rear vision, vehicle visibility and aesthetics. It also reduces the effectiveness of rear mounted cameras. Here, this problem is represented by a simple bluff body with a single sprayer mounted centrally under its rear trailing edge.

A Very Large Eddy Simulation (VLES) solver is used to simulate both the aerodynamics of the body and deposition of contaminant. Aerodynamic drag and lift coefficients were predicted to within +1.3% and −4.2% of their experimental values, respectively. Wake topology was also correctly captured, resulting in a credible prediction of the rear surface deposition pattern.

Contaminant deposition is mainly driven by the lower part of the wake ring vortex, which advects material back onto the rear surface. This leads to a maximum below the rear stagnation point and an association with regions of higher base pressure.

The accumulation of mass is linear with time; the relative distribution changing little as the simulation progresses, implying that shorter simulations can be compared to longer experiments. Further, the rate of accumulation quickly reaches a settled mean value, suggesting utility as a metric for assessing different vehicles.

1. Introduction

The following presents a numerical simulation of rear surface contamination for a simple bluff body, representing a road vehicle. It explores the interaction of an idealised tyre spray with a vehicle base wake and the resulting accumulation of material on the rear surface.

This models a significant issue: the accumulation of contaminants (soil, tyre debris, etc.) on the rear surfaces of cars diminishes both drivers' vision and vehicle visibility as material is deposited on lights and the rear screen. In addition, the aesthetic appeal of the vehicle may be reduced and soil transferred to users' hands and clothes as they access the rear load space via the tailgate. These processes have the potential to undermine customers' perceptions of product quality (Gaylard et al., 2014).

The main contaminant source for these surfaces is the spray generated by the vehicle's own rear tyres, as they move over wet road (Jilesen et al., 2013). This is advected into the base wake and subsequently deposited onto the vehicle's rear surfaces. The coupling with wake flows, and hence vehicle aerodynamic performance, means that this issue must be addressed concurrently with aerodynamic drag during the development process.

It has long been appreciated that square-backed vehicles such as hatchbacks, estates, and SUVs are particularly susceptible to this issue (Maycock, 1966) along with bus bodies (Lajos et al., 1986). Therefore this work uses a square-backed bluff body to represent vulnerable car designs.

Simplified bodies, which represent a few salient geometric features, are widely used in automotive aerodynamics, for an overview of this practise see Le Good and Garry (2004). They enable key processes to be investigated without the myriad interactions seen in production vehicles, or having to cope with their geometric complexity. In essence, they provide an improved signal-to-noise ratio, by omitting geometry responsible for generating flow features not significant for the class of problem under investigation.

However, this potentially useful approach has yet to be widely applied to the rear surface contamination problem. In one of the few

---

* Corresponding author at: Jaguar Land Rover, Banbury Road, Gaydon, Warwick CV35 ORR, UK.
E-mail addresses: agaylar1@jaguarlandrover.com, a.p.gaylard@warwick.ac.uk (A.P. Gaylard), A.Kabanovs@lboro.ac.uk (A. Kabanovs), jjilesen@exa.com (J. Jilesen), Kerry.Kirwan@warwick.ac.uk (K. Kirwan), D.Lockerby@warwick.ac.uk (D.A. Lockerby).

http://dx.doi.org/10.1016/j.jweia.2017.02.019
Received 27 January 2016; Received in revised form 10 February 2017; Accepted 15 February 2017
Available online 04 March 2017

examples published, Hu et al. (2015) demonstrate the use of a *modified* version of the MIRA Reference Model in computational fluid dynamics (CFD) simulations of the problem, but provide no comparative experimental data for either the aerodynamics of the model or deposition of the contaminant.

In contrast, the CFD investigation of Kabanovs et al. (2016) used a well-known simple bluff body and provided some contaminant deposition patterns obtained from wind tunnel experiments. However, their computational work did not account for realistic wake unsteadiness.

Hence, this work extends that of Kabanovs et al. (2016), applying an unsteady eddy-resolving CFD simulation to their simple test case. Doing so provides additional insights into spray advection into the wake, its distribution through the wake and the subsequent pattern of deposition. The latter permits some limited qualitative comparison of the numerical simulation against their experimental data. Data from the literature are also used to assess the degree to which the CFD simulation captures the aerodynamics behaviour of the bluff body, in terms of drag and lift force prediction along with wake topology.

In addition, guidance is provided on the numerical simulation of this issue; specifically coping with the mismatch between the sampling times available in experiments with those economically obtainable with unsteady CFD simulation.

2. Approach

2.1. Bluff body

The representative bluff body used in this study is illustrated in Fig. 1. This is the square-back version of the Windsor body; a simple design which has proportions typical of a small hatch-back car and has been used in a wide range of aerodynamics studies (See, for example, Volpe et al. (2014); Littlewood et al. (2011); Littlewood and Passmore (2010); Howell et al. (2013); Howell and Le Good (2008); Howell et al. (2003)). As shown, it is 1044 mm long, 389 mm high and 289 mm wide; with a stated projected frontal area (A) of 0.112 m^2.

It is usually mounted using four threaded bars (M8) at positions representative of front and rear axles, 15 mm inboard of the sides of the model. To maintain comparability with the available experimental data, ground clearance was set to 50 mm ($h_g/H = 0.17$).

One important advantage of using standard reference geometry is that experimental data is available to support correlation with the CFD simulation. In addition to limited qualitative data for surface contamination deposition (Kabanovs et al., 2016), zero-yaw drag and lift coefficients are available (Perry et al., 2015) along with the rear wake topology (Pavia et al., 2016).

Hence, the representation of a key vehicle type and the availability of experimental data for both aerodynamics and surface contamination make this a good initial system for the investigation of the interaction between a tyre spray and vehicle wake.

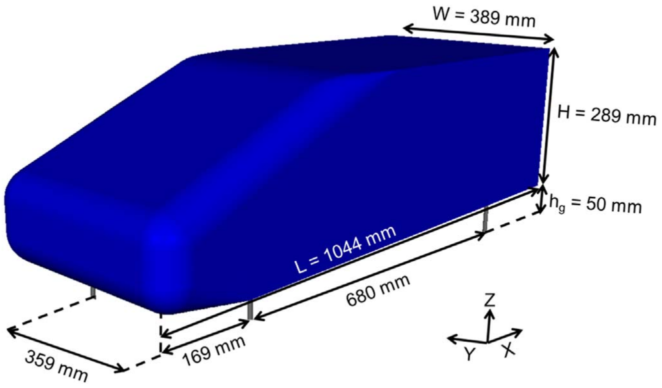

Fig. 1. Basic dimensions of the windsor body.

2.2. Mathematical models

Numerical simulations were performed with a commercially available CFD code, Exa PowerFLOW. This has been previously been applied to wind engineering (Mamou et al., 2008a, 2008b; Syms, 2008) as well as vehicle aerodynamics simulations (Chen et al., 2003). It is an inherently unsteady Lattice Boltzmann (LB) solver which uses what is essentially a Very Large Eddy Simulation (VLES) turbulence model (Chen et al., 1992, 1997, 2003), as when typically applied to bluff body aerodynamics simulations the spatial resolution used is too coarse to resolve more than 80% of the turbulent kinetic energy (Pope, 2013, p.575). Unresolved turbulence is accounted for by including an effective turbulent relaxation time, calculated via the RNG κ-ε transport equations (Chen et al., 2003).

The discrete airborne droplets of the spray were represented via a Lagrangian particle model. This technique has been previously applied to dispersed phase simulations, such as: wind-driven rain (Hangan, 1999; Persoon et al., 2008) and sand (Paz et al., 2015); water droplets falling under gravity (Meroney, 2006); pesticide spray (Xu et al., 1998); particulate atmospheric pollutants (Ahmadi and Li, 2000) and spray from vehicle tyres (Kuthada and Cyr, 2006). In this case, the particle model was run concurrently with the LB solver. Hence particle and flow time are coupled, enabling the particles to respond to the unsteady flow and allowing for two-way momentum transfer between the continuous and discrete phases. This has been extended to include standard models for splash (Mundo et al., 1995; O'Rourke and Amsden, 2000) and breakup (O'Rourke and Amsden, 1987). At the surface, particle mass, which is not lost via splash, is transferred into a thin surface film, represented by a model similar to that of O'Rourke & Amsden (1996). A re-entrainment model strips particles from the film if a user-set critical film thickness is exceeded. This continues until its thickness falls below a critical threshold, set at 0.3 mm in this work (Jilesen et al., 2015).

This combination of an eddy-resolving unsteady flow solver with extended particle and surface film sub-models provides a suitable tool for the investigation of the rear surface contamination problem. It is important to note that capturing the transport of droplets into a wake through the bounding shear layer requires the use of higher fidelity turbulence modelling than more widely used correlation-moment closure models provide, as these cannot capture the relevant unsteady structures in the shear (mixing) layer (Yang et al., 2004). Similarly, Paschkewitz (2006) demonstrated, while investigating the dispersion of a modelled tyre spray through the wake of a simplified lorry, that an LES turbulence model increased the vertical dispersion of the lowest inertia particles, compared to unsteady RANS (URANS). The use of LES increased the vertical dispersion distance by 35%, for particles with a diameter less than 5×10^{-5} m. This is twice the mean diameter of the particle distribution used here; hence, the use of an unsteady eddy-resolving approach is essential.

2.3. Simulation domain

The simulation domain was designed to replicate the environment provided by the test section of the Loughborough University Wind Tunnel, as this facility was used in the equivalent experiments. The wind tunnel, described in detail by Johl et al. (2004), is a semi-open return design with a closed working section measuring 1.92 m (wide) by 1.32 m (high).

Fig. 2 provides a cut-away view of the numerical domain, showing: inlet, outlet, floor and one of the two vertical walls (for the sake of clarity the ceiling and remaining vertical wall are not shown). The height and width of the working section match that of the wind tunnel, but the length of the domain has been extended both upstream and downstream to provide sufficient clearance between the bluff body, inlet and outlet. A prescribed flow velocity is set at the inlet, whilst the outlet is set to atmospheric pressure.