

Evaluation of Non-Uniform Upstream Flow Effects on Vehicle Aerodynamics

Adrian P. Gaylard and Nicholas Oettle
Jaguar Land Rover

Joaquin Gargoloff and Bradley Duncan
Exa Corp.

ABSTRACT

Historically vehicle aerodynamic development has focused on testing under idealised conditions; maintaining measurement repeatability and precision in the assessment of design changes. However, the on-road environment is far from ideal: natural wind is unsteady, roadside obstacles provide additional flow disturbance, as does the presence of other vehicles. On-road measurements indicate that turbulence with amplitudes up to 10% of vehicle speed and dominant length scales spanning typical vehicle sizes (1-10 m) occurs frequently.

These non-uniform flow conditions may change vehicle aerodynamic behaviour by interfering with separated turbulent flow structures and increasing local turbulence levels. Incremental improvements made to drag and lift during vehicle development may also be affected by this non-ideal flow environment.

On-road measurements show that the shape of the observed turbulence spectrum can be generalised, enabling the definition of representative wind conditions.

Here, unsteady Lattice-Boltzmann Method (LBM) simulations are used to evaluate the modification of the aerodynamics of a fast-back saloon by realistic on-road flow conditions. The turbulent conditions are added as fluctuations to the freestream flow, based on a prescribed turbulence spectrum, integral length scale, and turbulence intensity. The impact of these flow conditions on the separated flow structures and the incremental effects of design changes are evaluated. Further, the characteristics of anisotropic turbulence in the unsteady flow field around the vehicle are compared to show how freestream turbulence modifies the local turbulent field.

This approach enables the evaluation of design decisions against a broader range of operating conditions than can be achieved in traditional wind-tunnel testing.

CITATION: Gaylard, A., Oettle, N., Gargoloff, J., and Duncan, B., "Evaluation of Non-Uniform Upstream Flow Effects on Vehicle Aerodynamics," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(2):2014, doi:10.4271/2014-01-0614.

INTRODUCTION

This paper examines the effect of non-uniform (turbulent) onset flow on the aerodynamics of a saloon with a 'fast-back' coupe style rear. It builds on a recent publication by the same authors [1] by including an assessment of changes in the performance of two common aerodynamic devices: front and rear wheel deflectors.

Properties of on-road wind turbulence are specified using several standard quantities. Turbulence intensity (I) for a vehicle driving at constant speed is defined as the standard deviation of velocity component fluctuations, normalized by the vehicle speed. The intensity is typically computed for each

velocity component, I_x , I_y , I_z , using wind-measurements averaged into a single value, as follows (for the sake of brevity only the definition of I_x is shown):

$$I = \sqrt{\frac{1}{3} \cdot (I_x^2 + I_y^2 + I_z^2)}$$

where:

$$I_x = \frac{\sigma([V_x(t)])}{V_{ref}}$$

 **Documents**
Document Sharing Hub

V_{ref} is the speed of vehicle motion, and

$\sigma(a)$ is the standard deviation of a .

The integral turbulence length scale, L , is also determined separately for each velocity component, e.g., L_x , L_y , L_z , commonly using the time auto-correlation of the velocity component and the vehicle speed, for example used by Wordley and Saunders [2] and as described by the Engineering and Sciences Data Unit (ESDU) [3]. The turbulent flow structures have a dominant length scale in all three directions spanning the size range of most road vehicles, e.g., 1-10 m. Larger scales are also present in natural wind (up to several kilometres). However, these large-scale fluctuations are varying slowly relative to the vehicle and can be represented as a quasi-steady head, tail or crosswind lasting several seconds.

Most wind tunnels used for aerodynamic optimization have an inherently low level of turbulence intensity, of around 0.3 % or less. On the other hand, the typical atmospheric turbulence in regions where vehicles will be traveling is much higher, even on a relatively calm day. For natural wind, turbulence intensity levels of 1-5% are common [2]. Furthermore, natural wind is anisotropic, with values of L_z , typically measuring less than 50% of the L_x and L_y components. In the presence of upstream vehicles, turbulence intensity values of 5-15% are observed in on-road measurements, consistent with the amplitude of turbulent wake fluctuations produced by a single vehicle in wind-tunnel measurements (see Cogotti [4].)

Owing to interest in the effect of unsteady flow conditions on vehicle aerodynamic and aeroacoustic measurements, a number of passive and active systems have been developed that attempt to simulate the unsteady conditions that are experienced on-road in the wind tunnel. Many of these systems were reviewed by Sims-Williams [5], who assessed the scales of unsteadiness which could be simulated against those measured in the natural wind. It was found that passive grids were unable to practically produce turbulence at the most important scales for unsteady studies, with active turbulence generation systems best suited to the simulation of these scales. However, the main drawbacks of active devices for turbulence generator systems are the additional costs and complexity in the operation of the wind tunnel, the range of scales that can be generated, the addition of certain turbulence scales that may not be characteristic of natural wind and the addition of flow-induced and mechanical noise by the active turbulence generators. Consequently it was noted that CFD simulations did not suffer the same limitations as experimental approaches.

Using unsteady aerodynamics simulations, it is possible to consider the effect of unsteady upstream flow conditions, using an appropriate time- and space-varying boundary condition upstream of the vehicle. This approach is similar to an approach used in Large Eddy Simulation (LES) studies in which grid-resolved turbulence must be added into the

simulation via the boundary conditions so that it can be properly simulated. The theoretical approach for generating such turbulent conditions has been pioneered by Kraichnan [6] and others.

In this paper, the effect of non-uniform flow conditions coming from upstream turbulence is shown for a Jaguar XJ saloon in typical highway driving at zero degrees yaw. The aerodynamic features of the vehicle are evaluated in several design conditions with and without the addition of upstream turbulence. The anisotropic turbulence properties proximate to the vehicle are also shown, enabling the evaluation of the significance of the extrinsic upstream turbulence relative to the intrinsic (self-induced) turbulence produced in the wake of the vehicle. This type of analysis will allow vehicle aerodynamicists to consider the robustness of design changes to 'real world' operating conditions when determining aerodynamic efficiency and handling characteristics. The application of unsteady simulations using non-uniform flow conditions presented in this study provides a method for evaluating vehicle development decisions against a broader range of operating conditions than can be used in traditional wind-tunnel testing.

BACKGROUND

Upstream Turbulence Conditions

The study of the realistic flow environment for vehicle driving has received more focus in the literature lately, and is of increasing interest to aerodynamicists. The current authors recently provided a review of some of the relevant work [1], and for convenience the review is also summarized below.

The turbulence characteristics of the atmospheric wind can be measured on the road. For example, Watkins et al. [7] and Cooper and Watkins [8] report values measured on road using a hot wire anemometer installed on a car roof, showing the normal probability distribution of velocity fluctuations.

Carlino et al. [9] collected a dataset of on-road measurements using four different cars, demonstrating that the flow angle follows a normal distribution. This probability distribution varies with location and direction. The resultant wind speed and the resultant direction of the wind, the yaw angle, increase with height as well. Thus, the flow has a sheared, twisted profile over the height of a vehicle.

Terrain type and traffic conditions modify both the turbulence intensity and length scale. Different terrains can be represented as 'regions' on a chart of length scale versus component turbulence intensity (Wordley and Saunders [2], in their [Figure 1.](#)) These results show that:

- For smooth terrains, the average longitudinal and lateral intensities are of a similar magnitude, while the average vertical intensities were significantly lower. This is to be expected very close to the ground where the turbulence will be damped out and deflected (by the ground) into the

lateral and longitudinal directions

- Road-side obstacle terrains have the highest average proportion of lateral turbulence due to the influence of wakes from trees located upstream
- Highway traffic terrains show the highest average proportion of vertical turbulence (I_z) due to the influence of wakes generated by upstream vehicles. The highway traffic terrain also returned the lowest X and Y length scales. This is a result of traveling in close proximity to the highly turbulent wakes of other vehicles

These different flow environments can be characterized by three metrics: turbulence intensity, length scale, and the longitudinal wind speed.

Information on the characteristics of the turbulent flow generated by the presence of other vehicles running on the road is reported by Saunders and Mansour [10], in which two cars were tested on the road, with one car leading the other.

The literature points to efforts to include upstream turbulence in the wind tunnel. An initial approach was to use aerofoils to direct the flow, typically turning a component of the flow in the lateral, cross-flow direction. An early example of this approach is that of Bearman and Mullarkey [11], who positioned aerofoils on both sides of a model-scale wind tunnel nozzle which could oscillate at frequencies up to 15 Hz with an angle range of $\pm 8^\circ$.

This design influenced the model-scale system described by Passmore et al. [12], whose aerofoils could oscillate at up to 18 Hz. A similar approach is described by Schröck et al. [13], utilising four aerofoils positioned in the nozzle of a model-scale wind tunnel of FKFS. The Turbulence Generation System (TGS) installed in the Pininfarina wind tunnel consists of vertical vortex generator flaps which are hinged to a vertical mast, and oscillate to induce pseudo-random lateral motions up to 1 Hz [4].

Effects of Upstream Turbulence

From the vehicle design point of view, taking into account the influence of upstream turbulence can influence the outcome of design optimizations. Cooper and Campbell [14] show that the performance of a truck deflector for reducing aerodynamic drag coefficient (C_D) is modified by turbulence. The presence of turbulence reduced the drag benefit of the deflector by 19% and increased the optimum deployment angle. Morel [15] shows the sensitivity of the critical backlight angle on the drag of a slant-back car. In this study the drag levels were increased by turbulence and that the critical angle increased as well. Garry [16] presented the effects of turbulence in the forebody region of a truck. In this case the shear layer separating from the cab roof leading edge afforded less aerodynamic shielding to the trailer front face, thus increasing the overall forebody drag. In Cogotti [17] the effects of turbulence using the TGS are reported. For a wagon type vehicle, C_D and front lift coefficient (C_{LF}) increase with the turbulence

The presence of high levels of turbulence also affects the noise levels measured inside the cabin, the transient vehicle handling and aerodynamic lateral stability in the presence of crosswind. All these are important measures of vehicle perceived quality, and need to be assessed in the proper turbulent environment so that problems can be addressed early in the development process. Using a combination of on-road measured and steady wind tunnel measurements, Oettle et al. [18, 19] presented numerical and experimental results focussing on aerodynamic behaviour in the front side glass region and corresponding aeroacoustic effects measured in the cabin. This showed that the vehicle side glass surface pressure response generally remained quasi-steady up to 2-10 Hz, with the exception being the region closest to the A-pillar where the aerodynamic response at frequencies nearest 10 Hz was larger than predicted under steady conditions. However, this was not found to affect the overall cabin noise response (dominated by the front side glass) which remained quasi-steady. For frequencies greater than 5-10 Hz, both the cabin noise and surface pressure response was no longer quasi-steady, with the reduction in unsteady energy present in the smaller flow scales having a progressively smaller impact on the vehicle response.

This categorization of the flow field depending on the characteristic frequencies is also presented in [5], which groups the spectra in three classes: a) quasi-steady for frequencies lower than 1 Hz (thus the gust can be assumed static yaw), b) small scale turbulence for frequencies higher than 100 Hz (thus the effects are mostly local), and c) central range of turbulence scales, which is also the most challenging since the length scales become comparable to the vehicle size, and can interact more strongly with the vehicle flow structure.

Overall, the work reported in the literature provides examples that show that a turbulent environment typical of on-road conditions can have a significant effect on aerodynamic performance.

APPROACH

Aerodynamics Simulation Using the Lattice Boltzmann Method

In this study, numerical simulations were performed using the commercial software Exa PowerFLOW, which utilizes the Lattice-Boltzmann Method (LBM) approach to solve the unsteady aerodynamic, turbulent flow around the vehicle as represented by fully-detailed surface geometry. The Very Large Eddy Simulation (VLES) approach was used for modelling turbulence, in which the small scales are accounted for using a two-equation turbulence model, and the large scales are simulated on the computational grid. Kotopati et al. [20] provides a review of the LBM simulation approach using this turbulence model, and describes capabilities for aerodynamics, thermal and aeroacoustics applications for production vehicle development. Examples of published validation studies, including fully-detailed production vehicles, using this simulation approach can be found in the literature [21-22, 23].

[24, 25, 26]. In addition, several examples of validation studies using wind-tunnel data from reduced-scale models are shown in Fischer et al. [27, 28, 29, 30].

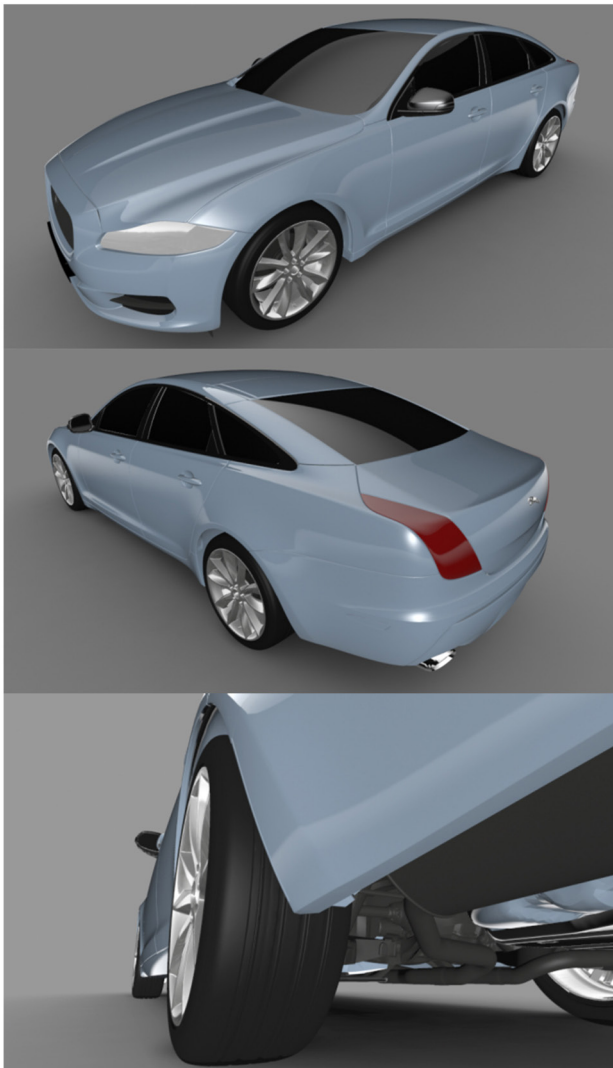


Figure 1. Geometry of the Jaguar XJ saloon.

For this study aerodynamics simulations were performed on a Jaguar XJ saloon with fully-detailed under body and engine bay. The vehicle is shown in [Figure 1](#). The XJ is an F-Segment saloon with a 'fast-back' coupe style rear. Its basic dimensions are 5.127 m (length) $\times$ 1.899 m (width - excluding mirrors) $\times$ 1.456 m (height). Projected frontal area (A) is 2.41 m² and its basic aerodynamic performance (as measured in the MIRA Full Scale Wind Tunnel) is $C_D = 0.30$, $C_{LF} = 0.08$; $C_{LR} = 0.09$.

The simulations were performed using a large rectangular domain with 0.3 % of the cross-sectional area blocked by the vehicle. The domain inlet was placed 12 m in front of the vehicle, in order to preserve the turbulence velocity fluctuations between the inlet and the vehicle. The domain outlet was placed 72 m behind the vehicle. The Cartesian grid for the LBM simulation was defined with the smallest cell size of 1.25 mm near the vehicle in critical regions with increasing cell size away from the vehicle according to high-fidelity "best practices"

grid setup for aerodynamics, consistent with the grid setup used in the studies cited previously [21, 22, 23, 24, 25, 26]. The vehicle was simulated over a moving ground plane, with rotating wheels, at a speed of 28.1 m/s.

The simulations were unsteady (time-resolved) with a time step of 6.9×10^{-6} seconds, performed for 2.0 seconds of physical time. They were initialized using an appropriate starting condition generated from an initial simulation at lower spatial resolution. The data was also averaged in time for post-processing. Both the unsteady and time-averaged results are shown in the following sections.

For the geometry changes and turbulence conditions tested, the simulation strategy is expected to have sensitivity of less than $0.002 \Delta C_D$ (2 counts).

Simulation of Moving Ground Configuration

For the current study, the wheels (rims) of the vehicle were rotated at a rate commensurate with the mean onset flow velocity using sliding mesh regions. The tyres were accurately modelled using scan data, and in order to represent rotating tyres, a surface of revolution was generated retaining only the axisymmetric grooves in the tread. The tyre rotation outside of the sliding mesh region was modelled using the usual angular velocity boundary conditions. A velocity was set on the ground plane to match the mean inlet flow.

Upstream Flow Conditions

Usually, for aerodynamics simulations in the on-road environment, the upstream conditions are specified via a velocity boundary condition sufficiently far in front of the vehicle that the velocity can be assumed to be uniform and equal to the vehicle speed. This is the approach taken for the baseline case. A $k-\epsilon$ turbulence model is active in the flow field and has a prescribed turbulence level of 0.01 % at the domain inlet.

In order to model a realistic turbulent environment, the velocity field far in front of the vehicle is replaced with a temporally and spatially varying velocity field composed of the mean uniform flow with fluctuating components of velocity in each spatial direction superimposed. These fluctuations are predetermined functions of time and space that satisfy the specified frequency range and kinetic energy spectrum in time, and specified wavenumber range and spectrum in each spatial direction. The energy is distributed across the three velocity components to achieve anisotropic turbulence consistent with wind measurements.

The resulting turbulent velocity fluctuations at a typical point are shown in [Figure 2](#). The resulting turbulence spectrum is shown in [Figure 3](#). The turbulence has an integral length scale of approximately 2 m in each direction. The amplitude of the velocity fluctuations can be scaled to any desired turbulence