

A Fully Coupled, 6 Degree-of-Freedom, Aerodynamic and Vehicle Handling Crosswind Simulation using the DrivAer Model

David C. Forbes, Gary J. Page, and Martin A. Passmore
Loughborough University

Adrian P. Gaylard
Jaguar Land Rover

ABSTRACT

In a real-world environment, a vehicle on the road is subjected to a range of flow yaw angles, the most severe of which can impact handling and stability. A fully coupled, six degrees-of-freedom CFD and vehicle handling simulation has modelled the complete closed loop system. Varying flow yaw angles are introduced via time dependent boundary conditions and aerodynamic loads predicted, whilst a handling model running simultaneously calculates the resulting vehicle response. Updates to the vehicle position and orientation within the CFD simulation are achieved using the overset grid method. Using this approach, a crosswind simulation that follows the parameters of ISO 12021:2010 (Sensitivity to lateral wind - Open-loop test method using wind generator input), was performed using the fastback variant of the DrivAer model. Fully coupled aerodynamic and vehicle response was compared to that obtained using the simplified quasi-steady and unsteady, one way coupled method. Between the quasi-steady and unsteady simulations, an overshoot in aerodynamic yaw moment for the latter resulted in a larger lateral deviation of approximately 8%. However, the differences in responses between the transient, one-way and fully coupled methods were small for this particular geometry. It is expected that by increasing gust length, differences will appear, as the vehicle is exposed to the larger flow yaw angle for a longer period.

CITATION: Forbes, D., Page, G., Passmore, M., and Gaylard, A., "A Fully Coupled, 6 Degree-of-Freedom, Aerodynamic and Vehicle Handling Crosswind Simulation using the DrivAer Model," *SAE Int. J. Passeng. Cars - Mech. Syst.* 9(2):2016, doi:10.4271/2016-01-1601.

INTRODUCTION

In the UK, average annual wind speeds fall in the range of 4-5m/s, which at typical vehicle speeds of 100km/h can produce flow yaw angles up to 10° [1]. In contrast, maximum mean hourly wind speeds of up to 24m/s over flat, open, central parts of the UK have been recorded, albeit with a probability of occurring once in 50-year period [1, 2]. Whilst data describing instantaneous peak values is rare, values greater than this mean can be expected. Nevertheless, these speeds correspond to a much larger maximum flow yaw angle of approximately 40°. Such high angles and speeds present a safety concern as it will be shown that at these values, the forces acting on the vehicle are large enough to cause a substantial course deviation and in some cases, are large enough for the vehicle to encroach upon adjacent lanes.

This issue will become even more prevalent with the current trend towards lighter, more fuel efficient vehicles, in a bid to meet emissions regulations brought about by the 'Worldwide Harmonized Light Vehicles Test Procedure' (WLTP) [3].

It is generally assumed that as any lateral or yaw accelerations are sensed a driver will be able to provide an adequate counter response to a gust to prevent a significant deviation. However, this is not always the case because when the frequency of the gust is combined

with the driver's inputs the driver's response can amplify the vehicle's deviation. This is shown by Wagner and Wiedemann [4], within a frequency range of 0.5-2Hz, to occur as the vehicle motion due to the crosswind and the driver's steering input approach an in-phase state, peaking at a frequency of 1.4Hz. For frequencies <0.5Hz, the driver typically has a positive influence on the vehicle's response, as the gust and loads acting on the vehicle are quasi-steady. Whereas at higher frequencies >2Hz, the driver has little to no influence, as the gust has passed before it is felt.

In order to assess the complete vehicle-driver response to such an event, a flow disturbance has to be generated, either naturally or artificially. The aerodynamic response of a vehicle during a crosswind has been investigated using numerous methods. On road vehicle testing [5, 6, 7, 8, 9] is often preferred as it places a representative driver and vehicle in a real-world environment, whilst crosswind generators beside test tracks [10, 11] offer a degree of control over gust parameters.

An International Standard ISO 12021:2010 [13] has been derived in an attempt to standardize such facilities. These guidelines appear to be based on the work of Howell [11], in which a trifurcated tail pipe attached to the exhaust of a jet engine was used to determine the behavior of a small Sports Utility Vehicle (SUV) when passing

through a severe crosswind. The experimental setup generated a flow yaw angle of 27.5° at a vehicle speed of 100km/h, with a disturbance frequency of 0.7Hz. Gyroscopic rate transducers and accelerometers were fitted to measure the chassis moment rates and lateral accelerations respectively, while course deviation was measured with a dye trail. With a fixed steering input, (steering wheel held in a fixed position) the vehicle reached an average lateral deviation of 2 metres, 2 seconds from the point of entering the crosswind. The downside to these tests is the limited flow field data that can be extracted, making it difficult to identify the flow structures responsible for the deviation. In addition, such tests cannot be performed without a prototype vehicle, at which stage, significant financial investment has been placed into the design and modifications can be very costly.



Figure 1. Crosswind Generator Facility at Applus+ IDIADA Proving Ground, Spain [12].

For this reason, full scale and model scale wind tunnel tests are an appealing alternative. The accepted method of testing such events has changed since the early tests of Beauvais [14], who traversed a vehicle across the width of the working section on a purpose built track. In more recent years, dynamically yawed flow has been generated using vertical oscillating airfoils upstream of the test section, as employed by Bearman and Mullarkey [15], Passmore, Richardson and Imam [16], Schroeck et al. [17], Mankowski, Sims-Williams and Dominy [18] and the system at FKFS Stuttgart [19]. Such devices are capable of generating the large length scales representative of the majority of on-road gusts. In addition, it has been shown that active drag-based devices such as the upstream deployable blades at Pininfarina [20, 21] can be used to generate dynamically yawing flow by controlling the relative phasing of the opening and closing of the blades [22].

Although these upstream devices make it possible to investigate a meaningful region of the aerodynamic coefficients frequency spectra, they are unable to reach the large flow yaw angles representative of an extreme gust that will expose any safety concerns. Ryan and Dominy [23] introduced an alternative arrangement, in which the model is kept stationary in a conventional wind tunnel and a crosswind jet scans along the model. This was achieved with a secondary tunnel placed at a 30° angle to the axis of the main working section. The model is gradually exposed to the secondary flow through a moving belt with variable aperture or in an updated configuration, an actuated shutter mechanism [24, 25]. Although this method allows for large flow yaw angles, the opening and closing mechanism is unable to produce a smooth gust profile with significant under and overshoots in flow yaw angle occurring.

An appealing alternative in the study of crosswind aerodynamics uses computational fluid dynamics (CFD). This type of simulation removes many of the restrictions imposed by physical testing, with parameters such as flow yaw angle and gust length simply given as boundary values. Many studies have reproduced realistic on road turbulence levels using CFD such as Gaylard et al. [26], D'Hooge et al. [27], and Theissen [28], with emphasis on accurate wind averaged drag values. These simulations limit flow yaw angles to a range of $\pm 5^\circ$ and thus can be classed separately to those of isolated extreme gusts which present a safety issue, however such conditions are important to manufacturers because they contribute to quality and refinement issues with respect to noise and ride comfort.

Simulating the impact of gusts on a vehicle's response requires a coupling between aerodynamics and vehicle handling. The coupling method can vary in complexity, and in practice, choice is normally driven by computational resources. However, it is necessary to quantify the differences between the methods and fully understand the shortcomings and advantages of one approach over another, before the choice of a particular method can be justified. Over the last ten years, research in this area has been led by Tsubokura, Nakashima et al. with the emphasis on heavy duty trucks [29, 30, 31, 32, 33, 34, 35]. Due to their large lateral area and extreme weight variations, this class of vehicle are highly susceptible to crosswinds. The coupling methods that have been identified are as follows.

1. **Quasi-Steady Static Coupling** - Lookup table of quasi - steady aerodynamic loads over a range of yaw angles. Used to generate approximate aerodynamic response during the gust for input into the handling model.
2. **Unsteady Static Coupling** - A transient history of aerodynamic loads during a convected crosswind event over a static model (equivalent to the shutter device of Dominy and Ryan) - temporal loads used as handling model inputs.
3. **Unsteady Dynamic Coupling** - A closed-loop system with CFD simulation and handling model running simultaneously. Real-time position of the vehicle used in CFD, calculated based on preceding aerodynamic loads. Closest representation of the on road system.

The simplest approach is a quasi-steady static coupling. This method consists of a set of steady state solutions at different flow yaw angles forming, through interpolation methods, an aerodynamic input for a handling model. In its simplest form this involves two solutions: one at zero yaw and one at the gust's maximum yaw angle, yielding a step input. However, Nakashima et al. [31] show that for a severe crosswind ($>30^\circ$), the quasi-steady approach is unable to capture the overshoot and undershoot of the aerodynamic yawing moment and drag force present in the transient, unsteady static coupling method. This is attributed to a difference in the yaw angle at the front and rear of the vehicle and delayed reaction of the flow passing over the vehicle as it enters and exits the crosswind [32]. This effect will be more pronounced for the long trucks used in their study, but the principle still applies to smaller road vehicles.

Hucho and Emmelmann [36] and Nakashima et al. [31] however, show that where long, shallow gust gradients in relation to the vehicle length and small flow yaw angles $<10^\circ$ exist, a quasi-steady behavior of the transient yawing moment occurs, due to a smaller difference in yaw angle at the front and rear of the vehicle, hence justifying the use of this coupling method. The vehicle dynamic response of Nakashima et al. [31] showed a similar relationship to the aerodynamic response. At small flow yaw angles $<10^\circ$, the quasi-steady coupling method captured the lateral deviation and yaw angle vehicle responses of the unsteady-static method to within an acceptable range. Increasing the yaw angle of the crosswind $>30^\circ$, the differences between the quasi-static and unsteady-static responses were magnified [31, 34]. As expected, reduced vehicle load (essentially a reduction in the weight) also enhanced the differences between a quasi-static and unsteady-static response, albeit for the large load variations typical of heavy duty trucks. This highlights the importance of reduced vehicle weight on stability. A useful reminder and consideration for the current design trend and subsequent testing procedures. The distinct advantage of the quasi-steady approach is the significant reduction in computational expense when compared to the time taken to obtain a corresponding, fully transient aerodynamic response.

In both the quasi-steady static and unsteady static coupling approaches, the vehicle is fixed in the CFD simulation and the resulting aerodynamics are provided to the handling model as inputs; two separate simulations performed sequentially and the coupling is one-way. It offers the advantage of static CFD geometry, negating the need for computationally expensive dynamic grid methods such as rapid-remeshing, grid deformation and overset (Chimera) grids, but neglects any positional change of the vehicle that results from the aerodynamic loads. Thus any damping or intensifying effects that the vehicle deviation may induce on the flow are not included.

An alternative coupling method, and one which can be considered the most representative of real world on road interactions, is unsteady-dynamic. This approach uses one of the aforementioned dynamic meshing models to continually update the vehicle position in the CFD simulation. The positional change is calculated by a handling model running simultaneously using instantaneous aerodynamic loads from the CFD. To date studies that investigate this final approach are limited [33, 35]. Such studies use the grid-deformation method based on the Arbitrary Lagrangian-Eulerian (ALE) formulation combined with a moving reference frame approach to include yaw rotation and lateral translation respectively. Thus these simulations are classed as having only two degrees of freedom (2DoF). In the published work comparisons are only made to the quasi-static coupling and display significant variations in both course deviation and the required driver input. However, these studies do not make comparisons between the two unsteady approaches, hence the value of the more computationally expensive unsteady-dynamic coupling method cannot be verified. There is evidence in the predicted loads to suggest that rolling moment and lift force have a sizeable impact on the vehicle response, thus a 6DoF simulation that includes rotation and translation about all three axes may yield a different response and hence should be investigated.

It is the purpose of this work to quantify the differences between the various coupling methods so that a chosen method can be used to assess crosswind sensitivity much earlier in the development of a production vehicle. In addition, the source of any differences in response will be identified in the flow structures. This will demonstrate the practicality of using such a simulation for shape modifications or testing of stability devices before prototype manufacture. The chosen vehicle for this study is a full-scale representation of the DrivAer model's fastback variant [37]. This variant was chosen over the notchback and estate back options as it has been shown to generate a larger aerodynamic yawing moment when subjected to crosswinds due to a reduction of rear side force in accordance with rear-end area and thus deemed more sensitive to gusts [38]. The specific configuration of the model includes a smooth underbody, closed body with mirrors, fixed non-rotating wheels with smooth tread-less tyres.

CROSSWIND PROFILE

The simulated crosswind event follows the conditions set out in the International Standard ISO 12021:2010 [13], in an attempt to ease any future experimental validation. This standard calls for a crosswind velocity of 20m/s and vehicle speed of 27.8m/s (100km/h) thus generating a flow yaw angle of approximately 36° (a severe gust). At its maximum, this arrangement generates a resultant velocity of 34m/s and Reynolds number of approximately 10 million (based on vehicle length). For a two second period after the vehicle enters the crosswind, there is no driver response and the steering wheel is held fixed in accordance with the standard. This condition removes the variability of a driver's response, advantageous for repeatability and simulation, but also gives a good indication of vehicle deviation when a driver is tired or distracted. A gust length of approximately 18.5m, (four car lengths, 4L) is above the minimum length prescribed by the standard, and at this vehicle speed, corresponds to an event duration of 0.66 seconds for a fixed point on the vehicle. This time period corresponds to a disturbance frequency of 1.5Hz. It has been shown that around this frequency, the driver's and vehicle reactions are in phase, thus intensifying the vehicle reactions to their full extent [4].

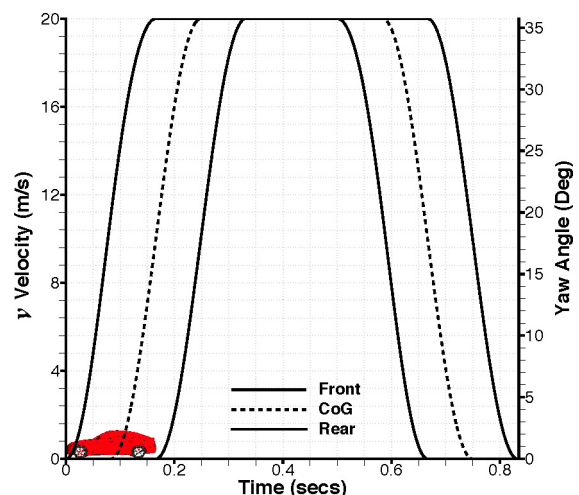


Figure 2. Crosswind Profile (Vehicle speed $u=100\text{km/h}$, disturbance frequency 1.5Hz).

The crosswind has a trapezoidal profile with maximum velocity of 20m/s and thus assumes the crosswind facility generates a constant band of flow. The two mixing layers either side of the generator are modelled by cosine functions based on the experimentally validated results of Hucho and Emmelman [36], with lengths of one vehicle length each. Thus the vehicle is only fully immersed in the crosswind for distance equal to one vehicle length. Although shorter in duration, the overall profile is similar to that used by Favre [39] and as shown by Wojciak et al. [40], the shape is representative of approximately a third of all on-road gusts (although on-road tests were only conducted in a single location over several days).

CFD BOUNDARY CONDITIONS

To impose the crosswind profile on the flow, time dependent boundary conditions are required. The method used by Demuth and Buck [41], uses periodic boundaries as compared to the more traditional method of multiple inlets and outlet used by Tsubokura, Nakashima et al. [29, 30, 31, 32, 33, 34, 35] and Favre [39]. Advantages of this boundary type are their unreflective nature and the single time-dependent velocity function requirement at the main inlet to convect the profile downstream. When using this method, it is important that disturbances re-entering the domain do not influence the flow over the vehicle, but this is rare at typical vehicle speeds and can always be resolved by widening the domain. This method uses a reference frame fixed on the vehicle and thus the u velocity component of the flow represents the reciprocal of the vehicle velocity, as is standard practice from a wind tunnel point of view. However, an issue will arise during the unsteady-dynamic simulation when the vehicle yaws in response to the crosswind: unless the u velocity component of the flow yaws at the same rate as the vehicle it will no longer represent the reciprocal of the vehicle velocity. It is not possible to impart this additional yaw angle to the flow, since any perturbation has to be defined at the inlet in advance, at which point the vehicle yaw angle is unknown.

The overset grid method provides a way around this difficulty by switching the reference frame and allowing a sub-mesh containing the vehicle to move at a constant forward speed through a fluid domain of still air, and static band of crossflow. Thus as the vehicle yaws in response to the crosswind, it will remain at a constant forward speed in its own reference frame. The actual boundaries for this method are similar to that used by Tsubokura, Nakashima et al. [29, 30, 31, 32, 33, 34, 35] and Favre [39] with two inlets and two outlets however there is now no time dependency and the only conditions are the v crosswind velocity profile on the crosswind inlet.

Consequently, the three coupling approaches each use a different set of boundary conditions. The quasi-steady static coupling uses two inlets and two outlets without any time dependency to impinge a constant flow yaw angle on the vehicle, Figure 3. The unsteady static coupling uses the periodic boundary conditions whilst the unsteady dynamic coupling uses the overset boundary conditions, with two inlets and outlets Figures 4 and 5. All methods use the slip wall condition on the top and bottom boundaries, with fixed, grounded, non-rotating wheels and tyres.

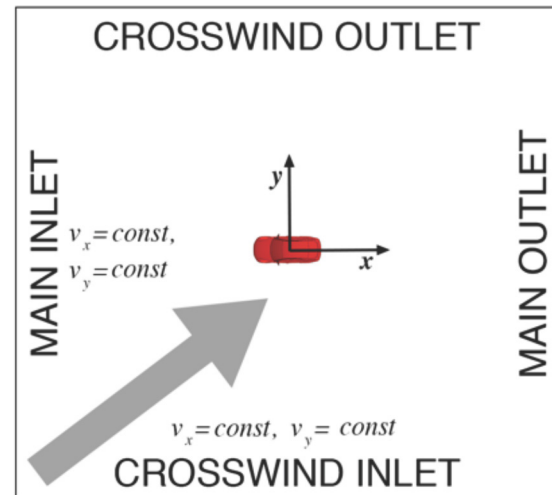


Figure 3. Quasi-Steady Static Boundary Conditions.

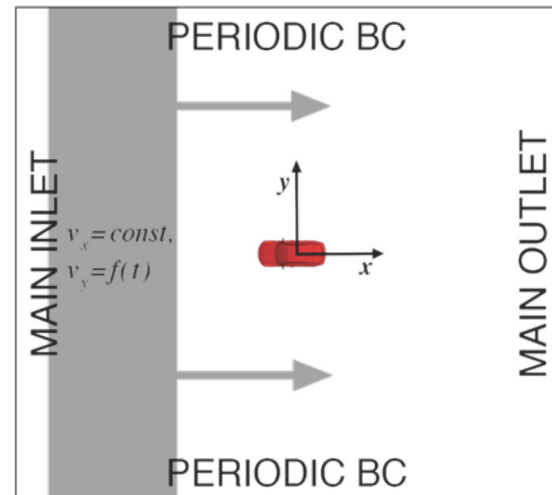


Figure 4. Unsteady Static Boundary Conditions.

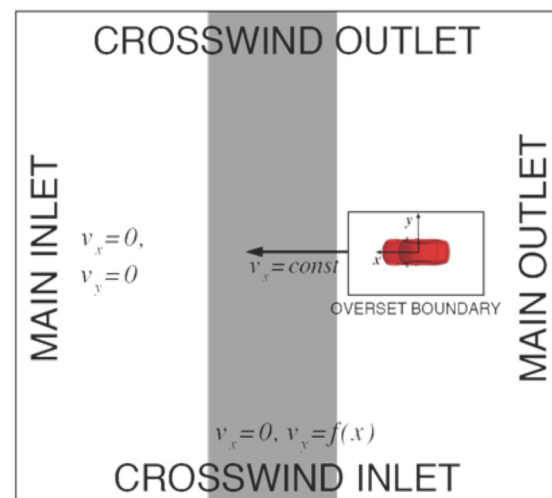


Figure 5. Unsteady Dynamic Boundary Conditions.