

The Effect of a Sheared Crosswind Flow on Car Aerodynamics

Jeff Howell, David Forbes, Martin Passmore, and Gary Page
Loughborough University

ABSTRACT

In the wind tunnel the effect of a wind input on the aerodynamic characteristics of any road vehicle is simulated by yawing the vehicle. This represents a wind input where the wind velocity is constant with height above the ground. In reality the natural wind is a boundary layer flow and is sheared so that the wind velocity will vary with height. A CFD simulation has been conducted to compare the aerodynamic characteristics of a DrivAer model, in fastback and squareback form, subject to a crosswind flow, with and without shear. The yaw simulation has been carried out at a yaw angle of 10° and with one shear flow exponent. It is shown that the car experiences almost identical forces and moments in the two cases when the mass flow in the crosswind over the height of the car is similar. Load distributions are presented for the two cases. The implications for wind averaged drag are discussed.

CITATION: Howell, J., Forbes, D., Passmore, M., and Page, G., "The Effect of a Sheared Crosswind Flow on Car Aerodynamics," *SAE Int. J. Passeng. Cars - Mech. Syst.* 10(1):2017, doi:10.4271/2017-01-1536.

INTRODUCTION

The aerodynamics of a car in a crosswind are only approximately modelled in conventional experimental or virtual simulations. This mainly occurs because an essentially transient event is studied using steady state techniques. In the wind tunnel the effect of a crosswind is represented by setting the vehicle at a yaw angle to the flow. CFD simulations usually mimic this arrangement. In both cases the flow field simulates a car travelling on an open road with a crosswind where the wind velocity is constant with height above the ground. In the real world a road vehicle in a crosswind is immersed in the lowest region of the earth's atmospheric boundary layer. This sheared flow, where the velocity is zero at the ground surface and increases with height above ground, gives rise to a vehicle flow field where both local yaw angle and resultant velocity increase over the height of the car.

The typical wind tunnel and CFD simulations of a crosswind also represent a wind input which is both steady and low turbulence, whereas the natural wind can be highly unsteady and can include significant levels of turbulence, as shown schematically in [Figure 1](#). For a general review of the effect of the natural wind on the aerodynamics of road vehicles, see Howell, [1], and Cooper and Watkins, [2]. The information regarding natural wind properties at height above ground relevant for cars, i.e. <2m, is extremely limited and the available data is extrapolated from that obtained by the building and wind turbine aerodynamics communities at much greater heights above ground. Meteorological data is typically obtained at a height of 10m.

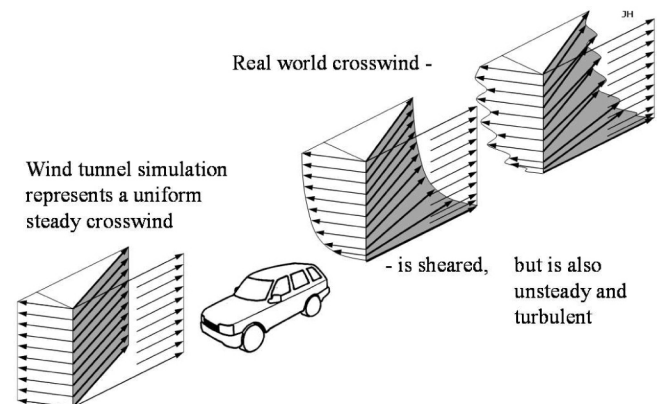


Figure 1. Comparison of natural wind and simulated flowfields.

The variation of the natural wind velocity, U_z , with height, z , above ground is defined by a power law relationship, such that:

$$U_z = U_{10} (z/10)^\alpha \quad (1)$$

where U_{10} is the wind velocity at 10m and the exponent, α , is dependent on the terrain. Typical values of α for different terrains are given in [Table 1](#), mainly based on Scruton,[3]. Z_G is the associated depth of the atmospheric boundary layer for each terrain.

Table 1. Natural wind characteristics for different terrains.

Terrain	Z_G (m)	α
Open Terrain, Few Trees	300	0.16
Rural, Many hedges	365	0.22
Woodland, Suburbs	430	0.28
City centre, Broken country	500	0.40

There have been almost no studies into the effect of a sheared flow on the aerodynamics of road vehicles. An early study by Maccabee, [4], introduced shear into the onset flow which is not really appropriate. The only relevant work known to the authors is by Macklin et al, [5], from Macklin, [6], which used a moving model set-up. The model was propelled across a wind tunnel working section, used for building aerodynamics studies, where the boundary layer profile could be varied over a significant range. By comparing the load distributions for the different wind profiles it was hoped to establish a relationship between sheared and non-sheared flows. With all the problems inherent in moving model testing, however, no firm rule could be established, but basing the lift and lateral aerodynamic coefficient data at yaw on the dynamic pressure integrated over the height of the car model gave an improved consistency for most of the shear flow cases investigated. No measurement of drag could be obtained from this experimental set-up.

In this paper only the effects of shear are investigated, as shown in the central flow diagram sketched in Figure 1. In the real world, as described above, a crosswind would also include a high degree of turbulence, both along and normal to the wind direction, but it is useful, initially, to separate these effects. As it is very difficult to set up an experimental investigation of a sheared crosswind flow without using a moving model, the problem has been studied using CFD.

COMPUTATIONAL GRID AND BOUNDARY CONDITIONS

The vehicle geometry used for these simulations is the DrivAer model as proposed by Heft et al. [7], which features a range of interchangeable rear end geometries. As a result, the effects of shear on both fastback and squareback vehicle types has been investigated. The smooth underbody and closed engine bay options were included to reduce computational cost. Important dimensions of the model are provided in Table 2.

Table 2. DrivAer model important dimensions.

Length (L)	4.613m
Wheelbase Length	2.786m
Frontal Area	2.168m ²
Height	1.418m
Width	1.753m
Re_L	8.9 x 10 ⁶

For each vehicle type, the full-scale geometry is positioned in a computational domain of size ($x=18L$, $y=13L$, $z=3L$) at a distance of $5L$ and $6L$ from the primary and secondary inlets respectively, as shown in Figure 2. On most of the vehicle surface, grid elements of size $0.001L$ are used, however smaller elements of size $0.0001L$ are required in certain locations in order to maintain a reasonable quality of highly curved, critical features such as A and C pillars. As a result, the total number of surface elements on the fastback and squareback is 2.7M and 2.8M respectively.

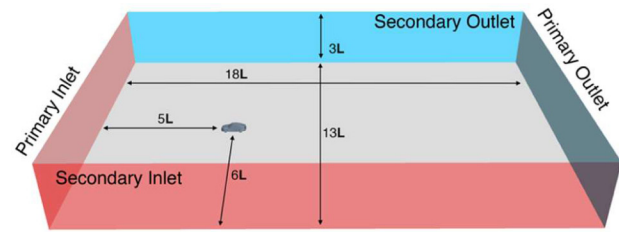


Figure 2. The computational domain.

Extruded from the surface, 8 prism layers to a total thickness of $0.001L$ are used to capture the boundary layer. A non-dimensional near wall spacing value $y^+ < 1$ over the entire surface ensures that the boundary layer is resolved and not modelled using wall functions.

Two hexahedral volume grids were generated using identical parameters: elements grow uniformly in size from the surface into a region of fixed $0.002L$ sized elements. This region stretches $0.5L$ downstream and $0.3L$ to the leeward side of the vehicle in order to capture the yawed wake. This first refinement region is enclosed by a second, of fixed sized $0.017L$ elements. This region stretches much further downstream, a length of $2.7L$ and leeward width of $1L$. These refinement regions are shown in Figures 3 and 4. Away from this second refinement, grid elements grow up to a maximum size of $0.14L$. As a result, the total number of volume elements is approximately 69M for both grids.

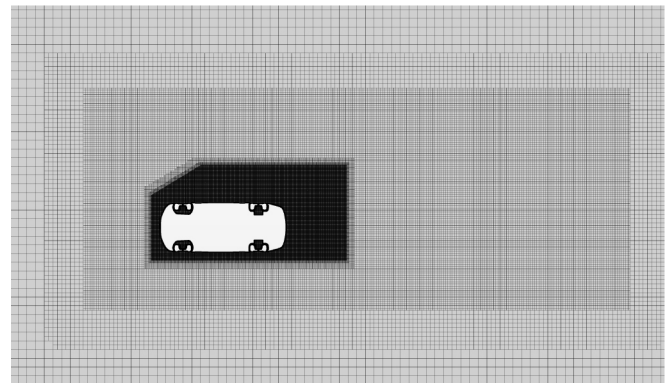


Figure 3. Horizontal slice through the fastback volume grid at axle height.

On the floor of the domain, a slip wall boundary condition is defined which may appear counter-intuitive in the simulation of a realistic atmospheric boundary layer. For these simulations, the intention is to maintain the same sheared wind velocity profile that is defined at the inlets at the vehicle. For this reason, the slip-wall boundary condition with 6 prism layers stretching to a total thickness of $0.04m$ is used.

A consequence of this approach is that any local ground boundary layer effects are not simulated. An alternative approach which uses a no-slip boundary condition and allows exact parameterization of the profile when it reaches the vehicle would require a modified shear profile that takes into account the boundary layer formed over the upstream distance from the vehicle to the inlet.

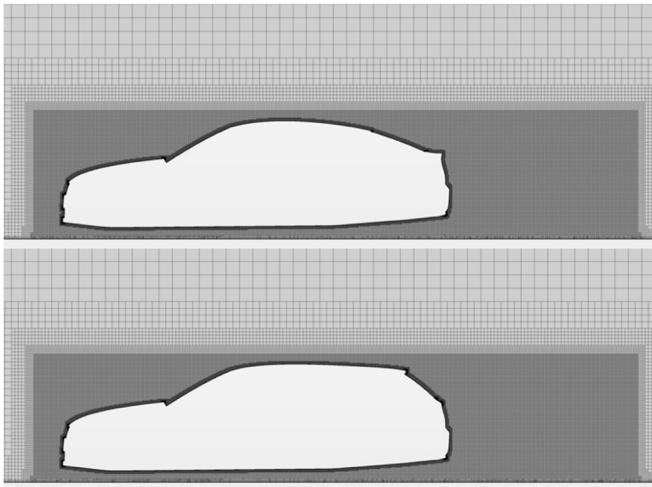


Figure 4. Vertical slice through the volume grids of the fastback and squareback along the centerline.

At the two inlets, the U_v velocity component is fixed at 27.8m/s (~62mph) and for the unsheared simulation, the U_w velocity component takes a value of 4.9m/s (~11mph). The wind input is assumed to act at right angles to the path of the car. This arrangement does not give the maximum loading, but is convenient to apply and generates a yaw angle of approximately 10° and a resultant velocity of $U_R = 28.23\text{m/s}$. For the sheared simulation, the U_w velocity component takes the profile shown in Figure 5 which has been calculated using Equation 1 and assumes the same mass flow over the height of the vehicle as found in the unsheared case. The power law exponent, $\alpha = 0.16$, implies open, smooth terrain.

For a wind velocity profile given by a power law, as equation (1), with exponent, α , the mean velocity, U_{WM} , over any height, H , is given by:

$$U_{WM} = U_{WH}/(\alpha+1) \quad (2)$$

where U_{WH} is the wind velocity at height, H . The height, z , at which this velocity occurs on the shear profile is given by:

$$z/H = (\alpha+1)^{-(1/\alpha)} \quad (3)$$

As a result, the dashed line, which represents the unsheared profile, has a velocity of 0.862 times the velocity at the roof of the car and is the velocity at 39.5% of the height of the car.

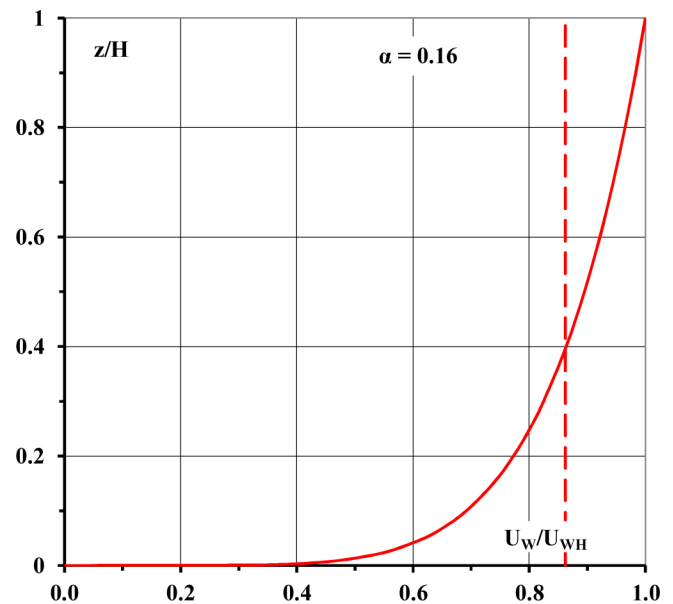


Figure 5. Sheared and unsheared crosswind profiles.

NUMERICAL METHODOLOGY

The numerical method used for these simulation is taken from Forbes et al. [8] who used the fastback geometry in the computational study of crosswind loadings on vehicle dynamics and is based on the thorough computational methodology assessment for the DrivAer model performed by Ashton et al. [9].

A segregated, incompressible, finite volume, semi-implicit, pressure based solver is chosen for all simulations with a hybrid 2nd order upwind/bounded central-differencing convection scheme. Turbulence is modelled with the IDDES variant of the Spalart-Allmaras Detached-Eddy model. A second-order temporal scheme limits the numerical dissipation of the sheared and unsheared velocity profiles and time-step of $1 \times 10^{-4}\text{s}$ ensures a Courant number below one within the LES region. Five inner iterations per time-step ensure consistent convergence of the residuals. All CFD simulations were carried out using Star-CCM+ software v10.04.009 from CD-Adapco.

All simulations were initialized using a steady state RANS solver, before switching to DES for a settling period of 1 second or 6 convective flow units; ($6 \times L/U_R$). The simulations were then allowed to run for 2 seconds (12 convective flow units) during which the flow field was averaged. Ideally, this averaging interval would be longer but due the number of simulations in this study, computational resources were limited. One 3 second simulation, took approximately 58 hours to run in parallel on 320 cores of the HPC-Midlands Facility, UK.