

# Towards realistic ‘on-road’ flow conditions for a simple vehicle model.

C. Wallace<sup>a</sup>, A. Garmory<sup>b</sup>, A. Gaylard<sup>c</sup>, D. Butcher<sup>d</sup>

<sup>a</sup> Loughborough University, Loughborough, Leicestershire, United Kingdom, [c.wallace@lboro.ac.uk](mailto:c.wallace@lboro.ac.uk)

<sup>b</sup> Loughborough University, Loughborough, Leicestershire, United Kingdom, [a.garmory@lboro.ac.uk](mailto:a.garmory@lboro.ac.uk)

<sup>c</sup> Jaguar Land Rover, Gaydon, Warwickshire, United Kingdom, [agaylar1@jaguarlandrover.com](mailto:agaylar1@jaguarlandrover.com)

<sup>d</sup> Loughborough University, Loughborough, Leicestershire, United Kingdom, [d.butcher@lboro.ac.uk](mailto:d.butcher@lboro.ac.uk)

## SUMMARY:

This study evaluates numerical modelling strategies for predicting underbody and wake aerodynamics of a simplified vehicle model, focusing on the performance of low  $y^+$  (‘wall-resolved’) and high  $y^+$  (‘wall-modelled’) LES approaches. Using a Windsor body, CFD predictions are compared against wind-tunnel measurements with detailed quantification of random uncertainty and statistical convergence. The results highlight the challenges in accurately capturing lift behaviour and the sensitivity of wake-driven unsteadiness. Building on the validated low  $y^+$  LES baseline, the study investigates progressively more realistic boundary conditions, first adding wheel rotation and then adopting a moving ground plane with reduced blockage, to assess their influence on ‘On-Road’ aerodynamic behaviour. The findings provide insight into modelling practices the accuracy of simulating wheel-ground interactions and the validity of simplified boundary conditions.

**Keywords:** Underbody Aerodynamics; Windsor Body; WMLES; Vehicle Wake; Rotating Wheels; Moving Ground; CFD Validation; Turbulent Wake Dynamics

## 1. INTRODUCTION AND OBJECTIVE

For accurate simulation of vehicle aerodynamics, the choice of numerical model and boundary conditions is crucial, particularly in relation to the wall-resolution strategy. This consideration has become increasingly important with the rapid development of battery electric vehicles (BEVs), where attention has increasingly shifted toward optimising underbody aerodynamics. The absence of conventional powertrain constraints provides unprecedented design freedom in the underfloor region, making accurate modelling of ground and wheel interactions essential. The smooth underbody also places more importance on the treatment of boundary layers in CFD. The aerodynamic characteristics of the Windsor body, both with and without rotating wheels, have been extensively documented in experimental and numerical studies (Pavia, Passmore, & Sardu, 2018) (Pavia, Passmore, Varney, & Hodgson, 2020). More recently, Page and Walle (Page & Walle, 2022) have provided a benchmark for assessing automotive CFD prediction capability across various modelling approaches. These studies collectively highlight both the progress and the limitations in current predictive methodologies, particularly regarding wake symmetry/asymmetry, vortex dynamics, and underbody flow development. The present work attempts to evaluate the effectiveness of LES wall-modelling strategies for simplified vehicle geometries. Low  $y^+$  (‘wall-resolved’) and high  $y^+$  (‘wall-modelled’) LES configurations are compared directly against wind-tunnel measurements of aerodynamic forces and surface pressures. The comparison accounts for both random and systematic measurement uncertainty to quantify the reliability of each numerical approach. Given the growing significance of the underbody region in overall vehicle performance, a study is also carried out to identify the most effective modelling practices for capturing true wheel–ground interactions and underfloor flow

behaviour. The study explores the extent to which the simplified model and dataset are representative of realistic ‘on-road’ conditions, with particular emphasis on the influence of wheel rotation and relative road motion on the underbody flow field.

## 2. SETUP AND METHODOLOGY

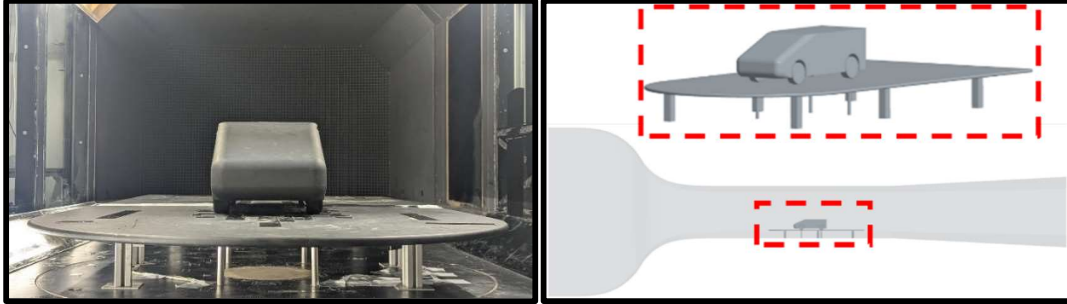
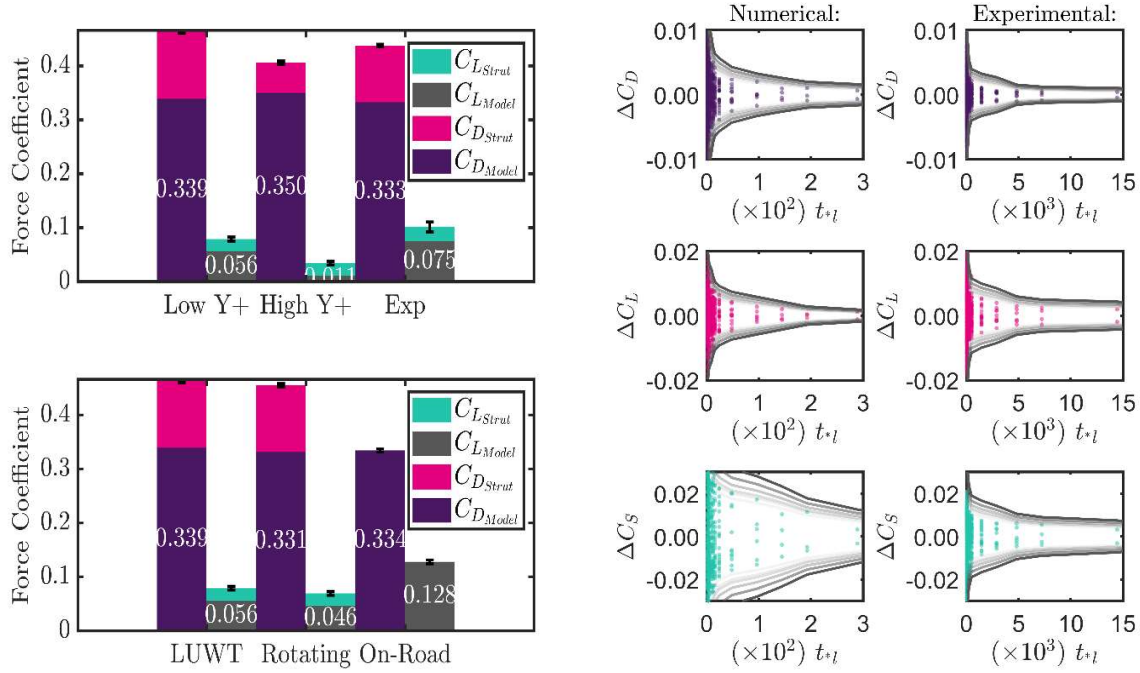


Figure 1: *Left* - Windsor model in the LUWT. *Right* – Numerical wind tunnel setup with details of the model and false floor.

The geometry used throughout this work is the square-back version of the Windsor Body, a simple, one-box bluff body which is effectively fifth scale compared to a full-sized SUV (Windsor, S. C., 1991). Chosen in part due to the availability of CFD studies compared in the Auto CFD workshop (Page & Walle, 2022) and experimental testing conducted (Varney, Pavia, & Passmore, 2024). Its design is relevant to modern sports utility vehicles (SUVs), a key automotive shape that seen a surge in market share as BEV technologies gain prominence. The experimental dataset was obtained in the Loughborough University Large Wind Tunnel (LUWT), an open-circuit wind tunnel with a closed working section measuring 3.6 m long by 1.32 m high by 1.92 m wide respectively. The ground plane of the wind tunnel is fixed, therefore, to reduce the height of the boundary layer at the model location a raised ground plane was introduced. Its top surface was 0.205 m above the wind tunnel floor which takes it out of the boundary layer. All tests were carried out at a freestream velocity of  $40 \text{ m/s}$  which gives a height-based Reynolds number ( $Re_h$ ) of approximately  $6 \times 10^5$ . The experimental setup provides a solid blockage of 4.8%. Force data from a six-component balance and pressure data for a 126 pressure taps have been taken for comparison to the numerical dataset. Both force and pressure measurements have been left uncorrected to allow direct comparison between datasets. The simulations within this work were conducted in Siemens StarCCM+ (Siemens, 2022) using both the low and high  $y^+$  wall modelled LES approaches with the Wall-Adapting Local Eddy-Viscosity (WALE) model. The meshing was conducted using mesh discretisation similar to the schemes prescribed in the Auto CFD workshop (Page & Walle, 2022) giving an overall cell count of around 57 million cells for the low  $y^+$  and 25 million cells for the high  $y^+$  mesh, respectively. The domain, shown in Figure 1 (*Right*), was modelled to accurately copy the real tunnel setup with the inclusion of the false floor and mounting structure to fully simulate the experiment. Velocity inlet was set to match the experimental set up. A time-step of  $2 \times 10^{-5} \text{ s}$  is used to ensure eddies were sufficiently resolved in time to give a Courant number  $\leq 1$ , and the simulation is run for 12 simulated seconds to allow assessment of random uncertainty. The boundary conditions were subsequently adapted to serve the needs of this work by adding a moving ground plane and lower blockage domain.

### 3. COMPARISON OF FORCES AND UNDERSTANDING UNCERTANTIES



**Figure 2: Left Top** - Comparison of numerical force coefficients simulated with ‘wall-modelled’ and ‘wall resolved’ wall treatment validated against experimental force coefficients. **Left Bottom** - Comparison of wind tunnel setup CFD to wind tunnel setup CFD with rotating wheels and full ‘On-Road’ conditions CFD (Wallace, Garmory, Gaylard, & Butcher, 2025). **Right** - Uncertainty of numerical and experimental forces time series with the rolling mean force coefficient delta presented in purple, pink and teal, respectively, and the 85% to 99% confidence bands depicted in grey.

An initial comparison of mean force coefficients is shown in Figure 2 (Left Top). The  $C_D$  value predicted by both the high and low  $y^+$  CFD over-predicts the drag on the model by  $\sim 0.01 C_D$  and  $\sim 0.02 C_D$ , respectively. The drag on the mounting struts shows a larger discrepancy, with the low  $y^+$  CFD over-predicting the strut drag by a further  $\sim 0.01 C_D$  and the high  $y^+$  CFD under-predicting the strut drag by  $\sim 0.05 C_D$ . This trend is also seen in the agreement of the  $C_L$  values. Both the high and low  $y^+$  CFD under-predict the  $C_L$  of the model and the mounting struts, a trend also reported in simulations by Page and Walle (Page & Walle, 2022). Considering just the model, the low  $y^+$  CFD shows agreement within  $\sim 0.02 C_L$  of the experimental values, which is marginally closer than the best agreement seen by Page and Walle (Page & Walle, 2022). The high  $y^+$  CFD shows poorer agreement, with a difference of  $\sim 0.06 C_L$ . The difference in strut lift is negligible, with the low  $y^+$  CFD under-predicting by  $\sim 0.004 C_L$  and the high  $y^+$  CFD by  $\sim 0.003 C_L$ . The high  $y^+$  CFD does not appear to accurately predict the lift shown in the experimental values or the expected drag on the mounting struts. Overall, the low  $y^+$  CFD appears to give a better prediction of the experimental force results. It should also be noted that the model-only forces are obtained differently in CFD and experiment. In the wind-tunnel measurements, the model load is derived by subtracting the strut-only forces from the combined model-plus-strut measurement. In contrast, the CFD determines the model and strut forces by integrating pressure and shear stresses over dedicated

surface groups. These approaches are not strictly equivalent and may contribute to some of the discrepancies observed, particularly in the strut loads and their influence on the overall  $C_L$  and  $C_D$ .

Random uncertainty was quantified for both datasets using rolling-mean confidence bands in Figure 2 (*Right*). The numerical  $C_D$  and  $C_L$  values converge within tight limits of  $C_D \pm 0.0015$  and  $\Delta C_L \pm 0.0018$ , respectively after approximately 200 characteristic time scales, while the experimental data require substantially longer to reach statistical convergence of sampling ( $\Delta C_D \pm 0.001$  and  $\Delta C_L \pm 0.005$  at approximately 5000 characteristic time scales) due to measurement noise. The larger uncertainty in  $C_L$  may contribute to the more significant discrepancy in the agreement of the numerical to the experimental lift predictions. Assessing the overall uncertainties, achieving statistical convergence in the confidence prediction bands requires both numerical and experimental datasets to be run for durations significantly longer than typical industry practice.

Using the low  $y^+$  CFD as the baseline, given its strong agreement with experimental data, the subsequent comparisons examine the effect of progressively more representative boundary conditions. First, the original wind-tunnel configuration is repeated with the addition of imposed wheel rotation. This is then followed by a comparison against the dataset of Wallace et al. (Wallace, Garmory, Gaylard, & Butcher, 2025), which employs a low-blockage domain with a moving ground plane and rotating wheel boundary conditions to approximate ‘*On-Road*’ conditions. As a method of predicting  $C_D$  the wind tunnel setup appears to present a good approximation of the more realistic boundary conditions (Figure 2 (*Left Bottom*)) with only a difference  $\sim 0.005 C_D$  to the true ‘*On-Road*’ conditions, however this doesn’t yet factor blockage effects. The largest discrepancy in the results is in the lift prediction which appears to minorly reduce in magnitude in the tunnel setup when rotating the wheels but doubles in value with the moving ground plane and rotating wheels. In the following paper/presentation, further analysis of the flow fields and component force distribution will be conducted to determine the sources of discrepancy between wind-tunnel data and CFD, as well as to further investigate the differences when comparing wind-tunnel experiments to ‘*On-Road*’ conditions.

#### ACKNOWLEDGEMENTS

Cameron Wallace is funded by a studentship provided by Jaguar Land Rover (JLR) and the UK Engineering and Physical Sciences Research Council (EPSRC), Project Reference:2812979.

#### REFERENCES

- C. Wallace, A. G. (2025). Investigation of the link between vehicle underbody and base unsteady wake aerodynamics. *Journal of Wind Engineering and Industrial Aerodynamics*.
- Page, G., & Walle, A. (2022). Towards a Standardized Assessment of Automotive Aerodynamic CFD Prediction Capability - AutoCFD 2: Windsor Body Test Case Summary. SAE Technical Paper.
- Pavia G, P. M. (2020). Salient three-dimensional features of the turbulent wake of a simplified square-back vehicle. *Journal of Fluid Mechanics*.
- Pavia, G., Passmore, M., & Sardu, C. (2018). Evolution of the bi-stable wake of a square-back automotive shape. *Exp Fluids*.
- Pavia, G., Passmore, M., Varney, M., & Hodgson, G. (2020). Salient three-dimensional features of the turbulent wake of a simplified square-back vehicle. *Journal of Fluid Mechanics*.
- Siemens . (2022). Siemens STAR CCM+ User Guide, V17.02.
- Varney, M., Pavia, G., & Passmore, M. (2024, 06 04). Windsor Automotive Reference Model. Retrieved from NWF: <https://www.nwtf.ac.uk/dataset/1709/>
- Wallace, C., Garmory, A., Gaylard, A., & Butcher, D. (2025). Investigation of the link between vehicle underbody and base unsteady wake aerodynamics. *Journal of Wind Engineering and Industrial Aerodynamics*.