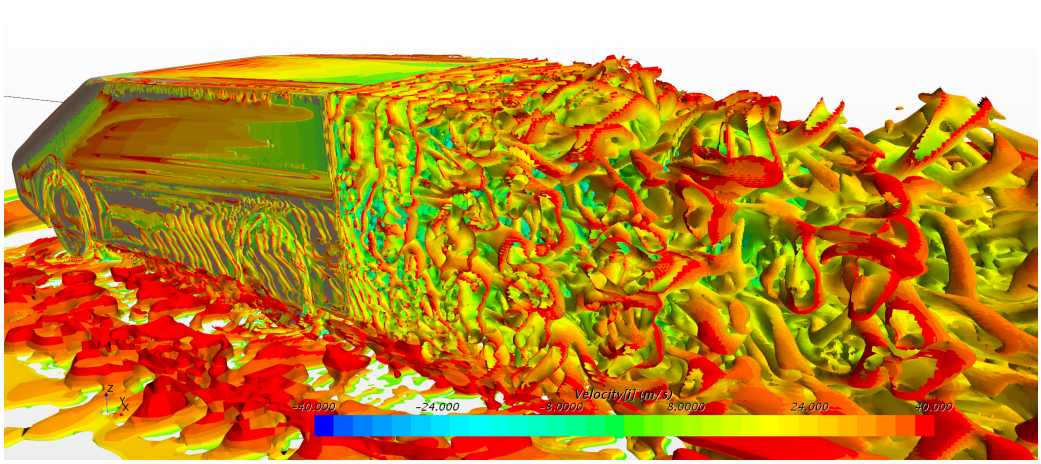
2nd Automotive CFD Prediction Workshop:
Case 1- Windsor Model

Gary J. Page
G.J.Page@lboro.ac.uk
Loughborough University
V0.3 13 April 2021



Introduction

This case requires the simulation of a simplified vehicle-like shape in wind tunnel conditions and is intended to capture the important flow-field structures without needing to model complex geometrical detail as is found in the DRIVAeR case.

The Windsor model, as developed by Steve Windsor of Jaguar Land Rover, used here has been modified to include a second version with wheels. Further details are given in the paper by Pavia et al. [1] and the PhD thesis of Varney[2].

These two cases have had extensive measurements taken at the Loughborough University wind tunnel at a Reynolds number of approximately 3 million (based on vehicle length). The full dataset is available on the Loughborough University data repository [3]

Two CAD definitions are provided for the ‘with wheels’ (1A) and ‘no wheels’ (1B) geometry. For each case, four standard meshes are provided: low Re Wall Resolved RANS (WRRANS), Wall Modelled RANS (WMRANS), low Re Wall Resolved Eddy Resolving (WRER), Wall Modelled Eddy Resolving (WMER). For the WRER case, the high aspect ratio near the wall makes this suitable for DES and similar hybrid RANS-LES approaches, but it is not recommended to be used for a classic eddy resolving LES that requires *both* a low y^+ and modest aspect ratios at the wall. The WMER has a target y^+ of around 100, with aspect ratios less than 1.5.

Geometry and Domain

The model geometry (with wheels case) is shown in Figure 1.

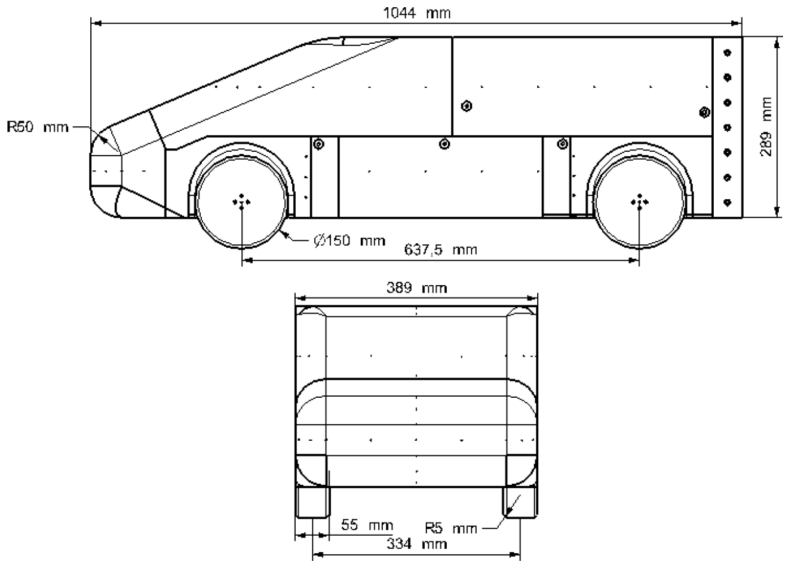


Figure 1 Windsor Model (with wheels) [Varney]

The reference frontal area is defined by the vehicle height and width and rounded to be $0.112m^2$. ($0.056m^2$ for half model)

The CAD geometry of the model has its origin on the ground plane, in the symmetry plane midway between the wheels. The coordinate system has x in the streamwise direction (hence the nose is negative x), z upwards and hence positive y is towards the right of the vehicle. The nose of the vehicle is at x=-0.56075m, the rear at x=0.4835m. (The sides of the car are at y==+/-0.1945m, the car underbody at z=0.05m and the car roof at z=0.339m).

The model is mounted in the wind tunnel with four pins at a ground clearance of 50mm and zero pitch. These pins should be included in the integration of the force coefficients.

For the with wheels (1A) model the wheels are physically connected to the model and are *not rotating*. The ground plane is fixed and to avoid touching, there is a small 'trough' under each wheel. To avoid this complication, the ground plane has been shifted down 3mm to provide a small gap under each wheel equal to that between the bottom of the wheel and the 'trough'. This makes the CFD ground clearance 53mm.

The experimental wind tunnel has a 3.2m long working section with a 1.92m wide x 1.32m high cross section expanding to 1.94m wide x 1.32m high at the end of the section. There is no moving ground plane and so boundary layers grow along the walls. Experimental measurements [4] at the centre of the working section quote a boundary layer thickness of 60mm, displacement thickness of 9.4mm and momentum thickness of 5.5mm. The maximum turbulence intensity was measured to be approximately 3% at the edge of the boundary layer.

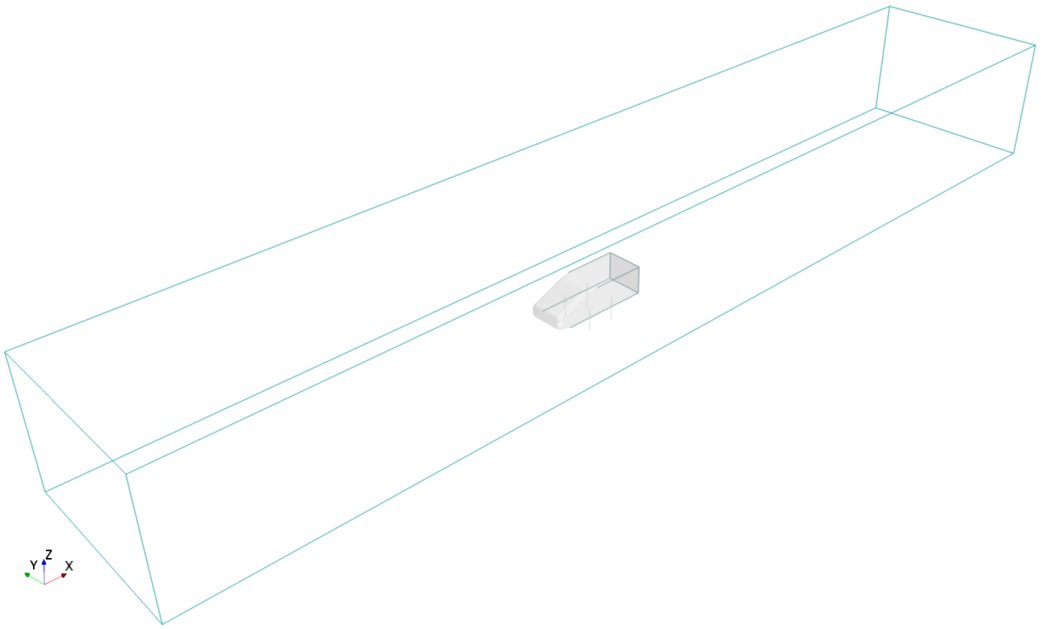


Figure 2: Computational wind tunnel domain

The domain required for this case (see Figure 2) represents the wind tunnel confinement but with the following modifications:

1. Parallel walls
2. Only the ground plane has a no slip condition and hence has boundary layer growth
3. A long parallel inlet run is used in order to grow a boundary layer on the ground plane of approximately the correct thickness.
4. A parallel exit run is added downstream to avoid interactions with the wake.

The domain extends upstream to x=-5m and downstream to x=+6m (the model is x=-0.56m nose to x=+0.48m base). The width and height of the CFD domain matches the wind tunnel.

An empty wind tunnel with these dimensions was set up in CFD with spacings identical to the low Re (WR) grids. When run with the SST k-ω turbulence model the simulated boundary layer height at the centre of the working section matched the experimental values well and had a turbulence intensity at the edge of the boundary layer of approximately 1.5%, increasing to a peak of 5% close to the wall.

Any user grids should use these same domain dimensions and boundary condition types.

Grids

The RANS grids exploit symmetry by just capturing half a car with a symmetry boundary condition at y=0. (For convenience we have chosen the negative y half so that when viewed the flow is left to righ). The Eddy Resolving grids capture the whole car. The grids have been designed to be as consistent as possible across the different variants. For example, the surface mesh spacing on the car is identical for all grids – they tend to vary in terms of the boundary resolution and resolution in the wake and under-vehicle region.

The grids are generated using a Cartesian trimmer mesh with prism layers on no-slip walls. For the Wall Resolved (WR) grids, the y^+ is 0.5-1.5 on the car, and 1-2 for the ground plane.

For the Wall Modelled (WM) grids, the y^+ is 60-90 on the car, and 60-140 for the ground plane.

The grid spacing on the ground plane and vehicle has been chosen to meet certain y+ and aspect ratio requirements as shown in Table 1. Two wake refinement regions are included downstream, and a third refinement region to control the prism layer added under the car and extending under the wake refinement. (See Figure 3 and Figure 4).

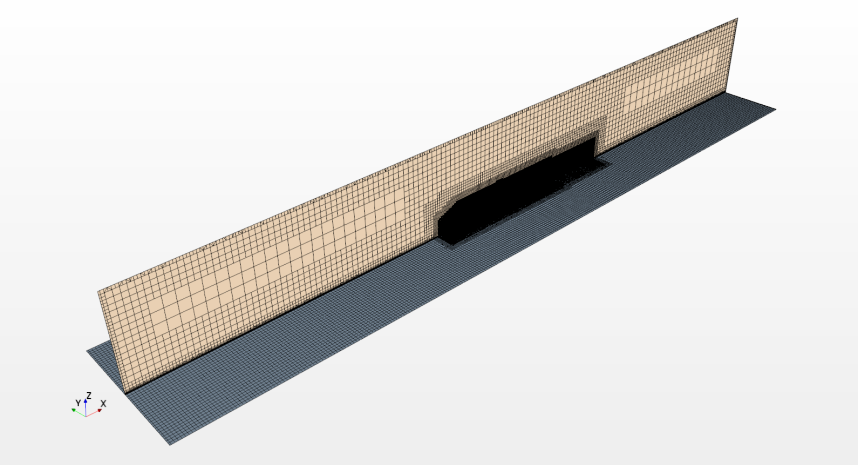


Figure 3: Overall grid (1B WRRANS)

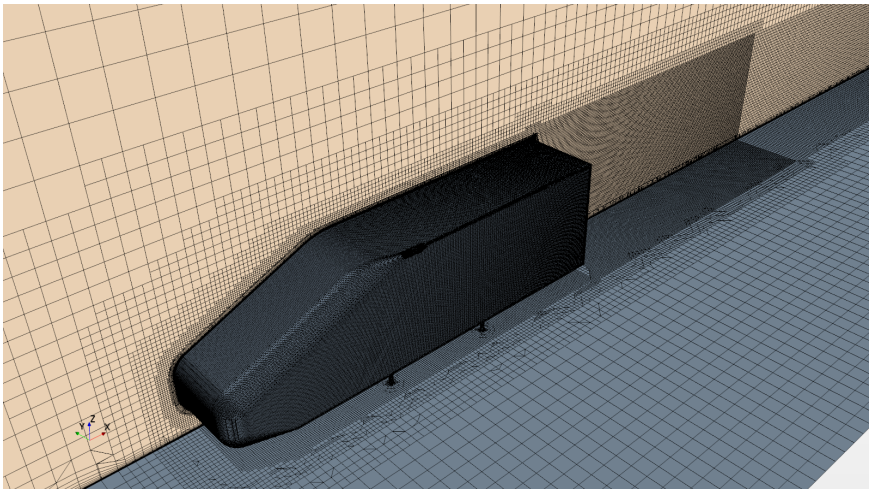


Figure 4: Refinement around vehicle (1B WRRANS)

Table 1: Grid Parameters

Component	Description	Item	WM RANS	WR RANS	WM ER	WR ER
Car	Surface	Size (m)	$2.4x10^{-3}$	$2.4x10^{-3}$	$2.4x10^{-3}$	$2.4x10^{-3}$
	Shoulder	Size (m)	$0.8x10^{-3}$	$0.8x10^{-3}$	$0.8x10^{-3}$	$0.8x10^{-3}$
	Prism Layer	Number	9	25	9	25
		Thickness (m)	0.013	0.013	0.013	0.013
		Near wall size (m)	$1.0x10^{-3}$	$20.0x10^{-6}$	$1.0x10^{-3}$	$20.0x10^{-6}$
Ground plane	Surface	Min Size (m)	0.0384	0.0384	0.0384	0.0384
	Prism Layer	Number	6	18	6	18
		Thickness (m)	0.05	0.05	0.05	0.05
		Near wall size (m)	$2.5x10^{-3}$	$50.0x10^{-6}$	$2.5x10^{-3}$	$50.0x10^{-6}$
Under car/wake	Surface	Min Size (m)	$4.8x10^{-3}$	$4.8x10^{-3}$	$2.4x10^{-3}$	$2.4x10^{-3}$
	Prism Layer	Number	9	18	9	18
		Thickness (m)	0.02	0.02	0.02	0.0125
		Near wall size (m)	$1.0x10^{-3}$	$30.0x10^{-6}$	$1.0x10^{-3}$	$30.0x10^{-6}$
Wake Refine	Near	Extent x= (m)	1.5	1.5	2	2
		Size (m)	$4.8x10^{-3}$	$4.8x10^{-3}$	$2.4x10^{-3}$	$2.4x10^{-3}$
	Far	Extent x= (m)	2.5	2.5	3	3
		Size (m)	$9.6x10^{-3}$	$9.6x10^{-3}$	$4.8x10^{-3}$	$4.8x10^{-3}$
Number of Cells	**With Wheels**	**1A**	**$3.87x10^{6}$**	**$7.46x10^{6}$**	**$37.45x10^{6}$**	**$50.08x10^{6}$**
	No Wheels	**1B**	**$3.34x10^{6}$**	**$5.63x10^{6}$**	**$35.92x10^{6}$**	**$46.76x10^{6}$**

Grids are supplied in CGNS, OpenFOAM, ANSYS Fluent .msh and Siemens Star-CCM+ .ccm formats.

Test Cases

A zero-yaw case at an inlet condition of 40 m/s, with a Reynolds number of 2.9×10^6 based on the vehicle length, should be run.

When presenting pressure and force coefficients, the experimental data uses a free stream probe approximately 2m forward of the origin mounted near the roof of the tunnel to determine total and static pressure. The same procedure should be used when presenting CFD data, the local static pressure and velocity magnitude at [2,0.0, 1.3]m should be used for normalisation of force and pressure coefficients. Note that no other forms of wind tunnel correction should be used for the data supplied to the workshop. As the mounting pins are connected to the balance, these need to be included when integrating force coefficients. For moment coefficients, the origin is mid-track, mid-wheelbase on the tunnel floor and corresponds to the origin in the coordinate system of the CAD/grid. The length used in the moment coefficient is the wheelbase (637.5mm) and positive pitching moment corresponds to a nose up force.

You should assume that the model is fully turbulent. Although it is likely that there are some regions of laminar flow at the nose of the vehicle, this is not documented in the experimental measurements.

The workshop requires that you use the standard grids. If your CFD methodology is unable to use the grids provided (e.g. LBM type code) then your grid (or lattice) should be set up to match the parameters provided in Table 1 as closely as possible.

Data Submission

This will be defined in a second document.

Grid Files

With Wheels 1A:

`wW1A_WMRANS.xxx`

`wW1A_WMER.xxx`

`wW1A_WRRANS.xxx`

`wW1A_WRER.xxx`

No Wheels 1B:

`nW1B_WMRANS.xxx`

`nW1B_WMER.xxx`

`nW1B_WRRANS.xxx`

`nW1B_WRER.xxx`

References

[1] G. Pavia and M. Passmore, “Characterisation of Wake Bi-stability for a Square-Back Geometry with Rotating Wheels,” in Prog. Veh. Aerodyn. Therm. Manag. (J. Wiedemann, ed.), (Cham), pp. 93–109, Springer International Publishing, 2018.

[2] Varney, M., “Base Drag Reduction for Squareback Road Vehicles,” Loughborough University, Feb, 2020. 10.26174/thesis.lboro.11823759.v1

[3] https://repository.lboro.ac.uk/articles/dataset/Windsor_Body_Experimental_Aerodynamic_Dataset/13161284

[4] Johl, G., Martin A. Passmore, and Peter M. Render. 2010. “Design Methodology and Performance of an Indraft Wind Tunnel,” The Aeronautical Journal, September 2004. https://hdl.handle.net/2134/6674