Case 1:
1^{st} Automotive CFD Prediction Workshop: SAE Reference Notchback Model

Gary J. Page
G.J.Page@lboro.ac.uk
Loughborough University
V0.6 7 November 2019

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. The case is based on the SAE 20-degree backlight angle notchback reference model at 1/5 scale (Cogotti 1988) with an experimental campaign undertaken at Loughborough University (Wood, Passmore and Perry 2014). The Loughborough institutional repository contains the underpinning PhD thesis (Wood 2015) and a 'green' open access version of the paper. This test-case description, geometry,meshes and example post-processing are available on the website transfer section (autocfd-transfer.eng.ox.ac.uk).

Geometry and Domain

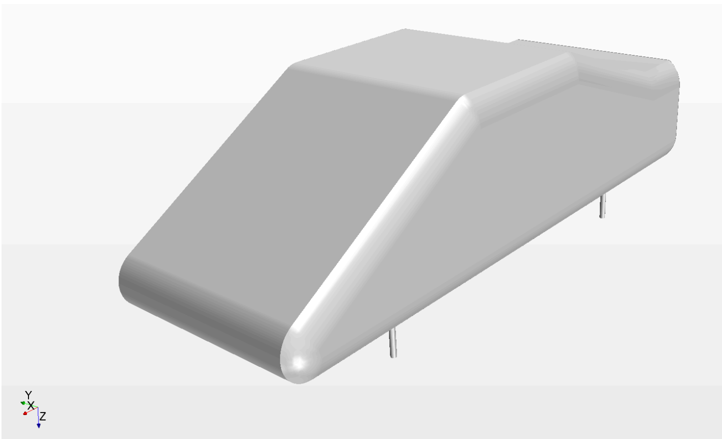

Figure 1 SAE Reference Model Geometry

This is a 1/5 scale model with a length of 840mm, a width of 320mm and a height (excluding pins) of 240mm. The reference frontal area is defined as $0.076m^2$ and the car length L and height H are used to non-dimensionalise distances. Note that Cogotti (1988) documents the full scale variant has a frontal area of $1.896m^2$, so that a 1/5 scale should be $0.07584m^2$, however the paper rounds this to **$0.076m^2$ and is the reference value that should be used**. The front of the model has a 30 degree slant and at the rear there is a 20 degree backlight leading to the notchback. The underside is flat until reaching a small 6 degree diffuser starting at the nominal location of the rear axle.

The model is mounted in the wind tunnel with four pins at a ground clearance of 40mm and zero pitch.

The CAD geometry of the model has its origin at the centre of the vehicle with the x-direction forward (nose +420mm, rear -420mm), y-direction to the right (left side -160mm, right side +160mm) and z-direction downwards (bottom -40mm, top -280mm). The pins have a 'wheelbase' of 490mm and are placed at x=-235mm and x=+255mm – hence the origin for pitching moment is at x=-10mm.

The supplied CAD geometry differs from that on the Loughborough University repository: the gap between the pins and the recessed holes in the model has been fared over and the pins extended to ensure that they reach down to the ground. Geometry is supplied as IGES, STEP, STL and Siemens NX part files. The Modified geometry should be used.

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 and the experimentalists estimate a δ^{99} boundary layer thickness of 40-60mm, with a momentum thickness of 9-10mm at the model location (note the ground clearance is 40mm).

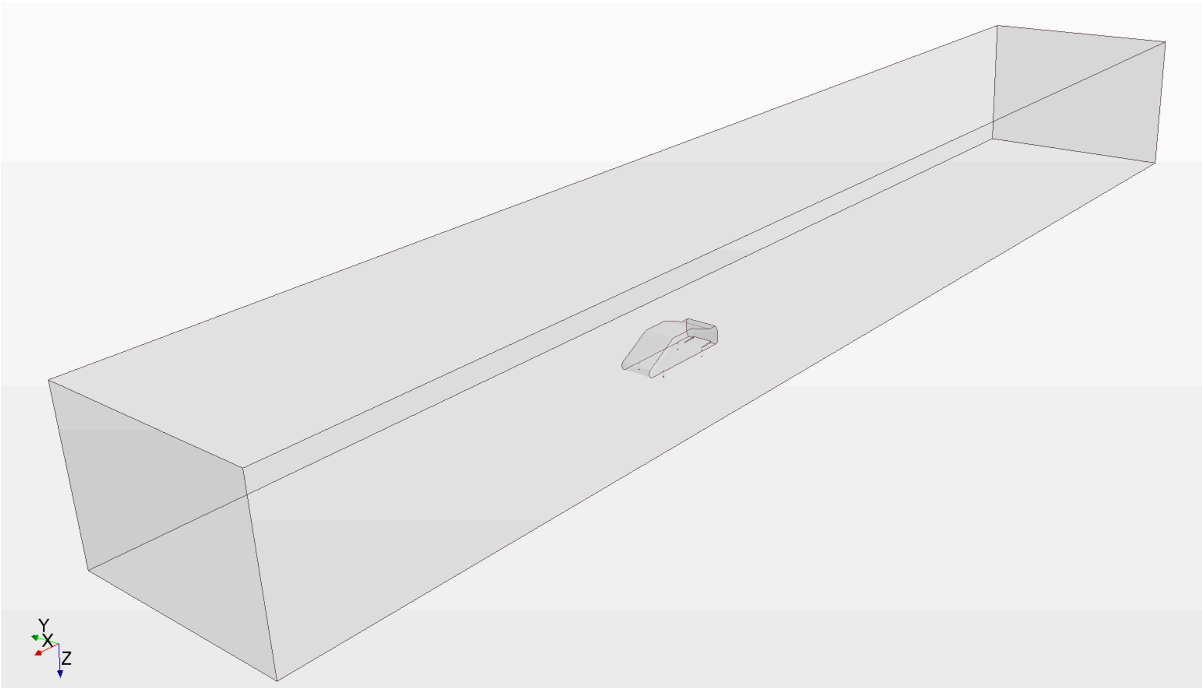

Figure 2: Computational wind tunnel domain

The domain required for this case 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 a 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 recommended domain has five car lengths (4.2m) forward of the nose and eight car lengths (6.72m) downstream giving a total length of 14L (11.76m). The working section is 1.94m wide and 1.32m high. Of the wind tunnel walls, only the ground plane should be treated as a no-slip wall.

Grids

Two grids are supplied: a medium grid suitable for (Unsteady) RANS with resolved boundary layers and a fine grid suitable for Detached Eddy Simulation. The first exploits symmetry so only half the model and domain is gridded, whilst the DES grid is of the full geometry.

The grids are generated using a Cartesian trimmer mesh with prism layers on no-slip walls. A single refinement zone extending 4L downstream is used to capture the wake for RANS grid. Additional refinement zones are added in the notch and close to the base for the DES grid. Table 1 summarises key parameters that defines these meshes – note that % refer to the vehicle height of 240mm as a reference.

Table 1: Grid Definitions

Parameter	**RANS**		**DES**	
Largest Cell	32%	76.8mm	32%	76.8mm
Smallest Cartesian Cell	1 %	2.4mm	1%	2.4mm
Wake refinement minimum size	2%	4.8mm	0.5%	1.2mm
Prism layer wall				
thickness	19%	4.8mm	19%	4.8mm
number	19		19	
stretching	1.15X		1.15X	
near wall cell size		0.52mm		0.52mm
Prism layer body				
thickness	5%	12mm	5%	12mm
number	23		23	
stretching	1.2X		1.15X	
near wall cell size		0.037mm		0.075mm
Total cells	4,115,762		29,145,776	

Grids are supplied in CGNS, OpenFOAM (v2.4, 3.0, 4.0), ANSYS Fluent .msh and Siemens Star-CCM+ .ccm formats.

Experimental Data

A large amount of experimental data is available in the SAE paper, online repository and Daniel Wood PhD thesis. For the purposes of the workshop we will be interested in:

1. Drag, lift and moment coefficients.
2. Mean pressure coefficient over the surface of the model, this will be compared to top surface centreline and backlight, boot-deck and base pressure distributions.
3. Slices of mean velocity for comparison with PIV data (x=-380mm, x=-290mm, y=0mm, y=-60mm, y=-90mm). See Figure 3.

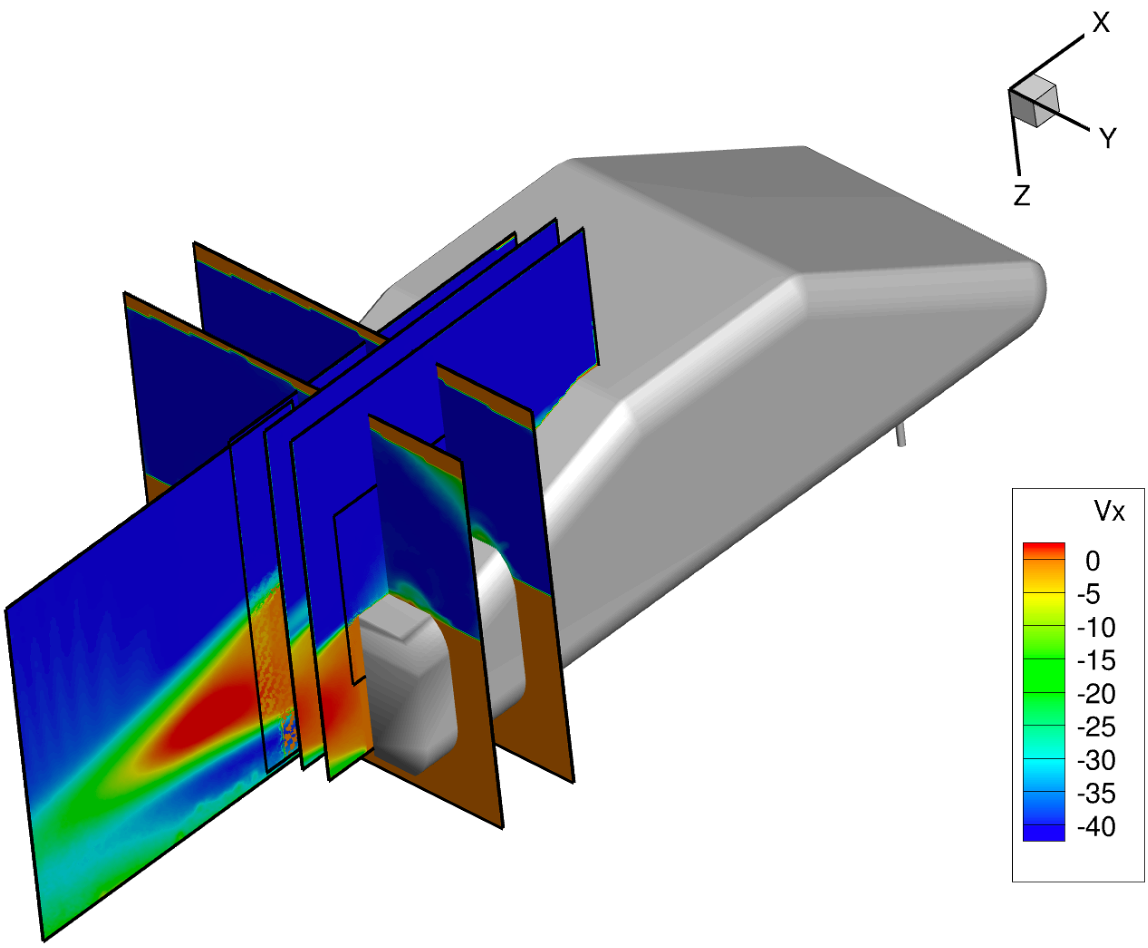

Figure 3: Experimental PIV locations of interest

Test Cases

A zero yaw case at an inlet condition of 40 m/s, with a Reynolds number of $2.3\text{x}10^6$ based on the vehicle length, should be run. A low turbulence intensity of 0.2% is reported in the experiment and should be used as a boundary condition for RANS inlet conditions alongside a sensible length scale. At this speed there is a characteristic time scale t* (vehicle length/freestream speed) of 0.021s which can be used to help guide your choice of CFD time step.

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,-0.002, -0.92]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 computing force coefficients. To be clear, positive lift would lift the vehicle from the surface of the road (and hence is in the negative z-direction); positive drag retards the vehicle (and hence is in the negative x-direction) and positive moment would lift the nose (and hence is in the positive y-moment rotation sense). For moment coefficient the 'wheelbase' is 0.49m and the origin for moments is at x=-0.1m (since the centre of the pins is not quite at the model origin).

As the PIV system directly measures velocity, no corrections are applied by the experimentalists, and in the SAE paper it is normalised by the assumed freestream velocity of 40 m/s.

For RANS solutions you should assume that the model is fully turbulent, and for LES/DES cases no inflow turbulence is required.

Data Submission

We require the following text files (and see examples provided):

1. Information on your code/mesh and set up: `Submission_info.dat`
2. Information on the hardware and speed of your simulation: `HPC_info.dat`
3. Final mean forces and moment coefficients (C_D,C_L,C_M): `Forces-mean.csv`
4. Time history for unsteady methods (iteration or time, C_D,C_L,C_M): `Forces-history.csv`
5. Centreline top and bottom surface pressure coefficient line plot (x, C_p): `CpCentreline.csv`
6. Pressure coefficient over surface of vehicle in tecplot, vtk or ensight format (x,y,z, C_p): `CpSurface.dat` (with an appropriate extension for vtk or ensight)
7. [Optional for unsteady eddy resolving simulations] RMS of pressure coefficient fluctuations over surface of vehicle in tecplot, vtk or ensight format (x,y,z, C_{pRMS}): `CpRMSSurface.dat` (with an appropriate extension for vtk or ensight)
8. Mean velocity field at several experimental stations (x,y,z,u,v,w): `velXNNN.dat` planes are x=-380mm, x=-290mm, y=0mm, y=-60mm, y=-90mm with files `velx380.dat, velx290.dat, vely000.dat, vely060.dat, vely090.dat`
9. [Optional for unsteady eddy resolving simulations] Image (in png format) of instantaneous isosurface of an eddy resolving quantity (e.g. Q-criterion or λ_2) from a similar viewpoint as Figure 3 in order to illustrate the scales that are being resolved.

All positions should be in metres, whilst the velocity field (item 8) should be normalized by the reference inlet condition of 40 m/s.

All the requested data should be uploaded to the OneDrive shared folder:

https://lunet-my.sharepoint.com/:f:/g/personal/ttgjp_lunet_lboro_ac_uk/EpWT2-YiJSRIrLlavJeAFmgBTGnbd4okDgbWB1cGTfSbBQ?e=flBUG1

You should create a subfolder with a name that contains your participant id (provided once you complete the intention to participate form on the website) and lastname of the participant (e.g. `002Page`) and email `g.j.page@lboro.ac.uk` with the subject line being "`AUTOCFD: XXX_Lastname`" to notify that the data has been uploaded. This will then be moved to a private folder and an email response to confirm that the data has been received. If your data files are small (less than ~20MB) then it is also possible to email these direct using the same format for the subject line.

The data files need to be uploaded/emailed by 25th November 2019.

The data should be tarred/zipped with the following convention:

`XXX_Lastname_Code_Model_grid_Case1_vN.tar.gz`

Where `XXX` is the submission ID, `Lastname` is the lead investigator last name, `Code` is the CFD code used and `Model` is the turbulence model, grid is either RANS, DES or USER (the latter being where you have generated your own). The version number is to allow you to send updates i.e

`002_Page_STARCCM+_SSTDDES_USER_Case1_v2.tar.gz`

The naming convention allows one participant to submit multiple results.

References

Cogotti, A. "A Parametric Study on the Ground Effect of a Simplified Car Model," SAE 980031, 1998.

Wood, D., Passmore, M., and Perry, A., "Experimental Data for the Validation of Numerical Methods - SAE Reference Notchback Model," *SAE Int. J. Passeng. Cars - Mech. Syst.* 7(1):2014, doi:10.4271/2014-01-0590. https://dspace.lboro.ac.uk/2134/15978

SAE reference model: 20 degree notchback validation dataset (reference SAE paper 2014-01-0590). https://dspace.lboro.ac.uk/2134/13886

Wood, D.," The effect of rear geometry changes on the notchback flow field," PhD Thesis, Loughborough University, 2015. https://dspace.lboro.ac.uk/2134/18889

Revisions:

0.5 5 November 2019:

1. Fix typo in 1/5 scale area

2. Add extra clarity to definitions of lift, drag and moment.

3. Add in requirement for moment.

4. Define data submission.

5. Add OneDrive upload.