

Introduction

External aerodynamics is the main application in the automotive industry. CFD is used to verify wind tunnel experiments, optimize car shapes in terms of lift and drag and study salient flow features. The simulation of aero-acoustics has become a major application, which helps the aerodynamicist to obtain fast and reliable results.

The data which the Assignment is based comes from Marco Lanfrit's paper on Automotive External Aerodynamics with Fluent. Fluent been a CFD software package in Ansys 16.9. The title been 'Best practice guidelines for handling Automatic External Aerodynamics with Fluent'. The document gives a described and reliable way to perform simulations and gather data.

mesh size & turbulence model	C_D	ΔC_D	CPUhs
wind-tunnel experiment	0.321	-	-
coarse mesh (5.5 M cells) realizable k- ϵ	0.336	4.7 %	450
fine mesh (11 M cells) realizable k- ϵ	0.328	2.1 %	750
fine mesh (11 M cells) RSM	0.322	0.3 %	1200

Table 1: Comparison of results for different modeling approaches [1]

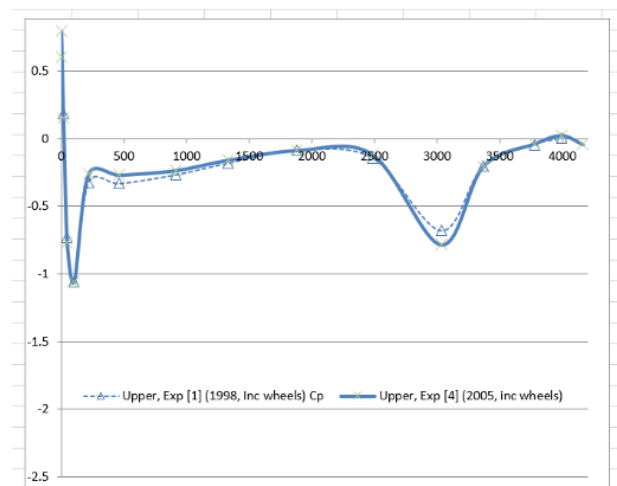
In table 1 above it shows the results of a simulation from Lanfrits paper of which I will try and get the M cell partition. The model viscous used will Realizable K-epsilon.

Data given

For the baseline MIRA Reference car and wind tunnel is supplied below both Upper and Lower surfaces.

Lower

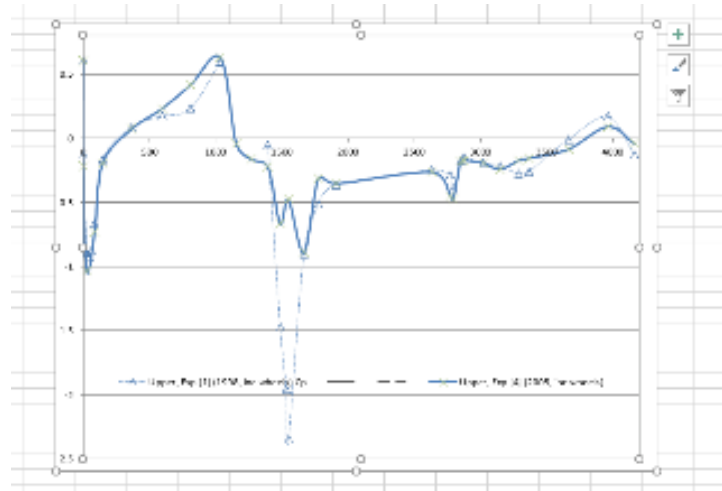
Upper, Exp [1] (1998, inc wheels)		Upper, Exp [1] (2005, inc v	
x (mm)	Cp	x (mm)	Cp
0	0.946	0	0.605
0	0.814	0	0.955
17.3	0.187	0	0.800
46.4	-0.731	17.3	0.155
102.5	-1.057	46.4	-0.766
217	-0.322	102.5	-1.055
464.1	-0.332	217	-0.262
911	-0.268	464.1	-0.268
1327.7	-0.183	911	-0.235
1878.2	-0.088	1327.7	-0.159
2482.6	-0.142	1878.2	-0.087
3033.1	-0.68	2482.6	-0.116
3367.7	-0.203	3033.1	-0.783
3775.7	-0.044	3367.7	-0.205
3991.6	0.004	3775.7	-0.038
		3991.6	0.024
		4164.3	-0.042



Michael Thame
Module: Advanced Computational Methods
Module code: M6X00408
Lecturer: Tim Tudor
Assignment Title: Aerodynamics Vehicle Testing and Analysis

Upper

Upper, Exp [1] (1998, Inc wheels)		Upper, Exp [4] (2005, Inc wheels)	
x (mm)	Cp	x (mm)	Cp
0	0.607	0	0.605
0	-0.13	0	-0.219
25.9	-0.891	25.9	-1.033
41	-0.924	86.4	-0.734
62.6	-0.876	145.7	-0.196
86.4	-0.664	367	0.074
145.7	-0.165	593.7	0.230
367	0.086	809.5	0.415
593.7	0.174	1036.2	0.630
809.5	0.233	1156.8	-0.045
1036.2	0.601	1268.3	-0.164
1156.8		1394.6	-0.224
1394.6	-0.055	1487.4	-0.664
1487.4	-1.471	1550	-0.476
1532.9	-1.954	1666.6	-0.891
1550	-2.347	1776.7	-0.312
1666.6	-0.899	1908.4	-0.353
1776.7	-0.51	2633.7	-0.263
1908.4	-0.365	2774.1	-0.448
2633.7	-0.249	2794.5	-0.478
2774.1	-0.285	2850.1	-0.179
2794.5	-0.395	2874.1	-0.171
2850.1	-0.173	3020.8	-0.191
2874.1	-0.147	3146.3	-0.241
3020.8	-0.191	3289.8	-0.171
3146.3	-0.22	3367.5	-0.155
3289.8	-0.275	3665.2	-0.084
3367.5	-0.26	3962.9	0.092
3665.2	-0.008	4164.3	-0.042
3962.9	0.169		
4164.3	-0.177		



Also noted is the Coefficient of lift, Coefficient of drag and Frontal area

- CL = -0.0845
- CD= 0.2963
- FA= 1.856m²

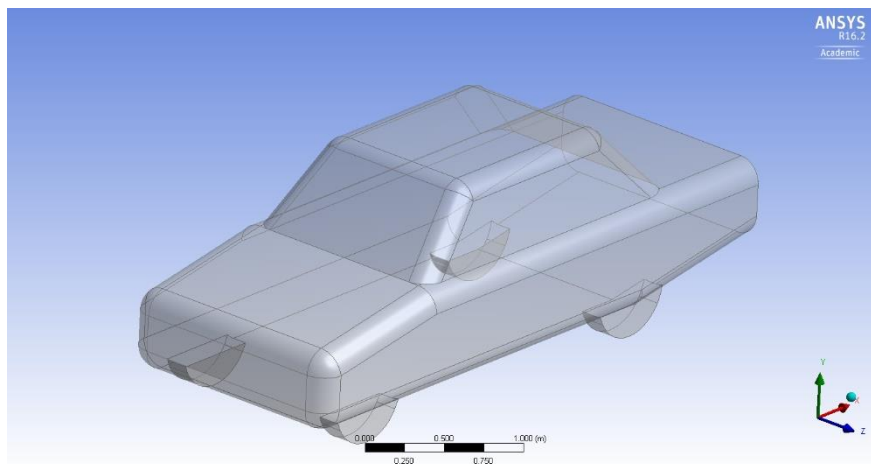
Michael Thame
Module: Advanced Computational Methods
Module code: M6X00408
Lecturer: Tim Tudor
Assignment Title: Aerodynamics Vehicle Testing and Analysis

Building geometry and meshing

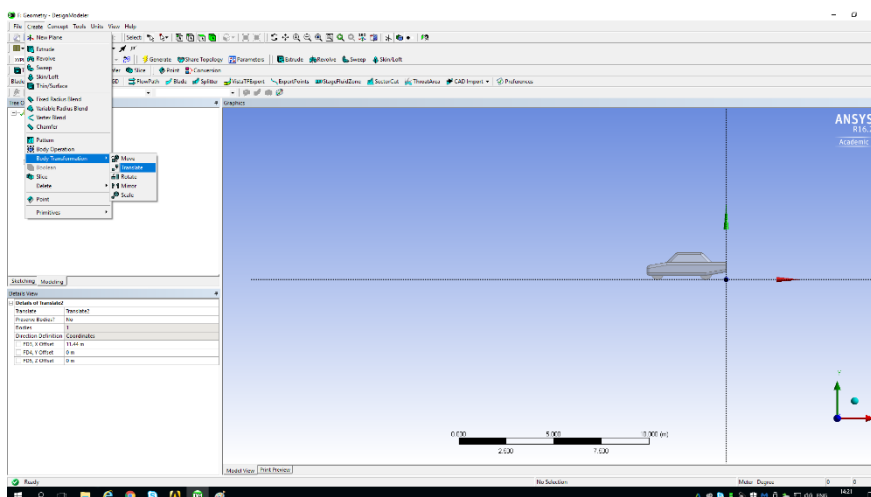
Using Ansys 16.9 as the software, in the toolbox section under Component systems 4 components will need to be included for a simulation these include:

- Geometry- Building and assembly of the model and MIRA windtunnel ready for Mesh
- Mesh- using ICEM CFD, which allows for a slightly co-operated and not clean geometry which the MIRA car is. Creates multiple layers, cells etc ready for the fluent flow.
- Fluent- simulates the car flow and creates a solution based on the mesh.
- Results

Below is MIRA car itself once imported (which is should of imported a solid body). Its also worth on the left hand side placing simplify geometries on as it will simplify the geometries although not really having affect as the MIRA car is already a simple geometry.



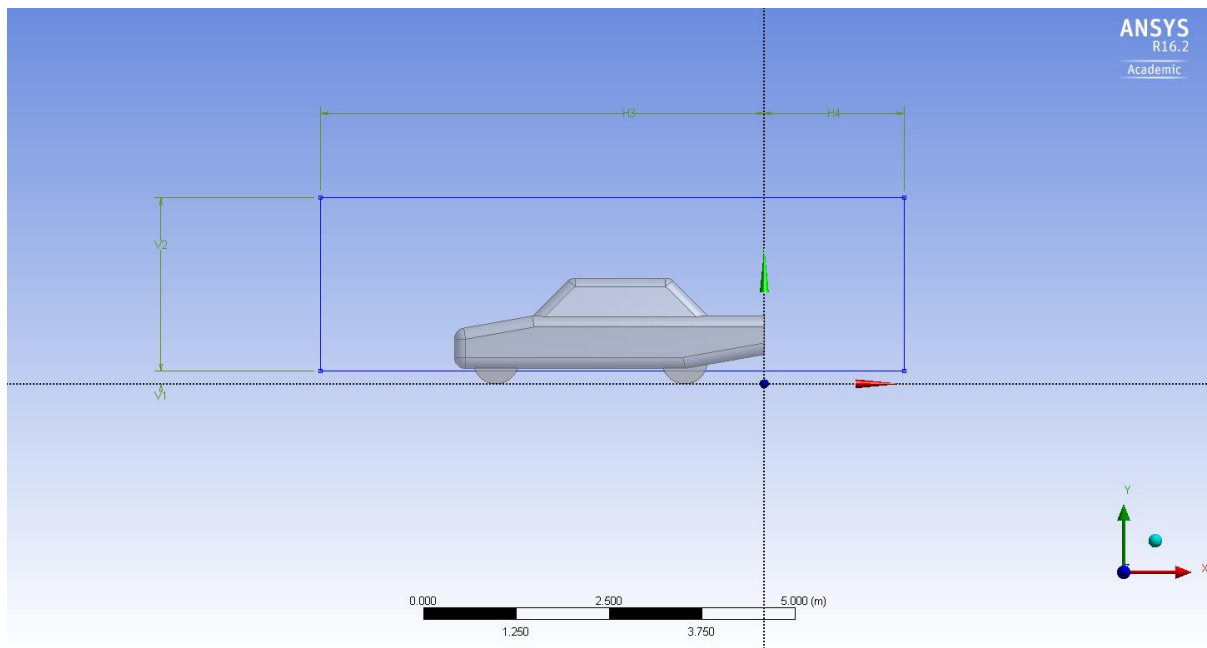
One problem I found was that the geometry was in the middle of the XY plane and not lining up on the ZY. One way I found was opening body operation and translating it by roughly 14.4mm. Deleting of course the previous geometry.



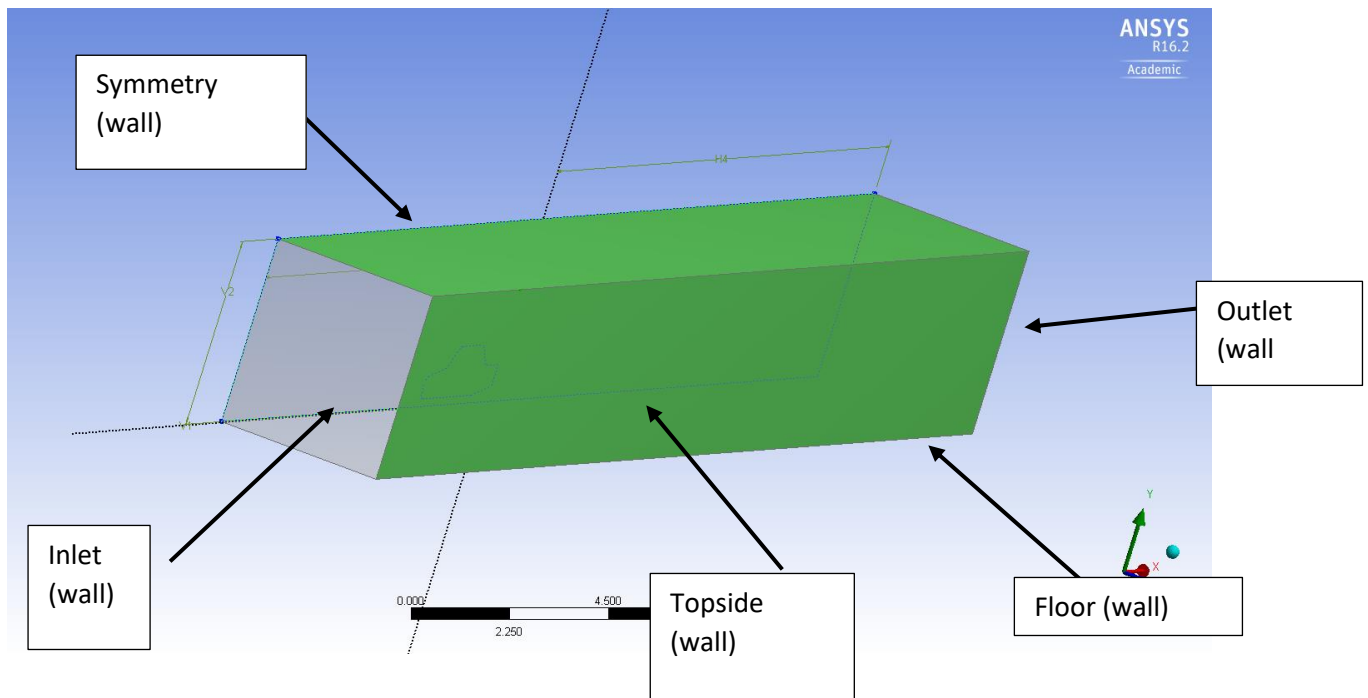
Michael Thame
Module: Advanced Computational Methods
Module code: M6X00408
Lecturer: Tim Tudor
Assignment Title: Aerodynamics Vehicle Testing and Analysis

For a Computational Domain (Wind Tunnel) we must first reference Lanfrits Paper stating that the domain should be 3 car lengths in front of the car, 5 car lengths behind and 3 car heights in height and 3 widths in width. However this is not applicable if the domain boundaries are of a real wind tunnel. This been of the MIRA wind tunnel will only use Lanfrits suggesting for forward lengths and rear lengths, and the rest the dimensions of MIRA wind tunnel. It's the important to know that the MIRA car was on Pneumatic

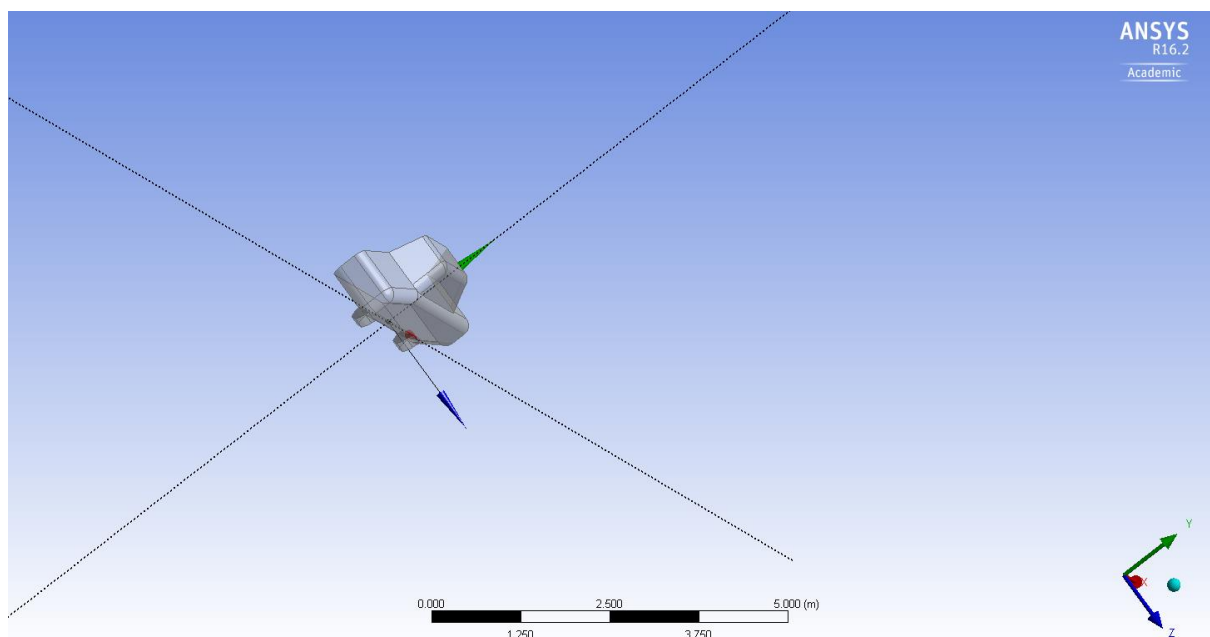
From the suggestions then setting dimensions H3 should be 16m in front of the car, H4 rear of the car should be 20m. The height should be that of the MIRA wind tunnel of 4.4m and because there on pneumatic tyres given this a assumption will be made I've assumed it as 20mm.



The next thing to do would be to create a void in the shape of the car by using a Boolean and the target body been the wind tunnel and tool body been the MIRA car. Its important to create faces to know which part is which listed below



Now with all the faces hidden you can select the faces for top of the car, Bottom and Front wheel and rear wheel creating each part as a separate part.

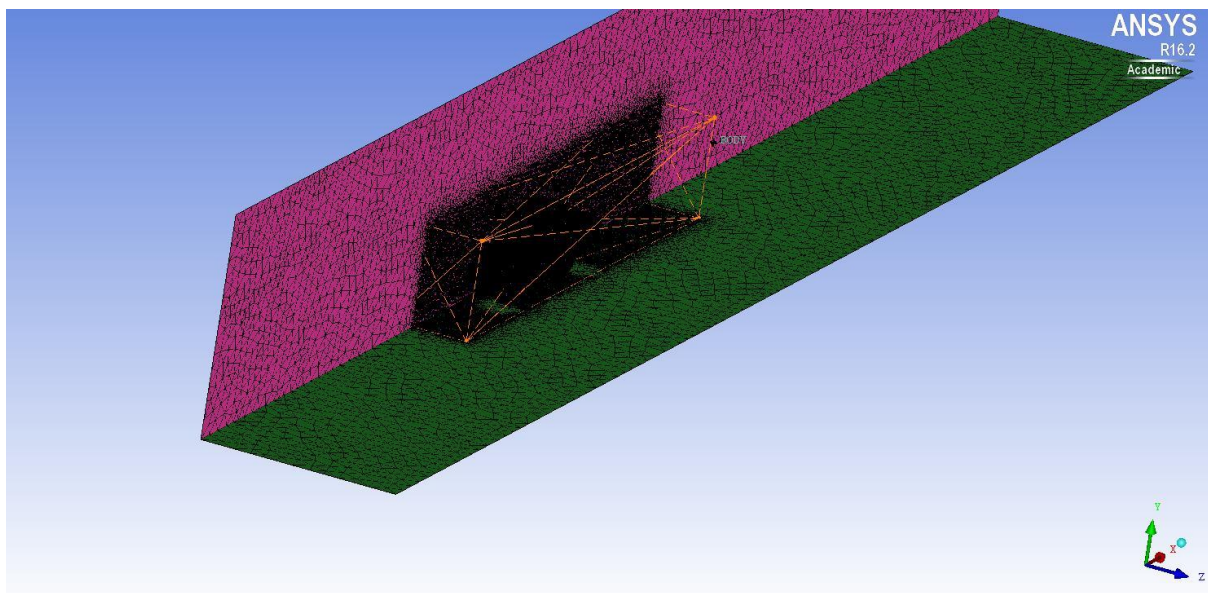


Michael Thame
Module: Advanced Computational Methods
Module code: M6X00408
Lecturer: Tim Tudor
Assignment Title: Aerodynamics Vehicle Testing and Analysis

Placing the geometry and domain in ICM CFD. In order to tell the software where we want the elements it's important to create a body in the wind tunnel so it can create a mesh around the mira car and Wind tunnel.

It's important next to set a maximum element size in Global Mesh setup which ive set as 1m. It was set thru all experiments carried out. Scale factor I've set as 0.5m for the second and third experiments, the first experiment was set at 1m. The scale will reduce the Element size such as if the size was 10m half of that would be 5m.

A box noted below is placed around the MIRA car itself this is called a refinement box or a wake box. This volume meshing strategy is a virtual box used for cell refinement. This approach is very accurate and avoids the creation of additional Surfaces prior steps.



Michael Thame
Module: Advanced Computational Methods
Module code: M6X00408
Lecturer: Tim Tudor
Assignment Title: Aerodynamics Vehicle Testing and Analysis

Below is the part mesh setup. It's important to apply something called Prismatic layers. Prism layers are layered elements providing good alignment with the flow near the boundaries. This reduces the numerical diffusion. Prism layers are extruded from the wind tunnel floor and most of the MIRA car features such as hood, back and side. The number of layers recommended are 5 layers defined below. The size depends on what mesh size is needed below is for a Coarse mesh.

- Fine mesh – MIRA Bottom 0.04m, MIRA front wheel 0.016m, MIRA rear wheel 0.016m, and MIRA top 0.046m.

Part	Prism	Hexa-core	Maximum size	Height	Height ratio	Num layers
BODY	☐	☐				
FLOOR	☐		0	0	0	0
FLUID__1_1	☐		0			
INLET	☐		0	0	0	0
MIRABOTTOM	☒		0.05	0	0	5
MIRAFRONTWHEEL	☒		0.02	0	0	5
MIRAREARWHEEL	☒		0.02	0	0	5
MIRATOP	☒		0.05	0	0	5
OUTLET	☐		0	0	0	0
SYMMETRY	☐		0	0	0	0
TOPSIDE	☐		0	0	0	0

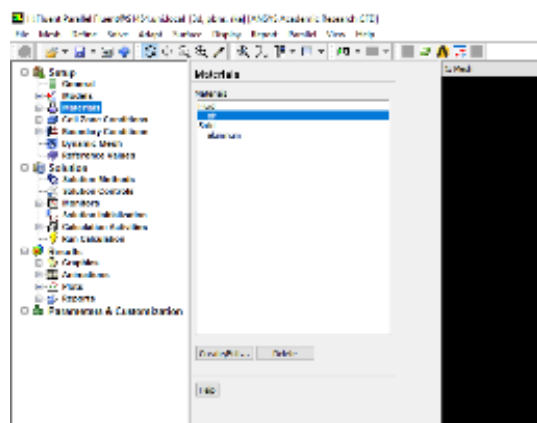
☒ Show size params using scale factor
☐ Apply inflation parameters to curves
☐ Remove inflation parameters from curves
Highlighted parts have at least one blank field because not all entities in that part have identical parameters
Existing workbench input parameters using a lightblue background

Submitted at the referencing is the full ICM commands and Prism command.

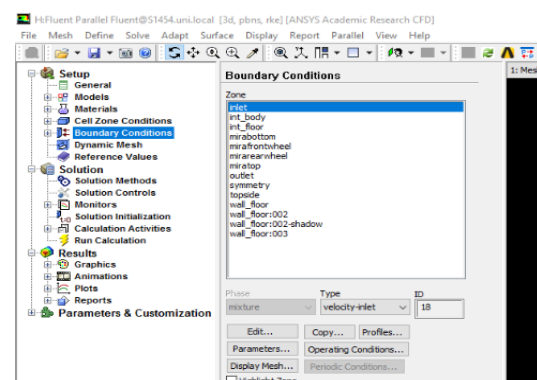
Assignment Title: Aerodynamics Vehicle Testing and Analysis

[illegible]

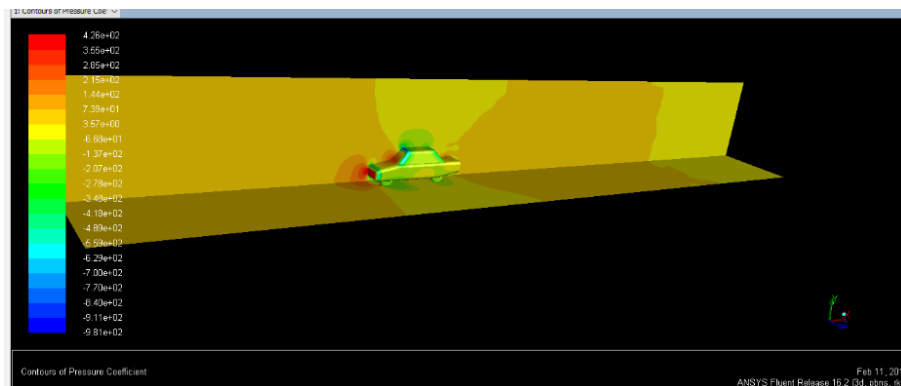
The turbulence model using is Realizable k-epsilon Model with this it's possible to achieve good results and creates a stable and fast converging. Because of this it allows a huge number of calculations in small time frame.



The boundary conditions should be close to the conditions in the wind tunnel as possible. Flow velocity and turbulent of the tunnel are known meaning, a velocity inlet boundary is used to model the incoming flow aka the inlet. Where boundary layer suction is present the areas are treated as separate wall zones and shear stress are set as zero. A symmetry domain will be applied for the sides.

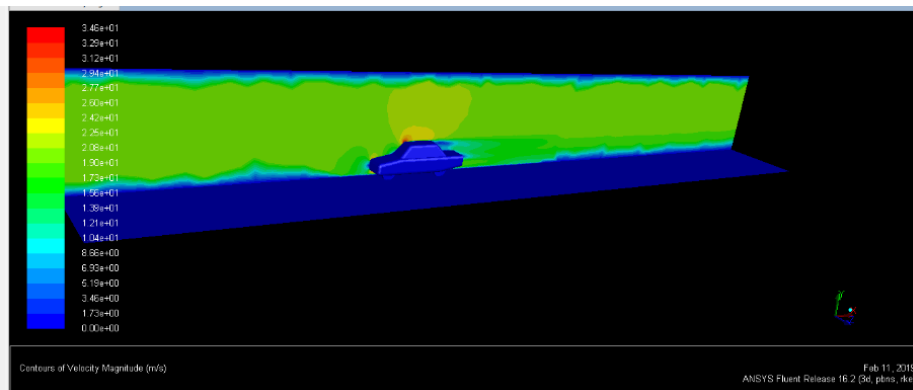


First results



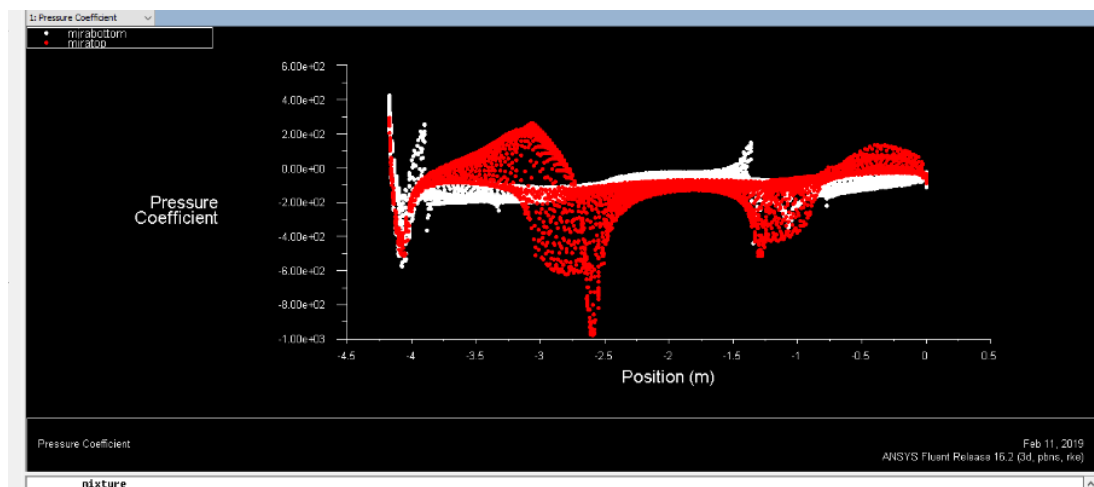
Contours of Pressure coefficient for the first mesh notice how varied it looks and unstructured compared to the third results. Note the pressure on the front of the car especially around the windscreen as high

pressure is around the windscreen and notice the low pressure on top of the roof meaning unstructured air flow.



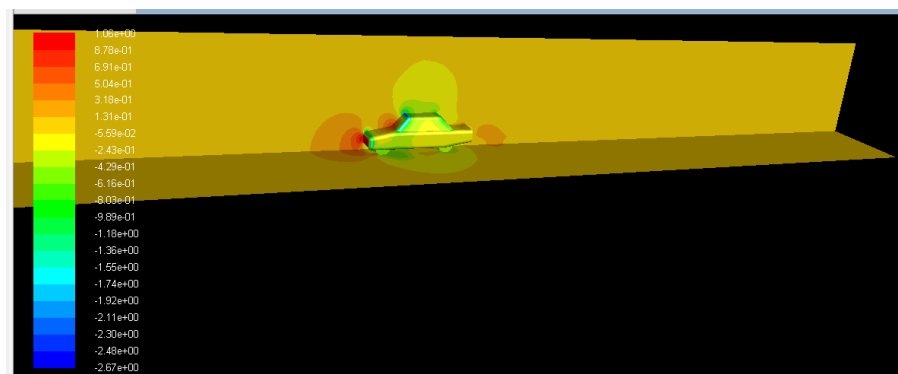
Shown is velocity magnitude aka speed of motion note on the symmetry wall especially the edges compared to the third results the velocity is unstructured and vary also at the back of the car the velocity is less vivid and displayed this

could be done to the mesh quality though or structure compared to third results.



Remembering that MIRA Bottom is white and MIRA red is Top. Coefficient of pressure is similar to dater supplied already. Looking at the position compared to the third or second results the dots are more scatter and viewer. Its also similar to that of the change of geometry. Its remembering this was run at 7.7M cell nodes.

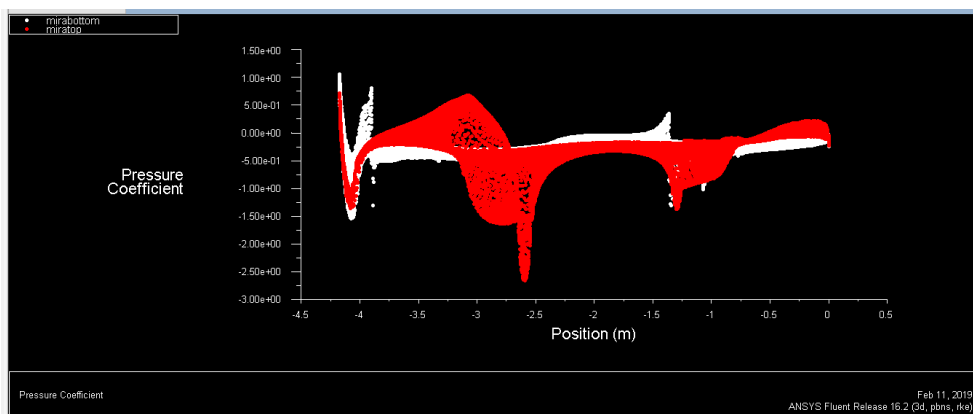
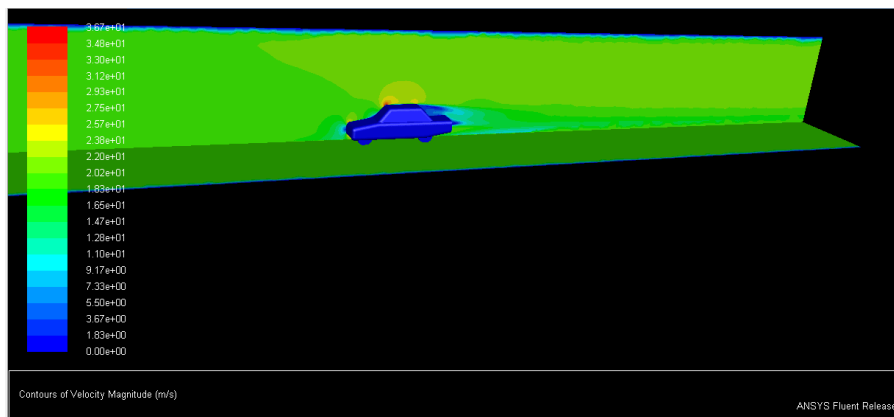
Second results



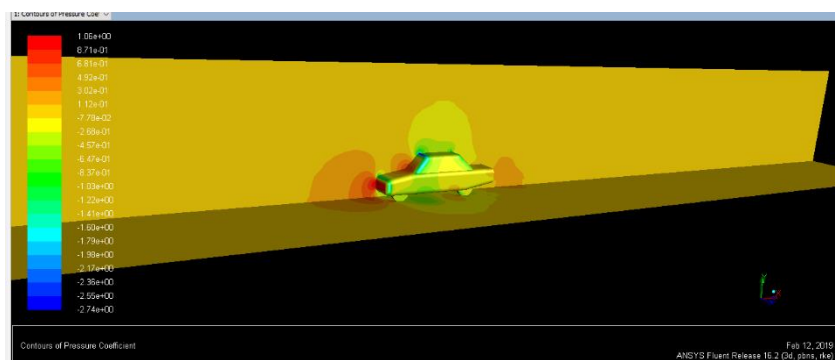
area at the rear of the car.

Shown in Figure 3 is the coefficient of pressure displayed as a contour. Clearly a much better mesh with the global mesh set up scale factor set as 0.5m. This is clear by looking at symmetry and rear pressure is built up in a specific

Michael Thame
Module: Advanced Computational Methods
Module code: M6X00408
Lecturer: Tim Tudor
Assignment Title: Aerodynamics Vehicle Testing and Analysis

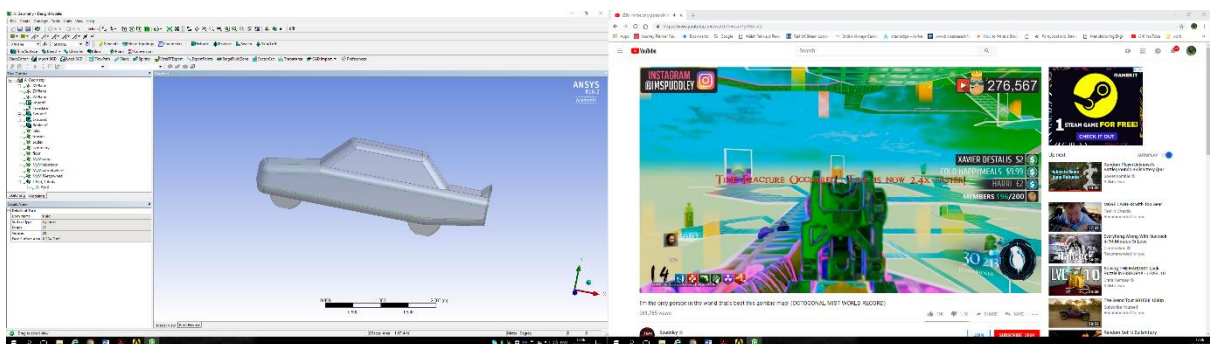
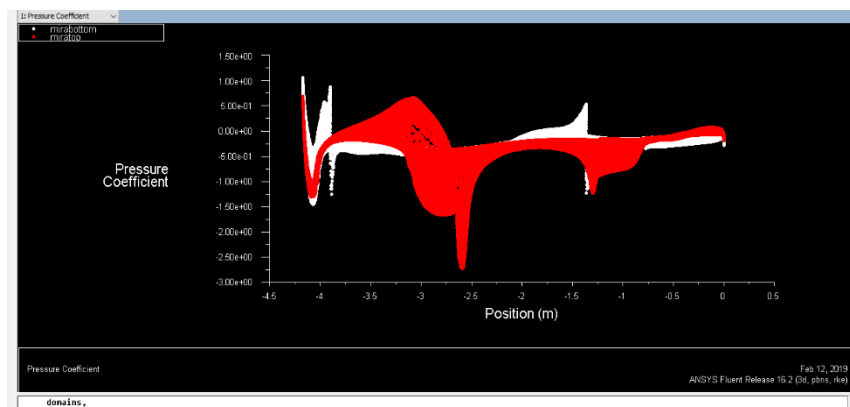
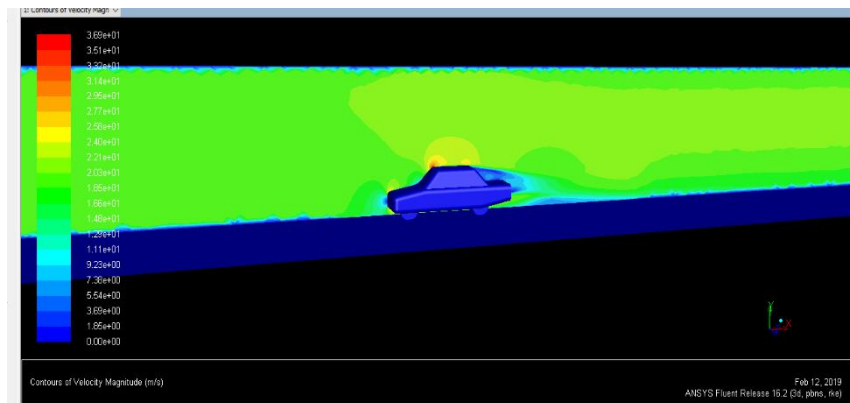


Third results

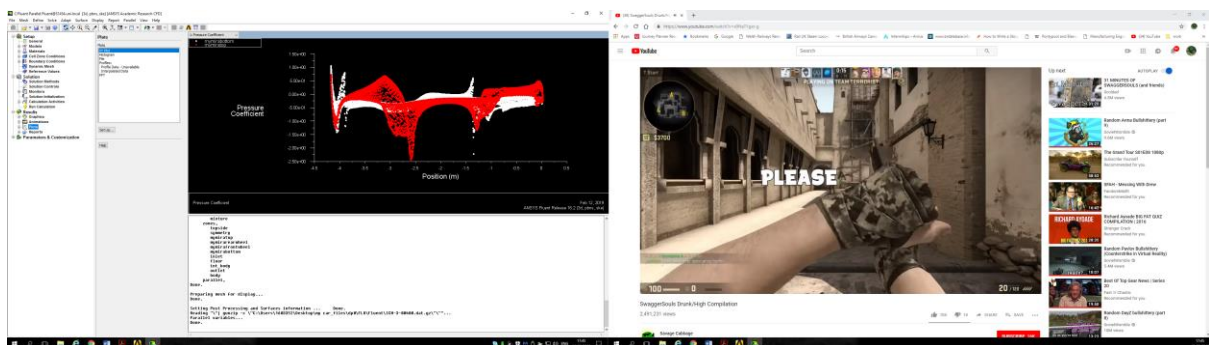
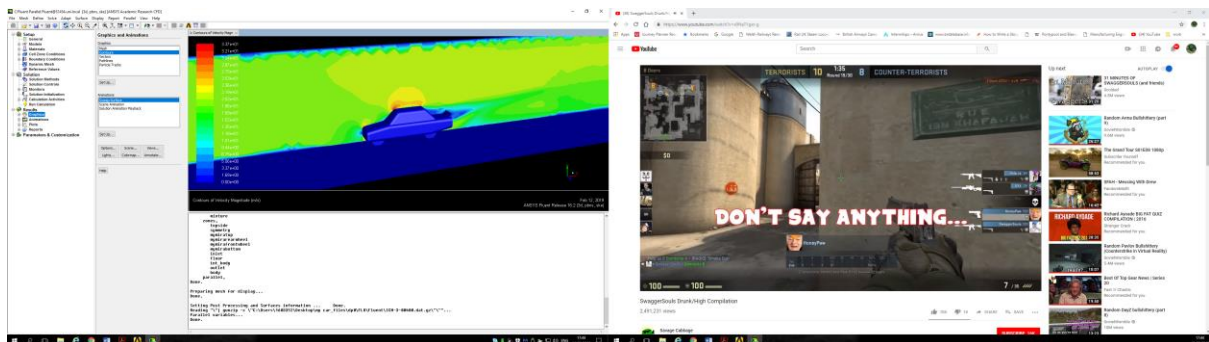
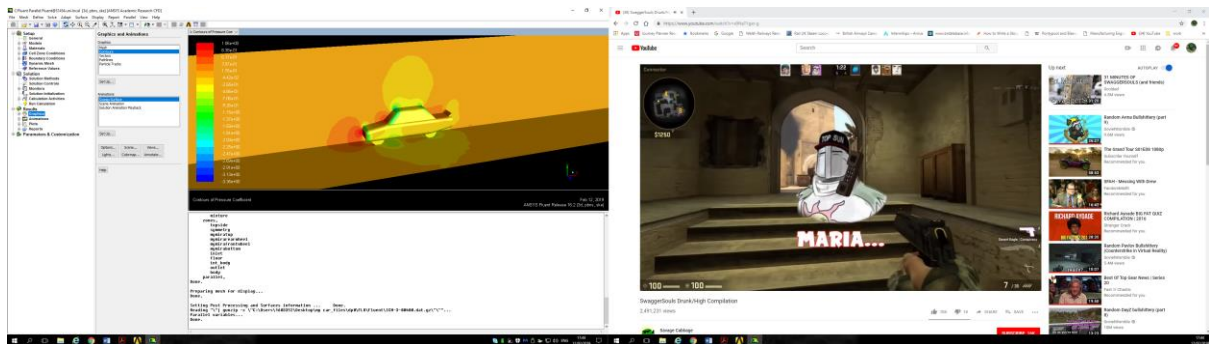
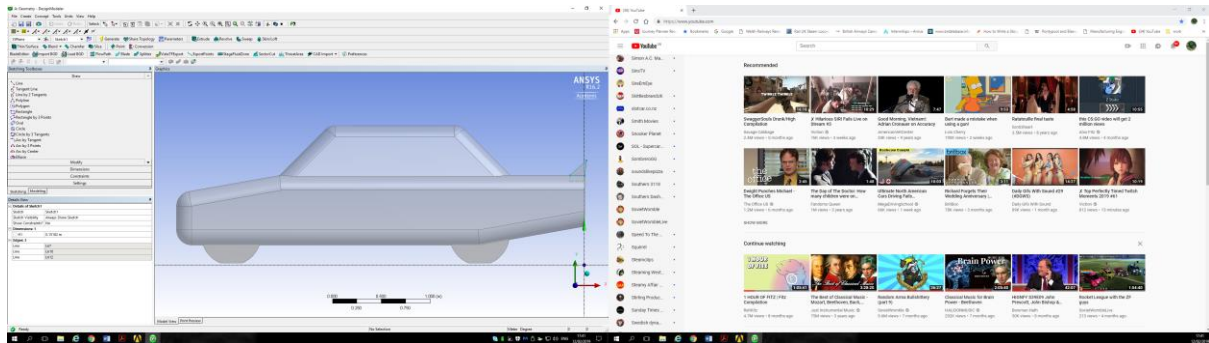


Contours of Pressure coefficient for the first mesh

Michael Thame
Module: Advanced Computational Methods
Module code: M6X00408
Lecturer: Tim Tudor
Assignment Title: Aerodynamics Vehicle Testing and Analysis



Michael Thame
Module: Advanced Computational Methods
Module code: M6X00408
Lecturer: Tim Tudor
Assignment Title: Aerodynamics Vehicle Testing and Analysis



Michael Thame

Module: Advanced Computational Methods

Module code: M6X00408

Lecturer: Tim Tudor

Assignment Title: Aerodynamics Vehicle Testing and Analysis