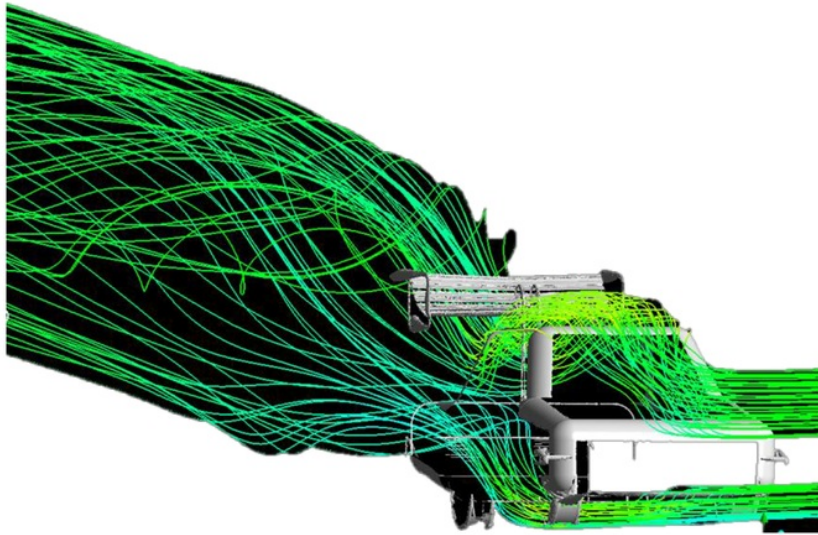


Correlating MIRA Notch Reference Geometry Wind Tunnel Data to ANSYS CFD Software

by Gavin Lee Quentin Spring-Benyon

FILE	9981_GAVIN_LEE_QUENTIN_SPRING-BENYON_CORRELATING_MIRA_NOTCH_REFERENCE_GEOMETRY_WIND_TUNNEL_DATA_TO_ANSYS_CFD_SOFTWARE__585139367.DOCX (210)		
TIME SUBMITTED	25-JAN-2019 10:18AM (UTC+0000)	WORD COUNT	10550
SUBMISSION ID	99794161	CHARACTER COUNT	64598

Correlating MIRA Notch Reference Geometry Wind Tunnel Data to ANSYS CFD Software



Gavin Spring-Benyon (P142997)

04/01/2019

Module: Advanced Computational Methods

Module Code: M6X00408

Degree: Motorsport Engineering BEng

School of Engineering

TABLE OF CONTENTS

TABLE OF FIGURES	3
TABLE OF TABLES	5
TABLE OF GRAPHS.....	5
1. INTRODUCTION.....	6
2. NOTCH REFERENCE GEOMETRY AND CFD RESEARCH	7
2.1 Notch Reference Geometry	7
2.1.1 MIRA notchback testing conditions	7
2.2 Finding A Initial CFD Approach.....	8
2.2.1 Domain size	8
2.2.2 Vehicle position.....	9
2.2.3 Meshing.....	9
2.2.4 Boundary conditions	12
2.2.5 Turbulence model	12
3. PARAMETER STUDIES METHODOLOGY.....	13
3.1 Apparatus For Studies	13
3.2 Testing Conditions.....	13
3.3 Notch Wind Tunnel Data.....	13
3.4 Initial CFD Standard Settings.....	13
3.4.1 Domain settings	13
3.4.2 Meshing settings	13
3.4.3 Boundary conditions settings	14
3.4.4 Turbulence model settings	15
3.5 Summary Of The Studies To Be Completed	15
4. PARAMETER STUDIES RESULTS, ANALYSIS AND DISCUSSION.....	16
4.1 1 st Run/Domain Size Results	16
4.1.1 The flow patterns around the 1 st CFD run	17
4.2 Rest Of Domain Study Results.....	19
4.2.1 Domain size 2	19
4.2.2 Domain size 3	19
4.2.3 Rest of the results	20
4.3 Mesher Results.....	22
4.4 Wheel Contact Patch Results	24
4.5 Model Progress	25

4.6 Mesh Refinement Results	27
4.6.1 Global element size	27
4.6.2 Car element size	27
4.6.3 Mesh density box size	28
4.6.4 Mesh density box element size	29
4.7 Prism Layers Study Results	30
4.8 Turbulence Models	32
4.9 Wheel Blocks Study Results	34
4.10 Polyhedra Mesh Run	35
4.11 MIRA Wind Tunnel Domain Run	36
4.12 Final Study Runs And Setting Conclusions	37
5. IMPROVED NOTCH CAR PERFORMANCE MODEL	43
5.1 CFD Results	45
6. CONCLUSIONS	50
7. FURTHER WORK	51
8. REFERENCES	52

TABLE OF FIGURES

Figure 1 MIRA's notch reference geometry car (Kumar & Vaidya, 2016)	6
Figure 2 MIRA's square-back reference geometry car (Kumar & Vaidya, 2016)	7
Figure 3 showing domain sizing	9
Figure 4 ANSYS meshing recommendations (ANSYS, 2012)	10
Figure 5 boundary layer's interaction with a wall (ANSYS, 2012)	11
Figure 6 ANSYS mesh quality scale (ANSYS, 2012)	12
Figure 7 graphical representation of parameter studies boundary conditions	14
Figure 8 symmetry wall pressure contour domain size 1	16
Figure 9 Velocity on symmetry wall domain size 1	16
Figure 10 floor with pressure contour on domain floor (domain 1)	17
Figure 11 Kumar and Vaidya experiment results of the rear wake region of the notch car (Kumar & Vaidya, 2016)	17
Figure 12 pressure contour on notch car from first run (domain 1)	18
Figure 13 pressure contour on notch car from first run (domain 1) side view	18
Figure 14 Kumar and Vaidya mesh refinement boxes used	19
Figure 15 Domain 2 pressure contours showing the height and width not cutting off the pressure changes of the notch car	19
Figure 16 Domain 2 symmetry wall velocity contour showing velocity changes not all being captured in the domain length	19
Figure 17 Domain size 3 contour results	20

Figure 18 case comparison between domain 3 and 4 size pressure contours	21
Figure 19 case comparison between domain 3 and 4 size velocity contours.....	22
Figure 20 Octree mesh	22
Figure 21 Delaunay mesh.....	23
Figure 22 Smooth and Fluent meshing	23
Figure 23 illustration of the contact amount changing	24
Figure 24 case comparison between the “just in contact” and 150mm velocity contours.....	25
Figure 25 current mesh quality	25
Figure 26 poor quality elements coming from the wheels.....	26
Figure 27 velocity flow streams on notch car	26
Figure 28 notch car vortices (white contours)	27
Figure 29 mesh density box	28
Figure 30 quality of mesh after refinement and wheel blocks added	30
Figure 31 case comparison between the 5 layers and 0 layers velocity contours	31
Figure 32 y+ and u+ flow graph (ANSYS, 2012).....	32
Figure 33 case comparison between the K-epsilon and SST model	33
Figure 34 case comparison between standard and 180X180 50mm wheel blocks with pressure contours	35
Figure 35 case comparison between standard and 180X180 50mm wheel blocks with velocity flow streams.....	35
Figure 36 visual representation of tetrahedron (standard) and polyhedral elements.....	36
Figure 37 illustration of the MIRA wind tunnel domain size run	37
Figure 38 comparing the wake region between best CFD run and Kumar and Vaidya findings	41
Figure 39 best run continuity level data	42
Figure 40 notch car pressure contour	43
Figure 41 notch car devises.....	43
Figure 42 notch car low pressure on bumper side	43
Figure 43 notch car flow streams bleeding under car	44
Figure 44 notch car with diffuser	44
Figure 45 notch car with vertex showing clean air	44
Figure 46 notch car with multi element wing	45
Figure 47 case comparison between the two cars show the pressure contour on the car and the velocity on the symmetry wall	46
Figure 48 case comparison between the two cars show the velocity on the symmetry wall	47
Figure 49 new notch car with pressure contour	47
Figure 50 new notch car with vectors on symmetry wall	48
Figure 51 new notch car rear wing	48
Figure 52 new notch car with free streams over car	49

TABLE OF TABLES

Table 1 testing conditions	13
Table 2 list of studies	15
Table 3 Domain size study sizing.....	21
Table 4 Car box size study sizing	29
Table 5 final CFD settings.....	42

TABLE OF GRAPHS

Graph 1 Domain size study Cl and Cd results	20
Graph 2 mesher types results	24
Graph 3 wheel contact patch results	24
Graph 4 global element size results	27
Graph 5 car element size results.....	28
Graph 6 car box size results	29
Graph 7 car box element size results.....	29
Graph 8 prism layers results	30
Graph 9 turbulence model results	33
Graph 10 wheel block study results	34
Graph 11 polyhedra mesh run results	36
Graph 12 MIRA wind tunnel run results	37
Graph 13 car surface top pressure tap data from the best CFD run over lay with the wind tunnel data	38
Graph 14 car surface bottom pressure tap data from the best CFD run over lay with the wind tunnel data	38
Graph 15 car surface top pressure tap data from the best CFD with domain as MIRA wind tunnel run over lay with the wind tunnel data	39
Graph 16 car surface bottom pressure tap data from the best CFD with domain as MIRA wind tunnel run over lay with the wind tunnel data	39
Graph 17 car surface top pressure tap data from the best CFD run with new mesh density box over lay with the wind tunnel data	40
Graph 18 car surface top pressure tap data from the best CFD run with new mesh density box over lay with the wind tunnel data	40
Graph 19 Gayland and Baxendal CFD findings (Gaylard & Baxendale, 1998).....	41
Graph 20 notch car performance coefficient results	45
Graph 21 notch car performance force results	46

1. INTRODUCTION

Aerodynamics in the automotive and motorsport industries is one of the most important aspects when considering designing a vehicle in the modern era as it can influence the car's performance dramatically. For example; Formula 1 allocates hundreds of resource hours developing the aerodynamic package. A substantial amount of that resource is applied to numerical methods using computational fluid dynamics (CFD) to simulate a body (aerodynamic package) moving through a fluid field. But with using these computational tools to predict the aerodynamic performance of a body, comes with a set of challenges to have confidence in the method and results. The output of CFD is worthless until the simulation methodology is validated against real life testing data.

This report documents the results obtained from carrying out a study to build a baseline CFD methodology and process to simulate automotive/motorsport shaped bodies. The study uses wind tunnel test data from MIRA's notch reference geometry car (see figure 1) as a validation tool when building the methodology. The study will encompass;

1. Obtaining data of the notch reference geometry car and understand what boundary conditions it was tested in.
2. Research into CFD best practice for simulating automotive body shapes
3. Build a CFD process from research as a starting point
4. Define the parameter studies to be completed
5. Carrying out parameter studies into testing domain size, meshing, and solving models.
6. Outline the best settings found from the parameter studies for simulating the reference car
7. Using the best CFD settings identified, test an improved performance model of the reference car.

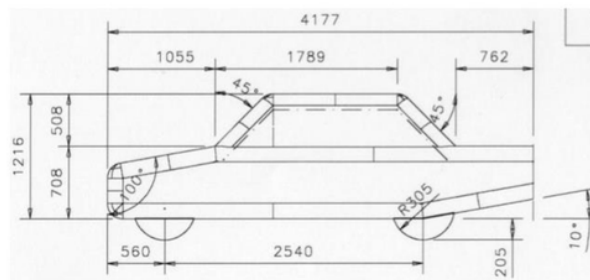


Figure 1 MIRA's notch reference geometry car (Kumar & Vaidya, 2016)

The initial step is examining the MIRA geometry and CFD parameters.

2. NOTCH REFERENCE GEOMETRY AND CFD RESEARCH

2.1 Notch Reference Geometry

The MIRA reference geometry is used to calibrate the wind tunnel facilities at MIRA. It comes in three configurations:

- A notchback style (see figure 1)
- A square-back style (see figure 2)
- And a fast-back style

The coefficient of lift (Cl) and coefficient of drag (Cd) data from MIRA's test within their wind tunnel has been supplied by Tim Tudor for the notchback style reference car, along with the upper and lower surface pressure data. As there is sufficient amount of data supplied from Tim Tudor, the study will only look at the notchback style configuration reference geometry. Another reason for using the notch MIRA car as described in Gaylard and Baxendale's CFD paper is that it is a more realistic simple model for automotive CFD calibration when compared to an alternative Ahmed body. (Gaylard & Baxendale, 1998) (Tudor, 2018)

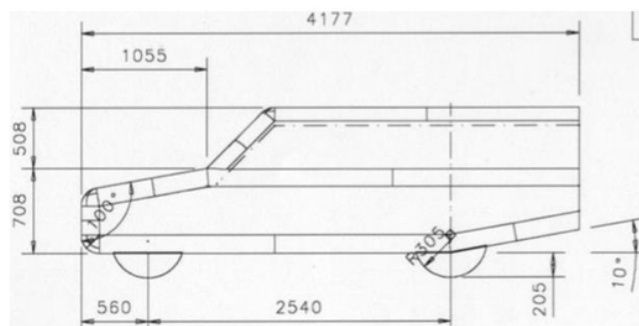


Figure 2 MIRA's square-back reference geometry car (Kumar & Vaidya, 2016)

2.1.1 MIRA notchback testing conditions

Gaylard and Baxendal's paper looking into use of CFD to predict aerodynamic characteristics of simple automotive shapes carried out testing on the notch car at MIRA's wind tunnel. They have listed the testing conditions as follows:

- The test was carried out in the full-scale open loop wind tunnel
- The working section of the tunnels was 7.92m wide X 4.42m high X 15.24m long
- Flow uniformity to $\pm 2\%$
- Turbulence intensity of 0.9%
- Fixed floor with a boundary layer fence installed to control boundary layer build up (Gaylard & Baxendale, 1998)

These testing conditions are also stated in a paper by Kumar & Vaidya looking into CFD methodology to predict drag and lift of passenger vehicles. (Kumar & Vaidya, 2016)

Air temperature, air density, and air speed of the test, will be adopted as shown in table 1 (section 3.2). The air speed of 20m/s has been provided by Tim Tudor as this was the testing conditions for the data recorded. (Tudor, 2018)

2.2 Finding an Initial CFD Approach

The first principle factor to address is whether to carry out a static or transient CFD study. The notch car data is collated while the air is at a contested speed so it would indicate that a static study would be sufficient. This reasoning is backed up by the studies carried out by Gayland/Baxendal and Kumar/Vaidya which were both done as a static CFD case. If the study was done in transient, it would consume a large amount of PC processing hours just to complete one run, thus taking an unreasonable amount of time to complete the study. (Gaylard & Baxendale, 1998) (Kumar & Vaidya, 2016).

The study will also be using a pressure based solver. The pressure solver is used for a range from low speed incompressible flow to high speed compressible flow. It requires less computational resource and less expensive compared to density based solver. As the notch car analysis is at fairly low speed (20m/s), it makes sense to use pressure based type particularly as it can achieve the same results in low speed as density based. (ANSYS, 2012)

Static CFD studies comprise a few important aspects for consideration when setting up a model, all of which all have a bearing on the output results. They are:

- Domain Size
- Vehicle position
- Meshing
- Boundary Conditions
- Turbulence Model

2.2.1 Domain size

A paper examining best practice for automotive external aerodynamics modelling by Lanfrit found that a good starting point for the domain size is to have a ratio of 3 car lengths in front of the car, and 5 lengths behind. The displacement of the car within the domain should be no greater than 1.5% of the total cross sectional area (see figure 3). However Lanfrit also goes on to say that this may not be appropriate when trying to model a car in a wind tunnel. In this case the simulation should take into account the wall effects (wind tunnel blockage) on the body. (Lanfrit, 2005)

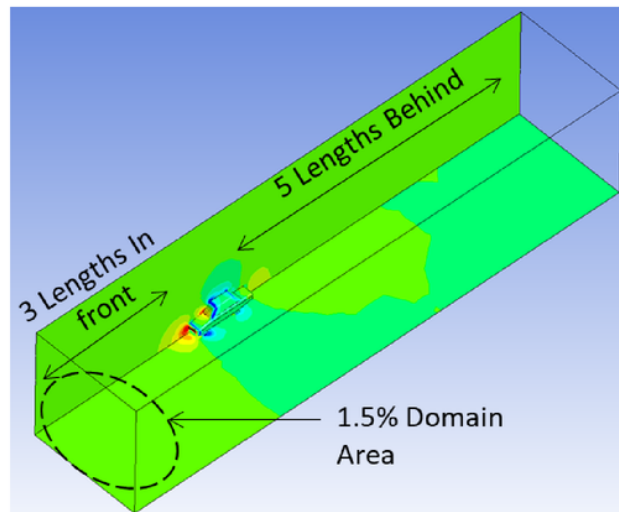


Figure 3 showing domain sizing

Kumar & Vaidya paper come to the same domain size conclusions as Lanfrit. They carried out the CFD study of the notch car with the domain size chosen being that of the MIRA wind tunnel working section. (Kumar & Vaidya, 2016) Gaylard and Baxendal also took the same CFD domain size approach. (Gaylard & Baxendale, 1998)

2.2.2 Vehicle position

Both Kumar/Vaidya and Gaylard/Baxendal modelled the notch car positioned in the X and Y coordinate centre of the domain to replicate MIRA testing conditions. Both papers also selected the ride height of the vehicle such that the wheels are just in contact with the floor. Examining this aspect further, there does not appear to be any credible information as to what exactly was the ride height of the notch car when being tested for the data obtained for this study. (Kumar & Vaidya, 2016) (Gaylard & Baxendale, 1998)

2.2.3 Meshing

There are a few meshing strategies to consider. ANSYS give the following (see figure 4) recommendations when meshing for simulating external aerodynamic bodies.

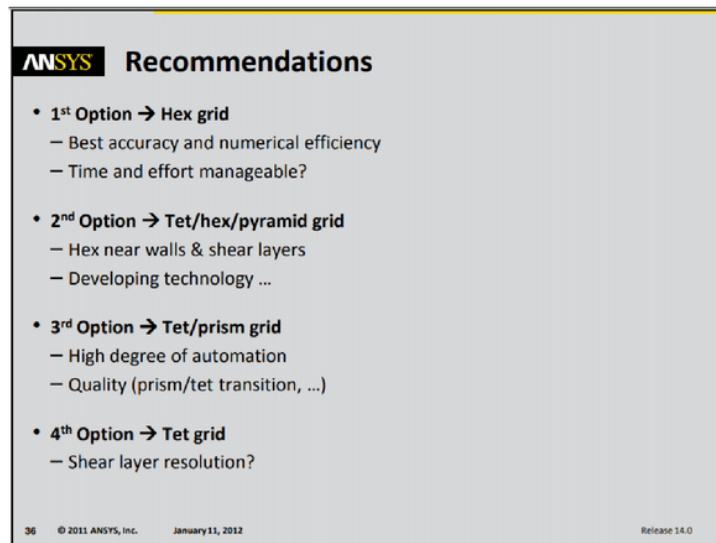


Figure 4 ANSYS meshing recommendations (ANSYS, 2012)

ANSYS found the best option is to use a hex grid mesh for the best accuracy. However, hex elements are difficult to generate on complex geometry, and most automotive shapes do have a high level of complexity.

Alternatively, the best all round option is to use a hybrid meshing strategy of tetrahedron elements making up the majority of the domain, together with prism layers at the surface of walls in shear contact with the fluid (air). Tetrahedron elements can be easily generated on complex geometry shapes whilst still maintaining a good level of accuracy. Prism layers better capture the shear layer (boundary layer) developing along a body in a fluid stream principally for two reasons:

1. Boundary layer is a complex phenomenon which is very thin and the solution gradients are very high, which means it requires minimization of skewed cells. Tetrahedron elements could be adequate but they would need to be extremely fine and this would result in a vast amount of elements to solve. Prism layers are very structured giving good quality cells without the need of having a significant amount of additional elements, thus saving computational time and power needed. (Chitale, 2014)
2. The number of layers, layer heights, and growth rate can all be controlled easily to account for the near wall treatments of a boundary layer. The y direction of the boundary has 3 main levels to it, sublayer + buffer layer, log-layer, and outer layer (see figure 5) which can be captured efficiently by having the control over the prism layers. Y plus method is a non-dimensional way of defining the different flow pattern layers within a boundary layer for different scenarios. Y plus can be useful when identifying the required height of the first cell to meet turbulence models needs to accurately capture the boundary layers. (Chitale, 2014) (ANSYS, 2012)

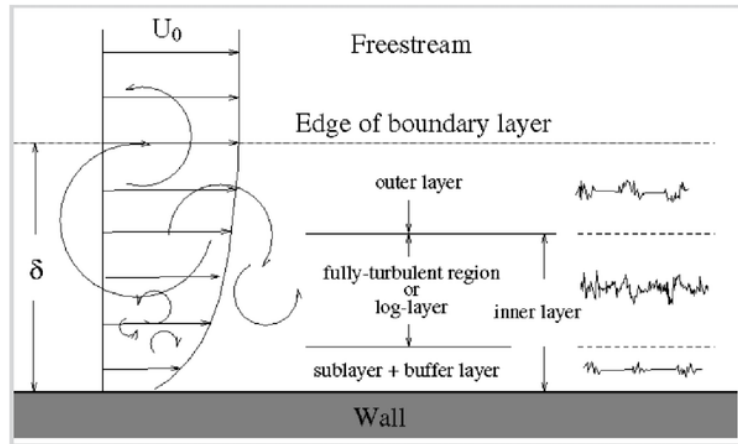


Figure 5 boundary layer's interaction with a wall (ANSYS, 2012)

Lanfrit's study recommends the same meshing strategy as being the best round all option for external automotive aerodynamics simulation. He recommends the prism layers should be 5 and the growth rate 1.2. But this is just a precondition; it is not clear what number of layers or growth rate there should be. y^+ plus can give the height of the first cell. After reviewing the literature, it appears to come down to experience as to what the number of cells and growth rate should be for a part flow pattern. (ANSYS, 2012) (Lanfrit, 2005)

Kumar & Vaidya used a similar meshing approach but had prism layers on the wind tunnel walls to capture the boundary layer developing. Gaylard & Baxendale took a different approach of using hex elements only. Along with Gaylard & Baxendale, a paper by Soares on the influence of CFD setup of a 3D realistic car model used hex elements. (Kumar & Vaidya, 2016) (Gaylard & Baxendale, 1998) (Soares, 2015)

Theoretically, the smaller the elements the greater amount of geometry and fluid flow detail will be captured resulting in more accurate results. But there is a point in which the % of accuracy does not increase substantially against the number of elements being generated. At that point it is considered to be wasting computational solving time. (Soares, 2015)

ANSYS also recommend a certain level of mesh quality. They give a scale of number of acceptable bad elements for cell skewness and orthogonal quality (see figure 6). (ANSYS, 2012)

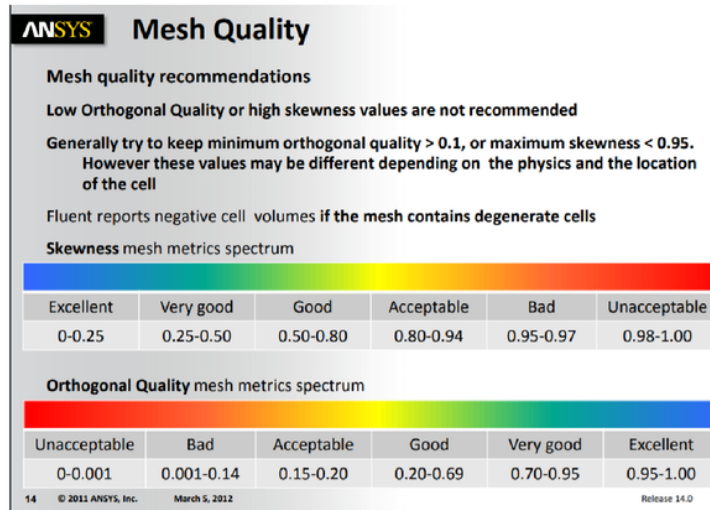


Figure 6 ANSYS mesh quality scale (ANSYS, 2012)

2.2.4 Boundary conditions

The boundary condition of the CFD simulation can be carried out in a variety of different ways. As the notch car is tested in a wind tunnel the front of the domain will be the velocity inlet, and the back will be the pressure outlet because the study is using a pressure based solver. Kumar & Vaidya came to the same reasoning. They treated the walls of the domain as 'no' slip walls with prism layers to capture any blockage effect of the wind tunnel domain. Because the notch car is stationary during testing, the floor is also treated as 'no' slip walls. (Kumar & Vaidya, 2016) Lanfrit takes a different approach with tunnel walls treated as a symmetry condition (a no viscous affect wall). (Lanfrit, 2005)

2.2.5 Turbulence model

A major issue in numerically modelling aerodynamic flows around vehicles is the representation of the boundary layer. The turbulence model is a numerical method of predicting the complex three-dimensional viscous layers within a fluid for CFD. There are many different models for different flow structure scenarios. For external automotive aerodynamic flow, Lanfrit recommends either K-Epsilon realizable, or Reynolds stress model (RSM). Both Kumar/Vaidya and Gaylard/Baxendal use K-Epsilon realizable for the notch car model. ANSYS also gives information on the different models. It finds K-Epsilon realizable a strong model in adverse pressure gradients areas and excels at solving complex geometry shapes. ANSYS recommends other models including K-Omega but for finding flow separation. (ANSYS, 2012) (Lanfrit, 2005) (Kumar & Vaidya, 2016) (Gaylard & Baxendale, 1998)

The K-Epsilon turbulence model has numerical wall treatment features for predicting the boundary layer interaction with wall behaviour. According to Lanfrit, it is recognized that resolving the near-wall region down to the wall is not practical in CFD. Lanfrit recommends using non-equilibrium wall-functions (NEWF) for external flow. NEWF, being sensitized to the effects of pressure gradients, account for the effects of local variation in the thickness of the viscous sublayer when computing the turbulent kinetic energy budget in the wall-adjacent cells. Again both Kumar/Vaidya and Gaylard/Baxendal use NEWF for their studies. Interestingly ANSYS recommends using enhanced wall treatment instead of NEWF. (Lanfrit, 2005) (Kumar & Vaidya, 2016) (Gaylard & Baxendale, 1998) (ANSYS, 2012)

3. PARAMETER STUDIES METHODOLOGY

Having now established an understanding of the notch reference geometry testing conditions and a better picture of the CFD approach, the next part of the study is to define the parameter studies methodology.

3.1 Apparatus For Studies

The studies will use the following:

- CFD package – ANSYS 16.2
- MIRA Notch Reference Geometry CAD. Supplied by Tim Tudor
- CAD Software – Solidworks

3.2 Testing Conditions

The following table sets out the testing conditions going to be used for all the CFD simulations.

Table 1 testing conditions

Test parameter	Notes
Air Speed 20m/s	Provided by Tim Tudor
Air Density 1.225kg/m ³	Assuming the MIRA test was done at sea level
Air Temperature 20°C	Assuming the ambient air temperature
Ambient Air Pressure is 1BAR	Assuming the MIRA test was done at sea level

3.3 Notch Wind Tunnel Data

From the notch car testing data collected, the C_l and C_d is found to be -0.0845 and 0.2963 respectfully. This is the target for the parameter studies to match. In addition, the end goal is also to match the upper and lower surface pressure data between the test and CFD. (Tudor, 2018)

3.4 Initial CFD Standard Settings

As outlined earlier on in this report, the CFD simulations will be static using the pressure based solver.

3.4.1 Domain settings

Following on from the review in section 2.2.1, the best domain sizing to use is modelling the MIRA wind tunnel section. Both Kumar/Vaidya and Gaylard/Baxendal achieved in their studies results to get a worst case C_l and C_d difference of 2% between the CFD and wind tunnel data. Gaylard/Baxendal also managed to get the upper and lower surface pressure data between the two sets very close. . However Lanfrit found overall the best sizing ratio for most applications and for the purpose of this exercise, the domain size and notch car position within the domain will be found from a parameter study. On conclusion of the all the parameter studies, a comparison of the best domain size and settings will be made against additional run using the MIRA wind tunnel dimensions.

3.4.2 Meshing settings

The meshing throughout parameter studies will use the hybrid meshing strategy of tetrahedron elements and prism layers discussed in section 2.2.3. This approach gives the best all round compromise of results accuracy and computational time.

Mesh sizing studies are necessary to find the optimum meshing settings for the notch car. This exercise also involves attempting to improve the quality of a mesh.

Prism layers will, at commencement be set to 5 and a growth rate of 1.2 as recommended by Lanfrit for the initial parameter studies until proven otherwise from a prism layer study.

ANSYS has capabilities of using different mesh approaches when meshing the geometry. For the initial domain studies, the Robust Octree mesher, which is a top-down approach, will be used. The Octree method creates a mesh from the boundaries of the domain using a grid generation process. This structured approach generally works for most CAD models even if poor quality and takes less computational time, hence is a good starting point. A study looking into the capabilities of the different approaches will be explored. (ANSYS, 2012) (Sharcnet, 2013)

3.4.3 Boundary conditions settings

The inlet will be air velocity inlet, and the outlet will be pressure outlet. The floor will be a 'no' slip walls with prism layers.

The domain walls should be treated as 'no' slip walls with prism layers to capture any blockage effect of the wind tunnel from the findings of other papers (see section 2.2.4). Again, Lanfrit has a secondary opinion but for the purpose of this exercise, the domain wall will be symmetry condition. That is until all the studies are completed, and then a comparison of using the output best domain size and settings will be made against additional runs using the MIRA wind tunnel dimensions with the walls treated as 'no' slip conditions. This will give the benefit of seeing how the wall conditions affect the results at the end of the studies.

In addition, as the notch car geometry is a symmetrical body and to save on computational time, all the parameter studies will be simulated as half a car. The mirror wall side of the car will be treated as a symmetry wall throughout all the studies.

See figure 7 for graphical representation of parameter studies boundary conditions.

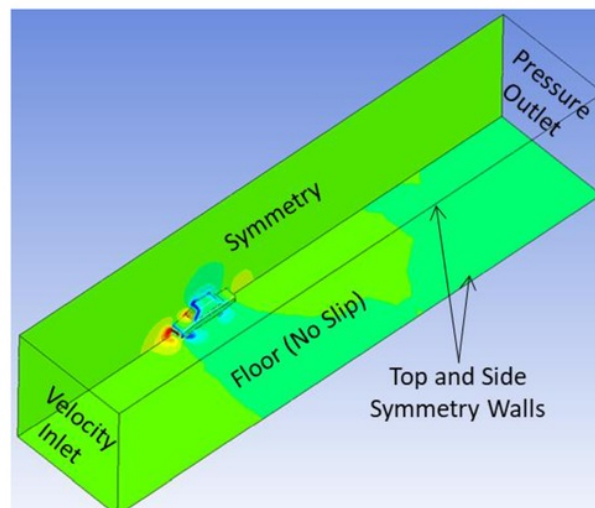


Figure 7 graphical representation of parameter studies boundary conditions

3.4.4 Turbulence model settings

As a starting point, the K-Epsilon realizable value with NWF model will be used as found from research (see section 2.2.5) to be the all-round best numerical model for predicting the C_l and C_d for external aerodynamics CFD simulation. From ANSYS and Lanfrit there is a case for testing different models and wall treatments to further improve results.

3.5 Summary Of The Studies To Be Completed

Table 2 below outlines the areas to be tested to find the optimum settings for simulating the MIRA reference geometry notch car.

Table 2 list of studies

No.	Study
1	Domain Size
2	Mesher Study
3	Wheel Contact Patch
4	Meshing Settings
5	Prism Layers
6	Turbulence Models
7	Extra Setting Runs

4. PARAMETER STUDIES RESULTS, ANALYSIS AND DISCUSSION

4.1 Initial Run/Domain Size Results

For this study, the domain size and car position is defined at the same time by using a method of scaling parameters between the domain and the dimensions of the car. This approach is identical as detailed by the papers discussed in section 2.2.1.

Applying the initial settings considered in section 3.4, the first run modelled the domain as just one car length in all directions. The results from that are as follows:

1. Observing at the pressure contour on the symmetry plane, it is apparent the size of the domain cuts off capturing the pressure changes within the freestream. This indicates the need for the domain to be higher (see red circle in figure 8).

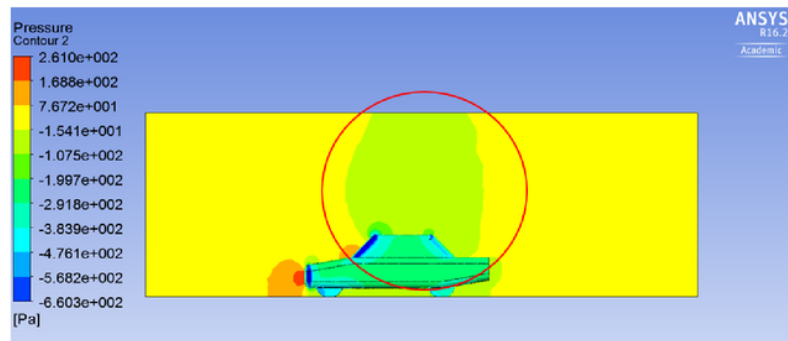


Figure 8 symmetry wall pressure contour domain size 1

2. By plotting the velocity on the symmetry wall, it can be seen that in front and behind the car, the change in freestream velocity again being cut off by the domain (see black circles in figure 9).

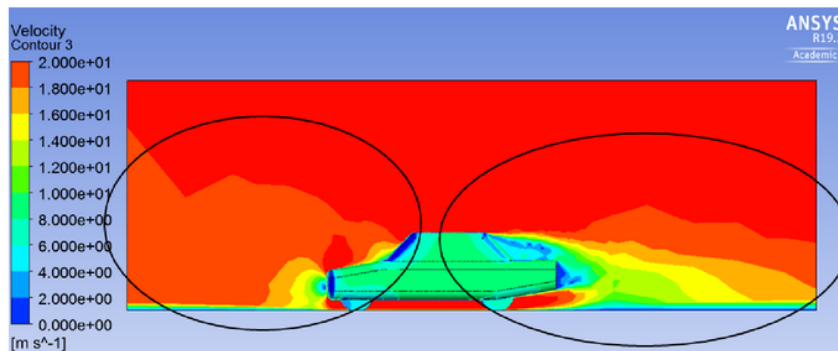


Figure 9 Velocity on symmetry wall domain size 1

3. The floor is experiencing the same problem with the domain cutting off capturing the pressure changes (see figure 10 black circle).

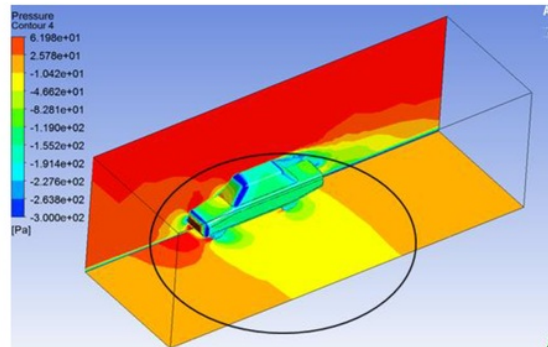


Figure 10 floor with pressure contour on domain floor (domain 1)

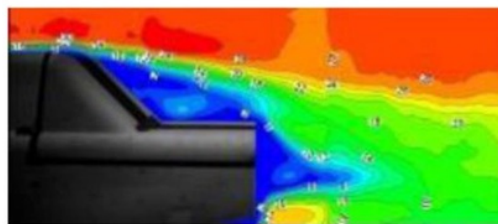
This first trial domain shows the current size is not capturing all of the flow patterns around the car, thus the domain needs to be bigger.

4.1.1 The flow patterns around the initial CFD run

The results from the first run are showing some flow patterns around the car which is building a picture of where the model settings might need to be refined to capture all of the flow characteristics. For example, looking at figure 9, the wake of the notch car has a lot of turbulence flowing off the rear windscreen pillar which indicates two things.

Firstly, the mesh should be refined at the slant and wake region to capture the boundary layer development and turbulent air as this impacts on the drag of the car.

Secondly, because there is such a big turbulent air region, selecting the right turbulence model to predict or capture flow separation could have a massive effect on the drag results, which in turn could have affect the lift results. Kumar and Vaidya paper found, through there wind tunnel experimental work, the notch car to have a substantial turbulent wake region (see figure 11), and in addition the wake area is key to getting the CFD to align with the data. Gaylard and Baxendal's paper also find the same. (Gaylard & Baxendale, 1998) (Kumar & Vaidya, 2016)



(a). Notchback, Exp[10] velocity field

Figure 11 Kumar and Vaidya experiment results of the rear wake region of the notch car (Kumar & Vaidya, 2016)

Other key parts to the notch car are as follows:

1. The red circle in figure 12 shows high pressure build up on the front face (red contour), with the flow bleeding underneath the car in the dark blue region, thus indicating the flow is indeed traveling beneath the car. Another point to note is the high pressure region on the front wheel. At the rear wheel area (black circle) there is a kick point (light blue contour) for the flow. With the rear underbody having a slant back together with the kick point, all points towards a diffuser effect. The negative lift (-0.0845 Cl) created is coming from the underbody flow of the notch car. In addition, the turbulent area on the top section of the car could also be contributing to the lower pressure region. This all points towards there being considerable flow underneath the car, hence increasing the mesh in this area might be considered appropriate in the future.

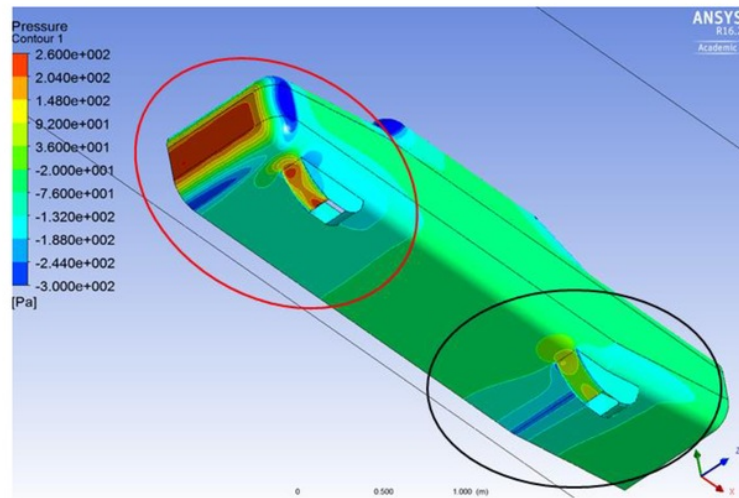


Figure 12 pressure contour on notch car from first run (domain 1)

2. The stagnation point on the front of the car within red pressure contour in figure 13 (red circle). This together with observing intense flow over the bonnet and front windscreen top marked by dark blue pressure contour (black box) suggests a finer mesh might be needed.

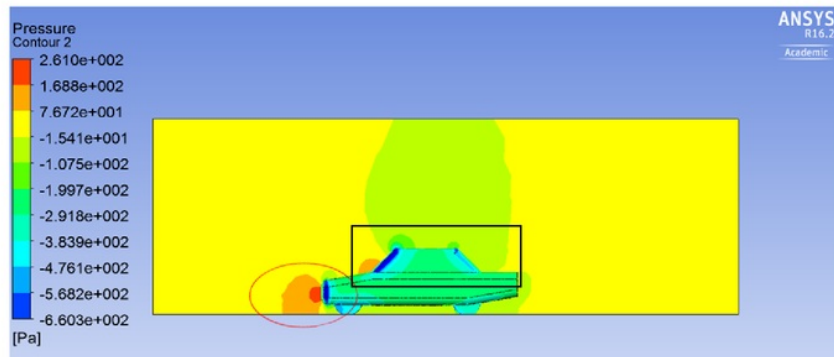


Figure 13 pressure contour on notch car from first run (domain 1) side view

3. All outputs on this initial run point towards the notch car having the flow patterns typical of a saloon shape type vehicle; high pressure build up at the front, fast flow regions on the vehicle top, and finally a separation/wake area coming off the rear wind screen slant. Kumar

and Vaidya adopted the same reasoning to refine the mesh in those regions (see figure 14). (Kumar & Vaidya, 2016)

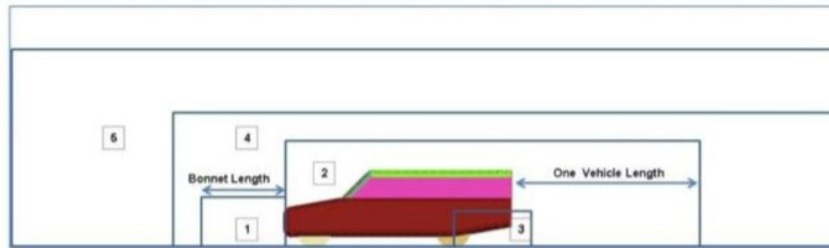


Figure 3. Refinement Boxes

Figure 14 Kumar and Vaidya mesh refinement boxes used

4.2 Further Domain Study Results

4.2.1 Domain size 2

From the initial results, the next run doubled the car lengths all round which resulted in pressure contours on the symmetry wall and floor providing adequate height and width dimensions of the domain (see figure 15; the pressure contours are not curtailed).

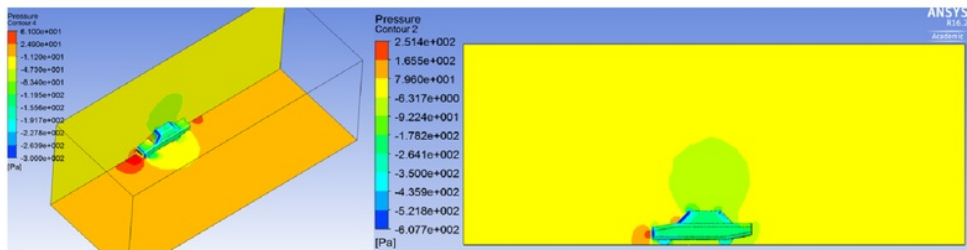


Figure 15 Domain 2 pressure contours showing the height and width not cutting off the pressure changes of the notch car

However, the domain length has not captured the velocity changes (see figure 16).

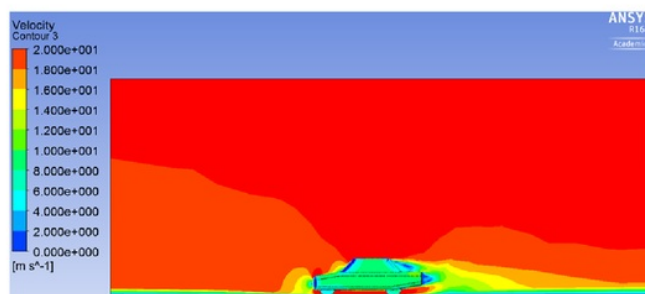


Figure 16 Domain 2 symmetry wall velocity contour showing velocity changes not all being captured in the domain length

4.2.2 Domain size 3

Domain size 3 retained the same height and width as Domain size 2. The length in front of the car was set to 3, and the rear to 5. The difference between the front and rear lengths meant the car is off-set and the influence of the car position is relevant and incorporated into the study.

The results from domain size 3 showed all the pressure and velocity contour changes due to the car being in free stream were being captured (see figure 17) on the domain walls (floor, symmetry). The C_l and C_d values calculated from size 3 had the best percentage difference to the wind tunnel data (C_l -8.28% and C_d 0.05%).

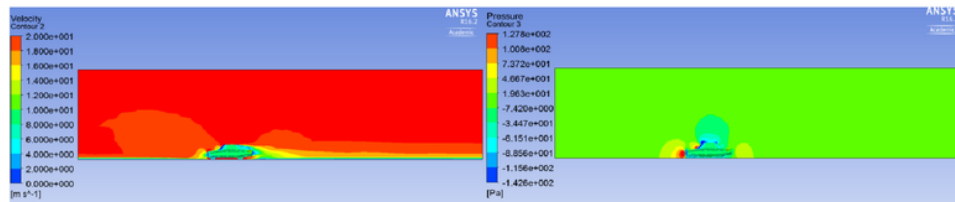
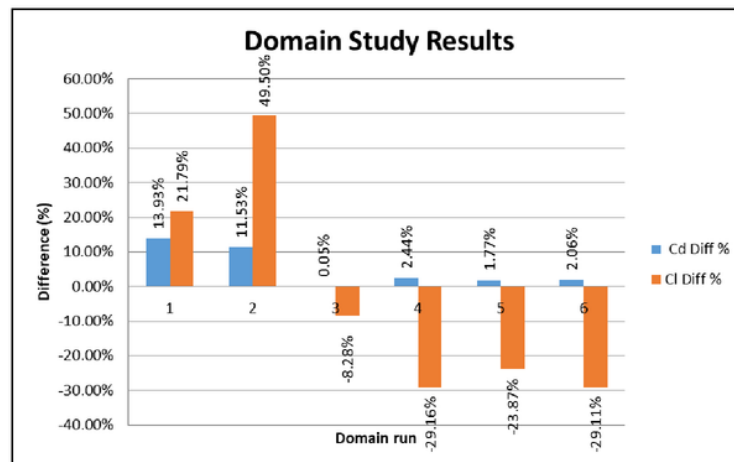


Figure 17 Domain size 3 contour results

4.2.3 Rest of the Domain size results

After trialling size 3, the domain dimensions were enlarged further, but without further improvements in the outputs. Two more sizes were tested with the same length adopted as domain 3 but with the width and height extended. The results were not as good in comparisons to size 3 (see graph 1).



Graph 1 Domain size study C_l and C_d results

A commentary on Graph 1 which shows outputs of 6 sizing runs. It is evident leading up to size 3 the results are improving which is understandable when considering the wall contours. However after size 3 the results of C_l deteriorate. The main different between 3 and 4/5/6 is the height and width size (see table 3). This is indicating that even though the top and side walls are symmetry and should have no boundary layers developing, there is a blockage effect in the domain. Section 2.2.1 discussed how this might be because the notch car was tested in a wind tunnel and the domain size should represent the same to account for blockage effect. It is interacting with the C_l from speeding increasing flow rate more or less in the freestream; this makes sense considering the C_d isn't changing dramatically.

Table 3 Domain size study sizing

Run	Forward (car Length)	Wake (car Length)	Side and Top (car Length)
Domain 1	1	1	1
Domain 2	2	2	2
Domain 3	3	5	2
Domain 4	4	6	3
Domain 5	3	5	2.5
Domain 6	3	5	1.5

The analysis of the domain results was carried out after completion of parameter studies. In hindsight the domain results should have been analysed more thoroughly before proceeding with the rest of the studies. The findings from this analysis show how important it is to model the working section of the wind tunnel when comparing data because of the interaction with the walls. Figure 18 and 19 is a case comparison between domain 3 and 4 size. Both comparisons show how the pressure and velocity patterns change because of wall interaction. The findings also indicate when using CFD for developing an aerodynamic package for an open environment, the domain should be as big as possible to reduce any blockage effect. In addition further runs should have been carried out to move the notch cars position further within the domain.

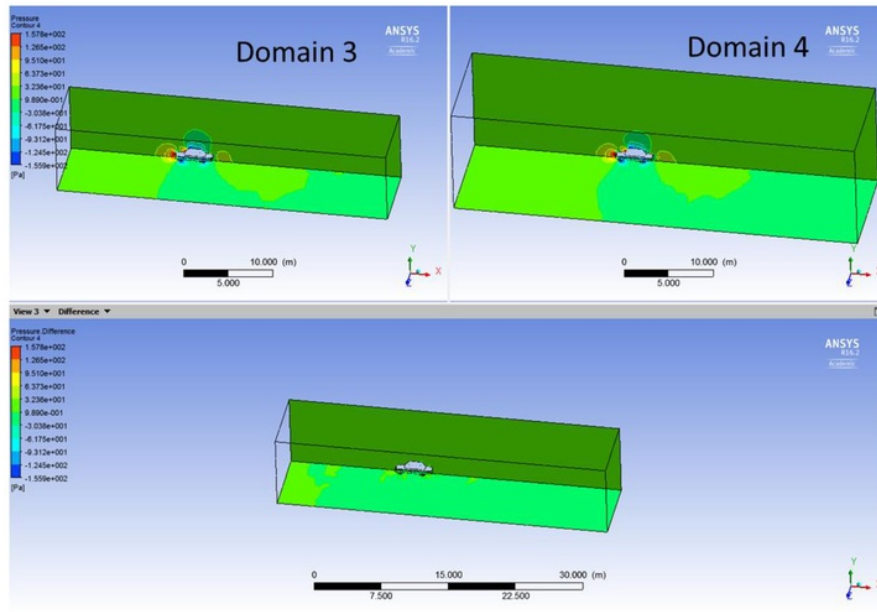


Figure 18 case comparison between domain 3 and 4 size pressure contours

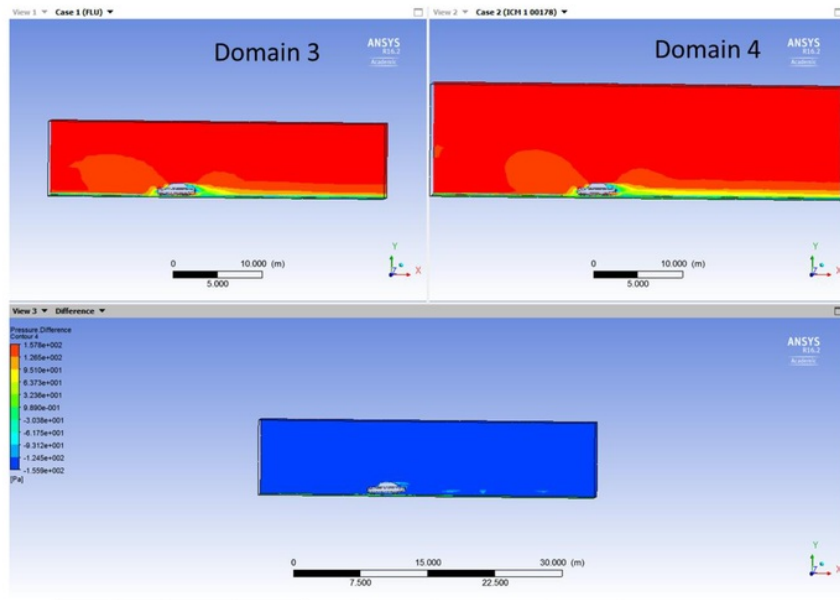


Figure 19 case comparison between domain 3 and 4 size velocity contours

Domain size 3 was adopted as it was found to have the best results for C_l and C_d .

4.3 Mesher Results

Up to this point, the studies have been conducted using the Octree mesher. Three other mesher will be tested.

Mesher types:

1. **Octree (Robust)** – top-down approach using a structured grid. Figure 20 is the Octree mesh, showing how the cells are very structured. As explained in section 3.4.2, it is great for poor geometries. (Sharconet, 2013)

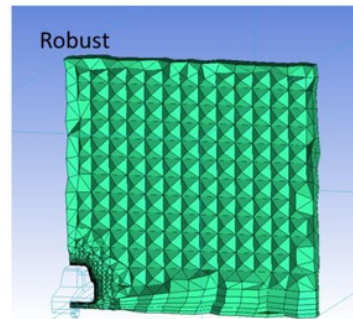


Figure 20 Octree mesh

2. **Delaunay** – Is a bottom-up approach and grows the mesh naturally from an existing surface mesh. The mesh is shown in figure 21. Comparing the Octree and Delaunay mesh type it can be observed the structure of the Delaunay is considerably more random. (Sharconet, 2013)

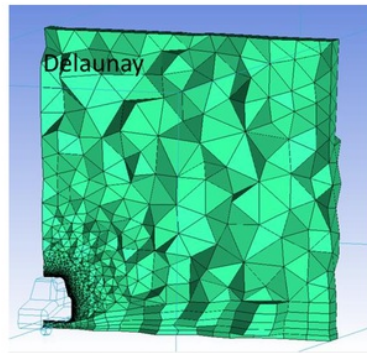


Figure 21 Delaunay mesh

3. **Smooth and Fluent Meshing** – Both mesh types are similar to Delaunay.

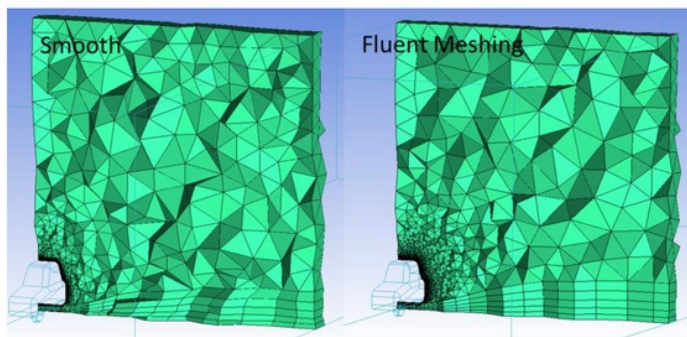
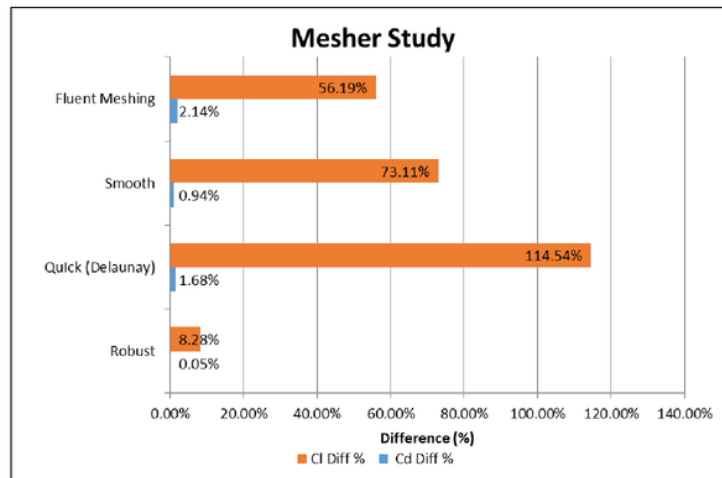


Figure 22 Smooth and Fluent meshing

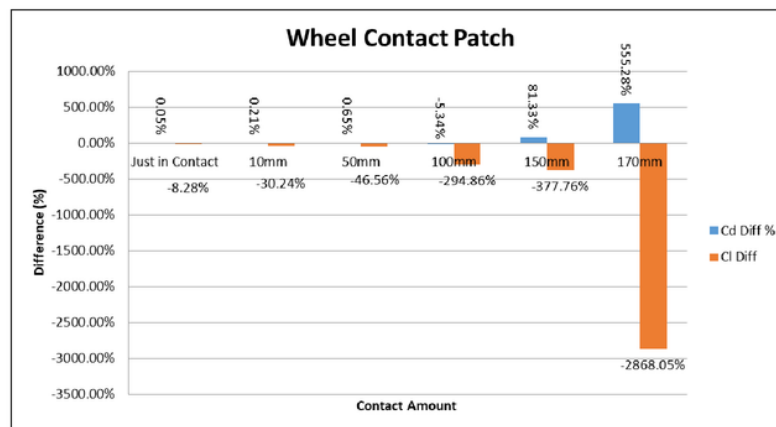
Graph 2 below is the results of the mesher study from using the different mesh types. Octree (Robust) gave the best results. Sharcent is a user guide for ANSYS ICEM mesher and it recommends Delaunay as the best type as it grows naturally which is more in line with fluid dynamics (FD) flow patterns (FD mechanics is not governed by strict laws). The Delaunay requires a lot of refinement to obtain best results and a descent geometry model. The Octree method was adopted but following subsequent additional reading about the mesh types, the model could have benefited from more refinement to obtain best results. (Sharcent, 2013)



Graph 2 mesher types results

4.4 Wheel Contact Patch Results

To run the geometry successfully in CFD, the wheels have to be intruding past the domain floor. In Kumar and Vaidya's report, they report the notch car model, when being tested, has the wheels just in contact with the floor. (Kumar & Vaidya, 2016) Changing contact patch could affect the results. A study changing the wheel contact patch amount has been carried out to find the effect on the results and confirm Kumar and Vaidys's report (see figure 23). The results are as follows:



Graph 3 wheel contact patch results

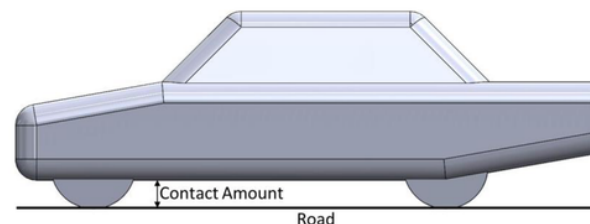


Figure 23 illustration of the contact amount changing

Immediately, the results are affected as the ride height of the car starts to lower and the contact patch expands. Lowering the ride height puts the car more into ground effect increasing the flow speed beneath the underbody and hence the CI value is greatly affected. Conducting a case comparison between the “just in contact” and 150mm lower shows the flow underneath increasing speed (see black circle figure 24). The increase is also changing the wake region as well, which could explain why the Cd is different. Outcome is to leave the wheels just in contact.

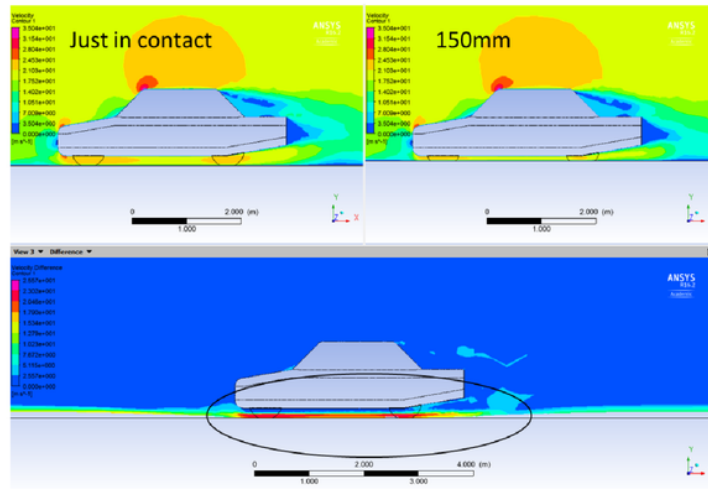


Figure 24 case comparison between the “just in contact” and 150mm velocity contours

4.5 Model Progress

Now considering the mesh quality; the skewness and orthogonal quality of the current mesh (see figure 25) are both outside ANSYS’s recommend scale (see section 2.2.3, note the ANSYS scale for skewness is the opposite way around with ICEM scale). There are a considerable amount of bad elements arising from the point where the wheels are in contact with the domain floor (see figure 26). These bad cells occur because of the sharp angles from the interaction of the wheels with the floor. To try and reduce the angle by moving forward will introduce 5mm high wheel blocks at the interaction point. Landrit also takes this approach. (Lanrit, 2005)

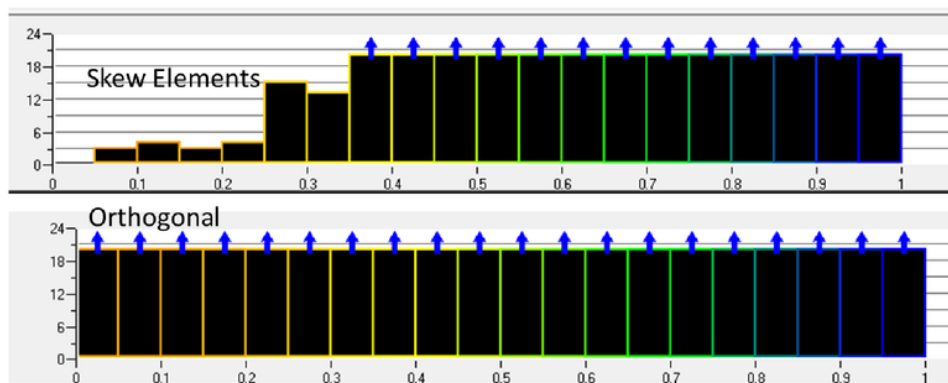


Figure 25 current mesh quality

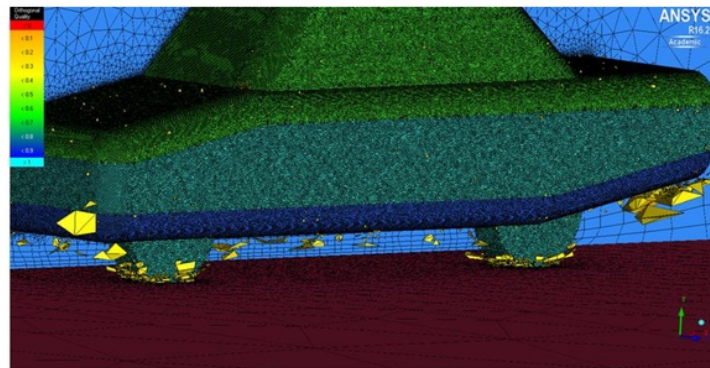


Figure 26 poor quality elements coming from the wheels

The next phase of the studies will refine the mesh and attempt to further reduce the bad cells.

Figure 27 below shows the velocity flow streams from the best run results so far. At the rear of the notch car, the streams become dark blue and recirculating behind the screen (black circle). This is further evidence of the flow separation and turbulent wake region discussed in section 4.1.1. The red circle shows how the underbody up wash is interacting with the flow coming off the rear slant profile. This interaction means capturing the separation point could be important predicting the lift of the car.

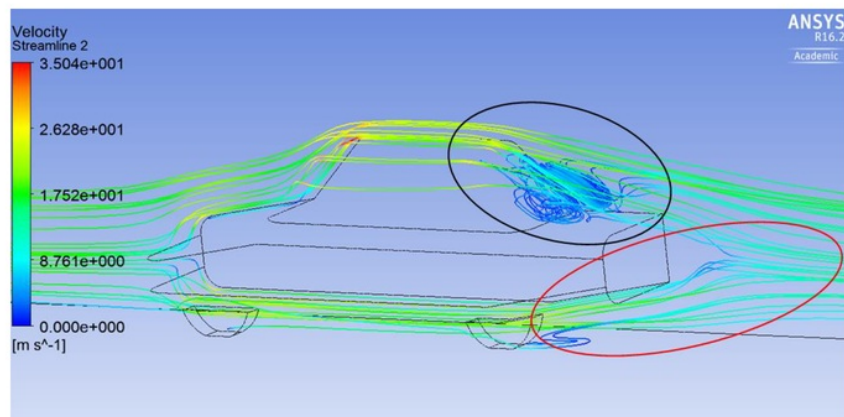


Figure 27 velocity flow streams on notch car

Figure 28 is displaying the vortices coming off the rear car region. Also from around the front wheel. The mesh needs to be refined in the rear area of the car

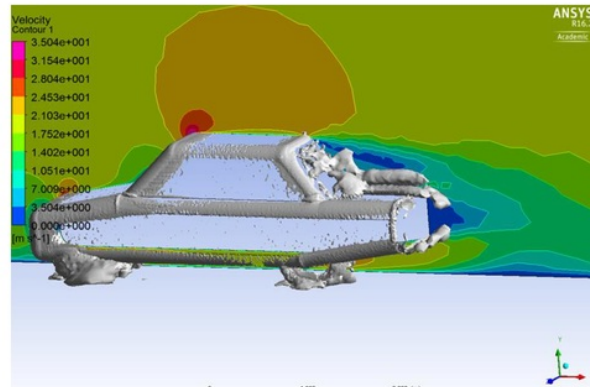
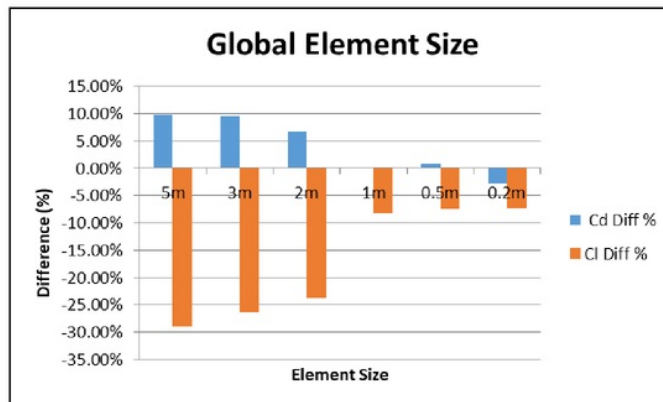


Figure 28 notch car vortices (white contours)

4.6 Mesh Refinement Results

4.6.1 Global element size

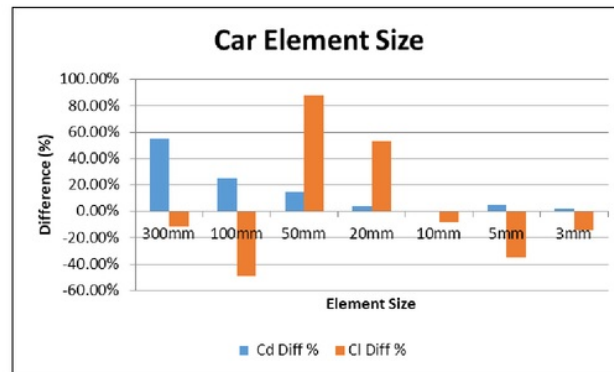
The global size is the max size of the elements in the domain. Graph 4 below is the result of trying different sizes. 1m size was found to be the optimum between accuracy of results and computation time. 1m will be used in the next phase of studies.



Graph 4 global element size results

4.6.2 Car element size

The element size on the car is important to accurately model the notch geometry. Graph 5 shows the results of changing the size. 10mm was found to be the best results accuracy and computation time.



Graph 5 car element size results

4.6.3 Mesh density box size

To further improve the results without greatly increasing the computation time, a mesh density box can be used (car box). Using a denser mesh only in the regions where complex flow is expected (see figure 29) means avoiding a very fine global mesh, thus saving computation time. In sections 4.5 and 4.1.1 it was identified that the area around the car would benefit from having a finer mesh due to the complex flow regions. Kumar and Vaidya used mesh box to refine their results. Landrit also used recommends using mesh boxes as well. (Kumar & Vaidya, 2016) (Lanfrit, 2005)

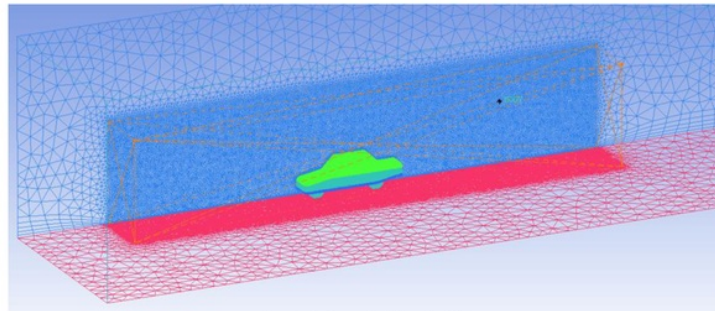
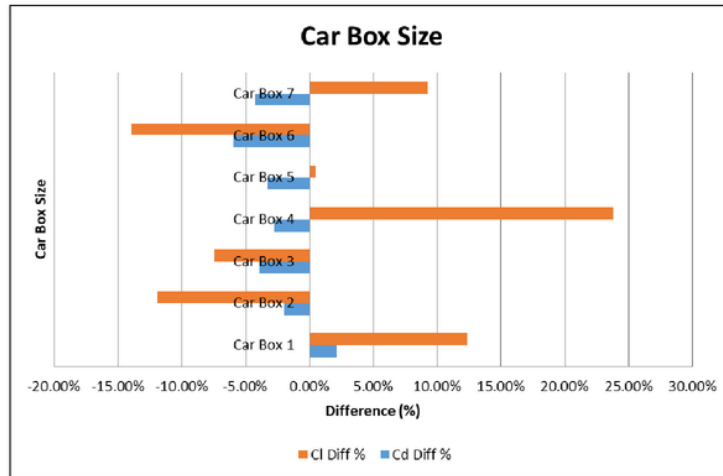


Figure 29 mesh density box

As with the domain study, the method for defining the size was to scale parameters of the box to the dimensions of the car and the element size in the box was set to 100mm. The study found the most affective size to be box 5 (see graph 6) of 2 car lengths all round (see table 4 for all box sizes). Upon reflection, the box position should have been considered in depth to optimize even further.



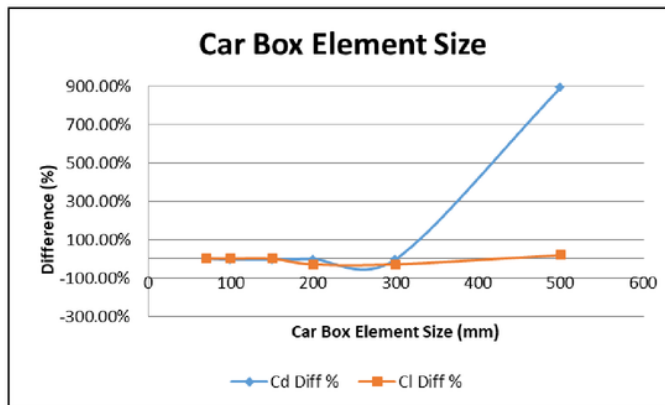
Graph 6 car box size results

Table 4 Car box size study sizing

Run	Forward (car Length)	Wake (car Length)	Side and Top (car Length)
Car Box 1	0	0	0
Car Box 2	0.2	0.2	0.2
Car Box 3	0.5	0.5	0.5
Car Box 4	1	1	1
Car Box 5	2	2	2
Car Box 6	0.5	1	0.5
Car Box 7	1.7	1.7	1.7

4.6.4 Mesh density box element size

With the box size defined, the next step was to find the best element size within the box. Graph 7 displays the results. Between 200mm to 100mm was found to be the best for accuracy of results and computation time. Below 100mm, the results did not improve but increased the time it took to resolve.



Graph 7 car box element size results

100mm size will be used for the next phase of study.

The mesh quality was examined again. From refining the mesh and using wheel blocks the quality has improved greatly (see figure 30). Both skewness and orthogonal quality have come into acceptable levels according to ANSYS scale.

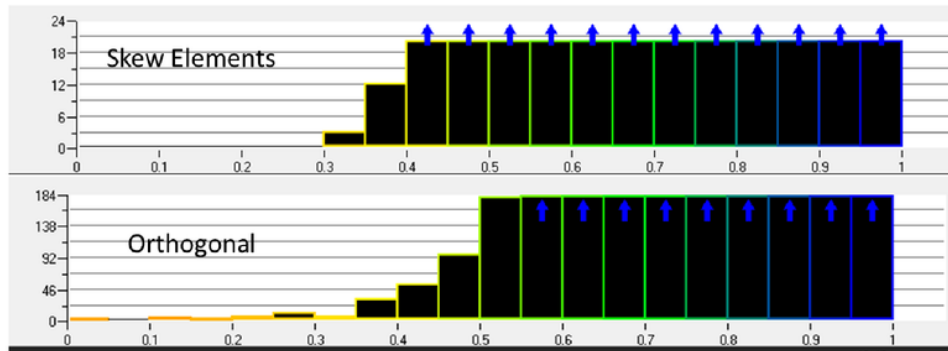
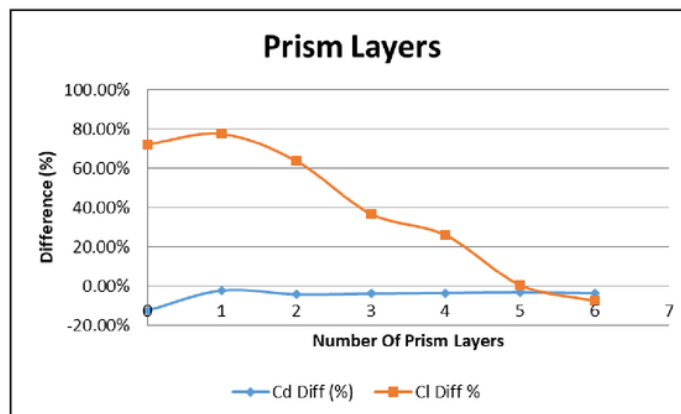


Figure 30 quality of mesh after refinement and wheel blocks added

4.7 Prism Layers Study Results

Section 2.2.3 discussed the reason prism layers are important for the boundary layer in CFD. This portion of the study looks at finding the best number of layers.



Graph 8 prism layers results

The study found that 5 layers was best (see graph 8) which matches up with Landrit's reconditions. Doing a case comparison between 5 and 0 layers, results showed the flow interaction with the car and floor changes. The red circle in figure 31 highlights the fluid floor shear interaction is not being captured. Examining the rear slant profile, the separation point is completely changing between the two. The results show the Cd has minimal change as the number of layers increase. However the Cl changes hugely. Considering the underbody and separation points are the big changes between the two but only the Cl is changing, it backs up what was discussed earlier in this report that capturing the separation point is vital to flow over the notch car. But there is still the question of why the 5 layers and 1.2 growth rate do get the best results.

To understand the prism layers settings, the turbulence model needs to be considered at this point. Up to now the K-epsilon Realizable with NEWF has been used with the benefit of predicting flow away from walls and reasonably accurate for a wide range of applications. It uses near wall

treatments (like NEWF) to improve K-epsilons ability of calculating boundary layer developing along walls. Back in section 2.2.3 it was established the boundary layer has 3 main levels to it; sublayer + buffer layer, log-layer, and outer layer (see figure 5). The turbulence near the wall is governed by the set of the boundary layer levels. By using a y^+ and u^+ (u^+ being a dimensionless form of velocity) graph to map out a flow profile, it shows there are two sections to consider mathematically. (ANSYS, 2012) (Chitale, 2014)

1. Below a y^+ value of 1 (see figure 32) the profile is linear and needs a fine mesh and low-Reynolds number turbulence model to actually resolve. The linear part is the viscous sublayer. (ANSYS, 2012)
2. From scaling the variables near the wall, the profile data takes on a logarithmic predictable form. The logarithmic part of the profile consists of the other parts of the boundary layer. Wall functions predict the buffer layer, log layer and outer layer rather than resolving them. For the wall function to work the first cell must be between a y^+ of $30 < 300$. (ANSYS, 2012)

The NEWF is the logarithmic predation approach. Wall functions don't need the a fine mesh at the wall for them to work, and having a wide y^+ range means it is easy to get a reasonable mesh height for accurate results. In hindsight the first cell height should have been calculated for all of the studies for the required y^+ value. If not specified the first cell height in the software, then it is controlled by the number of layers and growth rate settings. Considering all the factors the 5 layers work in theory because it is the minimum number of layers to achieve the required y^+ value as it is controlled by the growth rate. Past 6 layers the y^+ could go below a value of 30 making the wall functions ineffective. If the growth rate was changed then the first cell height would change, meaning past 6 layers could become effective again. (ANSYS, 2012) (Chitale, 2014)

The theory would be Lanfrit's reconditions work because they are achieving the right y^+ value for the K-epsilon Realizable with NEWF to be effective. If a different model was used then Lanfrit's settings wouldn't necessarily give the best results.

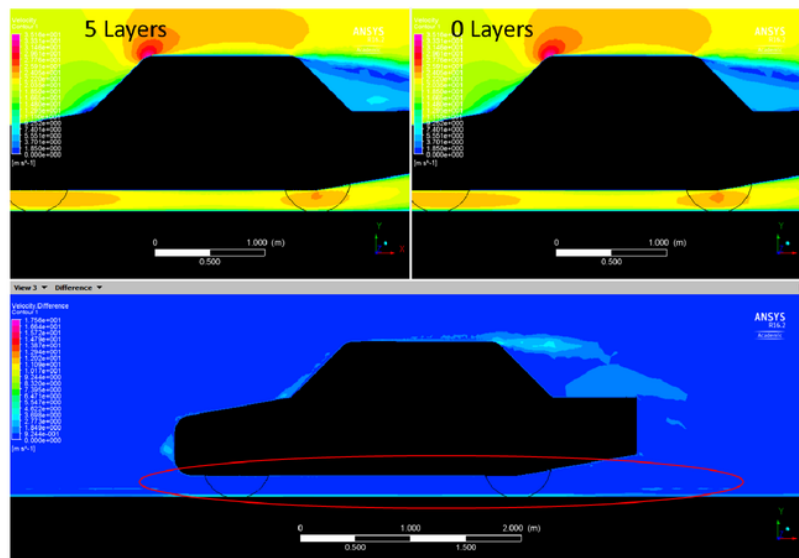


Figure 31 case comparison between the 5 layers and 0 layers velocity contours

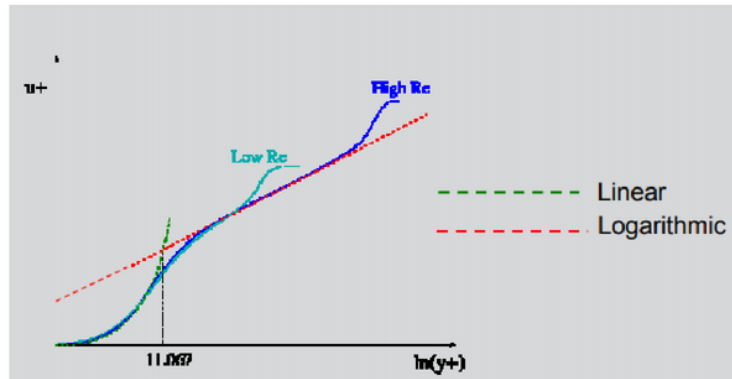


Figure 32 y^+ and u^+ flow graph (ANSYS, 2012)

4.8 Turbulence Models

At the time of the study, a number of turbulence models were tested. If there was further investigation into the turbulence models, then in hindsight only 3 would have been considered:

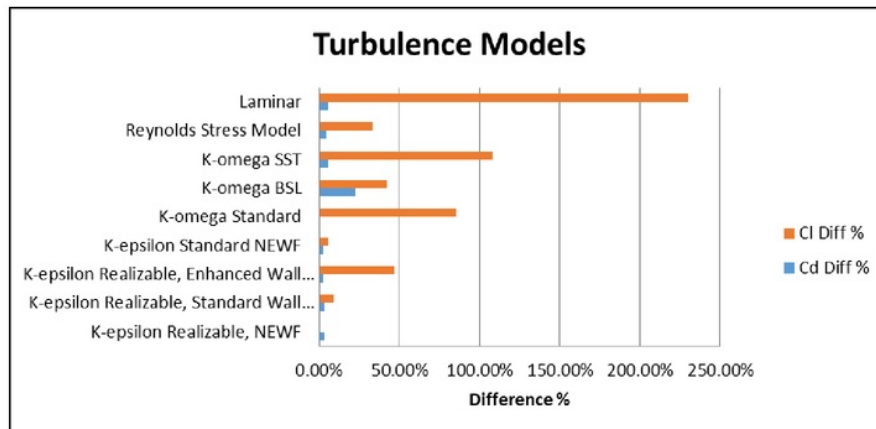
1. K-epsilon with wall function – A model for resolving flow away from walls
2. Standard K-omega – A low-Reynolds number model for resolving boundary layer flows
3. SST Model – The SST model takes the best parts of the K-omega and K-epsilon models to perform well in free stream, and resolving boundary layers without using a wall function.

Choosing the turbulence model comes down to understanding the flow scenario being tested. After examining some of the parameter studies results, the notch car C_d is found through the flow separation point and the C_l has two contributing flow factors.

1. Pressure gradients areas on complex geometry away from the walls
2. The rear windscreen slant flow separation point contributes to the lift of the car.

With there being two main factors to resolving the C_l , the SST model should be the best because it can resolve all of the boundary layer to find the correct separation point and still use the best parts from the K-epsilon model.

The results from testing the different turbulence models can be seen in graph 9. The SST model is far from reaching the correct C_l and C_d . However this is due to the prism layers not achieving a y^+ value of 1, when resolving the boundary layer a finer mesh is needed, and all of the settings up to now have been tailored to optimising the K-epsilon model. The y^+ value being wrong is highlighted by conducting a case comparison between the K-epsilon and SST model. The comparison shows how different the velocity is near the surface and the point of separation occurs (see figure 33). (Chitale, 2014)



Graph 9 turbulence model results

Another sign of the settings being tailored to the K-epsilon model is all the different variables are close to the CI and Cd answer. Apart from Enhanced wall function, because this requires a fine mesh near the wall, K-epsilon with Enhanced wall treatment might be a good alternative option to SST because it resolves the viscous sublayer better than other wall treatments. (Chitale, 2014)

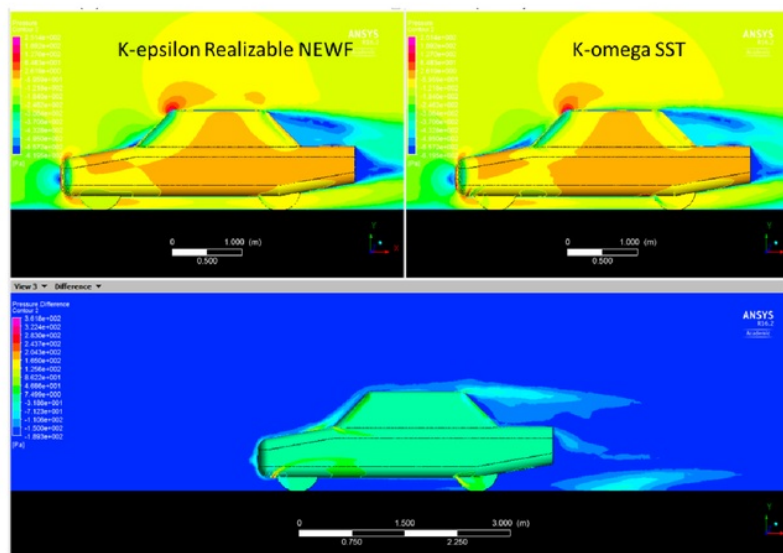


Figure 33 case comparison between the K-epsilon and SST model

The K-epsilon Realizable with NEWF was taken forward.

The meshing approach matches up with other documented approaches, but the y^+ method has been neglected. If more time was allocated on finding the best density boxes, then it is likely the results could get closer to Gayland/Baxendal and Kumar/Vaidya findings.

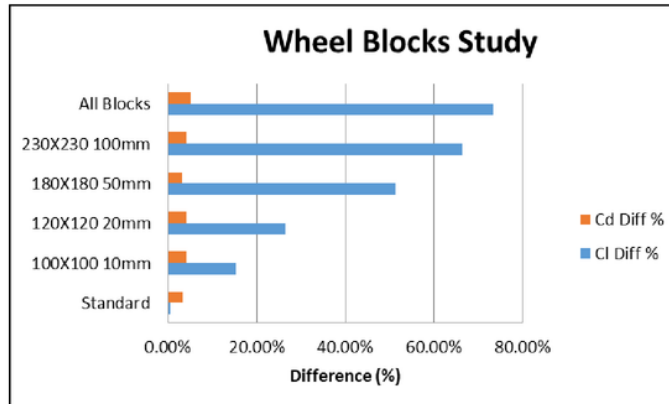
Using symmetry conisations for the domain sides works only if the K-epsilon Realizable with NEWF is used. If the SST was used, the results would be considerably at variance because the boundary layer would not be allowed to develop along the domain walls.

Overall, the K-epsilon Realizable with NEWF does work well for finding the Cd and Cl However, this is mainly due to the settings being tailored to fit. If the SST model was used with the correct mesh and prism layer y^+ value, the results could be very close both in Cl and Cd. In saying that though both Gayland/Baxendal and Kumar/Vaidya got very close with their studies using K-epsilon Realizable with NEWF, so it is possible.

Lanfirt's paper is very good, but it needs to be used with caution because as this paper has found, the turbulence model should be carefully considered with every case.

4.9 Wheel Blocks Study Results

As reported in section 4.5 and 4.6.4, wheel blocks can reduce the number of bad elements by a considerable amount. This part of the study was to find out, if by increasing the wheel block size, will also improve the element quality further without changing the flow around the wheels. For the study, the size of wheel block was controlled by increasing the height and the block 2D dimensions (length and width) in steps, going from the normal model wheels (standard) to the wheel being replaced with blocks all together. Graph 10 below are the results.



Graph 10 wheel block study results

The results found, as the wheel blocks grew bigger, changed the flow in front of the car which in turn changed the Cl value calculated. The Cl changed due to two factors:

1. The flat wheel block changing the high pressure static build up at the front of the car which altered the flow entering underneath the car (see figure 34).
2. The wake region behind the front wheel is reducing. The wake region acted as a method of sealing the sides of the notch car by creating vortices to stop pressure bleed (see figure).

The element quality did improve but does not outweigh the cost of changing the flow characteristics.

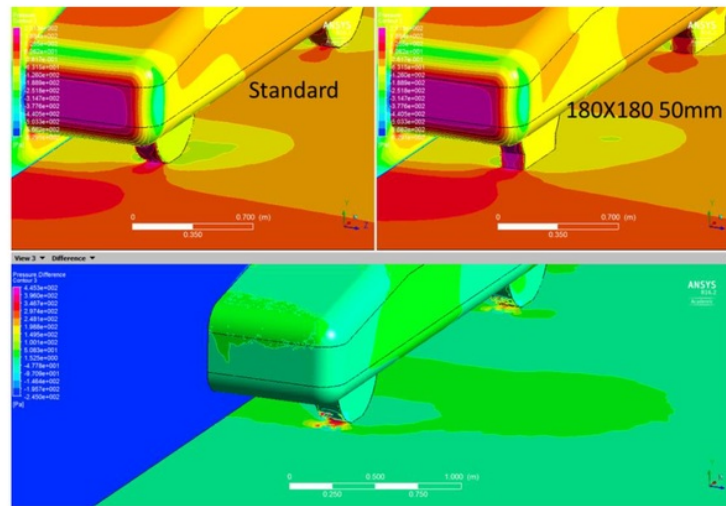


Figure 34 case comparison between standard and 180X180 50mm wheel blocks with pressure contours

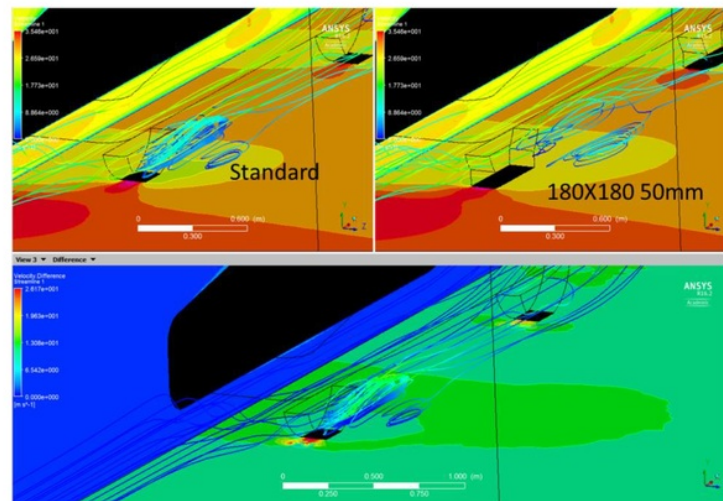


Figure 35 case comparison between standard and 180X180 50mm wheel blocks with velocity flow streams

4.10 Polyhedra Mesh Run

In ANSYS Fluent there is a feature for changing the tetrahedron elements to polyhedral (see figure 36 for visual representation) so as to reduce resolving time by lowering the amount of cells within the domain. The polyhedral elements might not always work if there are skewed cells being changed in complex flow regions. (Sharfnet, 2013)

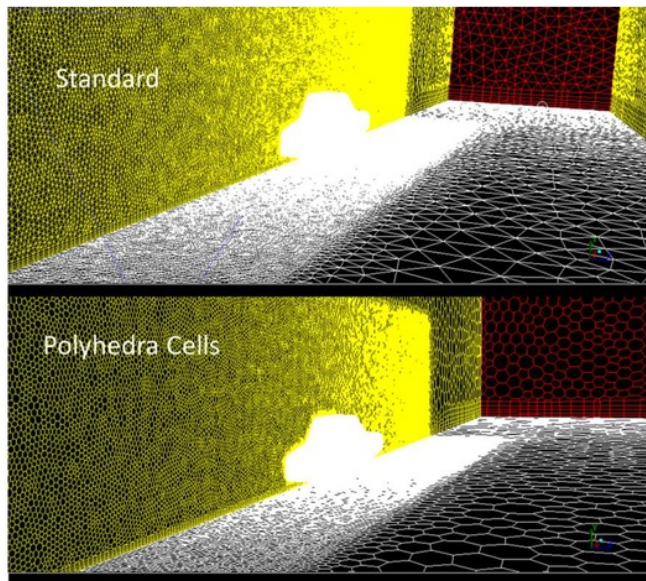
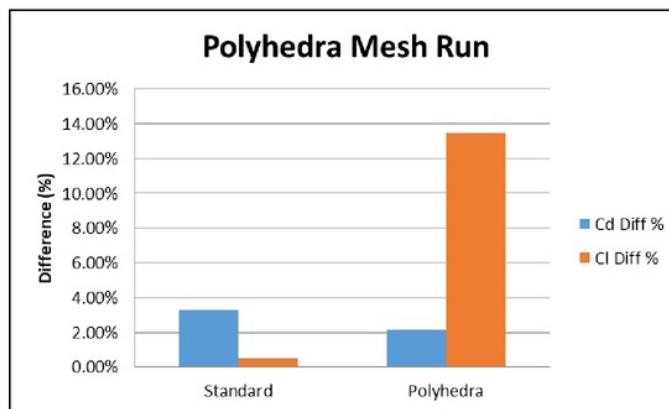


Figure 36 visual representation of tetrahedron (standard) and polyhedral elements



Graph 11 polyhedra mesh run results

The polyhedral elements do not work for this case (see graph 11). However, if the mesh quality was improved in complex flow regions to have no skewed cells, then in theory results would be improved. (Sharcnnet, 2013)

4.11 MIRA Wind Tunnel Domain Run

Adopting the settings established from the best run up to this point and applying them to a model with the domain size of MIRA wind tunnel (see figure 37). Graph 12 is the results from that run.

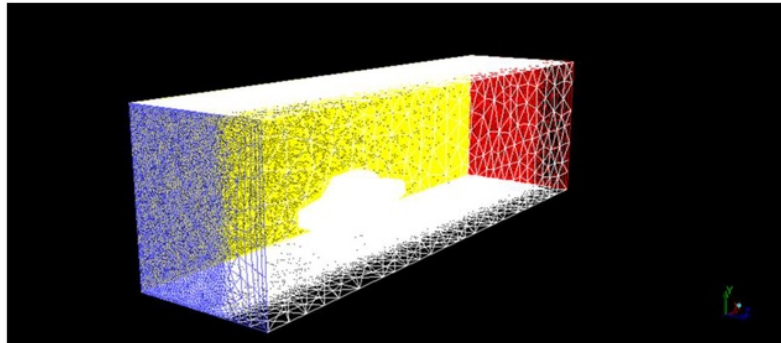
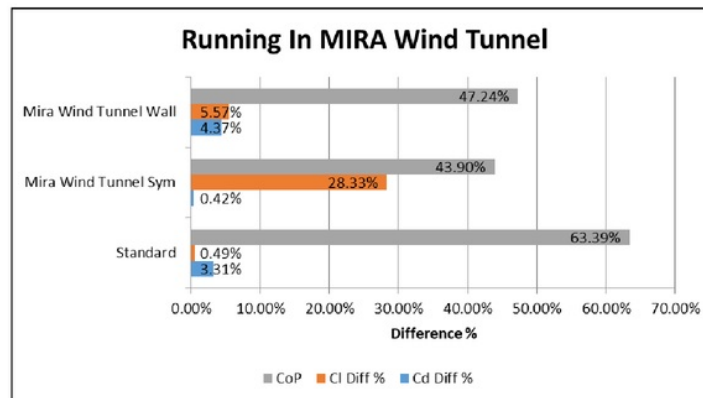


Figure 37 illustration of the MIRA wind tunnel domain size run



Graph 12 MIRA wind tunnel run results

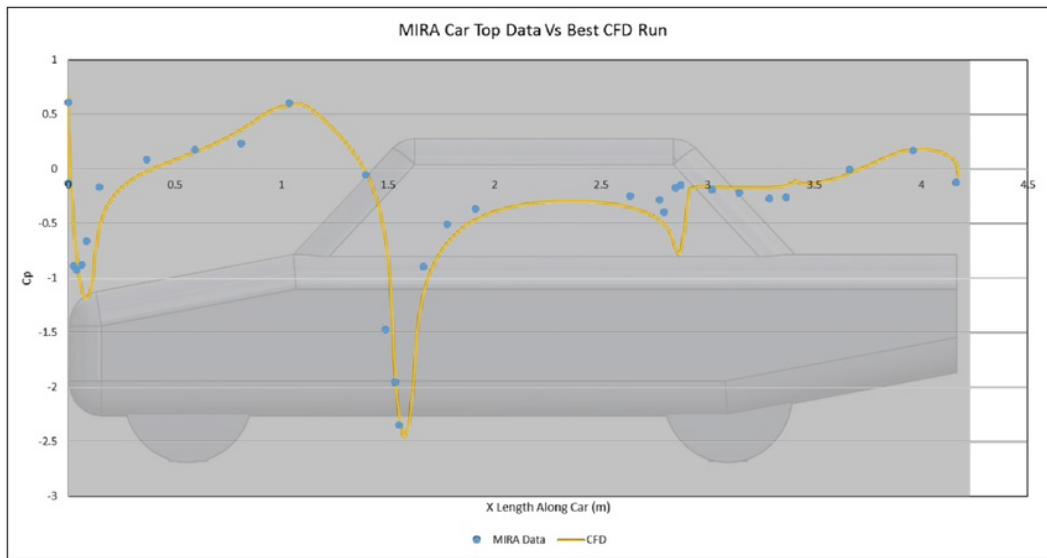
The first run used the wind tunnel size with the walls modelled as symmetry conditions. As graph 12 shows, the Cd was close but the CI far from target (28.33). The problem was with the settings being developed around a different domain size at the beginning and the symmetry walls not representing the blockage effect well enough.

To try and improve the results, use Gayland/Baxendal and Kumar/Vaidya method of putting prism layers on the walls and treat them as 'no' slip walls. This improved the CI to 5.57 (see graph 12). However, if taking the CoP calculated from the best run from the open domain series, there is a big difference between the two CoP points (see graph 12 standard being 63.39% and 47.24). Even though the Cd and CI are close between the two, the difference CoP is indicating unidentified anomaly. Further investigation is need. (Gaylard & Baxendale, 1998) (Kumar & Vaidya, 2016)

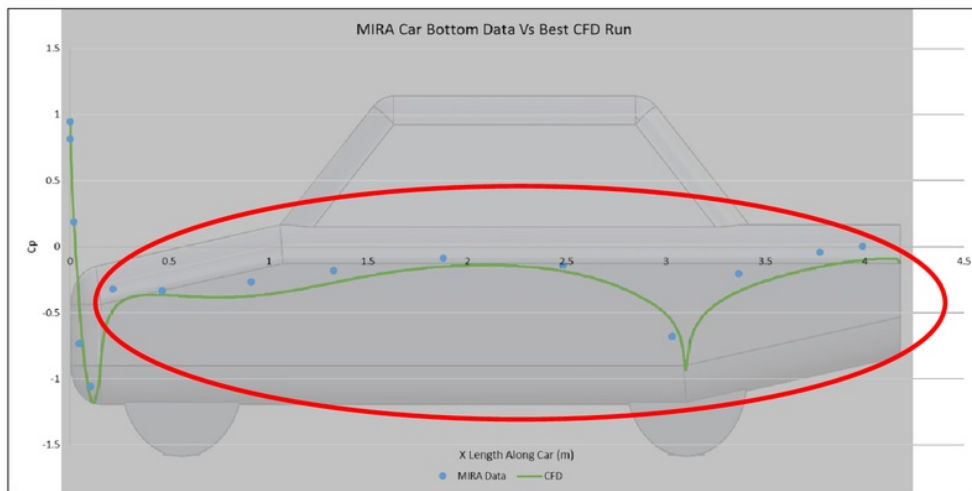
4.12 Final Study Runs And Setting Conclusions

The parameter studies has provided two sets of modelling criteria. In section 4.11, they are close with Cd and CI but have different CoP points. Another way of seeing how the results compare to the wind tunnel testing data is to over lay the surface pressure tap data (coefficient of pressure points Cp) from the CFD findings and the test.

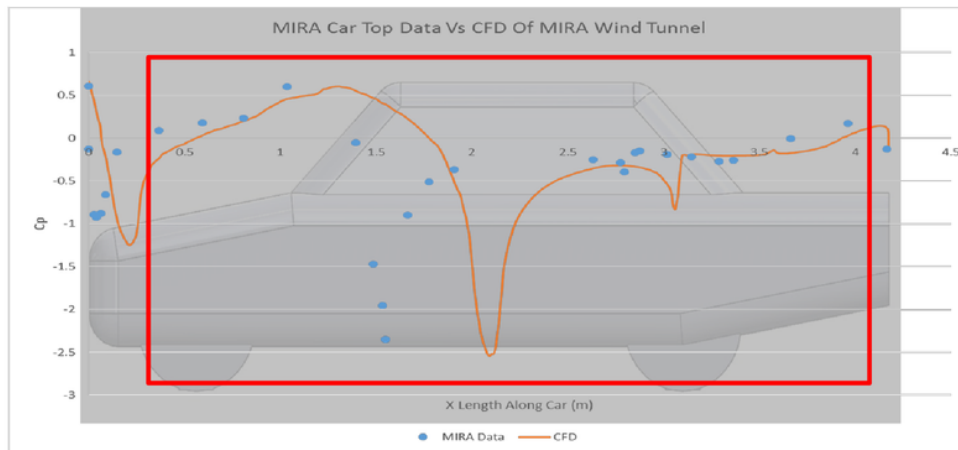
Graph 13 is the data compared for the notch car top surface. There is fairly good correlation between the top sets (see how the line is following the pots well). However, for the bottom surface, correlation not as good (see graph 14 red circle).



Graph 13 car surface top pressure tap data from the best CFD run over lay with the wind tunnel data

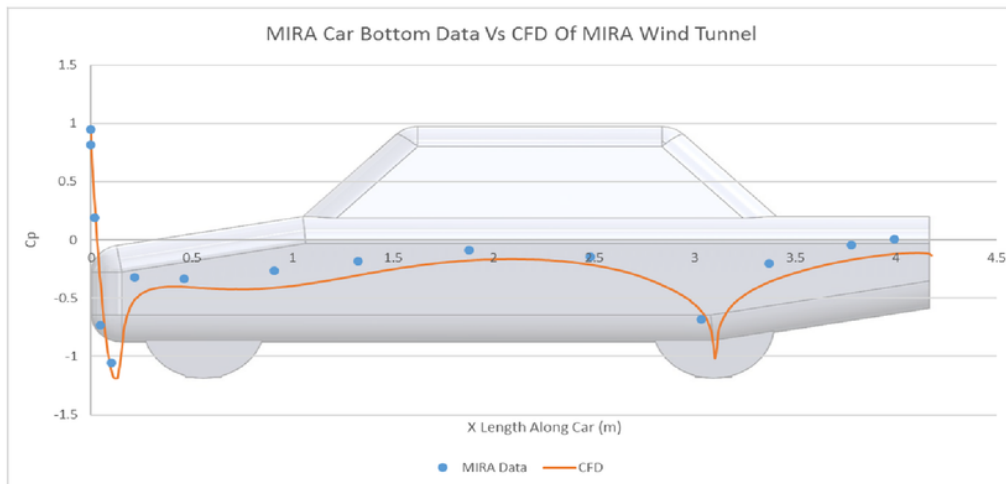


Graph 14 car surface bottom pressure tap data from the best CFD run over lay with the wind tunnel data



Graph 15 car surface top pressure tap data from the best CFD with domain as MIRA wind tunnel run over lay with the wind tunnel data

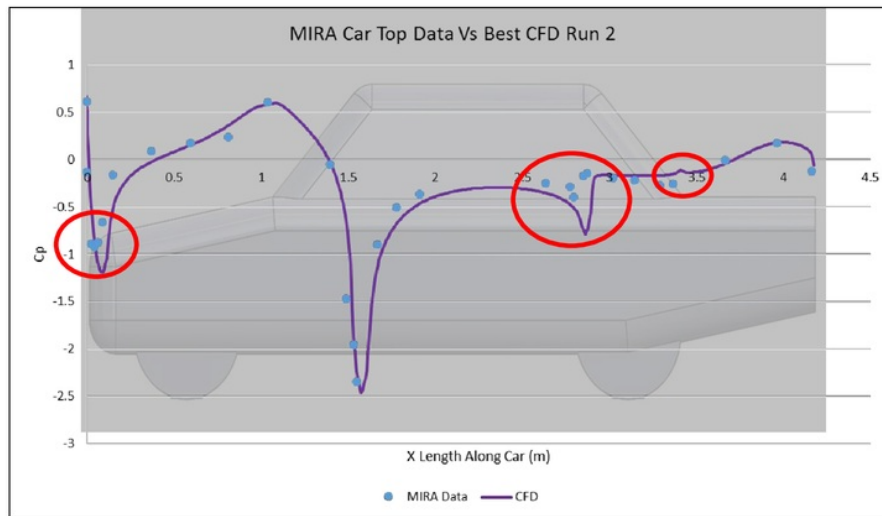
The MIRA wind tunnel domain best run saw the top surface pressure taps be very different in the mid-section of the car (see graph 15 red box). The bottom is very similar to graph 14 (see graph 16). This leads to the possibility that the top is seeing a blockage effect from the domain top wall influencing the flow over the top. This could mean the settings do not capture the boundary layer development adequately (hence why the CoP was so different in section 4.11). The settings are working for the bottom surface due to the other model being tailored with the floor being 'no' slip wall. This means for the wind tunnel domain size to work, it needs to be used from the beginning (to allow for the settings to be tailored) and the right y^+ with turbulence model to be used (to capture the boundary layer).



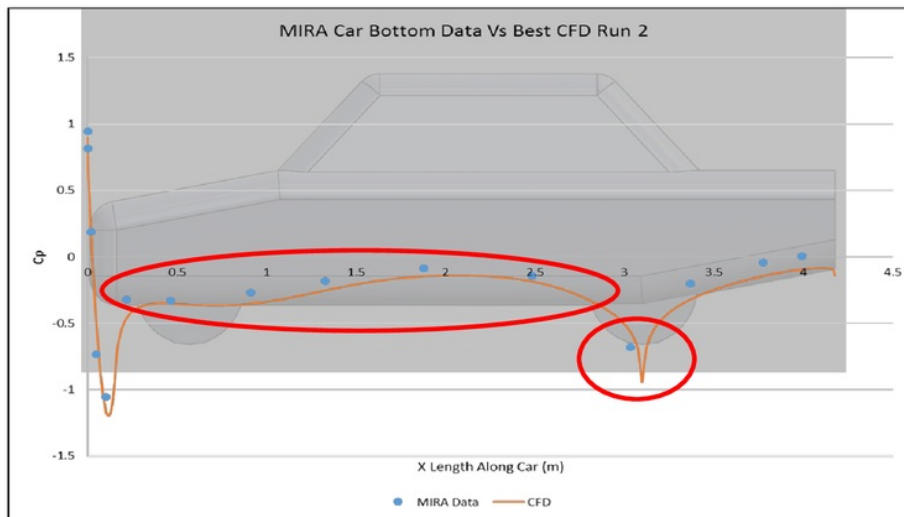
Graph 16 car surface bottom pressure tap data from the best CFD with domain as MIRA wind tunnel run over lay with the wind tunnel data

To try and improve the bottom surface pressure data with the open domain, the CFD run repeated but with an additional mesh density box on the underside of the car with smaller elements.

The new density box gave a slight improvement (see graph 17 and 18).



Graph 17 car surface top pressure tap data from the best CFD run with new mesh density box over lay with the wind tunnel data

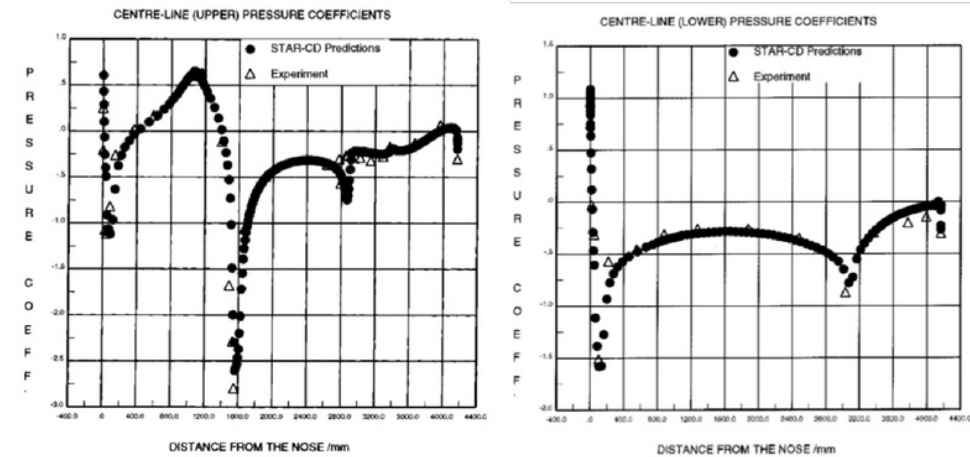


Graph 18 car surface top pressure tap data from the best CFD run with new mesh density box over lay with the wind tunnel data

Comparing these results to Gayland and Baxendal CFD findings, the new run with the underbody mesh density box car was very similar (compare graphs 19 to 17 and 18). The CFD settings with K-epsilon Realizable with NEWF worked well. However, there are still some differences between the wind tunnel data. For example, the underneath taps are not following the path exactly, and on the top surface there are some spikes in the data (see graphs 17 and 18 red circles). Conjecture as to why may be as follows;

- The spikes could be that there was not a pressure tap on the car surface capturing the point of the spike.
- In MIRA's wind tunnel, there is a boundary layer step on the floor which is not modelled in the CFD domain floor.

- The turbulence model needs either changing or improving the mesh with the y^+ .
- If there was a few pressure taps missing on the notch car surface causing to miss the spikes, then this in could change the flow further forward (or to the rear) on the car question if the CFD results car actually more accurate.
- Try refining the mesh to be a Delaunay mesh instead of an Octree.



Graph 19 Gayland and Baxendal CFD findings (Gaylard & Baxendale, 1998)

An alternative check is seeing how Kumar and Vaidya wake region compare. As figure 38 shows, there are still differences mostly starting from the flow separation points. This indicates the boundary layer isn't being captured and the turbulence model and mesh needs refining: also the continuity level when the best run converged is above ANSYS recommendation of being below $1E-04$ (see figure 39 red circle). This questions the validity of the results and points again towards the turbulence model and mesh needs further work. (ANSYS, 2012)

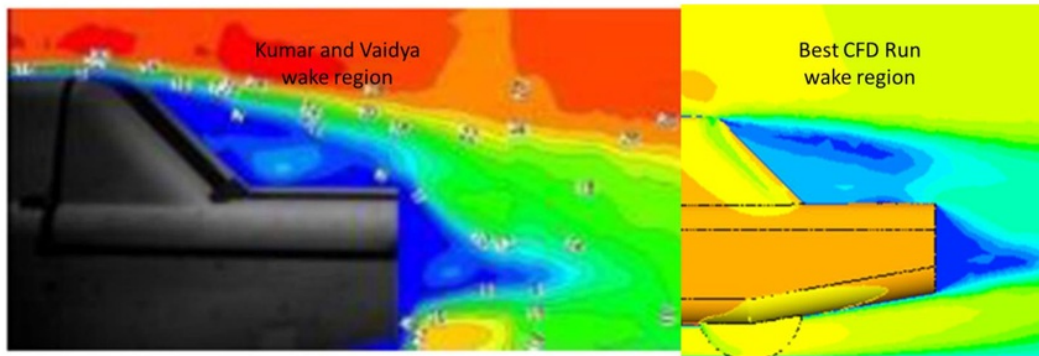


Figure 38 comparing the wake region between best CFD run and Kumar and Vaidya findings

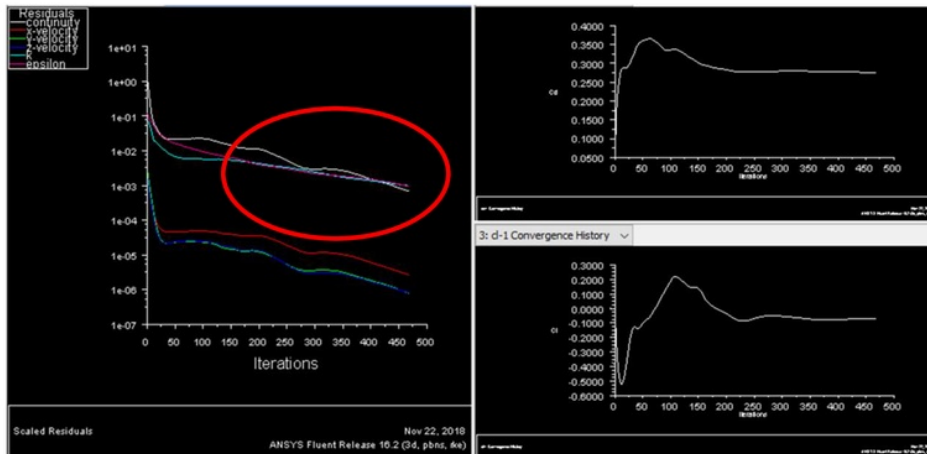


Figure 39 best run continuity level data

Overall, the final settings found from the initial set up gave a close correlation to the wind tunnel data. Improvements would be feasible by either getting the SST model to work with the right y^+ and the domain size to be the wind tunnel size at inception.

The best setting found was used for testing an improved performance model of the notch reference car in section 5. Settings found in table 5.

Table 5 final CFD settings

Domain Size		Mesh Settings	
3 car Lengths forward		Global Size:	1m
5 car lengths behind		Mehser:	Octree
2 car lengths up		Number of Prism:	5
2 car lengths to side		Height Ratio:	1.2
Notch car added features		Car Element Size:	0.01m
Wheels just in contact with floor		Mesh Density Box Size: 2 car lengths forward, 2 behind, 0.5 up and to the side	
Small 5mm wheel blocks			
		Box Element Size:	0.1m
Fluent Settings			
Slover:	Pressure-Based	Time:	Steady
Turbulence Model:	K-epsilon Realizable With NEWF	Scheme:	Coupled
Boundary Conditions			
Inlet:		Velocity-Inlet	
Outlet:		Pressure-Outlet	
Symmetry Side:		Symmetry Wall	
Top:		Symmetry Wall	
Side:		Symmetry Wall	
Floor:		Wall	

5. IMPROVED NOTCH CAR PERFORMANCE MODEL

The results of the normal notch car indicate there are some aerodynamic adjustments that can be made to improve the downforce performance of the geometry. They are:

1. Take advantage of the high pressure build up on the front face see (see figure 40 red high pressure point). Achieved by inputting a splitter to reduce the high static pressure bleeding to the underside and crate a Venturi. (see figure 41 red circle). (McBeath, 2006)

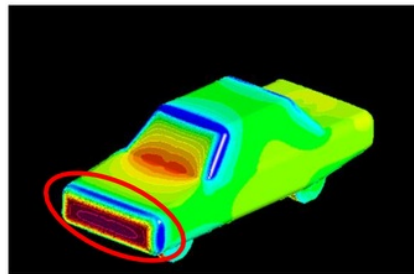


Figure 40 notch car pressure contour

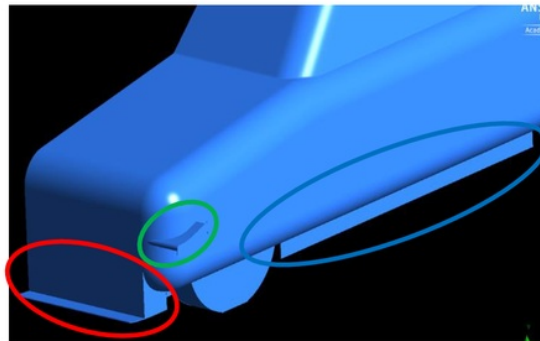


Figure 41 notch car devises

2. The low pressure on the front bumper sides indicates the high pressure at the front is bleeding causing an increase in flow rate (see figure 42 red circle). Option: a small wing-lit to take advantage of the fast moving air (see figure 41 green circle).

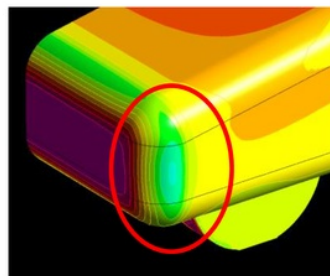


Figure 42 notch car low pressure on bumper side

3. Stop any high pressure bleeding into the low pressure region on the car underside (see figure 43 flow streams bleeding under car) by fixing side skirts (see figure 41 blue circle). (McBeath, 2006)

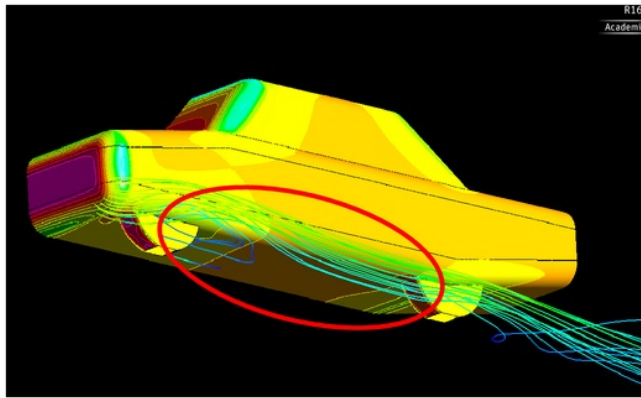


Figure 43 notch car flow streams bleeding under car

4. Place a diffuser at the rear to attempt and encourage the flow under car to further create a lower pressure region (see figure 44 red circle). (McBeath, 2006)



Figure 44 notch car with diffuser

5. There is a large amount of flow at the rear (see figure 45) which by placing a multi element wing in free stream is a good devise to take advantage of the clean air (see figure 46).

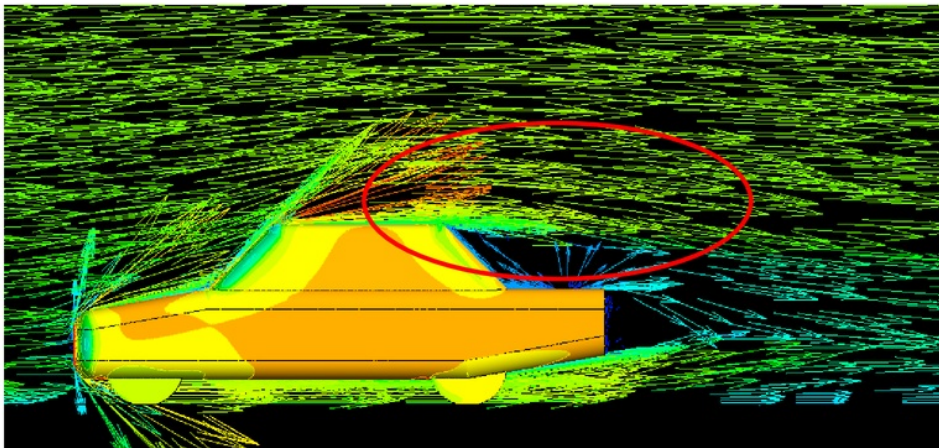


Figure 45 notch car with vertex showing clean air

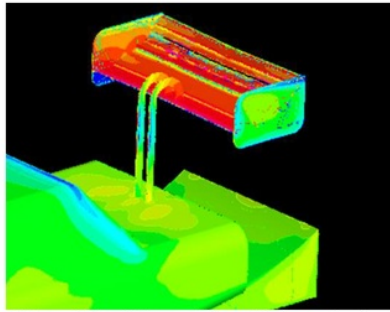
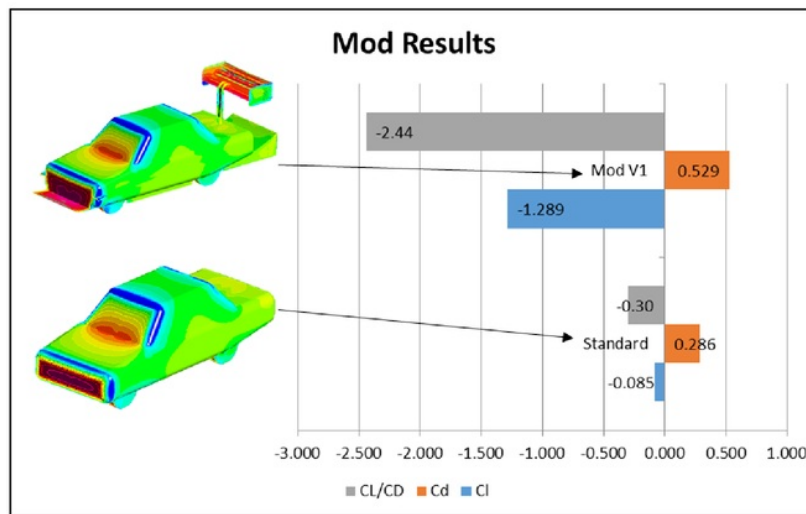


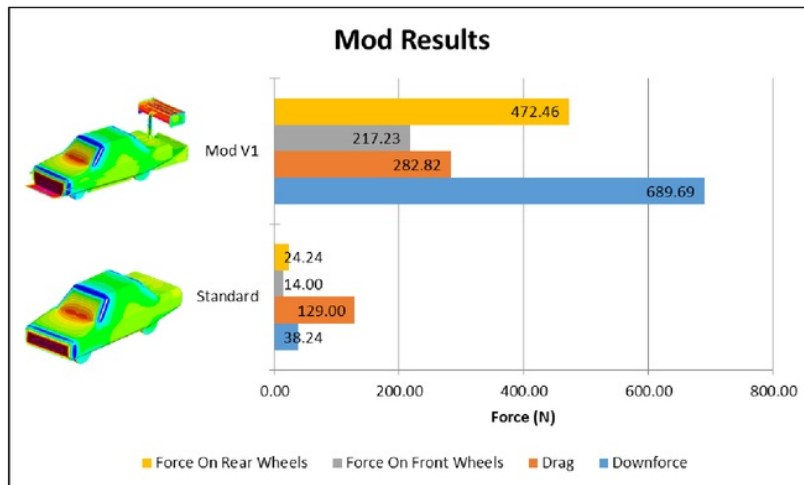
Figure 46 notch car with multi element wing

5.1 CFD Results

Graphs 20 and 21 are the CFD results of the new notch compared to the original. The upgraded notch car creates 15 times more downforce and only 1.8 times more drag, meaning it has an improved aerodynamic efficiency of -2.44 compared to -0.3.



Graph 20 notch car performance coefficient results



Graph 21 notch car performance force results

The ratio of force between the front and rear wheels hasn't change significantly (see graph 21) meaning it has a similar CoP point.

Figure 47 is a case comparison between the two. It shows the pressure contour on the car and the velocity on the symmetry wall.

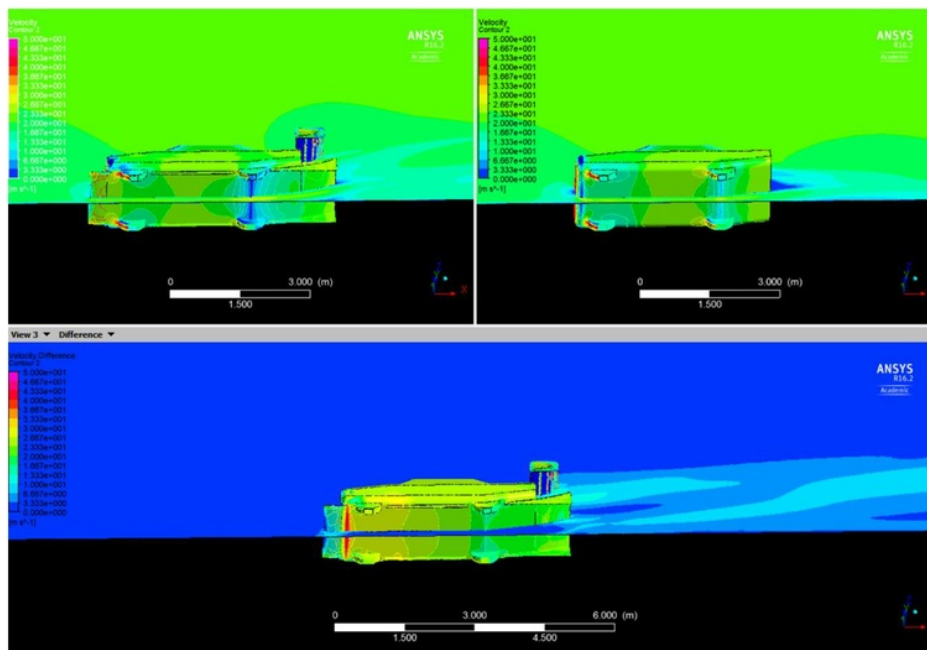


Figure 47 case comparison between the two cars show the pressure contour on the car and the velocity on the symmetry wall

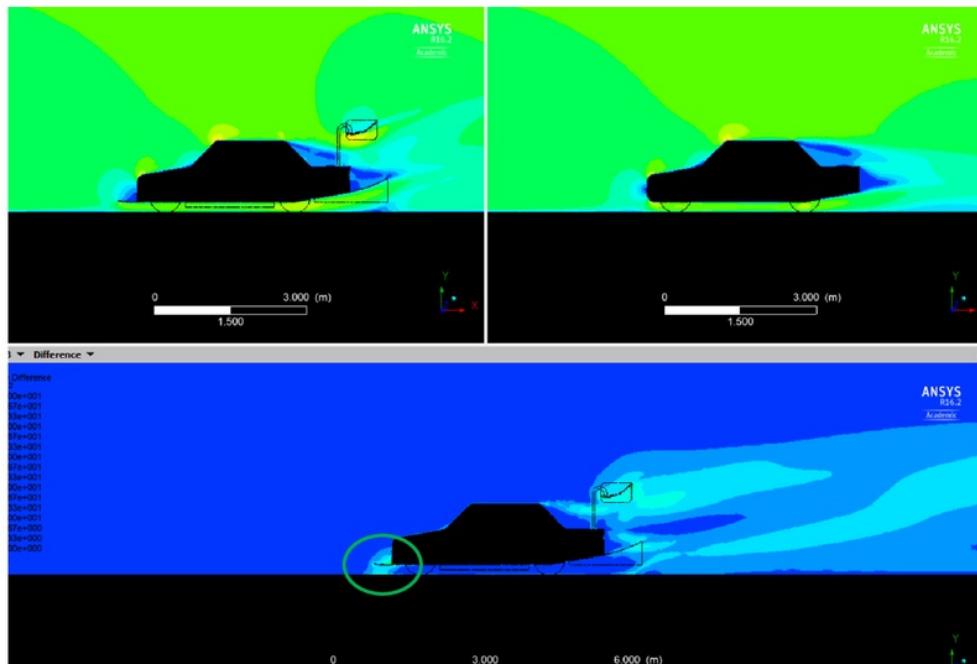


Figure 48 case comparison between the two cars show the velocity on the symmetry wall

Figure 48 shows how the air is traveling faster in the splitter region (see green circle), but once past the splitter the speed reverts to the same. This could mean the rear diffuser isn't having much of an effect. The rear wing is generating a waste wake region in comparison to the original which is contributing to the extra drag. The point of separation is also changing due to the interaction with the rear wing

Figure 49 is showing how the front splitter is effective reducing the flow entering under the car with high pressure accumulating on top (red circle red pressure contour). On the top surface, there is very similar pressure contours.

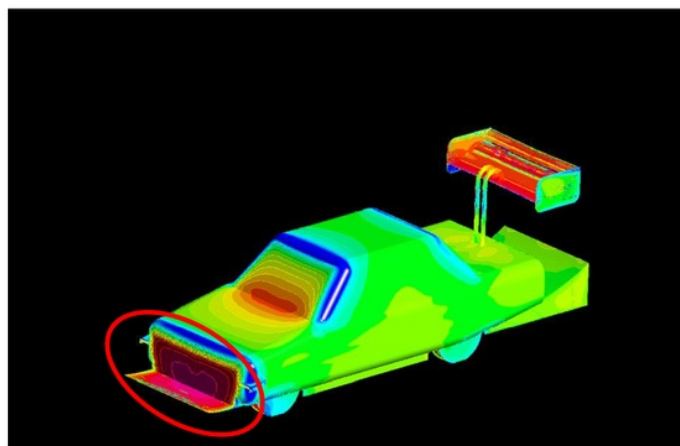


Figure 49 new notch car with pressure contour

A further indication the splitter is working well is the vectors curtailed at the front face (see figure 50 red circle). The rear is working efficiently as the vectors are staying attached up the wing (see green circle). The diffusers flow is also staying attached.

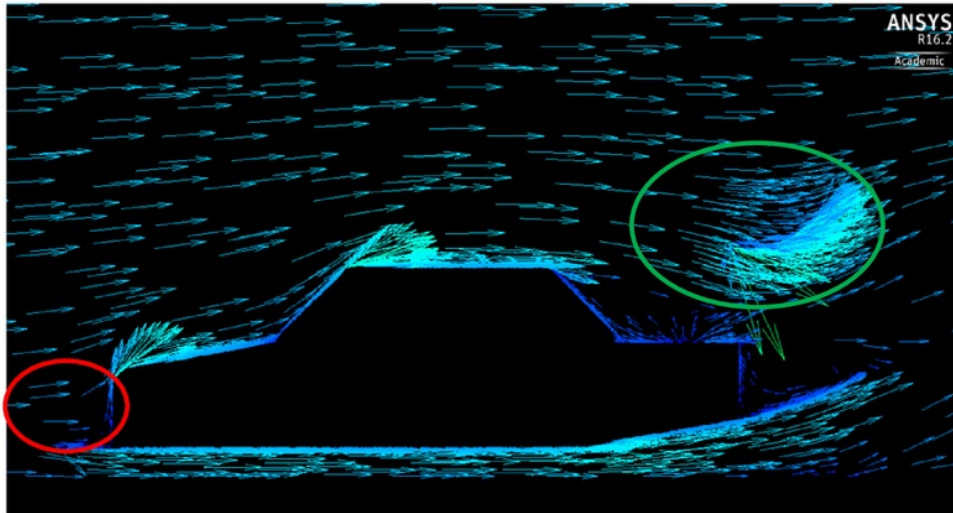


Figure 50 new notch car with vectors on symmetry wall

The flow on the rear wing is remaining attached indicated by the white turbulence created across the wing element surfaces (see figure 51 red circle). Notice how the end-plate is creating vortices at the trailing top and bottom edges with the turbulence streaming off.

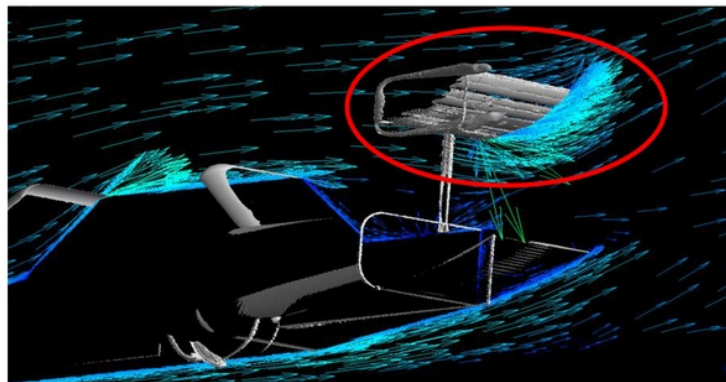


Figure 51 new notch car rear wing

The flow over the car itself is clean (see figure 52 free streams over car). The rear wing and diffuser are interacting with the upward flow from the wing driving the diffuser flow (see red circle).

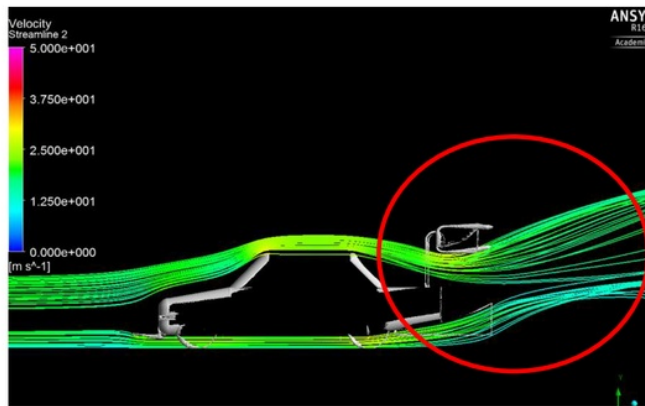


Figure 52 new notch car with free streams over car

To CFD the notch car with the added aerodynamic devises, an extra mesh density box was added to capture the flow through the rear wing and to pick up any interaction with the rear slant profile.

The results of the new notch car are questionable because of the turbulence model being used. As found earlier in this report, the separation point is key to finding the right C_l and as seen in figure 48 the slant point is interacting with the rear wing. This could mean the flow going towards the rear wing could be wrong and the wing is working better than it actually indicates. This could have a knock on effect with diffuser performance ultimately changing the flow at the rear completely.

6. CONCLUSIONS

The report has established a CFD process when considering new flow tasks. The top level settings for domain sizing, mesh and turbulence models have been explored. A large amount of data has been gathered and multiple settings tried. There is potentially a lot more scenarios to explore.

The results from the parameters studies can be challenged considering the later findings of how important the y^+ method is when modelling the geometry in CFD. In hindsight a test run should have been done at the beginning to establish the y^+ of the case and then refine the CFD model to achieve the right y^+ .

Using the open domain approach dose work to some extent when the settings and meshing are tailored, but the study should have started with sizing of the MIRA wind tunnel.

Reviewing all the turbulence models, the K-epsilon Realizable with NEWF is good when solving for flow away from walls and reasonable with near wall flows. With the separation point a key factor, moving forward would want to refine the mesh to use the SST model to further improve the surface pressure tap data correlation.

7. FURTHER WORK

The following is recommended future work:

- Re-do the parameters studies with the domain size as the MIRA wind tunnel and refine the mesh for the y^+ value needed for the SST turbulence model.
- Try refining the mesh to be a Delaunay mesh instead of an Octree.
- Model the MIRA boundary layer step on the floor to see the impacted on the results.

8. REFERENCES

- ANSYS. (2012). *Lecture 9 Best Practice Guidelines*. ANSYS.
- ANSYS. (2012). *Lecture 3 Introduction to ANSYS Meshing*. ANSYS.
- ANSYS. (2012). *Lecture 6 Turbulence*. ANSYS.
- ANSYS. (2012). *Lecture 4 Solver Settings*. ANSYS.
- ANSYS. (2012). *Lecture 2 Boundary Conditions*. ANSYS.
- Chitale, K. C. (2014). *BOUNDARY LAYER ADAPTIVITY FOR INCOMPRESSIBLE TURBULENT FLOWS*. USA: SCOREC.
- Gaylard, A., & Baxendale, A. (1998). *The Use of CFD to Predict the Aerodynamic Characteristics of Simple Automotive Shapes*. USA: SAE International.
- Kumar, V., & Vaidya, R. (2016). *Methodology Development to Accurately Predict Aerodynamic Drag and Lift for Passenger Vehicles Using CFD*. US: SAE International.
- Lanfrit, M. (2005). *Best practice guidelines for handling Automotive External Aerodynamics with FLUENT*. Germany: Fluent Deutschland GmbH .
- McBeath, S. (2006). *Competition car aerodynamics*. California: Haynes Publishing.
- Sharcnet. (2013). *Choosing the Tetrahedral Mesh Method*. Retrieved December 24, 2018, from Sharcnet: https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/icm_user/iuser_tetra_methods.html
- Soares, R. F. (2015). *Influence of CFD Setup and Brief Analysis of Flow over a 3D Realistic Car Model*. USA: SAE.
- Tudor, T. (2018). *CFD Lecture Notes*. Swansea: UWTSD.