

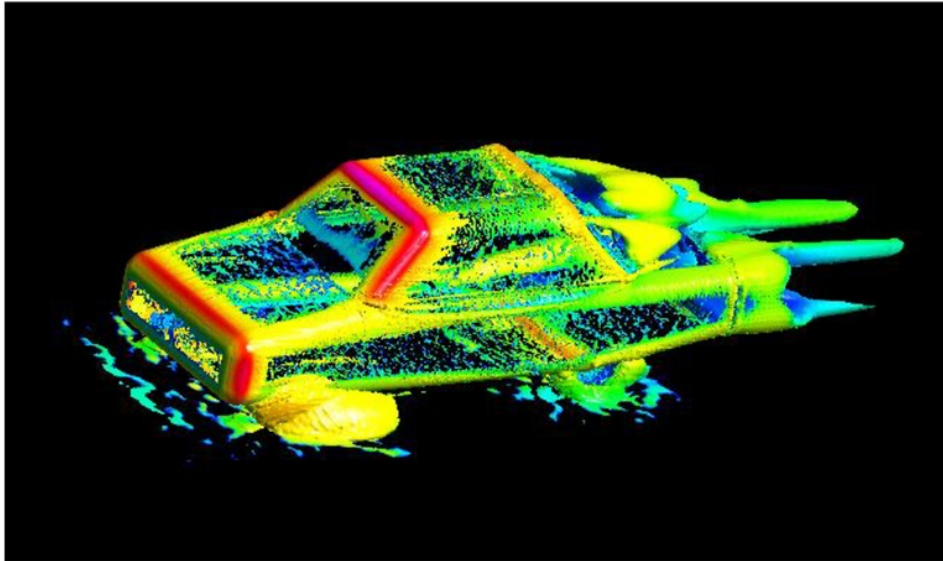
MIRA Validation Study

by John Michael Hughes

FILE	4209_JOHN_MICHAEL_HUGHES_MIRA_VALIDATION_STUDY_128450_113 4012469.DOC		
TIME SUBMITTED	13-JAN-2019 10:23PM (UTC+0000)	WORD COUNT	9929
SUBMISSION ID	98958956	CHARACTER COUNT	66307



Prifysgol Cymru
Y Drindod Dewi Sant
University of Wales
Trinity Saint David



External Computational Fluid Dynamics Simulation Validation Experiment – MIRA Wind Tunnel

Student – John Hughes (P146322)

Module – Advanced Computational Methods (External CFD)

Module Code – M6X00408

Lecturer – Tim Tudor

BEng Motorsport Engineering (Year 3)

Date – 28/10/18

Contents

List of Figures.....	3
1.0.0 Introduction	6
1.1.0 Aims.....	6
2.0.0 Methodology & Study Results.....	7
2.1.0 Initial Input Variables	7
2.2.0 Domain	8
2.2.1 Domain Sizing.....	9
2.2.2 Car Positioning within the Domain.....	10
3.0.0 Mesh.....	12
3.1.0 Global Element Seed Size.....	13
3.2.0 Density Box.....	15
3.2.1 Density Box Sizing & Positioning.....	15
3.2.2 Density Box Element Size	17
3.3.0 Surface Mesh.....	19
3.4.0 Prism Layers	21
3.5.0 Volume Meshing Method	23
3.6.0 Mesh Quality	26
4.0.0 Turbulence Models.....	28
4.1.0 What is Turbulence?	28
4.1.1 Reynolds Number	29
4.1.2 Euler & Navier-Stokes Equations	29
4.2.0 RANS Based Turbulence Models (Reynolds Averaged Navier-Stokes)	29
4.3.0 Y+ Method.....	34
4.3.0 Pressure Distribution.....	36
4.3.1 K-Epsilon, Realizable, Non-Equilibrium Wall Functions	37
4.3.2 K-Epsilon, Realizable, Enhanced Wall Treatment.....	38
4.4.0 Disputes	40
5.0.0 Analysis of the Implementation of a Rear Wing on the MIRA Car	41
5.0.0 Conclusion	43
6.0.0 References.....	44

List of Figures

Graph 1 - Domain Volume Results	9
Graph 2 - Distance to Origin	11
Graph 3 - Global Seed Size (Free Stream Elements)	13
Graph 4 - Density Box Positioning	16
Graph 5 - Density Box Element Size	17
Graph 6 - Part Element Size	19
Graph 7 - Prism Layers	21
Graph 8 - Volume Mesh Method	24
Graph 9 - Turbulence Model Study	30
Graph 10 - MIRA Car Wind Tunnel Pressure Distribution	36
Graph 11 - KE, Realizable, Non-Equilibrium Wall Functions Pressure Distribution	37
Graph 12 - K-Epsilon, Realizable, Enhanced Wall Treatment Pressure Distribution	38
Graph 13 - K-Epsilon, Enhanced Wall Treatment (Refined Mesh) Pressure Distribution	39
Graph 14 - Modified MIRA Car Pressure Distribution	42
Table 1 - Domain Size Results	9
Table 2 - Global Element Seed Size	13
Table 3 - Density Box Size	15
Table 4 - Density Box Element Size	17
Table 5 - Prism Layer Study	22
Table 6 - Skewness Values	26
Table 7 - K-Epsilon Refinement	31
Table 8 - Comparison of Maximum & Minimum CoP	37
Table 9 - Comparison of Maximum & Minimum CoP	38
Table 10 - Comparison of Maximum & Minimum CoP	40
Table 11 - Results from Standard & Modified Geometries	42
Table 12 - MIRA Car Aerodynamic Characteristics	42
Figure 1 - MIRA Car CAD Geometry	6
Figure 2 - Car Positioning within the Domain	7
Figure 3 - Baseline Mesh	8
Figure 4 - Ahmed Body with a Domain (Costa, 2015)	8

Figure 5 - Velocity Contours (Top Left = 5x, Top Right = 8x & Bottom = Difference).....	9
Figure 6 - Velocity Contours of MIRA Car (Palaskar, 2016)	10
Figure 7 - Pressure Field (Stagnation Zone).....	10
Figure 8 - 5m Frontal Distance Velocity Field.....	11
Figure 9 - 5m Frontal Distance Velocity Isosurface	12
Figure 10 - Volume & Surface Mesh	12
Figure 11 - Comparison of 0.4m (Top Left) vs 1m (Top Right) Volume Elements (Velocity Contours).....	14
Figure 12 - Velocity Vector Comparing Recirculation 0.4m vs 1m (0.4m Top – 1m Bottom)	14
Figure 13 - Density Box Positioning (1m Forward - Top Left, 2m Forward - Top Right)	16
Figure 14 - Density Box Elements (0.5m - Top Left, 0.1m Elements Top Right).....	18
Figure 15 - Density Box Elements Pressure Contours (0.5m - Top Left, 0.1m - Top Right)	18
Figure 16 - Surface Mesh.....	19
Figure 17 - Part Element Size Pressure Contours (0.04m Top Left, 0.1m – Top Right)	20
Figure 18 - Part Element Size Velocity Distribution (0.04m - Top Left, 0.1m - Top Right)	20
Figure 19 - Prism Element Lay Up	21
Figure 20 - Prism Layer Velocity Flow Field (6 Layers - Top Left, 3 Prism Layers - Top Right)	22
Figure 21 - Robust (Octree) Volume Method.....	23
Figure 22 - Quick (Delaunay) Volume Method.....	23
Figure 23 - Delaunay Surface Mesh (Left), Advancing Front Surface Mesh (Right) (www.pointwise.com, 2012)	24
Figure 24 - Robust Octree Method (Top), Quick Delaunay Method (Bottom) (Velocity Vectors).....	25
Figure 25 - Element Skewness	26
Figure 26 - Element Orthogonal Quality (www.sharcnet.ca, 2015)	26
Figure 27 – Showing Elements of Poor Orthogonal Quality (3 Prism Layers – Left) (6 Prism Layers – Right)	27
Figure 28 - Orthogonal Quality Histogram (3 Prism Layers).....	27
Figure 29 - Orthogonal Quality Histogram (6 Prism Layers).....	27
Figure 30 - Added Wheel Blocks to Geometry.....	27
Figure 31 - Turbulent Eddie Structures (Ansys Inc, 2006).....	28
Figure 32 - Boundary Layer Conditions (Frei, 2017).....	31
Figure 33 - Enhanced Wall Treatment Modification (Ansys Inc, 2006).....	32
Figure 34 - Enhanced Wall Treatment (Top) & Non-Equilibrium Wall Functions (Bottom) Vortex Core Velocity.....	32
Figure 35 - Enhanced Wall Treatment (Top) & Non-Equilibrium Wall Functions (Bottom) (Velocity Streamlines)	33
Figure 36 - MIRA Car Streamlines (Palaskar, 2016)	33
Figure 37 - (Buscariolo, et al., 2016) Found Turbulence Model Results	33

Figure 38 - Y+ Method (Leap CFD Team, 2012)	34
Figure 39 - Y+ Figures within Boundary Layer Conditions (Mali, 2016)	35
Figure 40 - Viscous Sublayer Resolving (Left) & Log Law Resolving (Right) (Jouse, 2018)	35
Figure 41 - Mesh Refinement.....	39
Figure 42 - MIRA Car Rear Wing Geometry	41
Figure 43 - Comparison (With & Without Wing) (Velocity Contours)	41

1.0.0 Introduction

The Horiba Mira wind tunnel is the only full-scale aerodynamic open loop wind tunnel in the United Kingdom. It offers a working volume of 521.4m³ with a variable air velocity control (maximum of 36m/s). Force measurements of drag, lift and side are obtainable, along with the capability to study the affects of yaw, pitch and roll. (www.horiba-mira.com, 2018)

However, with the rise of computational power, an influx of highly complex software has become available to the public. This has resulted in computational fluid dynamics becoming increasingly popular compared to that of wind tunnel testing. (Linfield, Ph.D., P.E. & Mudry, P.E., 2008).

As CFD testing uses a basis of mathematical equations to predict turbulence and flow structures with a volume body, the question most frequently asked is, "*Can you trust your results?*", as inevitably, mathematics follows a series of strict paths to arrive at an outcome. However, as air can be defined as a fluid, it can be unpredictable and may not react in the way that the software predicts it should, so wind tunnels are still commonly used to validate CFD results. (Qin, 2003).

1.1.0 Aims

The purpose of this report is to examine how accurately computational fluid dynamics can represent a real-world flow structure over a bluff body, and how settings can be adjusted to achieve a thorough representation of the potential complex fluid flow.

Data obtained from the Horiba Mira wind tunnel describes the static pressure field over the upper and lower surface of their test bed vehicle (The Mira Car - Fig.1) along with C_l and C_d figures and is often used to validate CFD phenomenon.

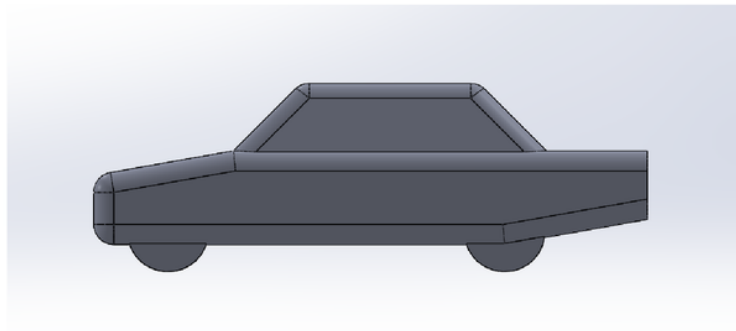


Figure 1 - MIRA Car CAD Geometry

Once a validation study is complete, the basic study of an aerodynamic device transposed onto the geometry shall be conducted and a discussion of its effects on the aerodynamic characteristics of the car itself shall take place.

2.0.0 Methodology & Study Results

Throughout the process of developing a CFD case, certain criteria need to be considered. These criteria include:

- Fluid Domain
- Mesh Construction
- Turbulence Script

Each of these areas may have a significant impact on the results produced, it is therefore imperative that each result is fully understood in order to facilitate the accurate use of the software.

For the purposes of this research, Ansys simulation software has been employed and a number of test simulation sequences have been run in order to provide the results that are needed to describe how changing variable inputs can affect the outcome results.

At the start of the procedure, the Ansys settings were adjusted to correspond with the overall C_I and C_d figures provided by Mira. However, the pressure distributions were not considered until the resulting C_I and C_d figures became representative, as it was believed that once the coefficient values become more representative, the pressure distributions will do likewise. The coefficients to be achieved are as follows.

- $C_I = -0.0845$
- $C_d = 0.2963$

2.1.0 Initial Input Variables

As the purpose of this experiment is to obtain the optimum settings by which a CFD case can be made comparable to that of a wind tunnel experiment, a baseline for the settings had to be determined in order to afford a common reference point from which subsequent measurements could be taken.

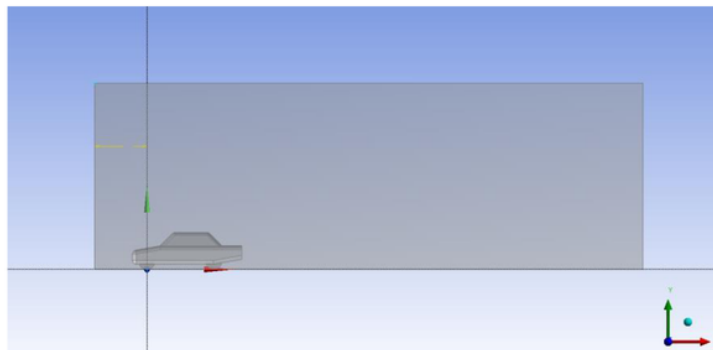


Figure 2 - Car Positioning within the Domain

As *Figure 2* indicates, the geometry, of the MIRA car must have a set positioning within its set domain. In this instance, the origin positioned at the midpoint of the front wheel was set to a distance of 2m to the rear of the velocity inlet in the X direction.

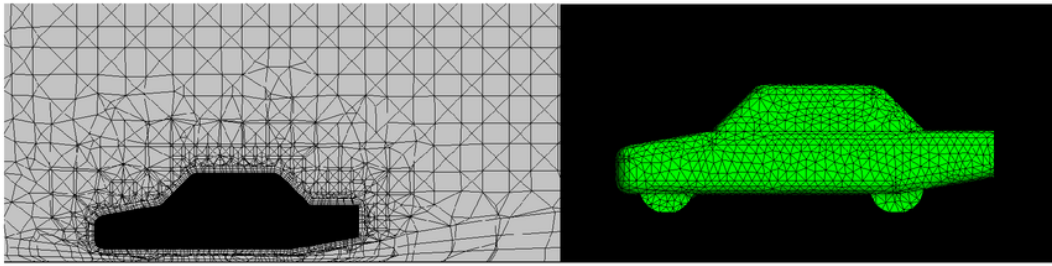


Figure 3 - Baseline Mesh

A maximum global element size of 1m was selected to represent the free stream mesh, and 0.1m elements were selected to represent the part (MIRA car) mesh. As a default, the ICEM mesher utilises 3 prism layers to capture the 'close to wall flow', therefore 3 prism layers were used over the surfaces of the car and road. A Robust (Octree) volume method was also applied to the initial cases.

K-Epsilon turbulence models are the most commonly used for fluid flow simulation over a bluff body such as a car and are described as being "*Robust and reasonably accurate*" (Ansys Inc, 2006). As a turbulence model is a base of algorithmic coding to predict turbulent flows, a number of subsets are available within the main body of the K-Epsilon model. A realizable K-Epsilon model is suggested, (Llanfrit, 2005), due to its superior nature for capturing rotational turbulence and areas of adverse pressure gradients compared to that of the Standard K-Epsilon model. A Non-Equilibrium Wall Function method was also chosen as it satisfies mathematical ideals to predict close to wall behaviours and attempts to improve predicted flows of higher-pressure gradients and greater areas of separation.

All models were run at 20m/s.

2.2.0 Domain

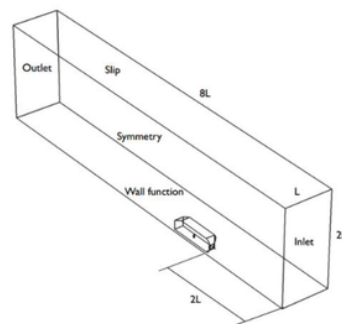


Figure 4 - Ahmed Body with a Domain (Costa, 2015)

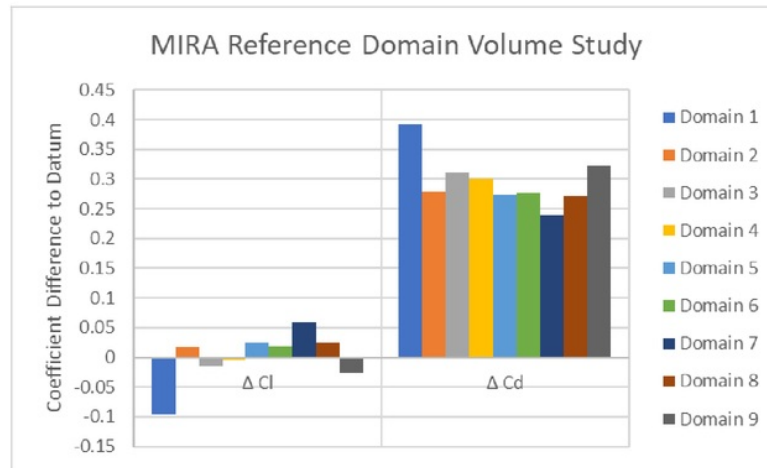
A domain is the interpretation of the working area. In this case, it represents the working area of the wind tunnel. As illustrated in Fig.2, the domain is set up to be of certain dimensions, often described in lengths of the object of interest and the sides are named. The names given to the sides of the domain are important as they represent the applied boundary conditions to the walls, whether they are a non-slip condition (where shear forces are applied) or are symmetrical (where shear forces are disregarded), (Utyuzhnikov, 2004). In some cases, if the representative object to be studied is symmetrical, it can be halved within the domain, as indicated in Fig.2, with an Ahmed body. A symmetry plane simulates turbulence symmetry and can be used effectively to reduce computational time, but should only be used if the geometry is symmetrical. It can also be effectively used to represent a body of fluid in open atmosphere rather than in wind tunnel test section conditions.

2.2.1 Domain Sizing

Table 1 - Domain Size Results

Study	Notes	Domain Dimensions (m)	Volume (m ³)	CI	Cd	Δ CI	Δ Cd
Domain 1	2x Car Dimensions (2m from origin)	(X = 8.36) (Y = 2.84) (Z = 3.24)	76.93	0.0122	0.498	-0.0967	0.393
Domain 2	3x Car Dimensions (2m from origin)	(X = 12.54) (Y = 4.26) (Z = 4.86)	259.62	-0.1014	0.4948	0.0169	0.2794
Domain 3	4x Car Dimensions (2m from origin)	(X = 16.72) (Y = 5.68) (Z = 6.48)	615.4	-0.0695	0.5122	-0.015	0.3113
Domain 4	5x Car Dimensions (2m from origin)	(X = 20.9) (Y = 7.1) (Z = 8.1)	1201.96	-0.0803	0.5023	-0.0042	0.3005
Domain 5	6x Car Dimensions (2m from origin)	(X = 25.08) (Y = 8.52) (Z = 9.72)	2076.99	-0.108	0.499	0.0235	0.2728
Domain 6	7x Car Dimensions (2m from origin)	(X = 29.26) (Y = 9.94) (Z = 11.34)	3298.18	-0.1033	0.4906	0.0188	0.2775
Domain 7	8x Car Dimensions (2m from origin)	(X = 33.44) (Y = 11.36) (Z = 12.96)	4923.22	-0.1423	0.4809	0.0578	0.2385
Domain 8	9x Car Dimensions (2m from origin)	(X = 37.62) (Y = 12.78) (Z = 14.58)	7009.83	-0.109	0.4884	0.0245	0.2718
Domain 9	10x Car Dimensions (2m from origin)	(X = 41.8) (Y = 14.2) (Z = 16.2)	9615.67	-0.0578	0.4825	-0.0267	0.323

Nine domain studies were conducted, each with increasing volume as indicated in Table 1. Highlighted in yellow, a domain with 5x the car dimensions in the X, Y & Z directions provided the closest CI result.



Graph 1 - Domain Volume Results

However, as Graph 1 indicates, a domain 8x the car dimensions provide the closest Cd results to the datum.

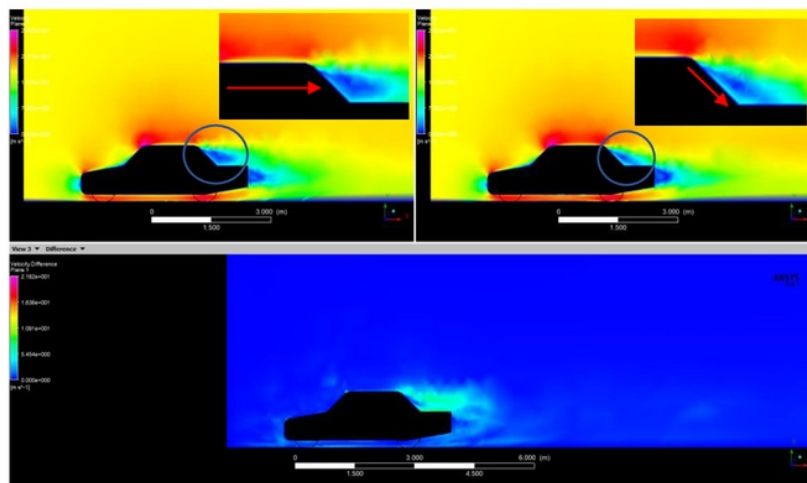


Figure 5 - Velocity Contours (Top Left = 5x, Top Right = 8x & Bottom = Difference)

A comparison of the two domains shows some interesting features, as can be observed in Figure 5. A domain with dimensions 8x the car develops a smaller wake region behind the rear slant. Whether its under predicting this particular area of separation remains to be seen.

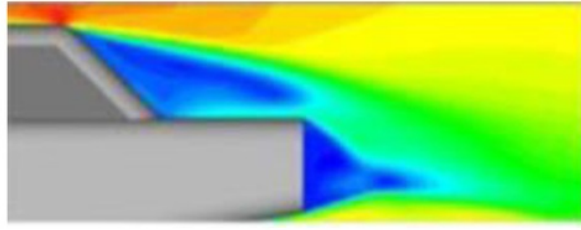


Figure 6 - Velocity Contours of MIRA Car (Palaskar, 2016)

The velocity field produced (Palaskar, 2016), illustrated in *Figure 6* provides a close representation to that conducted in the present experiment using a domain 5x the car dimensions, although there are differences to be observed. However, *Palaskar* produced a C_d value of 0.277 whereas the current experiment produced a value almost twice that of 0.502, which is 0.206 over the datum. This would suggest that the CFD cases were using different boundary conditions.

2.2.2 Car Positioning within the Domain

The positioning of the geometry of interest within the domain is critical in capturing all of the flow field representation data. If the car is placed too close to the velocity inlet, a significant amount of data could be lost at a crucial point on the geometry. The forward face of the vehicle is the first face to make contact with the fluid (air). This region is presented as a high-pressure area, with the highest pressure being present on the surface, and subsequently dissipated forwards as an equal and opposite reaction to the fluid flow in the X direction, which is essential to the overall amount of drag produced

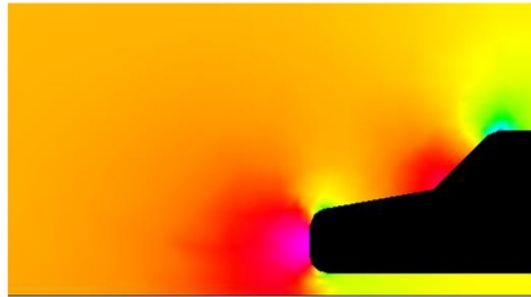
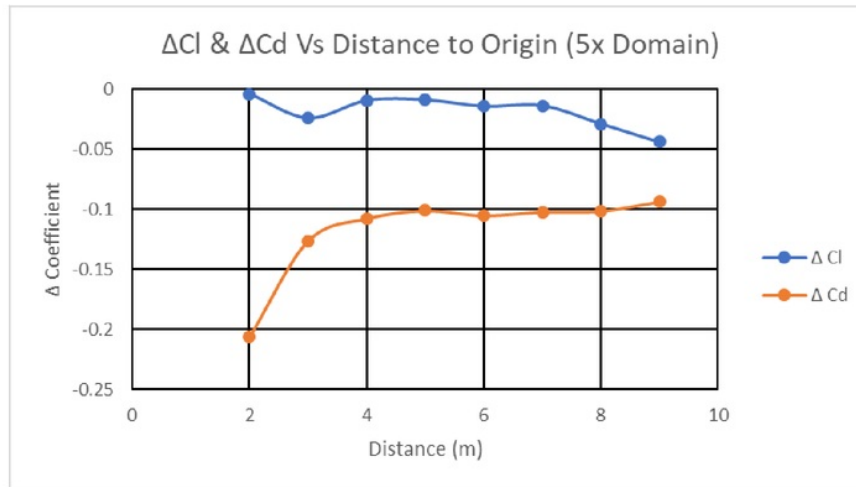


Figure 7 - Pressure Field (Stagnation Zone)

As can be observed in *Figure 7*, if the car is positioned too far forward within the domain, a crucial pressure field may not be fully calculated and thus producing inaccurate results. A length of at least 3 vehicle lengths is suggested (Llanfrit, 2005), while (Franck, et al., 2009) recommends 2.4 vehicle lengths to capture the entire frontal pressure field.



Graph 2 - Distance to Origin

The distance experiment carried out was conducted in meter lengths rather than car lengths. In retrospect this could be misleading, especially when compared to other recommendations. The reasoning behind this was the consideration that had to be given to the unusually small domain size produced, and the possibility of a failure to predict the majority of the wake region produced from the rear of the vehicle if it were to be positioned too far rearwards. However, extra considerations could have been made in regard to overall domain size to account for this issue.

Graph 2 illustrates the results from the car positioning. A distance of 5m from the velocity inlet was found to produce the best overall average C_l and C_d figures.

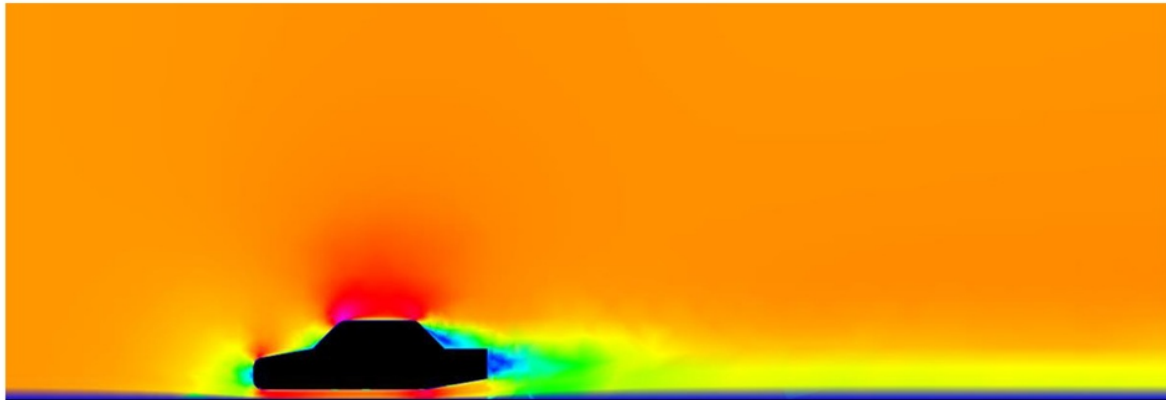


Figure 8 - 5m Frontal Distance Velocity Field

Moving the car rearward in the x direction has had an unusual impact on the rear separation zone. Its visible that the wake region on the rear slant has again changed its form and has now become similar to that produced in a domain of 8x the vehicle dimensions, though the frontal stagnation appears to have been fully captured.

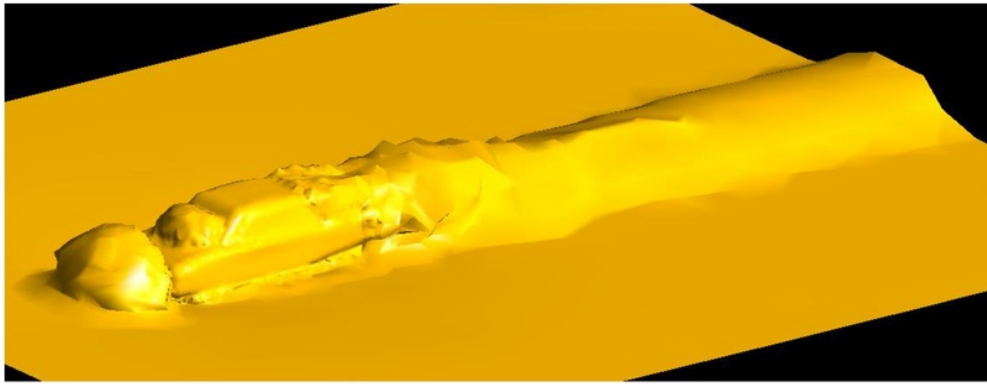


Figure 9 - 5m Frontal Distance Velocity Isosurface

An isosurface displaying areas with a velocity range of 0m/s to 17m/s clearly shows that the full rear wake region produced from the MIRA car has not been captured. This suggests that critical wake pressure field information has not been calculated that could potentially have a severe impact on the resulting figures and pressure characteristics.

However, the relativity of this study is questionable. As the purpose is to accurately replicate the known aerodynamic characteristics of the MIRA car, it would be advisable to then accurately replicate the conditions in which the MIRA car was placed during the time of the test. This would therefore invalidate any domain size testing as the test area used in the wind tunnel is of fixed dimensions in the X, Y and Z. It is also unknown where in the wind tunnel the car is placed relative to the velocity inlet, so an overall accurate representation of the wind tunnel modelling cannot be simulated.

3.0.0 Mesh

A mesh is considered to be the build grid that is referencing the domain and any objects within it. It is very important that a mesh of a suitable quality is produced. However, an economy of computational time must be understood and maintained.

To produce accurate results, it is imperative that the mesh generation process is fully understood. As the flow form around a vehicle is highly complex and relatively large, a fine mesh would be beneficial, although this would drastically increase computational time.

To avoid this eventuality, only certain areas of the fluid body need to be defined with a denser mesh. This avoids high element numbers and subsequently, lengthy computational times. (Ahmad, et al., 2010)

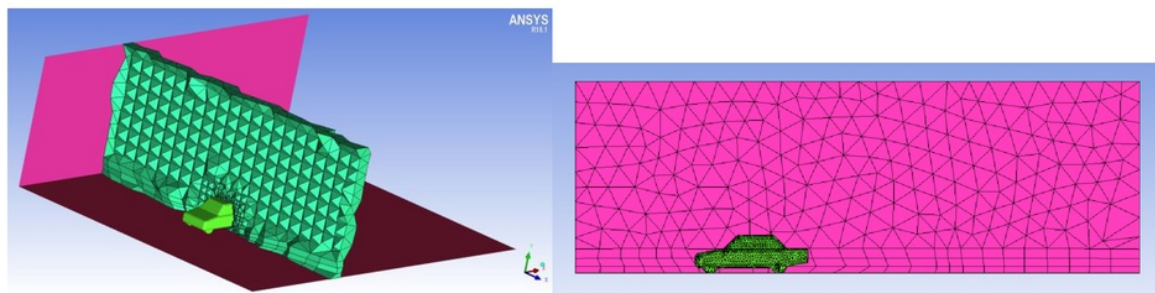
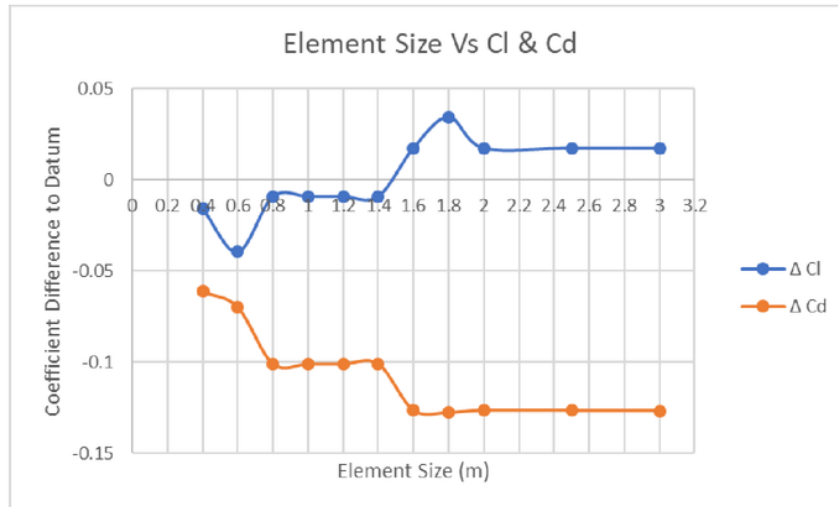


Figure 10 - Volume & Surface Mesh

For the **purpose** of this **study**, tetrahedral elements were selected to capture flow interactions within the **free stream** and prisms **were** used to obtain boundary layer performance.

3.1.0 Global Element Seed Size



Graph 3 - Global Seed Size (Free Stream Elements)

Within ICEM mesher, the Global Element Seed Size sets the maximum allowable element size within the build grid. A scaling factor of one was used throughout the test. It is advisable, however, to utilise this option for quick amendments to all mesh parameter inputs, as all mesh parameters are treated individually within the study. Surface elements were set to a continuous value and no density boxes were present. (Ansys Inc, 2009).

Global element sizes were amended to fall within a range of 0.4m to 3m, and results were examined to consider the affects that altering the volume mesh size had on the overall results within the case study. Studying *Graph 3*, it can be noted that within a range of 0.8m to 1.4m, the Cl and Cd figures remain constant and but begin to deviate when a coarser mesh size is applied.

Table 2 - Global Element Seed Size

Free Stream Element Size					
Size (m)	No. of Elements	Cl	Cd	ΔCl	ΔCd
3	81741	-0.1019	0.4229	0.0174	-0.1266
2.5	81741	-0.1019	0.4228	0.0174	-0.1265
2	81741	-0.1018	0.4228	0.0173	-0.1265
1.8	81944	-0.1189	0.424	0.0344	-0.1277
1.6	81741	-0.1019	0.4228	0.0174	-0.1265
1.4	132261	-0.0754	0.3973	-0.0091	-0.101
1.2	132261	-0.0754	0.3973	-0.0091	-0.101
1	132261	-0.0754	0.3973	-0.0091	-0.101
0.8	132261	-0.0754	0.3973	-0.0091	-0.101
0.6	593424	-0.045	0.3662	-0.0395	-0.0699
0.4	592974	-0.0687	0.3576	-0.0158	-0.0613

As previously stated, as the mesh element count rises, the computational time begins to appreciate. This fact was taken into consideration, so a maximum element size of 1m was chosen to be carried forward. If maximum precision was required at this stage, an element size of 0.4m could have been selected (*Graph 3*), however, as can be noted in *Table 2*, the element count is 4 times that of the 1m elements when they are employed.

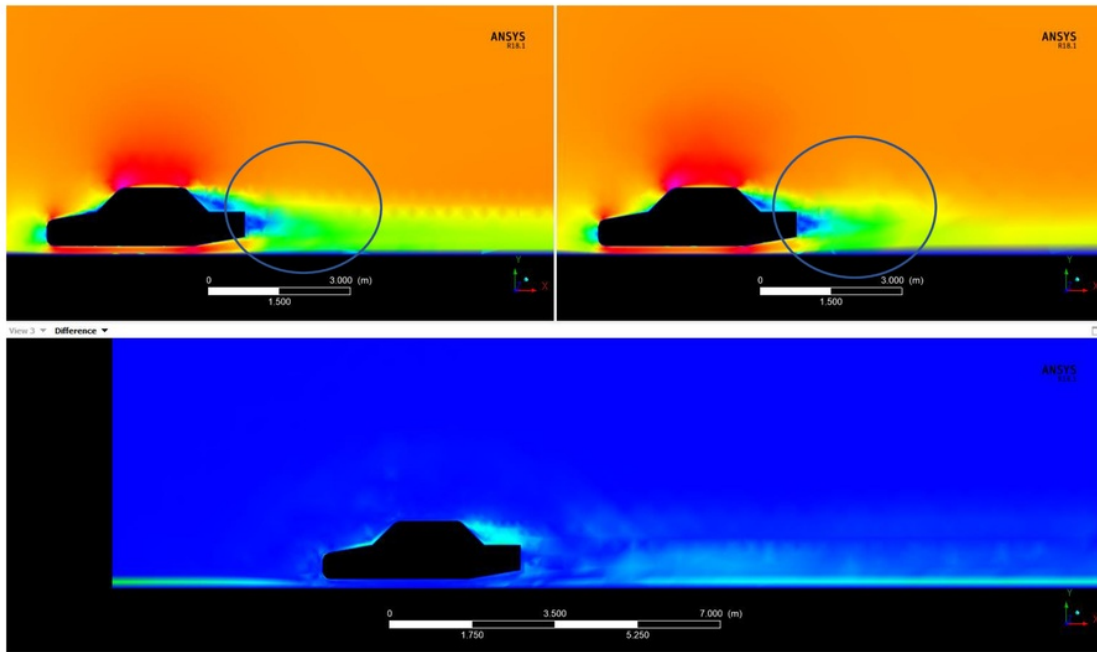


Figure 11 - Comparison of 0.4m (Top Left) vs 1m (Top Right) Volume Elements (Velocity Contours)

As the accuracy of the results is related to the overall finite method used, it is apparent that a finer mesh will provide superior results. This is as a result of a mathematical model being overlaid with the ability to predict the turbulence structures produced. It achieves this by obtaining pressure and velocity predictions on a point located on each apex of the node. Therefore, the greater the number of elements produce more nodes and increased accuracy in flow structure prediction. (www.comsol.com, 2017)

Figure 11 displays cases using 0.4m volume elements and 1m elements. The majority of flow structure difference can be observed within the wake region highlighted by the circles. Using finer elements predicts a much smoother velocity transition, whereas, the use of coarser elements indicates a much harsher transition. It can also be observed that the separation areas behind the rear slant have a much larger recirculation area as shown in Figure 12.

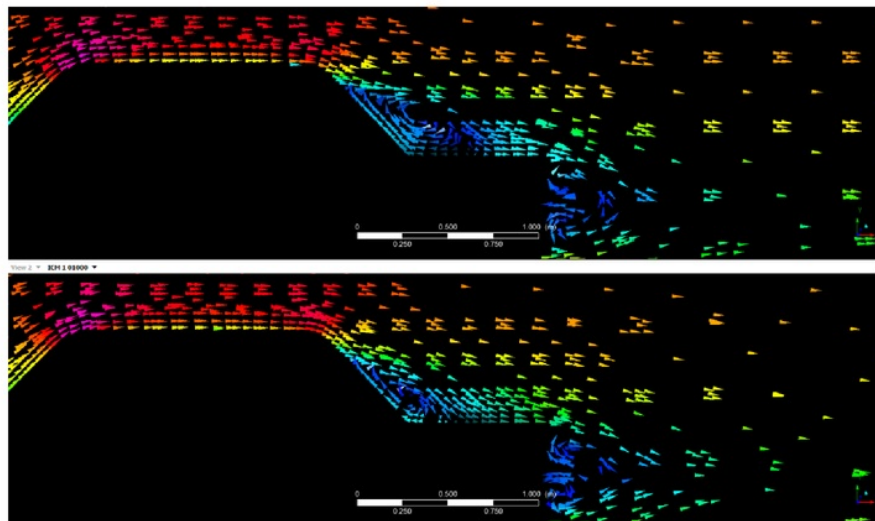


Figure 12 - Velocity Vector Comparing Recirculation 0.4m vs 1m (0.4m Top – 1m Bottom)

Using larger elements predicts a recirculation area much closer to the wall boundary of the car. This is possibly due to the number of available nodes within the rear section.

3.2.0 Density Box

A method used to capture areas of interest in substantial detail, while keeping the element count low. A density box is placed over an area of interest where the element size can be set independently from other mesh parameters, resulting in an increase in the mesh density.

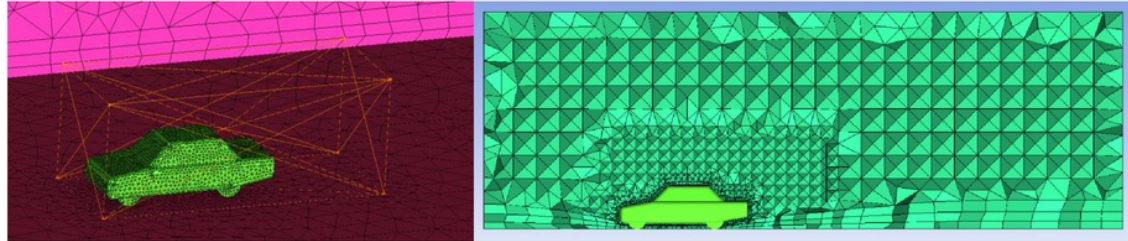


Figure 13 - Volume Mesh Density Box

3.2.1 Density Box Sizing & Positioning

The primary intentions of this experiment were, rather than examining small individual areas of interest around the car, the entire car was subjected to a density box to which further refinements could be made to include smaller areas of particular interest.

The density box needed to be of sufficient size, enough to capture the required detail within the flow structure over the car.

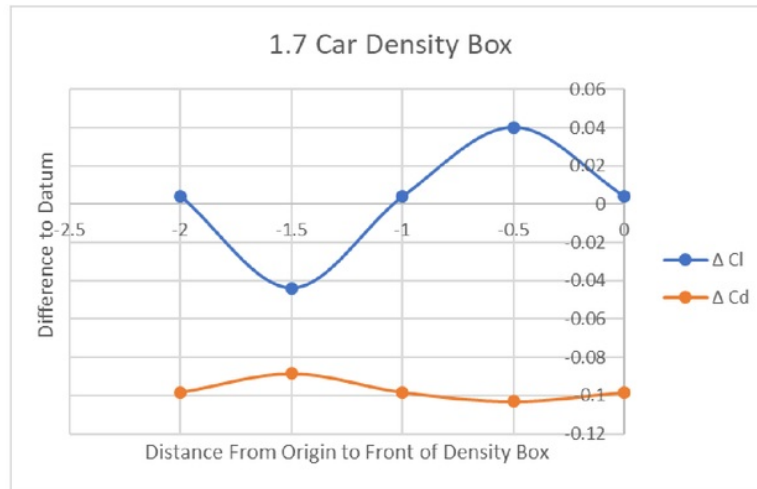
For simplicity, the sizing was conducted via a scaling of the X, Y and Z dimensions of the car.

Table 3 - Density Box Size

MIRA Reference Car Density Box						
Box Size (x Car)	Element Size (m)	Total Elements	Cl	Cd	Δ Cl	Δ Cd
1.5	0.5	138845	-0.1106	0.3886	0.0261	-0.0923
1.7	0.5	145083	-0.0885	0.3945	0.004	-0.0982
1.9	0.5	150286	-0.0734	0.3978	-0.0111	-0.1015
2.1	0.5	158155	-0.0982	0.3877	0.0137	-0.0914
2.3	0.5	165498	-0.1153	0.3942	0.0308	-0.0979
2.5	0.5	177080	-0.0993	0.4049	0.0148	-0.1086

As Table 3 indicates, a density box with the dimensions of 1.7x the car (Figure 13) provided the best overall Cl compared to that provided by wind tunnel data, however, the calculated Cd is 0.0983 too high. It has been found that with a finer mesh closer to the wall boundary, the skin friction produced was captured and provided comparable results (Ahmad, et al., 2010). As indicated in Table 3, a density element size of 0.5m was used throughout each sizing study.

Along with the total domain, the density box needs to be positioned around the car in such a way that will allow it to capture the greatest amount of detail.



Graph 4 - Density Box Positioning

It was found that at a distance of 1m from the front of the density box to the origin (positioned in the centre point of the front axle on the road) produced the closest results, however, as can be observed in *Graph 4*, large variances in C_l developed depending on the distance, whereas C_d remained relatively stable. As a 2m distance provided the same results as 1m, it was decided that having a greater distance of density to the rear of the car would provide superior accuracy, due to the nature of the rear wake region.

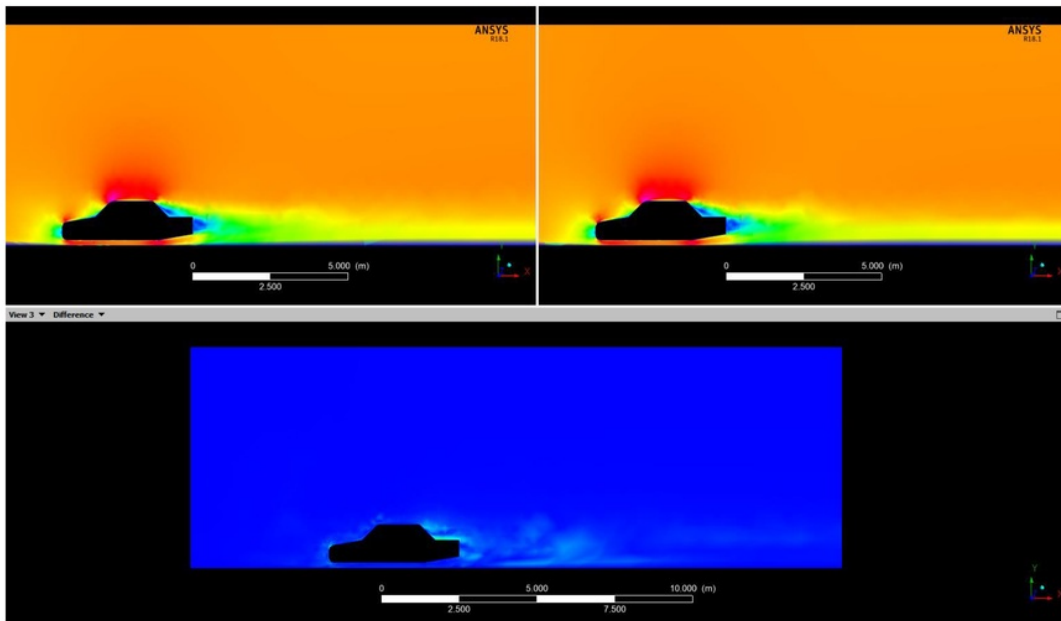


Figure 14 - Density Box Positioning (1m Forward - Top Left, 2m Forward - Top Right)

As illustrated in *Figure 14*, moving the density box forward 2m alters the way in which the flow structure in the wake at the rear is captured. Based on these findings, a decision to have the density box positioned 1m forward of the origin was made.

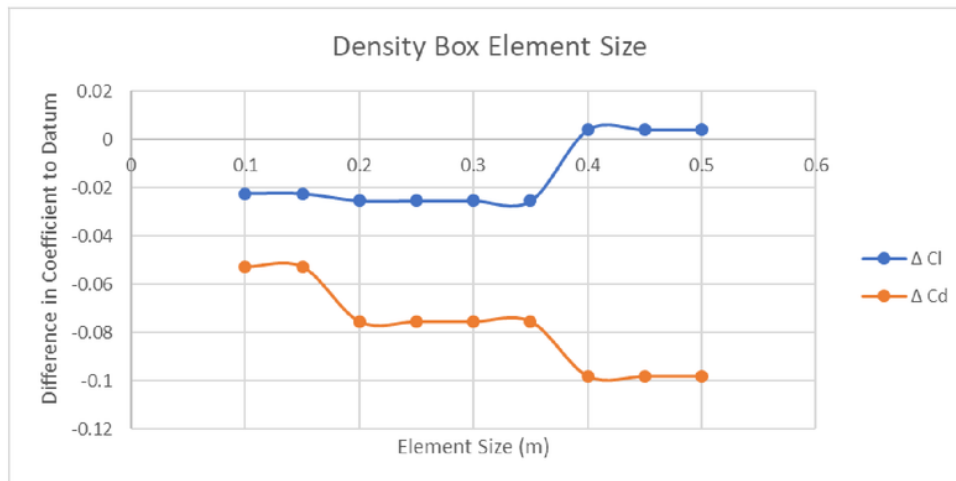
3.2.2 Density Box Element Size

As identified in the previous chapter, C_d very much depends on the density of the mesh located close to the wall in order to capture the shear forces produced from the viscous sub-layer within a boundary layer. This means close to wall element sizes need to be established and defined.

Table 4 - Density Box Element Size

1.7 Car Density Box Element Size					
Element Size (m)	Total Elements	Cl	Cd	Δ Cl	Δ Cd
0.5	144473	-0.0885	0.3945	0.004	-0.0982
0.45	144473	-0.0885	0.3945	0.004	-0.0982
0.4	144473	-0.0885	0.3945	0.004	-0.0982
0.35	246318	-0.0591	0.3717	-0.0254	-0.0754
0.3	246318	-0.0591	0.3717	-0.0254	-0.0754
0.25	246318	-0.0591	0.3717	-0.0254	-0.0754
0.2	246318	-0.0591	0.3717	-0.0254	-0.0754
0.15	1008667	-0.0621	0.3489	-0.0224	-0.0526
0.1	1008667	-0.0621	0.3489	-0.0224	-0.0526

Density box elements were reduced in size from 0.5m to 0.1m. As the size reduced, the C_d values also reduced closer to that of the datum value. However, C_l values also reduced, moving further away from the datum value as indicated in Graph 5.



Graph 5 - Density Box Element Size

A decision was made to use 0.5m elements within the density box for the remaining duration of the experiment as it provided the closest C_l values to the datum. On further examination and evaluation, by using a mean value for the resulting coefficients to choose the best suited element size may have been a hasty decision as it was based on a lack of information.

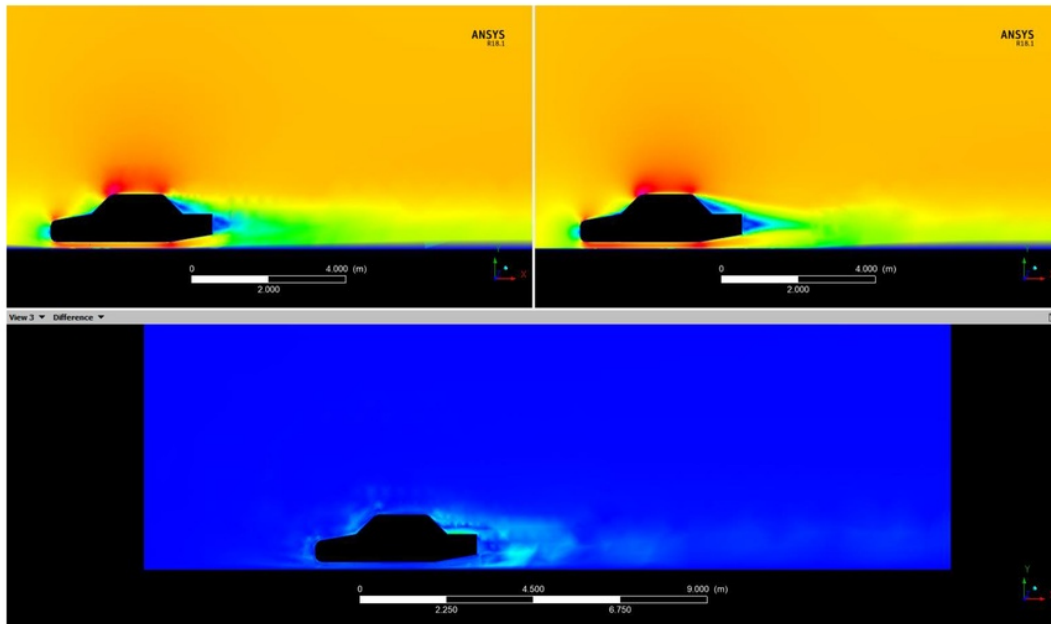


Figure 15 - Density Box Elements (0.5m - Top Left, 0.1m Elements Top Right)

As illustrated in Figure 15, once again the flow structure has been altered within the wake region and to a lesser degree around the front stagnation zone. Without comparing static pressure distribution figures across the surface of the car with the provided wind tunnel data, it is probable that a denser mesh would provide an overall greater accuracy within resultant outcomes.

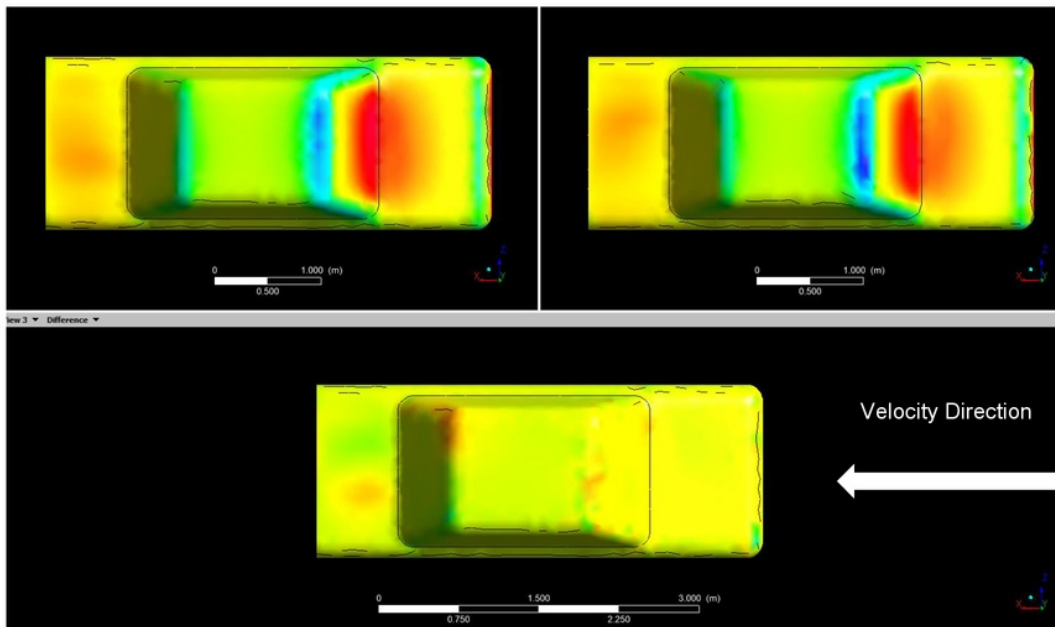


Figure 16 - Density Box Elements Pressure Contours (0.5m - Top Left, 0.1m - Top Right)

It can be observed that the pressure flow fields across the car have altered. Examining the top surface, it is noticeable that the smaller elements have modelled a lower pressure across the leading edge of the roof line, and the high-pressure distribution across the rear of the car has also changed.

3.3.0 Surface Mesh

When geometries are created, they can contain areas of high complexity. A surface mesh is used to capture the geometry, but may reduce its overall complexity depending on the mesh element size. As with a density box, separate surfaces on a single geometry can have independent element sizes assigned to them. This can assist in breaking up the geometry and recreating areas of intricate detail whilst having a reduced element size on simple surfaces.

(Bakker, 2006).

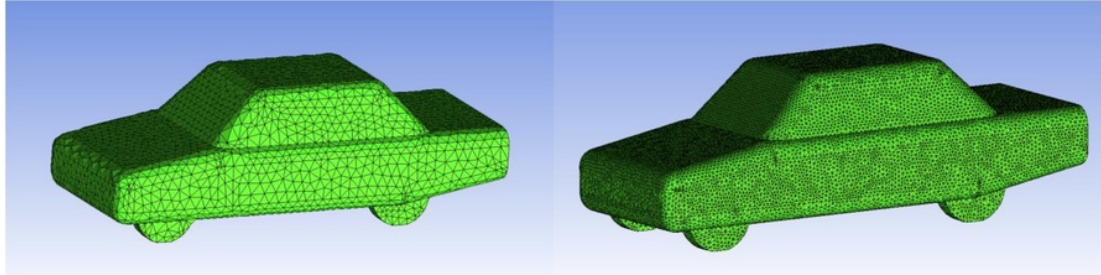
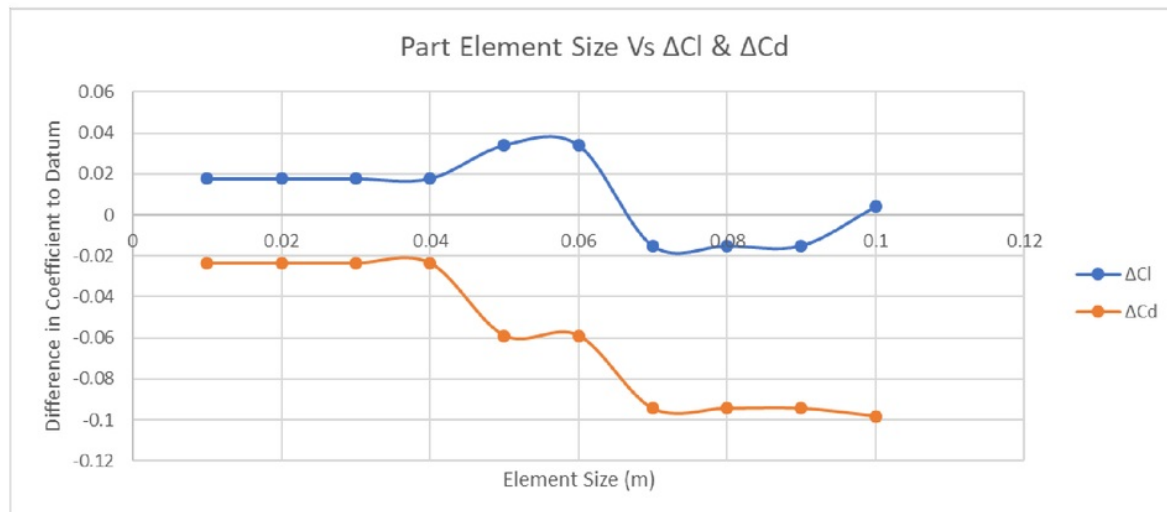


Figure 17 - Surface Mesh

The above figure illustrates the visible difference between a coarse mesh (left) and a fine mesh (right).



Graph 6 - Part Element Size

Surface element sizes were produced in a range from 0.1m to 0.01m. *Graph 6* indicates that once an element size of 0.04m had been reached, the C_l and C_d figures stabilised.

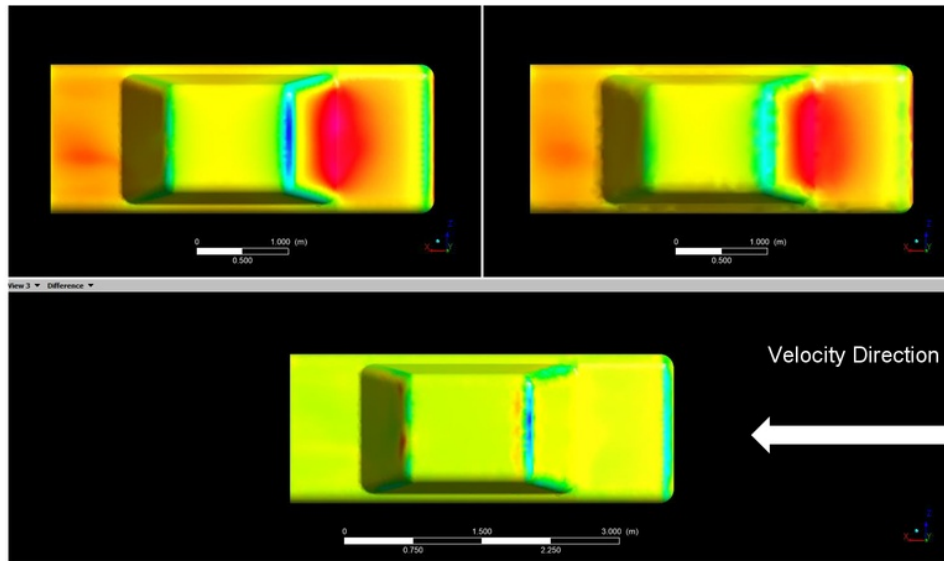


Figure 18 - Part Element Size Pressure Contours (0.04m Top Left, 0.1m – Top Right)

The pressure contours displayed in *Figure 18* indicate some of the same traits produced by reducing the density box element size. The most notable difference can be observed in the leading edge of the roof line, with 0.04m elements capturing a much lower pressure. The distribution of pressure across the leading edge of the bonnet also became significantly more uniform using a finer surface mesh.

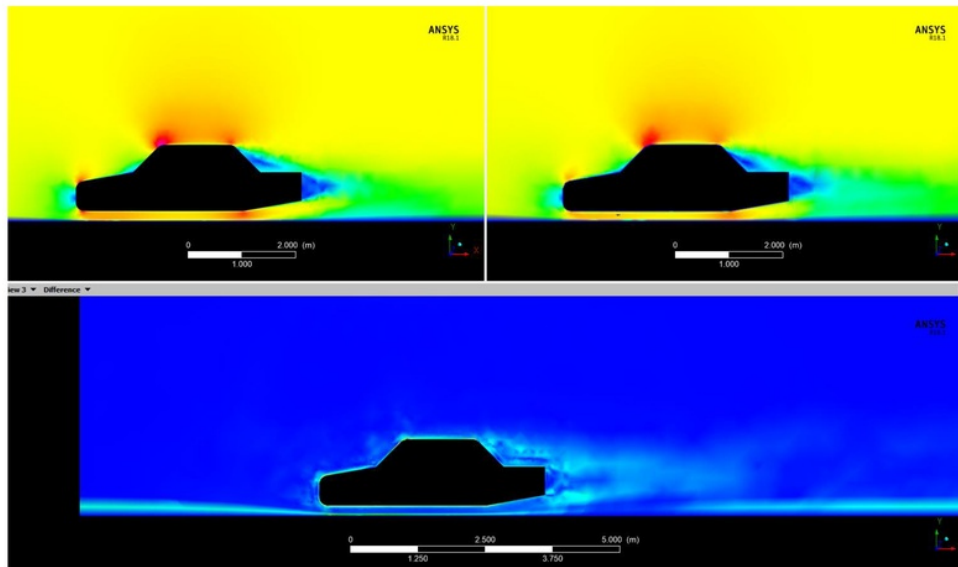


Figure 19 - Part Element Size Velocity Distribution (0.04m - Top Left, 0.1m - Top Right)

Due to the reduced size in surface mesh, the growth rate from 0.04m on the surface of the car to 0.5m in the density box has changed compared to that of 0.1m to 0.5m. This, therefore, has altered the way in which the volume flow structure has been predicted. Based on these discussions, a surface element size of 0.04m has been carried forward.

3.4.0 Prism Layers

Prism layers are used to capture the flow velocity and pressures close to walls as skin friction is produced within a boundary layer. A default growth rate of 20% was applied. This allows the prism layers to expand in areas of lesser complexity and contract in areas of high flow complexity, such as the underside of the vehicle. (Ahmad, et al., 2010) (Bakker, 2006)

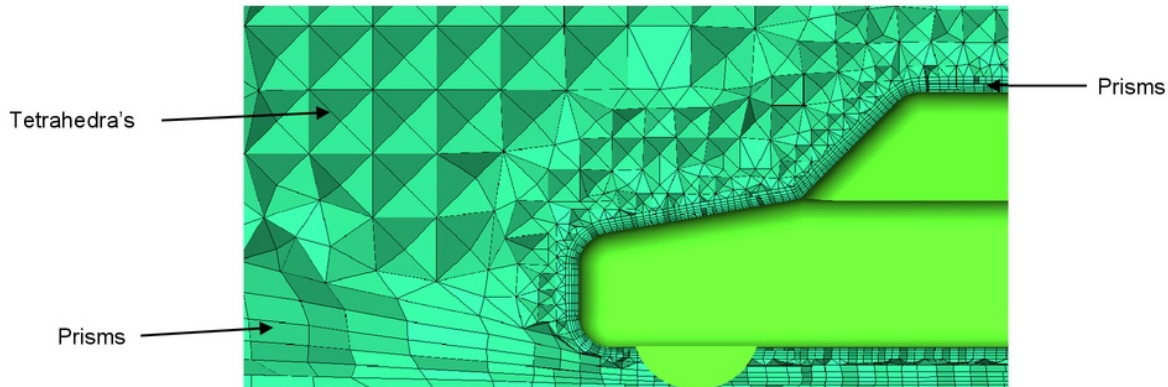
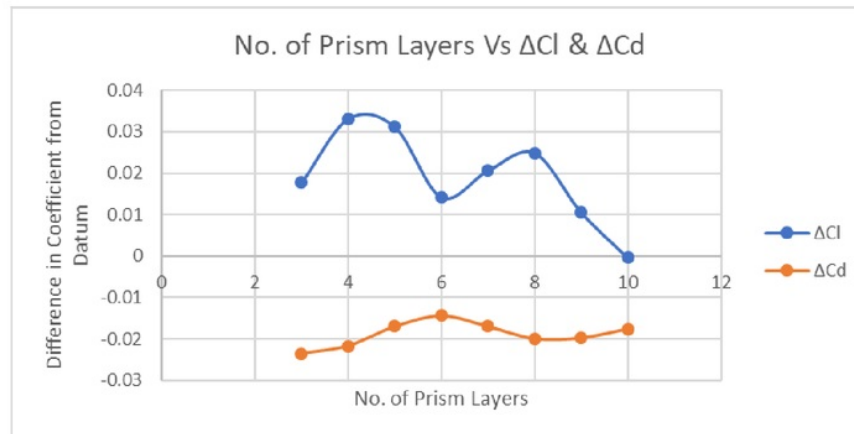


Figure 20 - Prism Element Lay Up

Unstructured meshes, such as those used within this experiment, can use tetrahedrons to resolve the close to wall pressure gradients, however extreme refinement is required, therefore prism layers are used to best capture the boundary layer as less numerical complexity is required due to flow alignment, as demonstrated in Figure 20. (Komarla, 2016). A boundary layer inflates along its surface of travel. To be able to portray it in its entirety, it is important that a suitable number of prism layers are used.

Studies using a range of 3 prism layers to 10 prism layers were simulated and the results analysed.



Graph 7 - Prism Layers

As Graph 7 indicates, as the number of prism layers reaches 6, both C_l and C_d equalise. However, 10 prism layers produces the optimum overall C_l result, although C_d is still not as closely matched as the result produced from 6 prism layers.

Table 5 - Prism Layer Study

Prism Layer Study						
No. of Prism Layers	Total No. of Elements	Cl	Cd	ΔCl	ΔCd	
3	532839	-0.1022	0.3199	0.0177	-0.0236	
4	542968	-0.1176	0.318	0.0331	-0.0217	
5	561450	-0.1157	0.3133	0.0312	-0.017	
6	591433	-0.0987	0.3107	0.0142	-0.0144	
7	625461	-0.1051	0.3133	0.0206	-0.017	
8	657290	-0.1094	0.3163	0.0249	-0.02	
9	693221	-0.0951	0.3161	0.0106	-0.0198	
10	730963	-0.0841	0.3139	-0.0004	-0.0176	

6 prism layers produced a Cl result with a difference of 0.142 from the datum, whereas 10 prism layers produced a 0.0004 difference, but with a small sacrifice of Cd accuracy. Based on these results, using 6 prism layers produces Cd results of greater accuracy with less computational time being required to be resolved, compared to that of using 10 layers.

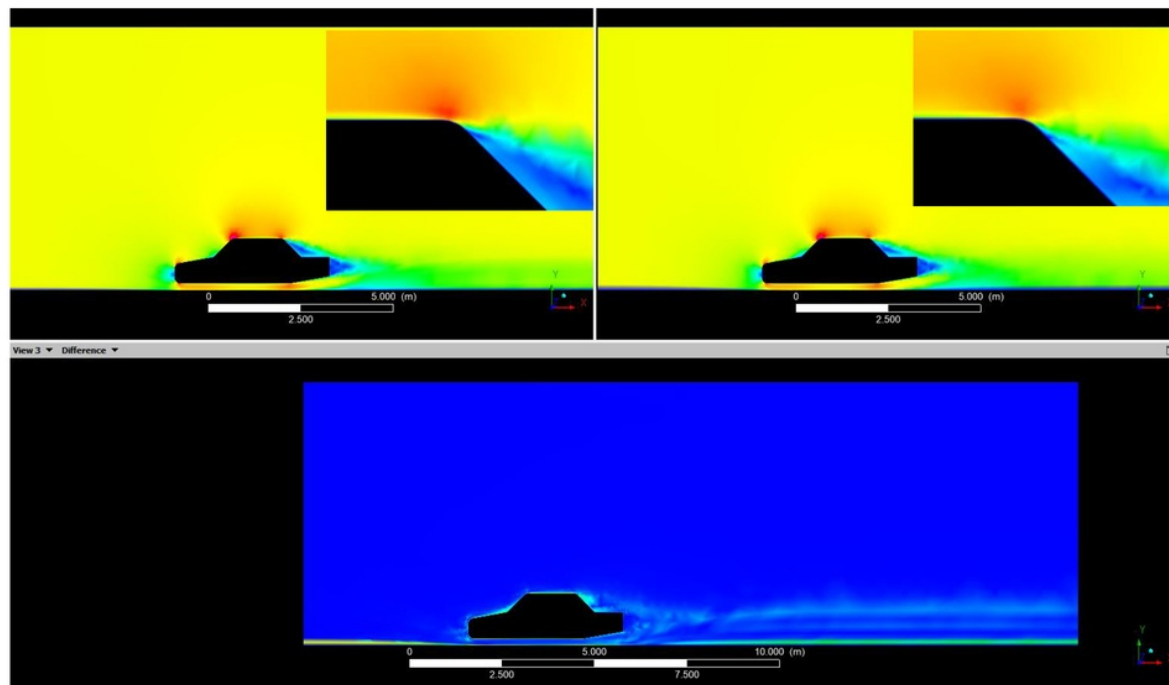


Figure 21 - Prism Layer Velocity Flow Field (6 Layers - Top Left, 3 Prism Layers - Top Right)

The above figure demonstrates that using 6 prism layers predicts a thinner boundary layer when assessed against 3 prism layers. This would suggest that previous results have been over predicting the severity of the boundary layer and its influence across the wall boundaries. It is also visible that the use of 6 prism layers predicts that separation occurs earlier on the rear slant along with a larger adverse pressure gradient.

As a boundary layer is directly related to static pressure, it would be beneficial to compare the static pressure distribution across the car and compare it to the provided data to confirm the results of the current study. However, rather than implement this, a series of assumptions were made based entirely on the Cl and Cd figures. This could prove to be problematic within this experiment, but it is suggested that 5 prism layers are a representative basis to capture a boundary layer around a bluff body. (Llanfrit, 2005)

3.5.0 Volume Meshing Method

Three types of volume meshing method exist within ICEM CFD;

- Robust (Octree)
- Quick (Delaunay)
- Smooth (Advancing Front)

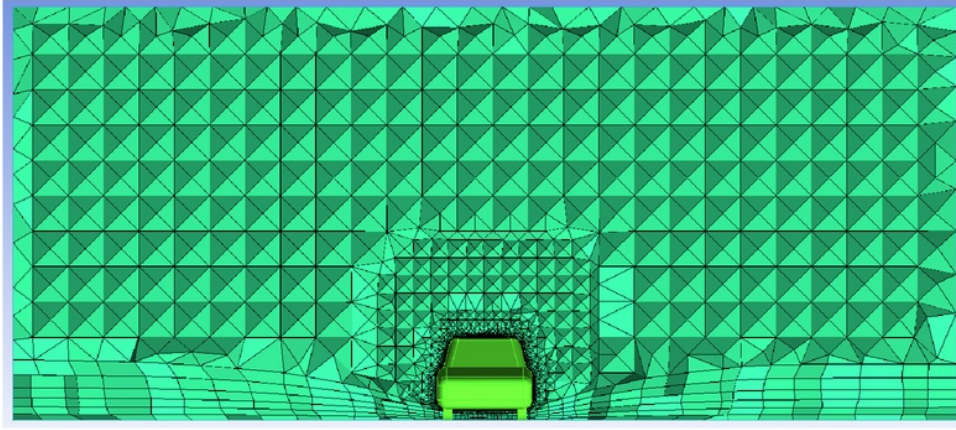


Figure 22 - Robust (Octree) Volume Method

A Robust Octree method uses a “*top down*” approach in the way it applies the volume structure. An input volume element size is subjected to the outer walls of the domain and then extruded inward towards the geometry of interest. Once the geometry is enclosed, the constructed mesh is subjected to a “cutter” running along the geometry curves, resulting in a mesh that is representative of the geometry. This volume method balances elements sharing a face or edge by not allowing them to differ in size by more than a factor of 2. (www.sharcnet.ca, 2015). This creates a mesh that appears very well structured and “inorganic”.

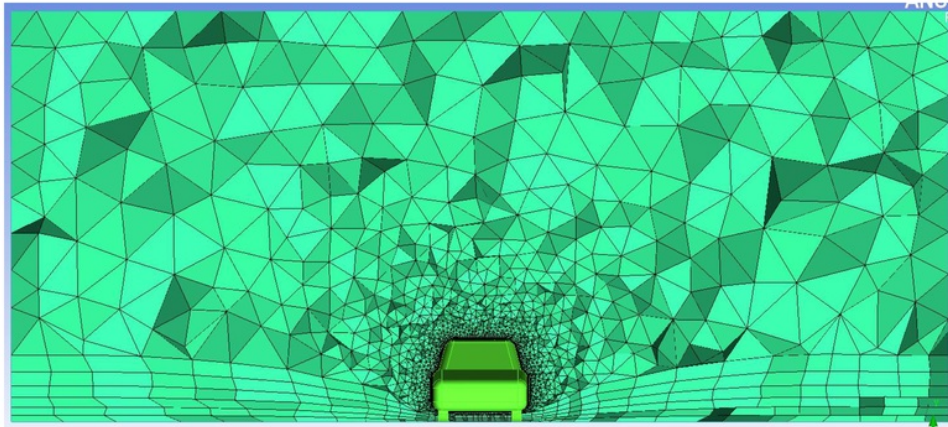


Figure 23 - Quick (Delaunay) Volume Method

Unlike the Octree method, the Delaunay algorithm produces a volume mesh based on a “*bottom up*” approach. This process requires the initial input of a generated surface mesh on the geometry and existing walls and then extrudes elements into the body volume, with respect to input global element settings. This technique produces a gradual expansion in element sizes, which visually is not as “structured” as an Octree mesh. (www.sharcnet.ca, 2015)

The Smooth Advancing Front meshing technique starts by prescribing points equally spaced along a boundary curve to discretize the geometry, from this, triangles are formed in layers (fronts) and are “advanced” into the centre of the domain. (www.pointwise.com, 2012)

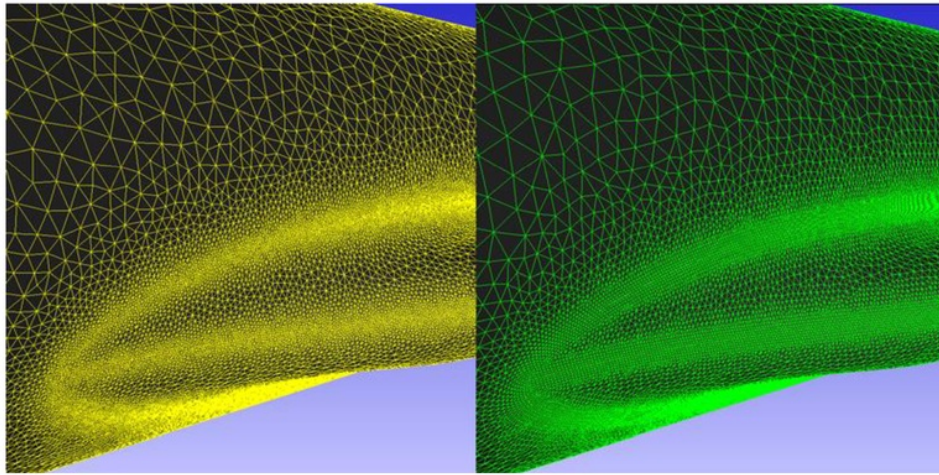
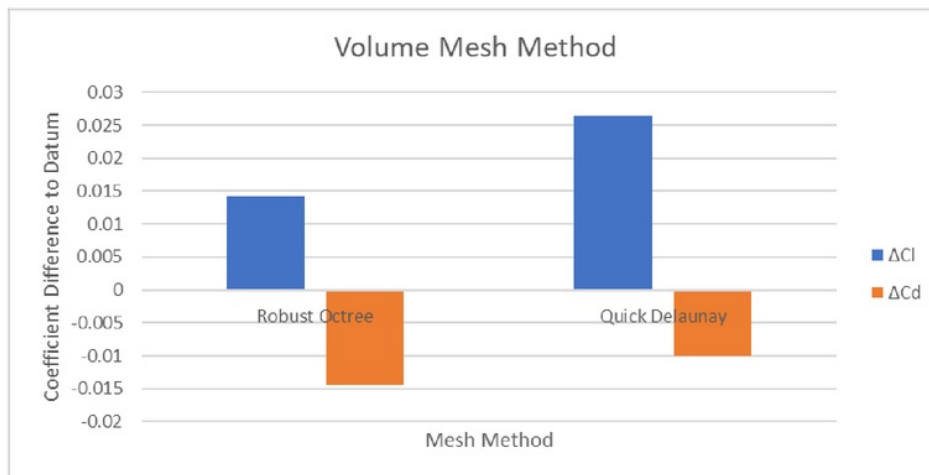


Figure 24 - Delaunay Surface Mesh (Left), Advancing Front Surface Mesh (Right) (www.pointwise.com, 2012)

This approach creates further uniformity compared to that of the Delaunay method, as illustrated in Figure 23. Using this method produces high quality boundary meshes, however, the geometry provided needs to be of an exceptional quality in order to allow this method to work (Mokri's, 2012). In the case of the MIRA car, this method was not feasible, due to the quality of the geometry.

Both Robust Octree and Quick Delaunay meshes were produced with exacting global settings and simulated.



Graph 8 - Volume Mesh Method

The Robust Octree method produced consistent Cl and Cd results with respect to the datum values. However, the Delaunay method produced an offset in the results, with Cd being 0.044 closer to the datum than the Octree method, although Cl was 0.0122 further away from the datum value. It is also noted that the element count for the Octree method contained 4300 more elements than the Delaunay method. This could be the reason for the change in the results produced.

A further comparison to be noted between the two methods is the way in which both predict the rear wake region produced at the rear of the car.

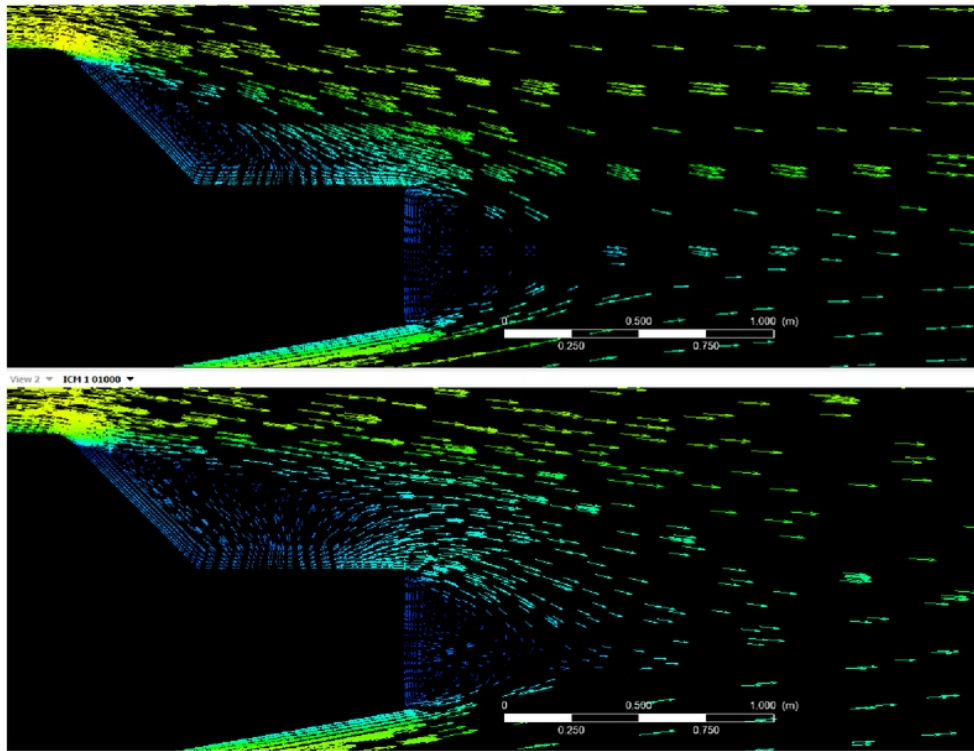


Figure 25 - Robust Octree Method (Top), Quick Delaunay Method (Bottom) (Velocity Vectors)

The Delaunay method is predicting a much greater recirculation zone at the rear slant of the car, with the flow field travelling in both clockwise and anti-clockwise directions. Pressure contours show little variance within the distribution of pressure within this region, however, static pressure distribution was not monitored in a way which could accurately compare both methods. Observing *Figure 25*, the construction of varying mesh methods alters the way in which the turbulence model replicates the turbulent flow field around the object of interest.

3.6.0 Mesh Quality

The quality of the produced mesh is important to the accuracy of the results produced by the solver. One way of measuring mesh quality is to analyse its element skewness.

Skewness is the measure of how much a single element deviates from its optimal shape (equilateral). (Bakker, 2006)

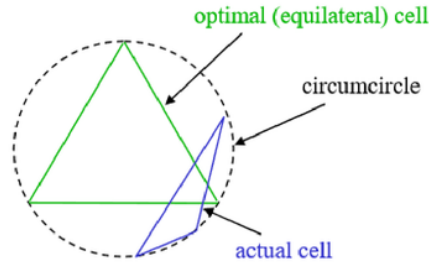


Figure 13 - Element Skewness

Table 6 - Skewness Values

Value of Skewness	0-0.25	0.25-0.50	0.50-0.80	0.80-0.95	0.95-0.99	0.99-1.00
Cell Quality	excellent	good	acceptable	poor	sliver	degenerate

The above table compares a skewness value against the element quality. A value of 0 denotes an element of good quality, suggesting its shape is optimum, with a value of 1 denoting poor element quality. High levels of skewness are not acceptable within the solving software, as the turbulence models used to solve the simulation assume that the elements within the mesh are of equilateral shape, or near to equilateral, as the nodes based upon the elements need to exist in a non-coplanar state. (www.sharcnet.ca, 2015)

Orthogonal quality is based on vector travel from each element centroid node to the next element centroid node. For good orthogonal quality, each vector should be adjacent to the element face of which it travels through (Figure 26).

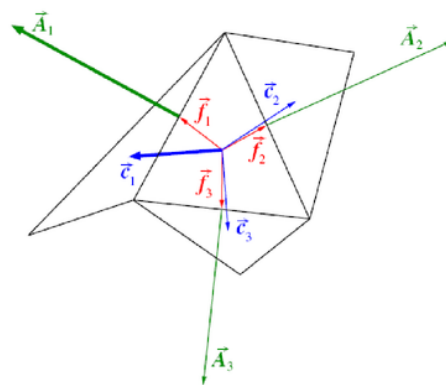


Figure 14 - Element Orthogonal Quality (www.sharcnet.ca, 2015)

The means by which orthogonal quality is measured is in reverse to that of element skewness. A value of 1 represents an excellent level of orthogonal quality, whereas 0 suggests very poor quality. (Smith, 2017)

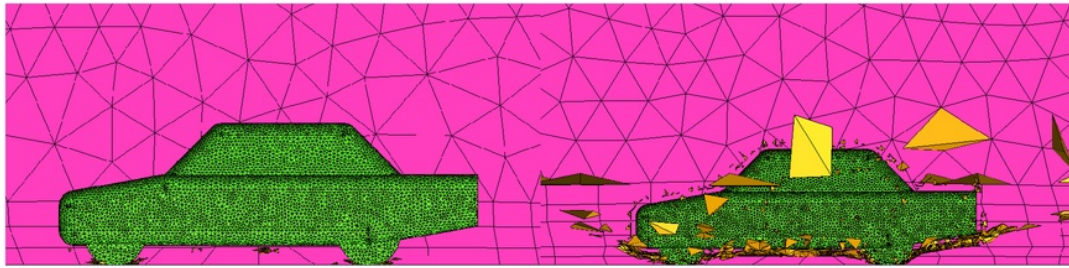


Figure 15 – Showing Elements of Poor Orthogonal Quality (3 Prism Layers – Left) (6 Prism Layers – Right)

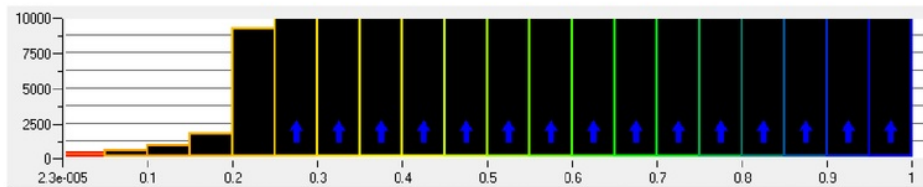


Figure 16 - Orthogonal Quality Histogram (3 Prism Layers)

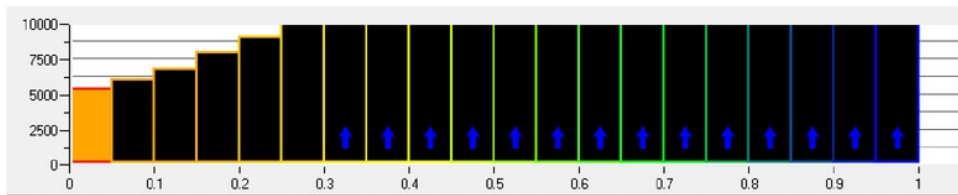


Figure 17 - Orthogonal Quality Histogram (6 Prism Layers)

The difference in orthogonal quality between 3 prism layers and 6 prism layers can be identified in *Figure 27*. The majority of the poor quality is produced by the compressing nature of the tetrahedron's forward and aft of the wheels. This progressively worsens when more prism layers are applied.



Figure 18 - Added Wheel Blocks to Geometry

Blocking the wheels is a technique that could have been used to eliminate some of the poor quality elements around the wheels. However, this method was not employed due to the requirement to maintain the most accurate representative geometry to that of the real car. Making the blocks oval rather than square eliminates the majority of non-existent high pressure areas that square blocks would produce, and begin to replicate a tyre contact patch upon the road surface.

It is not known whether having a higher level of orthogonal quality produces a lesser degree of accuracy within the results than that of introducing wheel blocks to the geometry. A study of these cases would have been beneficial in order to assess how much either scenario would affect the overall outcome of results.

Another popular method of reducing poor orthogonal quality in such areas is to introduce individual density boxes having a reduction in element size. However, as previously stated, this will considerably increase the computational time for potentially very little gain in modelling accuracy.

4.0.0 Turbulence Models

Before case solving can begin, it needs to be understood what the function of turbulence models is and how they operate. A turbulence model can be described as a numerical method that predicts and simulates turbulence in and around an object.

As fluid travels across a surface, several interactions take place. The turbulence model predicts these interactions in relation to velocity, pressure, time and distance, then produces an overall analytical flow field simulation.

Within Ansys Fluent, two types of simulation can be conducted, these being steady and transient. Steady provides a fully converged flow field that is time averaged, whereas a transient simulation provides a flow field over time-based iterations.

For the purposes of the current experiment, only steady simulations were conducted, as only the final result of each case was considered relevant for comparison to the wind tunnel data.

4.1.0 What is Turbulence?

Turbulence is an occurrence of irregularity within a flow structure, when previously connected layers of atoms begin to break apart as inertial forces overwhelm viscous forces (viscosity – resistance to shear). This produces unsteady and irregular fluid motions, from which swirling structures (eddies) can be identified.

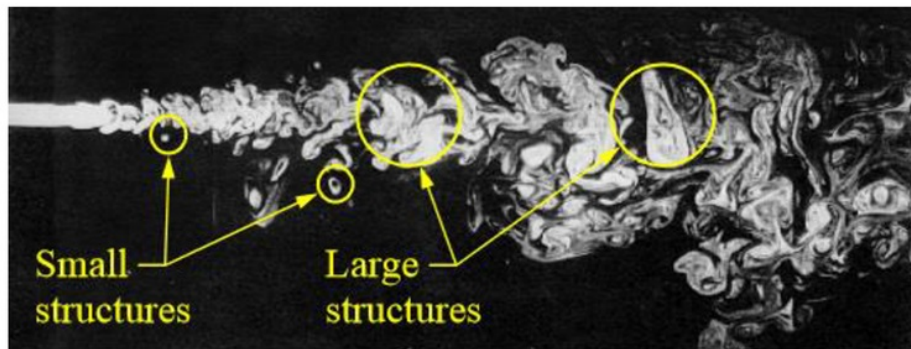


Figure 19 - Turbulent Eddie Structures (Ansys Inc, 2006)

The interaction between a fluid and a surface results in kinetic energy being depleted as heat resulting from the friction created between the fluid and the surface. The outcome resulting in fluid separation as previously discussed.

4.1.1 Reynolds Number

A Reynolds number is scalar figure that describes the condition of a fluid flow, whether it be laminar or turbulent. The number performs a significant role in many fluid modelling equations as it is one of the “*main controlling parameters in all viscous flows*”. (www.simscale.com, 2018)

The Reynolds number is based on a ratio of inertial forces to viscous forces, and is described as;

$$Re = \frac{\rho VL}{\mu}$$

Where ρ is the density, V is the velocity, L is the length and μ is the dynamic viscosity of the fluid. For a flat plate,

Reynolds numbers can be described as;

- $Re > 500,000$ = Turbulent
- $Re < 500,000$ = Laminar

However, Reynolds numbers for laminar and turbulent internal flows will be reduced.

(Subramanian, 2014) (Reynolds, 1883)

4.1.2 Euler & Navier-Stokes Equations

The Navier-Stokes equations describe the conservation of mass, momentum and energy equations for Newtonian fluids, based upon the commonly known Euler's equation, which are mathematical models describing fluid flow conditions. Euler's equation is a two-dimensional model as is Bernoulli's equation ($P_{total} = P_{static} + P_{dynamic}$), but

translated into another form of J/Kg, and is a description of how velocity, pressure and density relate within a fluid flow, neglecting any viscous forces, therefore it is only an approximation of energy. Navier-Stokes is a three-dimensional equation and provides the viscosity formulation for the Euler's equation, greater complexity being employed and is, therefore, a far superior computation for turbulent flows.

The coupled differential equations are solved simultaneously in the X, Y and Z directions, giving energy properties using a cartesian axis over a period of time.

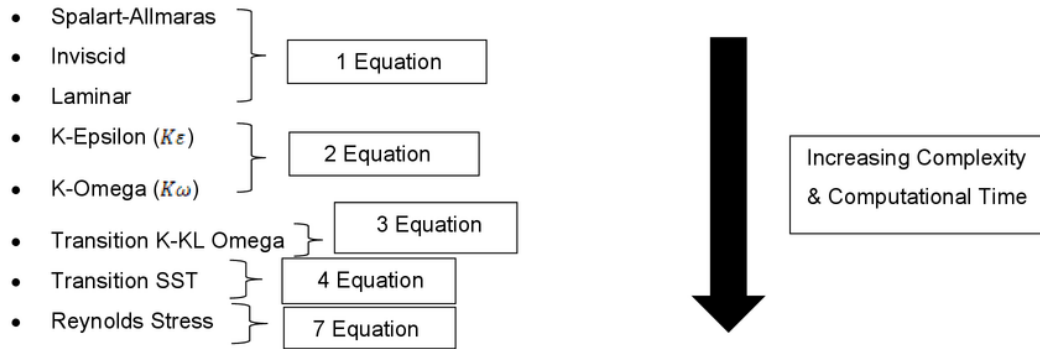
(www.simscale.com, 2018) (www.grc.nasa.gov, 2015) (Versteeg & Malalasekera, 2007) (Sodja, 2007)

4.2.0 RANS Based Turbulence Models (Reynolds Averaged Navier-Stokes)

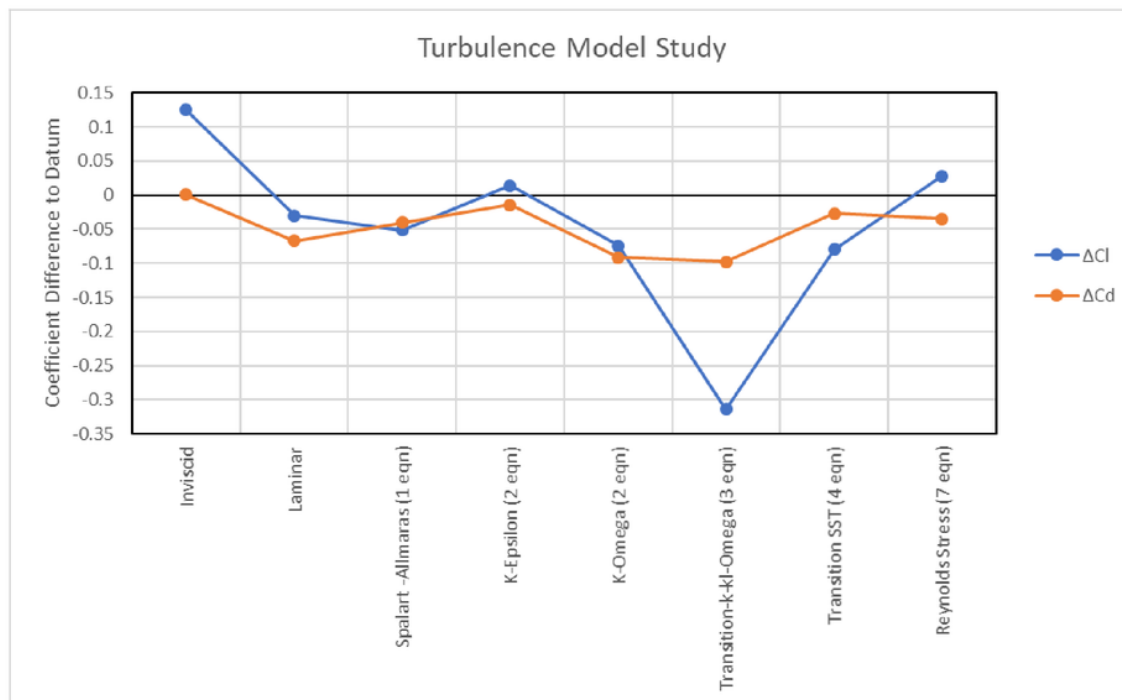
RANS turbulence models are based on a time average and are the most commonly used industry standard model for solving fluid flow around a bluff body, such as a car. They offer economical superiority over other models by neglecting the full-time dependant flow field (Bakker, 2006).

In the majority of situations, fluid dynamicists will only need to consider the mean properties of flow (velocities, pressures and stresses) rather than the instantaneous changes that occur within a fluids' regime. However, with respect to this, extra transport equations have to be added to satisfy the Reynolds Stresses and the scalar transport terms (e.g. – kinetic energy and turbulence / specific dissipation rate). (Versteeg & Malalasekera, 2007)

There are several RANS models, each calculating differing transport equations.



Each of the above turbulence models were applied to the MIRA car in individual cases.



Graph 9 - Turbulence Model Study

K-Epsilon has been observed to have notable advantages in predicting flow around a bluff body. Observed in the above graph, K-Epsilon produced the closest mean values of Cl and Cd to the wind tunnel datum. The case in question used an evolved version of the $K\epsilon$ model, known as "Realizable, Non-Equilibrium Wall Functions". This model satisfies certain criteria unseen with other variations of the $K\epsilon$ model.

In general terms, $K\epsilon$ models struggle to fulfil close to wall conditions, resulting in areas of large adverse pressure gradients and separation failing to be simulated accordingly. The reason for this is the lack of improper coding that will enable the accurate modelling of the viscous sub-layer and thus buffer regions within a boundary layer (Figure 32).

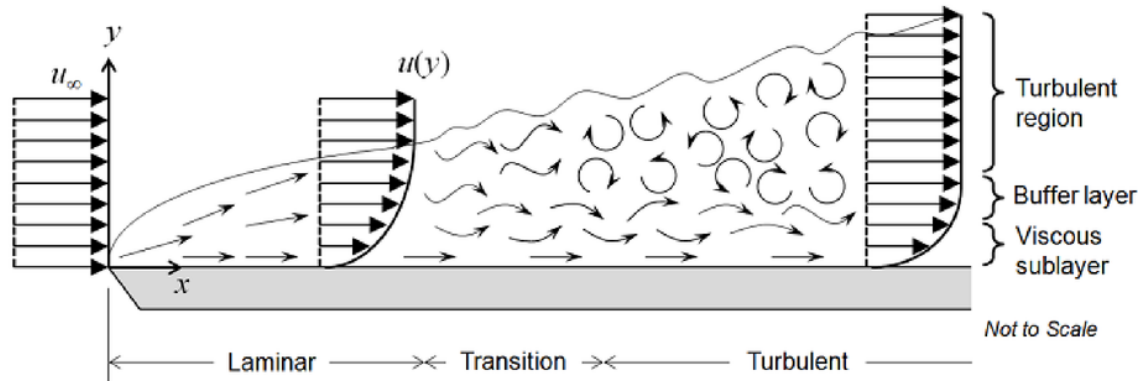


Figure 20 - Boundary Layer Conditions (Frei, 2017)

As a result of this, wall functions are used, which neglect the buffer layer and effectively ignore the 0 velocity at the wall.

The Realizable term begins to evaluate the Reynolds shear stresses and apply them, while the Non-Equilibrium Wall Functions process provides a greater accuracy in areas of larger pressure gradients. (Frei, 2017) (Ansys Inc, 2006)

Table 7 - K-Epsilon Refinement

Turbulence Model	CI	Cd	ΔCI	ΔCd
Standard, Standard Wall Functions			-0.0845	0.2963
Standard, Scalable Wall Functions			-0.0845	0.2963
Standard, Non-Equilibrium Wall Functions			-0.0845	0.2963
Standard, Enhanced Wall Treatment			-0.0845	0.2963
Standard, Menter Lechner			-0.0845	0.2963
RNG, Standard Wall Functions	-0.0324	0.2987	-0.0521	-0.0024
RNG, Scalable Wall Functions	0.0232	0.3072	-0.1077	-0.0109
RNG, Non-Equilibrium Wall Functions			-0.0845	0.2963
RNG, Enhanced Wall Treatment	-0.0136	0.3051	-0.0709	-0.0088
RNG, Menter Lechner			-0.0845	0.2963
Realisable, Standard Wall Functions	-0.1185	0.2848	0.0340	0.0115
Realisable, Scalable Wall Functions	-0.1184	0.2847	0.0339	0.0116
Realisable, Non-Equilibrium Wall Functions	-0.0987	0.3107	0.0142	-0.0144
Realisable, Enhanced Wall Treatment	-0.0991	0.289	0.0146	0.0073
Realisable, Menter Lechner	-0.1079	0.2891	0.0234	0.0072

Increasing
Computational
Time

As several evolutions of the $k\epsilon$ model now exist, a refinement study on the model took place.

Table 7 presents the findings from each "sub-model" that can be applied to $k\epsilon$. The blank fields found within the table denote non-convergence (divergence) conditions and, therefore, the case is not solvable with the current case development settings. Post processing settings (such as mesh grid build) were not altered to satisfy constant relativity between each case.

As indicated in the table of results, none of the Standard $k\epsilon$ models and 3 of 5 RNG models could be solved. The RNG (Re-normalisation Group Method) based model out-performs the Standard within the shear flow region, however, it still does not converge at the rate of the Realizable model.

It could also be assumed that as the Realizable model was used throughout each of the conducted studies, that the potential for the post processing inputs could have been developed to appease its particular needs, which then brings into debate whether the variable inputs have been developed to validate a fluid case against wind tunnel data.

However, from the current findings, the Enhanced Wall treatment performed with superiority by producing the closest mean values of C_l and C_d . The model contains a modification to its transport equations, in the way it predicts close to wall interactions within the inner layer (Viscous and Buffer),

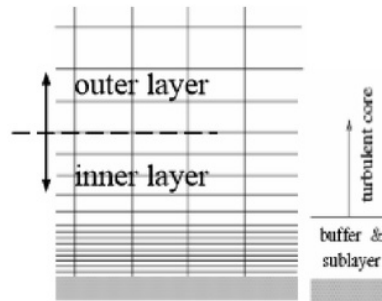


Figure 21 - Enhanced Wall Treatment Modification (Ansys Inc, 2006)

The model subsequently combines both viscous and buffer layer with benefits from both reasonable computational cost and modelling accuracy as it takes into account close to wall interactions. As a result of these qualities, the modelling capabilities for significant separation zones and swirling motion are achieved. However, in order to capture these affects, the mesh build grid must ultimately be refined around the wall regions of the model. (Al-Witry & Es-Saheb, 2013)

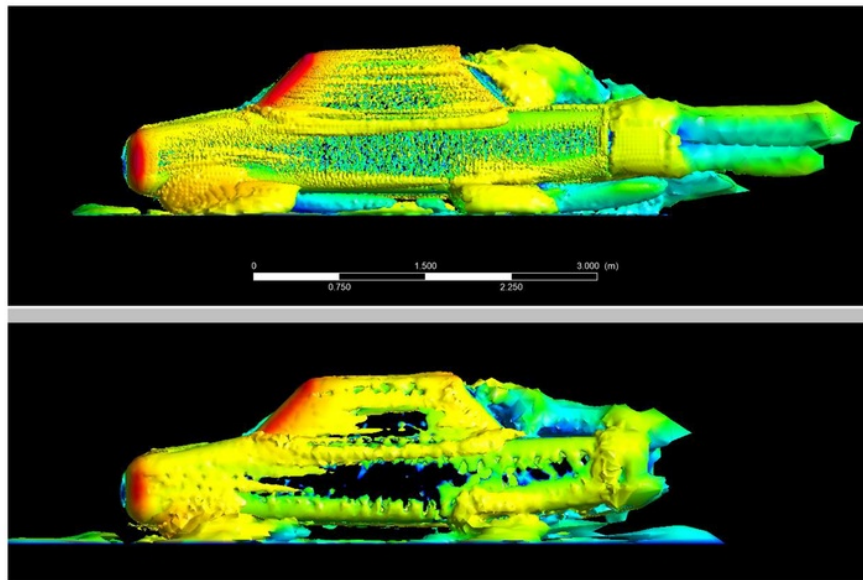


Figure 22 - Enhanced Wall Treatment (Top) & Non-Equilibrium Wall Functions (Bottom) Vortex Core Velocity

The modification made by the Enhanced Wall Treatment model upon the main body of the geometry can clearly be observed in Figure 34. The model has begun to interpret the viscous and buffer layers and model the interactions made. A comparison can be made on the frontal section, where both models have begun to predict similar turbulence structures, however, the Enhanced Wall Treatment has otherwise refined these structures, and has produced a much greater quantity of turbulent regimes.

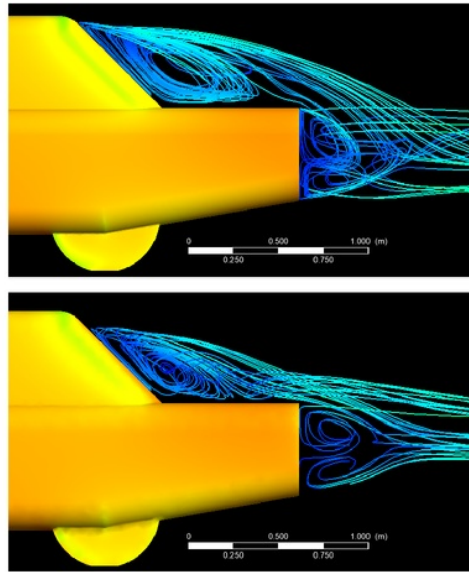


Figure 23 - Enhanced Wall Treatment (Top) & Non-Equilibrium Wall Functions (Bottom) (Velocity Streamlines)

Whilst flow structures remain similar in both cases, the Enhanced Wall Treatment has generated a much larger area of recirculation in the wake region, aft of the rear slant.

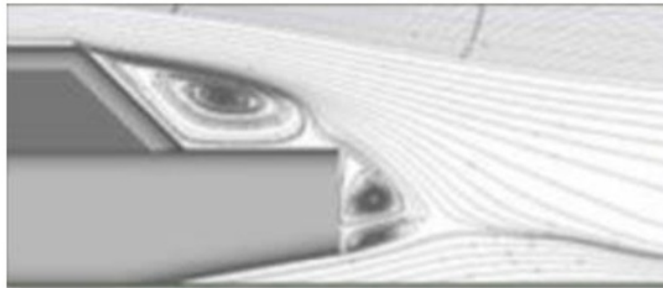


Figure 24 - MIRA Car Streamlines (Palaskar, 2016)

From Palaskars (2016) experimental CFD findings, it can be concluded that the Enhanced Wall Treatment model has produced the most comparable flow structures to that of his own experiments. However, Buscariolo (Buscariolo, et al., 2016) found that turbulent model results depend greatly on the mesh grid refinement whilst conducting a similar experiment based around a hatchback vehicle, with $k\epsilon$ producing comparable Cd results to a datum when using a coarser mesh, although a $k\omega$ SST model produced closer results when using a refined mesh (Figure 37).

RANS	~80M Mesh	~340M Mesh
k- ϵ	-0.011	+0.030
k- ω SST	-0.021	+0.011

Figure 25 - (Buscariolo, et al., 2016) Found Turbulence Model Results

4.3.0 Y+ Method

A Y+ value is a dimensionless distance of the first cell node away from the adjacent wall. It is based on the fluid velocity (wall shear stress) within the element of interest.

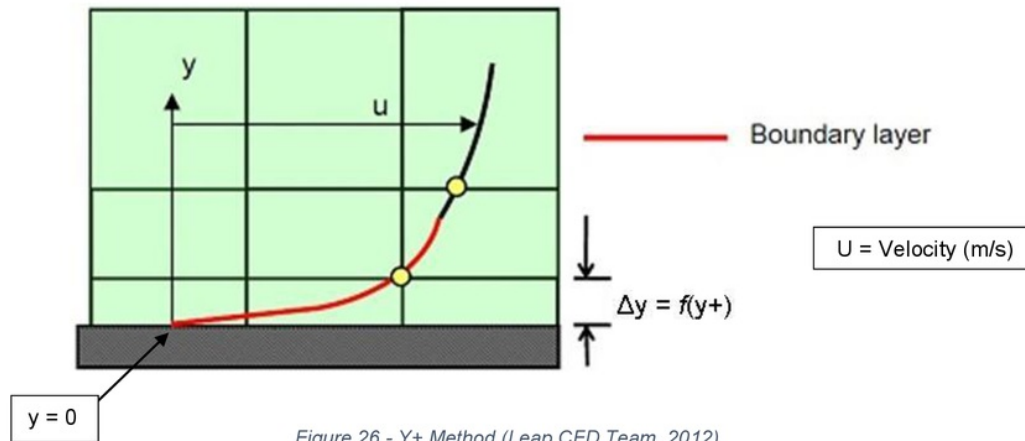


Figure 26 - Y+ Method (Leap CFD Team, 2012)

Consideration needs to be given when in relation to this height, if it were too great, then the employed turbulence model (dependant on wall function employment) may not be able to accurately predict the conditions within the boundary layer and thus further inaccuracies can develop.

$$y+ = \frac{\Delta y}{\mu} \left(\frac{\sqrt{\tau_w}}{\rho} \right)$$

- Δy = Height of first element
- μ = Kinematic Viscosity
- τ_w = Wall Shear Stress (at $y = 0$)
- ρ = Fluid Density

$$u+ = \frac{ut}{\frac{\sqrt{\tau_w}}{\rho}}$$

- ut = Velocity tangential to the wall (Gradient)
- τ_w = Wall Shear Stress (at $y = 0$)
- ρ = Fluid Density

From these two equations, an assumption can be made on the internal characteristics within a boundary layer and educated considerations can be made in regard to turbulence modelling.

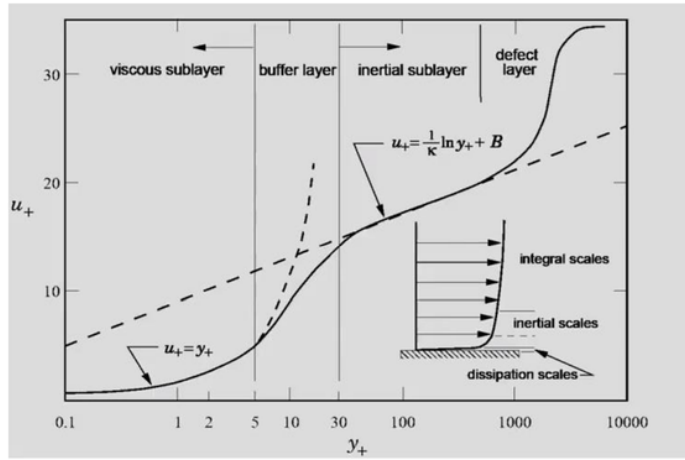


Figure 27 - y^+ Figures within Boundary Layer Conditions (Mali, 2016)

The above figure describes the y^+ values to be used according to the characteristics within a boundary layer. It is recommended that a $k\omega$ model use a y^+ value of $y^+ < 1$ where as a $k\varepsilon$ model use $y^+ > 11$.



Figure 28 - Viscous Sublayer Resolving (Left) & Log Law Resolving (Right) (Jouse, 2018)

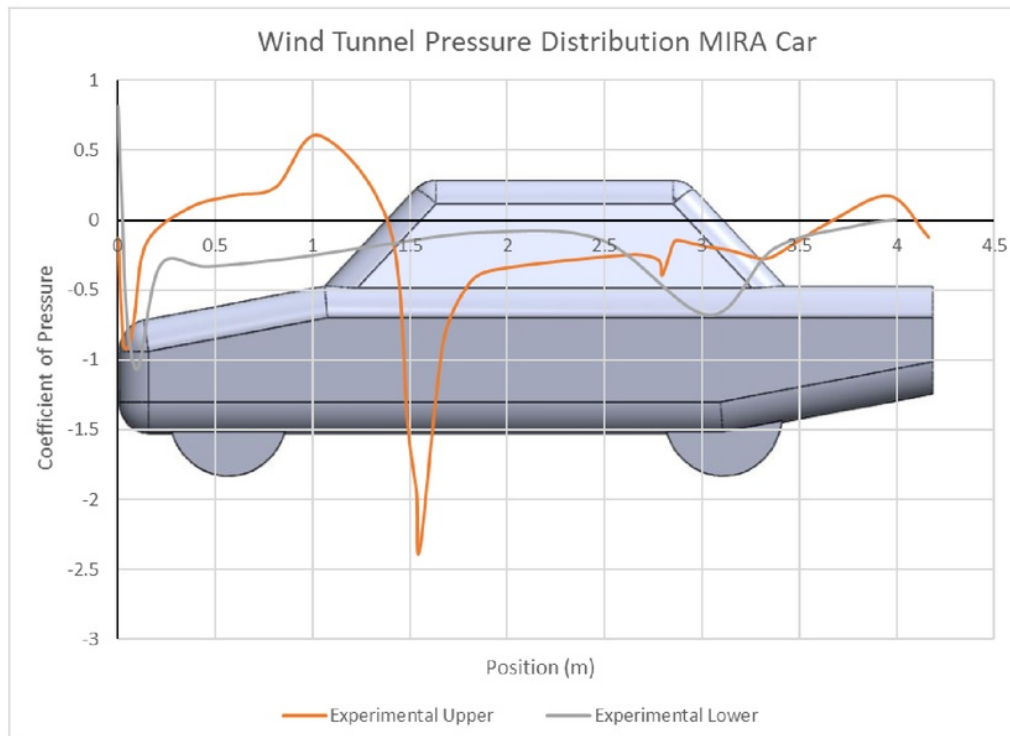
For accurate modelling, considerations need to be made in the turbulence model. A standard $k\varepsilon$ model employs the use of wall functions as previously stated, therefore the Log Law region (Inertial Sublayer) is as defined as the model can achieve close to the wall. If the first node were to fall above this region, full boundary layer definition would not be achieved, and thus inaccurate results would be the outcome. The same, applies to a $k\omega$ model which does not employ the use of wall functions and can define the viscous sublayer, therefore mesh density needs to be increased close to the wall, thus a decrease in y^+ is preferable.

(Ariff, et al., 2009) (Kobeissi, 2017) (www.autodesk.com, 2018)

However, this method was never utilised throughout the experimentation process as a default undefined prism growth law was used throughout prism inflation extrusion. Therefore, it is unknown whether the close to wall characteristics within the CFD cases have been modelled correctly. However, referring to surface pressure distribution graphs will describe the interaction between the fluid and geometry, providing an estimate of the accuracy of the close to wall modelling.

4.3.0 Pressure Distribution

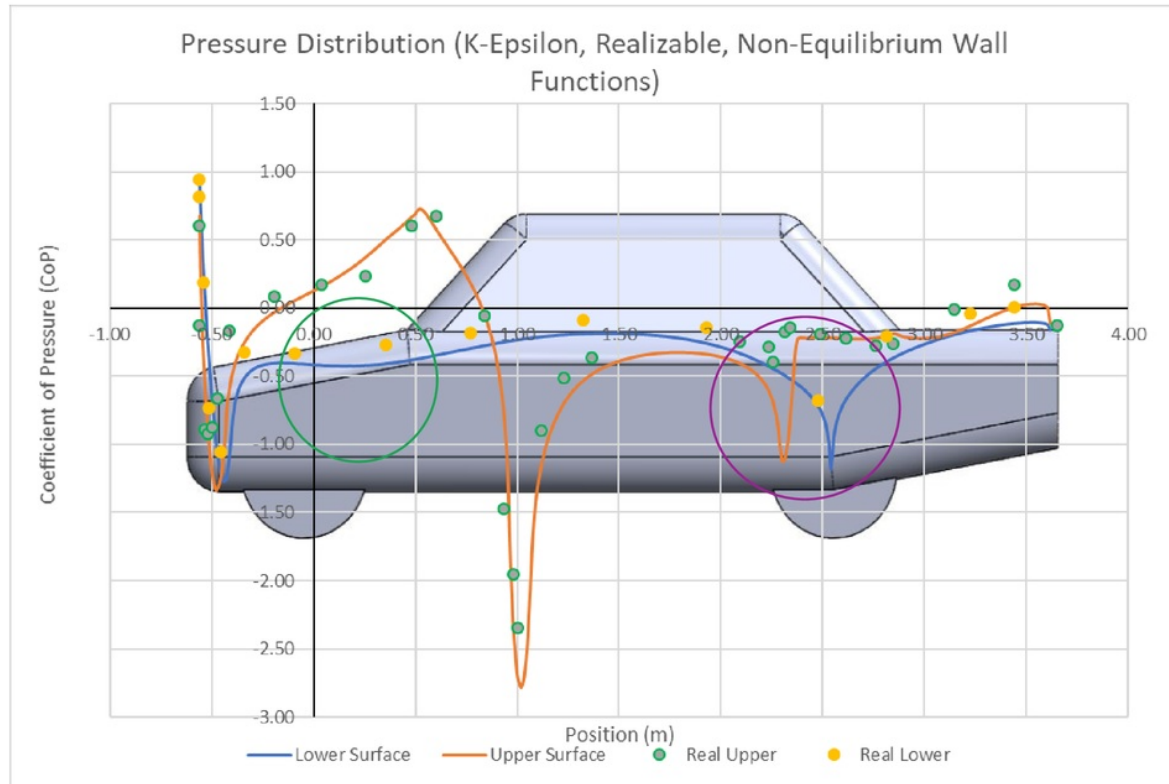
Further means of validation were conducted using the production of pressure distribution graphs.



Graph 10 - MIRA Car Wind Tunnel Pressure Distribution

The above graph displays the static pressure distribution across the surface of the car. This data was produced with the use of a wind tunnel, so therefore represents real pressure rather than predicted pressure. As a result of this, it was decided that to produce an even greater representation of the real-world scenario, using the domain dimensions acquired through the presented experiment process would skew the resulting outcomes, therefore the dimensions of the MIRA wind tunnel were implemented along with wall boundary conditions.

4.3.1 K-Epsilon, Realizable, Non-Equilibrium Wall Functions



Graph 11 - KE, Realizable, Non-Equilibrium Wall Functions Pressure Distribution

As $K\epsilon$, Realizable, Non-Equilibrium Wall Functions was the base model throughout the majority of the conducted testing, it was decided that a further direct comparison to the wind tunnel data was required.

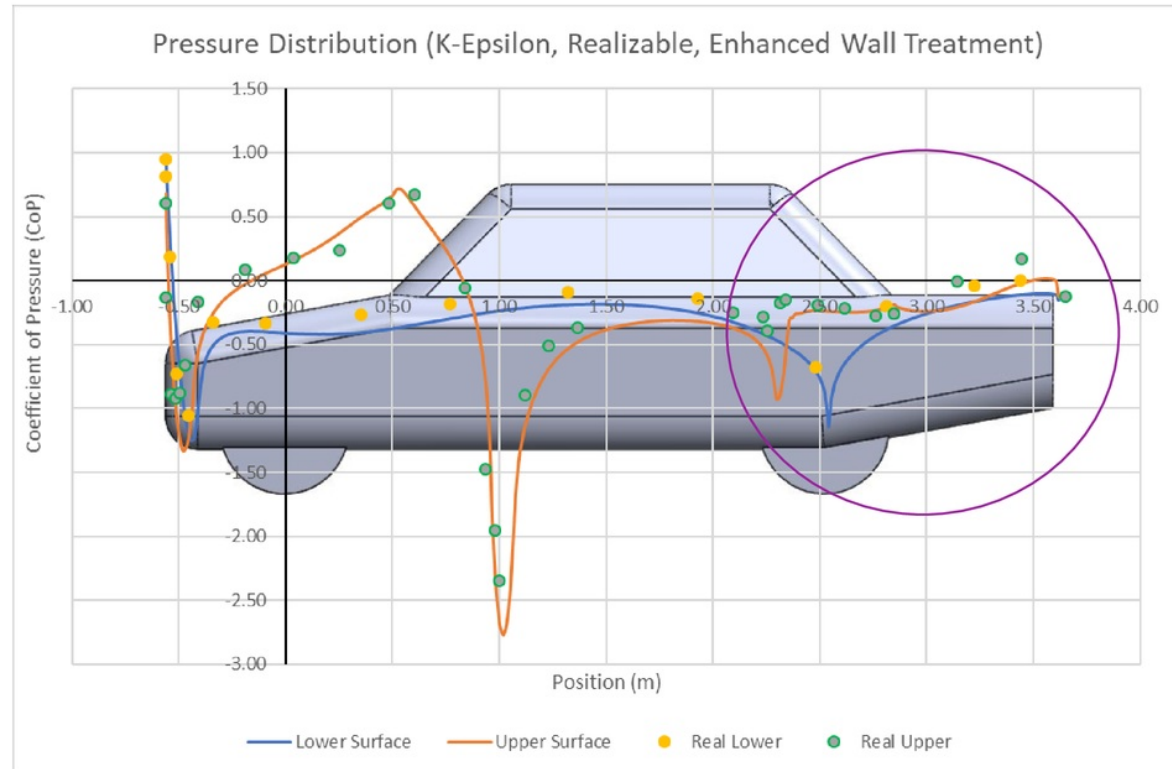
Regardless of whether the model produced close to datum C_l and C_d results, it showed variations in the produced pressure distribution across the body of the car. However, it does seem apparent that although the pressure distribution from CFD to wind tunnel does not line up, the overall pressure trend is followed, but significant variations do exist. Highlighted in the purple circle, it can be observed that the CFD case has dramatically over predicted the produced negative pressure when separation at the rear slant begins. Compared to that of the wind tunnel, however, the forward stagnation zone correlates well. This would suggest that the model struggles to accurately represent areas of separation and also struggles to model the correct boundary layer interactions.

Table 8 - Comparison of Max & Min CoP

	Lower CFD	Real Lower	Upper CFD	Real Upper
Max	0.96	0.95	0.73	0.67
Min	-1.27	-1.06	-2.78	-2.35

As a comparison, there is little difference in maximum coefficient of pressure (CoP), however, greater variance is noticeable within the minimum CoP.

4.3.2 K-Epsilon, Realizable, Enhanced Wall Treatment



Graph 12 - K-Epsilon, Realizable, Enhanced Wall Treatment Pressure Distribution

Evidentially, adjusting the $K\epsilon$ model has made very little visual impact on the pressure distribution. Even though the Enhanced Wall Treatment may have modelled the turbulent flow structures to be more comparable with other studies, it has had little impact on the static pressure, showing the same traits as Non-Equilibrium Wall Functions. Again, trends between the calculated and real pressure distributions on both upper and lower surfaces remain, with the rear of the car suffering with greater variation (purple circle). As the majority of the variation can be found within the location of a high separation area, this would therefore suggest that regardless of the Enhanced Wall Treatments superiority in these conditions, it still suffers when modelling and calculating in strong pressure gradients.

Table 9 - Comparison of Max & Min CoP

	Lower CFD	Real Lower	Upper CFD	Real Upper
Max	0.96	0.95	0.72	0.67
Min	-1.25	-1.06	-2.77	-2.35

Table 9 indicates the maximum and minimum CoP for both the CFD and wind tunnel data. In common with the Non-Equilibrium Wall Functions, the majority of the difference exists within the lower surface of the car. This would suggest that potentially, the turbulence model used has had little bearing on pressure distribution and potentially a mesh refinement is required.

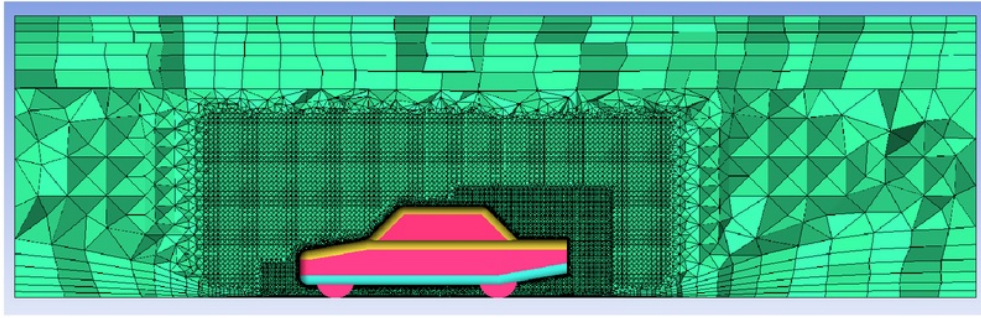
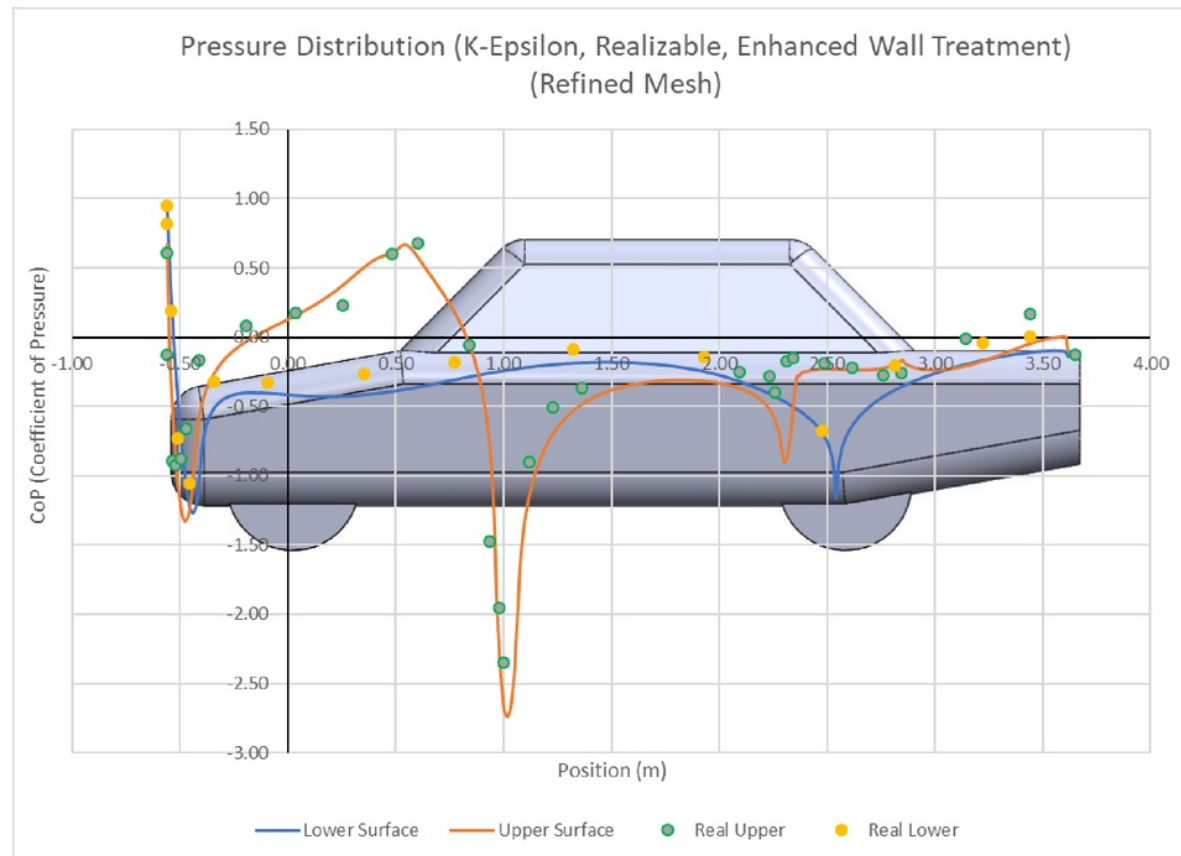


Figure 29 - Mesh Refinement

As previously stated, a turbulence model, and thus the Navier-Stokes equations will only predict and calculate as well as the build grid allows. As there will almost certainly be a significant amount of interaction between the lower surface of the car and the road, an additional density employing 0.05m elements was placed in this region in addition to the rear of the vehicle in order to allow the model to better predict the strong separation and wake region (Figure 39).



Graph 13 - K-Epsilon, Enhanced Wall Treatment (Refined Mesh) Pressure Distribution

Graph 13 visually displays very little change in pressure distribution compared to that seen in Graph 12. It is still noticeable, however, that the majority of variation occurs at the rear of the car where strong pressure gradients exist.

Table 10 - Comparison of Max & Min CoP

	Lower CFD	Real Lower	Upper CFD	Real Upper
Max	0.95	0.95	0.67	0.67
Min	-1.27	-1.06	-2.74	-2.35

The results in *Table 10* show very little change is apparent in the minimum CoP following a mesh refinement around the areas of most CoP variance. However, the maximum and minimum CoP on the upper surface at this point exactly match that provided by the wind tunnel data. Regardless of this, it is evident in *Graph 13* that the CoP does not correspond across the car's entire longitudinal axis.

4.4.0 Disputes

As the lower surface of the car and the road are in such close proximity of each other, a considerable amount of interaction occurs, as evident in *Figure 34*. As these two surfaces react with each other, vehicle ride height could have a dramatic affect on the pressure distribution along the lower surface. As the actual ride height of the MIRA car was not known, an estimation of ride height was used. If this height was set too low, it could explain why the CFD results consistently showed a lower pressure on the lower surface than that seen in the wind tunnel data.

The static pressures captured along the surface of the car within the wind tunnel would have employed the use of pressure taps set at a specific distance apart from each other. 31 pressure taps were used along the upper surface and only 15 along the lower surface. This difference becomes significant when compared to the number of virtual pressure taps the CFD software utilizes, employing 1211 along the top surface and 1047 along the lower surface. This would then permit large areas of static pressure to become obscured within the wind tunnel data when overlaid with the CFD data, manifesting in large differences.

5.0.0 Analysis of the Implementation of a Rear Wing on the MIRA Car

As basic aerodynamic characterisation of the MIRA has now been conducted with its original form, a rear wing was then applied to the geometry, together with end plates, in order to observe the effects this aerodynamic device has upon the car.

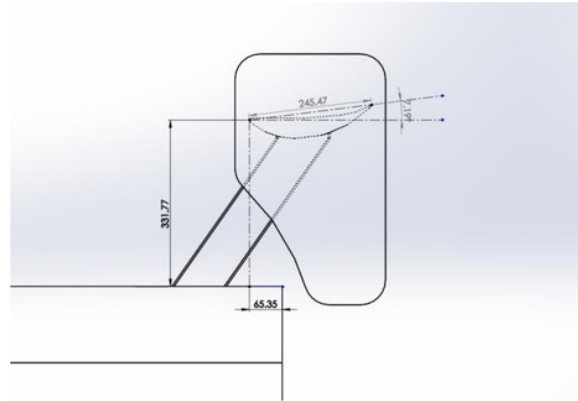


Figure 30 - MIRA Car Rear Wing Geometry

The rear wing was set at a height of 331.37mm, with an overhang of 65.35mm and an angle of attack of 7.19°. Ride height remained the same from standard geometry to modified geometry.

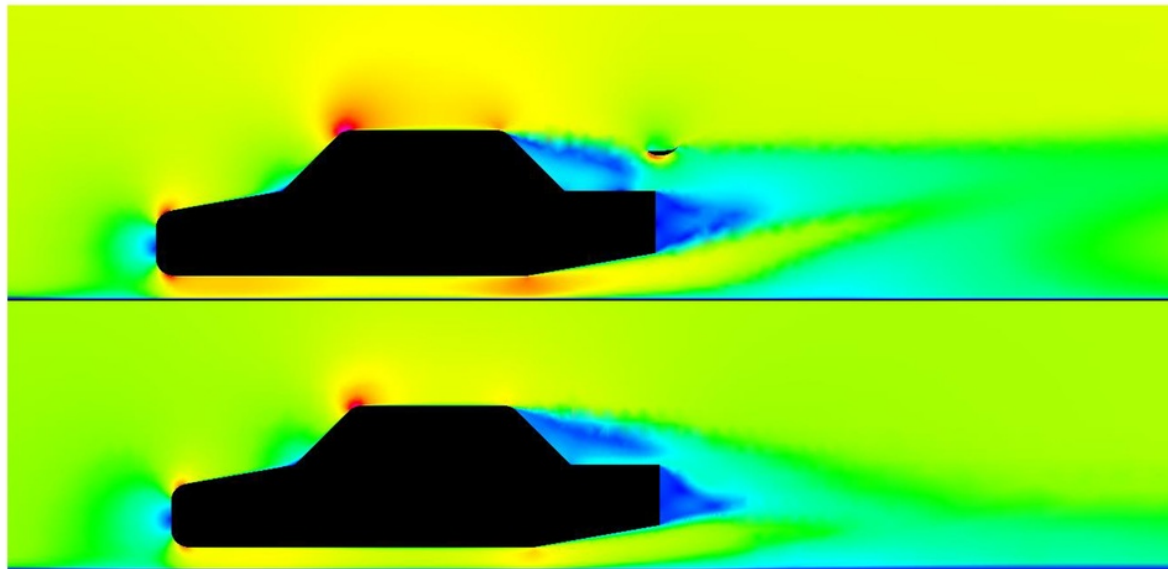


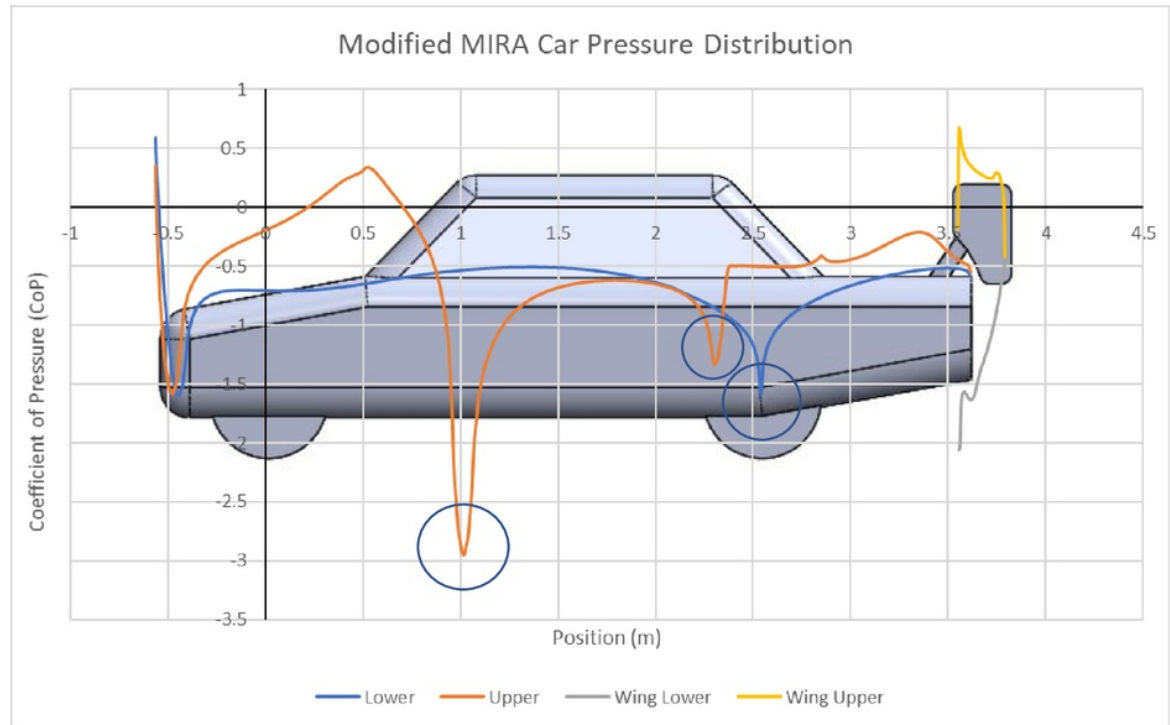
Figure 31 - Comparison (With & Without Wing) (Velocity Contours)

Applying a rear wing to the geometry has had an immediate affect on the flow field around the car (Figure 43). It is apparent that the inclusion of a rear wing has enabled the diffuser section of the car to work much harder. The upwash generated from the wing is now encouraging a greater amount of upwash from the diffuser. The low pressure generated between the underside of the wing and rear of the vehicle generates a much lower pressure field, encouraging the air under the car to travel at a greater velocity. This effect is also evident across the top surface of the car, with a visible increased velocity over the leading edge of the roof line. (McBeath, 2000)

Table 11 - Results from Standard & Modified Geometries

	Cl	Cd
Standard Geometry	-0.0889	0.3129
Modified Geometry	-0.5871	0.2544

Introducing a wing has created over six times the negative lift originally produced, and has reduced the drag by 0.0585. The reduction in drag is as a result of the wing beginning to re-energise the wake region, allowing a greater flow attachment.



Graph 14 - Modified MIRA Car Pressure Distribution

The affect of the introduction of a rear wing is apparent when examining Graph 14. Each circled peak indicates a considerable drop in pressure when compared to the same points on Graph 13. The reduction in pressure points to a higher velocity being produced by the energising effects of the wing.

Table 12 - MIRA Car Aerodynamic Characteristics

	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)
Standard MIRA Car	140.98	40.06	4.1	0.28	72.24	28.94	11.12
Modified MIRA Car	114.62	264.53	27.0	2.31	-24.26	-64.18	328.71

The introduction of the rear wing has been a powerful addition. As can be observed in Table 12, the balance of negative lift has been moved rearward, with the front now generating 64.18N of lift compared to that of its standard geometry which produces 28.94N of negative lift. As the modified car now has a centre of pressure so far rearwards, it is predicted that if the car was real, it would be almost undrivable, requiring further adjustments if this were an aerodynamic optimisation study.

5.0.0 Conclusion

It remains to be seen how accurately the wind tunnel data generated for the MIRA car can be replicated within a CFD study. From the findings produced in this report, overall C_l and C_d figures can be generated within a 6% difference, however, the pressure distributions vary greatly.

Computational fluid dynamics is an important tool in the generation of aerodynamic devices, however, its sole dependency is still in question and it would be recommended that wind tunnel testing was used to correlate with CFD usage.

Considering everything, the areas focussed on within this report could be evaluated in further depth to reach the end goal of validating the input variables used. However, these input variables will vary from case to case, depending on the area of study and the amount of allowable computational time and power.

However, once a baseline case was established, the software generated the expected results when a modification was applied to the geometry in question. Therefore, CFD can be used as a constructive and evaluative tool.

It has to be emphasised that the settings used in this case would unlikely be transferable to anything other than a study of flow around a bluff body.

6.0.0 References

- Ahmad, N. E., Abo-Serie, E. & Gaylard, A., 2010. *Mesh Optimization for Ground Vehicle Aerodynamics*, Coventry: Coventry University.
- Al-Witry , A. & Es-Saheb, M., 2013. *Benchmarking the Performance of the ANSYS-FLUENT Standard k- ϵ Turbulence Model in Fluid Flow and Heat Transfer Predictions for Complex Flows around Circular Pin-Fins Using Various near Wall Functions*, Riyadh: King Saud University.
- Ansys Inc, 2006. *Modeling Turbulent Flows*. [Online]
Available at: www.fluentusers.com
[Accessed 29 11 2018].
- Ansys Inc, 2009. *ICEM CFD Introductory Course*. [Online]
Available at: slideplayer.com/slide/13146156/
[Accessed 1 12 2018].
- Ariff, M., Salim, S. M. & Cheah, S. C., 2009. *WALL Y+ APPROACH FOR DEALING WITH TURBULENT FLOW OVER A SURFACE MOUNTED CUBE: PART 1 – LOW REYNOLDS NUMBER*, Selangor: University of Nottingham (Malaysia Campus).
- Bakker, A., 2006. *Lecture 10 - Turbulence Models*. [Online]
Available at: <http://www.bakker.org/dartmouth06/engs150/10-rans.pdf>
[Accessed 5 12 2018].
- Bakker, A., 2006. *Lecture 7 -Meshing _ Applied Computational Fluid Dynamics*. [Online]
Available at: www.bakker.org
[Accessed 1 12 2018].
- Buscariolo, F. F. et al., 2016. *Analysis of Turbulence Models Applied to CFD Drag Simulations of a Small Hatchback Vehicle*, Sao Paulo: SAE International.
- Costa, B., 2015. *Studying the Airflow Over a Car Using an Ahmed Body*. [Online]
Available at: www.comsol.com/blogs/studying-the-airflow-over-a-car-using-an-ahmed-body/
[Accessed 27 11 2018].
- Franck, G., Nigro, N., Storti, M. & D'Elia, J., 2009. *Numerical simulation of the flow around the ahmed vehicle model*, Santa-Fe: Bahía Blanca.
- Frei, W., 2017. *Which Turbulence Model Should I Choose for My CFD Application?*. [Online]
Available at: uk.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application/
[Accessed 28 11 2018].
- Jouse, F. M., 2018. *What is y+ (yplus)?*. [Online]
Available at: <https://www.simscale.com/forum/t/what-is-y-plus/82394>
[Accessed 12 12 2018].
- Kobeissi, R., 2017. *CFD ANSYS Tutorial - Y+ value in mesh for flow over flat plate | Fluent*. [Online]
Available at: <https://www.youtube.com/watch?v=kecgPw-VmPU>
[Accessed 12 12 2018].

Komarla, S., 2016. *In CFD, why prism layers are needed?*. [Online]
Available at: www.quora.com/In-CFD-why-prism-layers-are-needed
[Accessed 2 12 2018].

Leap CFD Team, 2012. *Tips & Tricks: Turbulence Part 2 - Wall Functions and Y+ requirements*. [Online]
Available at: <https://www.computationalfluidynamics.com.au/tips-tricks-turbulence-wall-functions-and-y-requirements/>
[Accessed 12 12 2018].

Linfield, Ph.D., P.E., K. W. & Mudry, P.E., R. G., 2008. *Pros and Cons of CFD and Physical Flow Modeling*, Livonia:
Airflow Sciences Corporation.

Llanfrit, M., 2005. *Best practice guidelines for handling Automotive External Aerodynamics with FLUENT*, Birkenweg :
Fluent Deutschland GmbH.

Mali, V., 2016. *Basics of Y Plus, Boundary Layer and Wall Function in Turbulent Flows*. [Online]
Available at: www.learncax.com/knowledge-base/blog/by-category/cfd/basics-of-y-plus-boundary-layer-and-wall-function-in-turbulent-flows
[Accessed 12 12 2018].

McBeath, S., 2000. *Competition Car Downforce*. 1 ed. Nr Yeovil: Haynes.

Mokri's, D., 2012. *Mesh Generation and Optimisation*, Prague: Mathematical Institute of Charles University.

Palaskar, P. M., 2016. *Effect of Side Taper on Aerodynamics Drag of a Simple Body Shape with Diffuser and without Diffuser*, Pune: SAE International.

Qin, N., 2003. *CFD for Better Understanding of Wind Tunnel Tests*, Sheffield: The University of Sheffield.

Reynolds, O., 1883. An Experimental Investigation of the Circumstances Which Determine Whether the Motion of Water Shall be Direct or Sinuous, and the Law of Resistance in Parallel Channels. *Philosophical Transactions of the Royal Society of London*, Volume 174, pp. 935-982.

Smith, T., 2017. *When the going gets tough, the tough use ANSYS for CFD Meshing*. [Online]
Available at: <http://www.padtinc.com/blog/the-focus/when-the-going-gets-tough-the-tough-use-ansys-for-cfd-meshing>
[Accessed 4 12 2018].

Sodja, J., 2007. *Turbulence models in CFD*, Ljubljana : University of Ljubljana .

Subramanian, R. S., 2014. *Reynolds Number*, New York: Clarkson University.

Utyuzhnikov, S. V., 2004. Generalized wall functions and their application for simulation of turbulent flows. *INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN FLUIDS*, 1(1), pp. 1-6.

Versteeg, H. K. & Malalasekera, W., 2007. *An Introduction to Computational Fluid Dynamics - The Finite Volume Method*. 2 ed. Essex: Pearson Education Limited.

www.autodesk.com, 2018. *Two Equation Turbulence Models (TKE & TED)*. [Online]
Available at: <http://help.autodesk.com/view/SCDSE/2019/ENU/?guid=GUID-61C4EB55-362C-48A0-8B22-20F9148D190D>
[Accessed 12 12 2018].

www.comsol.com, 2017. *Finite Element Mesh Refinement*. [Online]

Available at: www.comsol.com/multiphysics/mesh-refinement

[Accessed 1 12 2018].

www.grc.nasa.gov, 2015. *Euler Equations*. [Online]

Available at: <https://www.grc.nasa.gov/www/k-12/airplane/eulereqs.html>

[Accessed 5 12 2018].

www.horiba-mira.com, 2018. *Full Scale Aerodynamic Wind Tunnel*. [Online]

Available at: [www.horiba-mira.com/our-services/full-scale-wind-tunnel-\(fswt\)](http://www.horiba-mira.com/our-services/full-scale-wind-tunnel-(fswt))

[Accessed 27 11 2018].

www.pointwise.com, 2012. *Advancing Front Mesher and Faster I/O Coming to Pointwise*. [Online]

Available at: <https://www.pointwise.com/theconnector/2012-November/Advancing-Front-Mesher-Faster-IO-Coming-Pointwise.html>

[Accessed 3 12 2018].

www.sharcnet.ca, 2015. *Orthogonal Quality*. [Online]

Available at: https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/wb_msh/msh_orthogonal_quality.html

[Accessed 4 12 2018].

www.sharcnet.ca, 2015. *Quick (Delaunay)*. [Online]

Available at: https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/icm_help/imesh_tetramixed

[Accessed 3 12 2018].

www.sharcnet.ca, 2015. *Robust (Octree)*. [Online]

Available at: www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/icm_help/imesh_tetramixed

[Accessed 3 12 2018].

www.sharcnet.ca, 2015. *Skewness*. [Online]

Available at: https://www.sharcnet.ca/Software/Ansys/17.0/en-us/help/wb_msh/msh_skewness.html

[Accessed 4 12 18].

www.simscale.com, 2018. *What are the Navier-Stokes Equations?*. [Online]

Available at: <https://www.simscale.com/docs/content/simwiki/numerics/what-are-the-navier-stokes-equations.html>

[Accessed 5 12 2018].

www.simscale.com, 2018. *What is the Reynolds Number?*. [Online]

Available at: <https://www.simscale.com/docs/content/simwiki/numerics/what-is-the-reynolds-number.html>

[Accessed 5 12 2018].