

Aerodynamic Vehicle Testing and analysis

by Thomas Edds

FILE	4206_THOMAS_EDDS_AERODYNAMIC_VEHICLE_TESTING_AND_ANALYSIS IS_128450_31090863.DOCX (19)		
TIME SUBMITTED	28-JAN-2019 03:37 AM (UTC+0000)	WORD COUNT	4000
SUBMISSION ID	99894391	CHARACTER COUNT	25250

Aerodynamic Vehicle Testing and analysis

University of Wales Trinity Saint David

School of Engineering

Thomas Edds

Date: 25/01/2019

Module Name and Code: M6X00408

Lecturer's Name: Tim Tudor

Contents

1	Introduction	1
2	Background Research.....	Error! Bookmark not defined.
2.1	Viscosity.....	2
2.2	Boundary Conditions.....	3
3	Domain, Mesh setting and Turbulence model Study	4
3.1	Study Domain and Mesh Settings	4
3.2	Domain and Geometry Preparation.....	7
3.2.1	Geometry.....	7
3.2.2	Domain.....	8
3.2.3	Front Domain Length.....	8
3.2.4	Rear Domain Length.....	13
3.3	Meshing.....	16
3.3.1	Scale Factor.....	16
3.3.2	Prism Layers	16
3.3.3	Mesh Types.....	17
3.4	Turbulence Models.....	18
4	Solving and analysing optimum Settings	20
4.1	First Run.....	20
4.2	Refinement	27
4.2.1	Additional named faces.....	27

4.2.2	Additional Density boxes.....	29
5	Geometry Change.....	31
6	Conclusions	34
7	Bibliography	35

Figure 1 - Named Face Selections	8
Figure 2 - Cl as Domain in Front of Axle Changes	9
Figure 3 - Cd as Domain in Front of Front axle Changes.....	10
Figure 4 - Domain Front Changing.....	11
Figure 5 - Graph Showing Percentage Difference from MIRA Reference Data to collected data as domain size changes	12
Figure 6 - Distance behind Vehicle Changing.....	13
Figure 7 - Vortices in Wake Region	14
Figure 8 - Cd % From MIRA against Domain length	14
Figure 9 - Graph Shown Mesh Dependency	16
Figure 10 - Cd and Cl % Difference as Mesh Type Changes.....	18
Figure 11 - Graph showing Difference in Cd and Cl for different Turbulence Models	19
Figure 12 - Showing Optimum Mesh	24
Figure 13 - Results from Optimum Settings.....	25
Figure 14 - Coefficient of Pressure Chart for the upper surface.....	26
Figure 15 - Coefficient of Pressure for the lower Surface	27
Figure 16 - New Named Face Selection	28
Figure 17 - Named faces Part Mesh Parameter's.....	28
Figure 18 - Visualisation of pressure and velocity contours from named faces meshing strategy	29
Figure 19 - Mesh created using Density Boxes	30
Figure 20 - Visualisation Produced From Density Box Meshing Strategy	31
Figure 21 - Estate Model.....	32
Figure 22 - Estate Car Visualisation	33

Table 1- Showing Meshing and Solver setting for dependency studies	6
Table 2 - Showing Setting derived from the dependency study	22
Table 3 - Coefficient of Drag and Lift	23
Table 4 - Results from New Named Faces Run.....	29
Table 5 - Data Collected from the Density Box Meshing strategy	30
Table 6 - Coefficient of lift and drag data for estate car	32

1 Introduction

Computational fluid dynamic (CFD) involves the analysis of system involving fluid flow, heat transfer and other associated properties using computer-based simulations. The use of CFD for production car and competition car aerodynamics has now been around for several years and software has seen produce that is usable on the home PC.

CFD work by solving the basic underlying principle equations of:

- The Conservation of Mass
- The Conservation of momentum
- The Conservation of energy

The conservation of momentum states that the rate of changes of a body linear momentum is directly related to the force acting on it, so as the air hits a surface its direction changes, hence its velocity changes and the surface feels a force. This force contributes towards the overall aerodynamic load. The conservation for mass requires that air enter a system will equal the air leaving and together with Bernoulli's equation which states the overall constant pressure changes with respect to the velocity hence, changes in velocity and pressure changes can be calculated. Together with the conservation of energy theoretically the overall aerodynamic load can be calculated but viscosity poses a problem (McBeath, 2015).

$$\rho V_1 A_1 = \rho V_2 A_2 = \rho V_3 A_3$$

Equation 1 - Conservation of Mass

$$F = (m_2 V_2 - m_1 V_1) / t$$

Equation 2 - Conservation of Momentum

$$P_s + \frac{1}{2} \rho V^2 = \text{"Total Pressure"}$$

1.1 Viscosity

When air passes over a body it can be said to be traveling in thin layers parallel to the body. In question, the layer closest to the surface will “grab on” or “stick” to the surface. Lower the velocity of that layer and subsequently the layer next to it and so on until the force is dissipated. This area from the surface to the last layer that feels the velocity gradient is known as the boundary layer. This area steals kinetic energy from the moving air body and this can lead to the boundary layer thickening up and causing turbulent flow.

When all the air particles are moving parallel to a surface and moving in the same direction, the flow is said to be laminar. If on the other hand particles become erratic and trace out non-parallel paths the flow is said to be turbulent. This turbulent effect can occur over distance as the viscosity gradually removes energy from the air building up the boundary layers causing turbulent flow. The thickness of a boundary layer across a car depends on how fast the vehicle is traveling and the distance along the body of the vehicle. Typically starts a few millimetres thick at the front of the vehicle and thickening to a few centimetres at the rear of the vehicle (McBeath, 2015).

Navier-Stokes equations derived from the basic underlying principles of mass, momentum and energy are used as the governing equations in CFD. They consist of the continuity equation, the momentum equation and the energy equation and used to model the viscous forces within the boundary layer and alongside them so-called turbulence models are used to bolster the calculation done by the Navier-Stokes equations.

1.2 Boundary Conditions

Boundary conditions are used to specify the way different parts of the model act on the air flow. They are known as Dirichlet or von Neuman condition, Dirichlet are used when specific function are required such as the inlet velocity of a wind tunnel, while von Neuman condition are where there are zero gradients specifically symmetry boundaries. No slip wall conditions are used to specific wall boundaries like the cars surface and also outflow conditions are used.

The report details the modelling process of an external aerodynamic study of the MIRA reference car within the MIRA reference tunnel using the computational fluid dynamics software FLUENT. The geometry was set up in the fluent design modeller then dependences study were carried out on the effects of the domain size changing, mesh scaling, prism layers, mesh types and turbulence models. The details of each study are laid out in the sections below.

The results from these studies were then used to create an initial domain and meshing strategy. Solutions were then carried out using this strategy and further refinements were made.

2 Domain, Mesh setting and Turbulence model Study

2.1 Study Domain and Mesh Settings

Table show the mesh settings and solver setting for the dependency study, the setting will be at each step change with regard to what element is being studied.

<u>Global Mesh Parameters</u>					
<u>Max Mesh</u>	<u>Scale</u>	<u>Shell Mesh</u>		<u>Volume Mesh</u>	
<u>Size</u>	<u>Factor</u>	<u>Para</u>	-	<u>Para</u>	-
		Type	Method	<u>Type</u>	<u>Tetra Mesh</u>
1m	1	All Tri	Patch independent	Tetra/Mixed	Robust (octree)
<u>Prism Mesh Para</u>					
Growth	Initial		Number of		
Law	Height	Height Ratio	Layers	Total Height	
Exponential	0	1.2	3	0	

Parts Mesh Parameters					
MIRA Bottom			MIRA TOP		
Prism	Max Size	Prism Layers	Prism	Max Size	Prism Layers
Y	0.05	3	Y	0.05	3
MIRA FW			MIRA RW		
Prism	Max Size	Prism Layers	Prism	Max Size	Prism Layers

Y	0.02	3	Y	0.02	3
---	------	---	---	------	---

<u>Fluent Settings</u>		
Viscous	Materials	
	Fluid	Solid
Realizable K-e, Non-EQ		
Wall Fn	air	Aluminium
<u>Cell Zone Condition</u>		
Zone	Type	
Body	Fluid	
-	-	-
<u>Boundary Conditions</u>		
Floor	Inlet	INT_Body
	Velocity inlet (20m/s in	
Wall	X)	Interior

MIRA_Top	Outlet	Symmetry
Wall	Pressure Outlet (0)	Symmetry
MIRA_Bottom	MIRA_FW	MIRA_RW
Wall	Wall	Wall
Topside		
Wall		
<u>Reference Values</u>		<u>Solution Methods</u>
Compute From	Area	Scheme
Inlet	0.9164265m ²	Coupled
<u>Solution Controls</u>	<u>Explicit Relaxation Factors</u>	
Flow Courant Number	Momentum	Pressure
50	0.35	0.65
-	-	
<u>Under Relaxation Factors</u>		
Density	Body Forces	Turbulent Kinetic Energy
1	1	0.5
Turbulent Dissipation Rate	Turbulent Velocity	
0.5	1	

Table 1- Showing Meshing and Solver setting for dependency studies

2.2 Domain and Geometry Preparation

2.2.1 Geometry

In today's age most cars are analysed using a CFD software package, and with geometry close to the actual production geometry. The geometry of production car can often be quite complex especially in areas such as the underbody, this can cause issues during the meshing process which will lead to undesirable solutions or further convergence problems. It is advisable to assess the chosen geometry before a mesh strategy is chosen to identify any potential problematic areas such as sharp angles and decide whether the geometry can be simplified. An example would be smoothing off headlamp area to remove sharp edges often around the headlamp housing area. Modifications to the geometry should be kept to as low as possible but are in some cases unavoidable. Care should be taken when creating simplifications to ensure a closed geometry. The surfaces should be kept as smooth as possible to allow for the growth of prism layers, additional faces can be made to reduce sharp angles such in places like where the wheel meet the ground as this is often an area of skewed cells. Once the geometry has been assessed and simplified the next step would be to check the geometry quality's using the repair function within ICEM Meshing software (Lanfrut, 2005).

The selected geometry from this study which represents the MIRA reference car, this is actually quite a simple geometry with smooth surfaces all around which will help in the development of the meshing strategy. The geometry could be simplified by added contact plates by the wheel to floor contact areas to reduce skewed cells in this area.

To assist with the mesh generation it can be helpful to name faces on the selected geometry these faces can then be mesh as individual parts, also boundary conditions can be set up for each face and this aids with the visualisation in the post processor. The named face for the geometry in this study are shown in figure 1.

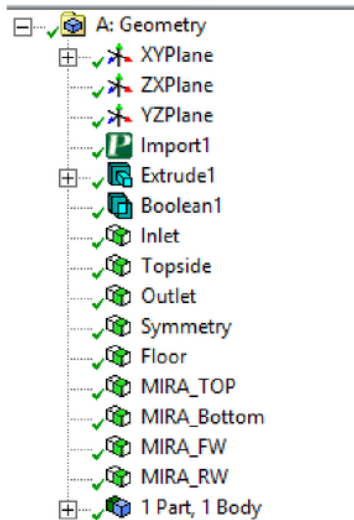


Figure 1 - Named Face Selections

2.2.2 Domain

The domain in this study will be based on the MIRA wind tunnel as the reference data supplied was collected from there. The wind tunnels it's self is 7.92m wide, 4.42m high and 15m long. Previous studies suggest the computational domain should extend at least 3 car lengths in front and 5 car lengths to the rear (Lanfrit, 2005). So the decision was made to not use the MIRA length as a dimension in the computational domain but to investigate into the optimum length. The investigation was carried out in 2 parts firstly by studying how the distance from the inlet face to the front axle of the vehicle effects the solution and then secondly the distance from the outlet to the front axle of the vehicle effects the solutions. The width and height of the domain was kept to the MIRA wind tunnel size for this study.

2.2.3 Front Domain Length

The figures below show the results from the domain change investigations. Figure one shows how the coefficient of lift changes with the change of distance from front axle to the inlet

face of the domain. It can be seen from figure 1 that from 2m to 20m in front of the front axle the coefficient of lift is found to be between -0.15 and -0.1 this suggests that the distance in front of the car should be kept between 2m and 20m.

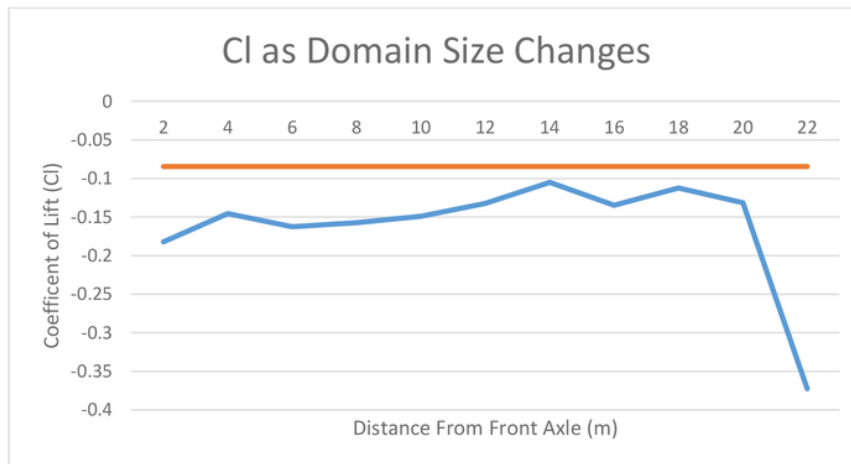


Figure 2 - Cl as Domain in Front of Axle Changes

Figure 2 shows how the coefficient of drag (C_d) changes in respect to the domain size changing, this also shows the optimum domain front length is between 4m and 22m. These results were then loaded into the post processor for further investigations.

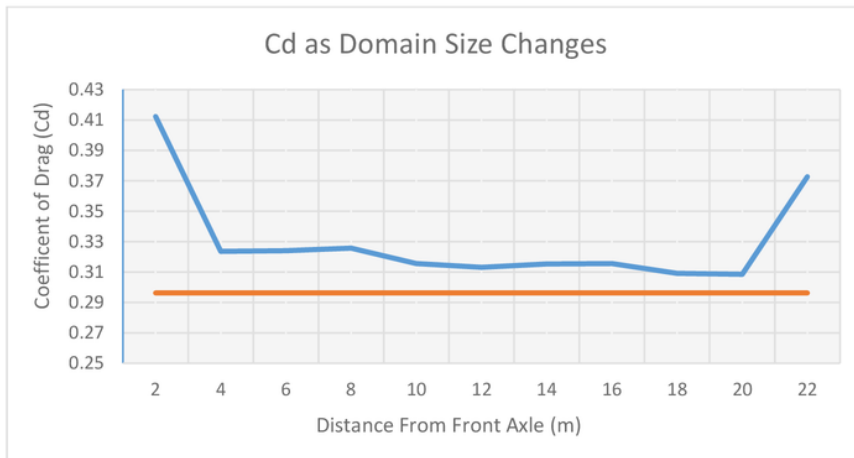


Figure 3 - Cd as Domain in Front of Front axle Changes

A graphical visualisation was produce for all the test runs showing the change in velocity ion the symmetry plane and pressure contour on the car surfaces and floor. The visualisations show that when the front length is at 2m the pressure contours on the front of the vehicle and the floor in front of the vehicle are not very well represented. Four examples are shown in figure 3. As this length is increased it can been seen that there is an improvement in the representation of the pressure contours and velocity contours in front of the vehicle and also behind.

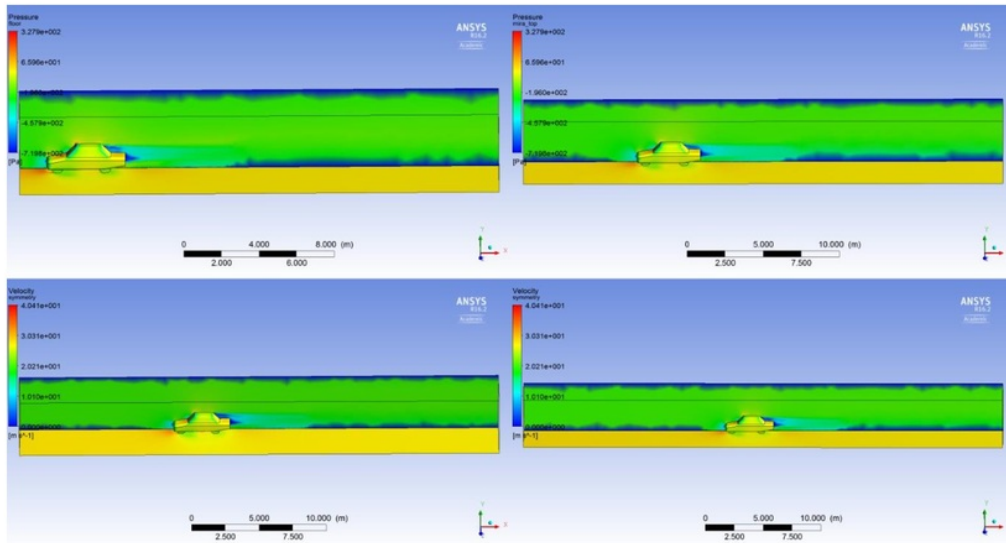


Figure 4 - Domain Front Changing

The length was then investigated between 12m and 14m in 0.5m steps, figure 4 shows a graph representing the percentage changes as the domain size changes. The decision was then made to set the length in front of the front to 12.5m as this captures the pressure and velocity contours well and the difference from the reference data is one of the closest but also matches what is suggested in other studies (Lanfrit, 2005) (Ahmad, Abo-Serie, & Gaylard, 2010).

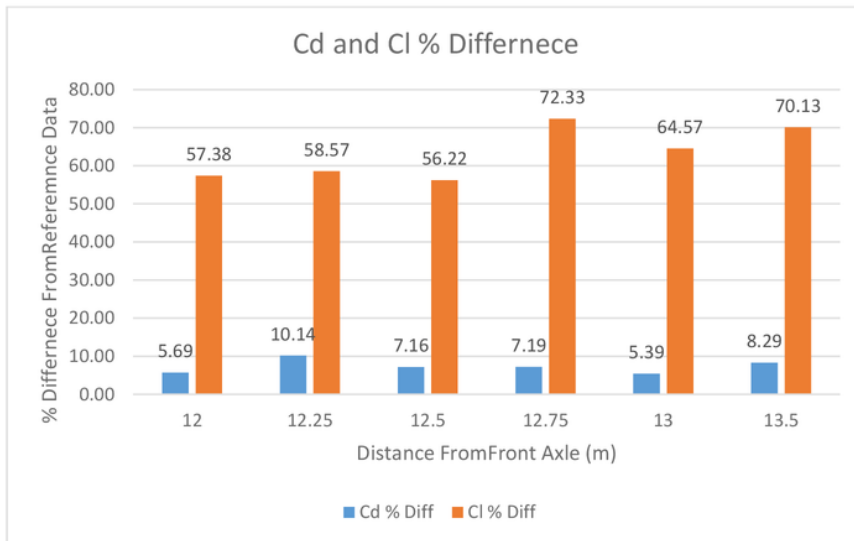


Figure 5 - Graph Showing Percentage Difference from MIRA Reference Data to collected data as domain size changes

2.2.4 Rear Domain Length

The length of the rear of the domain was then evaluate in a similar fashion, Figure 5 shows the visualisation of pressure and velocity contours at different domain length. It can be seen that the short domain length does not represent the forces with in the wake region. As this length is increased more of the wake region becomes more visible.

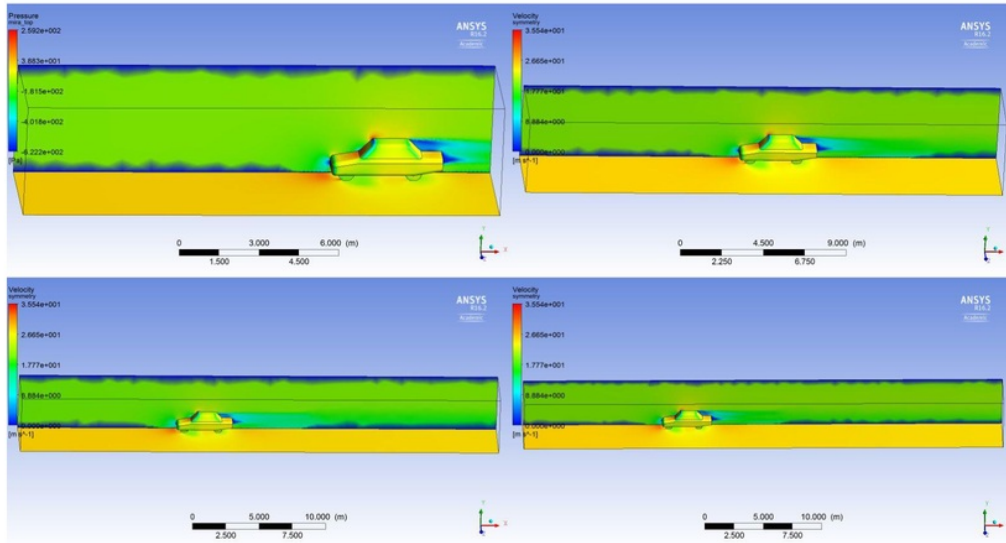


Figure 6 - Distance behind Vehicle Changing

Furthermore figure 6 shows how the vortices in the wake region are not captured with the short domain length and how the vortices are captured with the longer domain length.

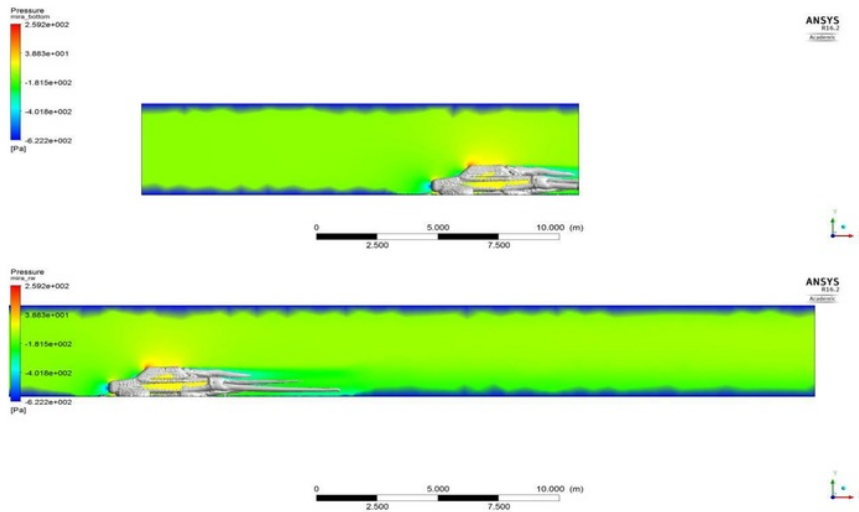


Figure 7 - Vortices in Wake Region

A graph (figure 7) was also produce showing how the Cd changes with the increased length of the rear of the domain.

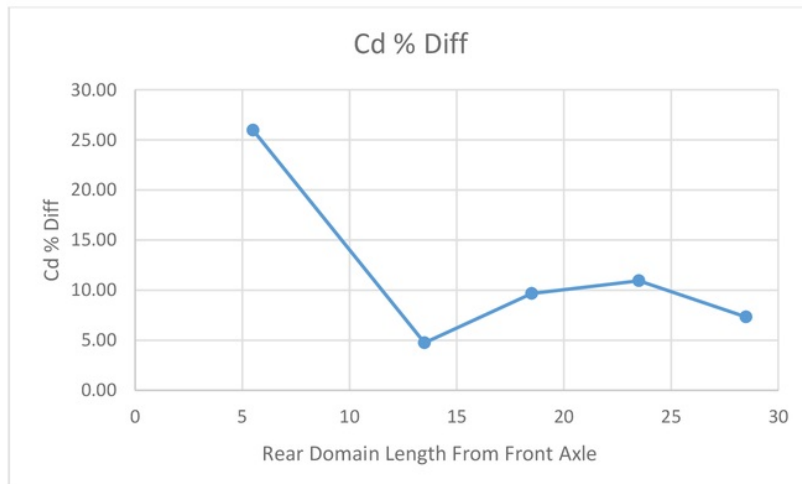


Figure 8 - Cd % From MIRA against Domain length

Following these investigations the decision was made to set the rear domain length to be 23.5m from the front axle. At this length the software is able to capture the wake region

forces well and adequately predict the C_d . Also this ties up with other studies as this around 5 car lengths behind the vehicle (Lanfrit, 2005) (Ahmad, Abo-Serie, & Gaylard, 2010).

2.3 Meshing

2.3.1 Scale Factor

Within ICEM meshing software the global scale factor setting can be used to alter all elements within a mesh by a certain factor. This was used to study how the results are effected by the number of elements with a mesh. Using the All Tri patch independent mesh type and Octree method the graph show in figure 8 was produces. This clearly a shows that when the number of elements is increased the accuracy of the results is increased. This is also what is suggested in previous studies (Ahmad, Abo-Serie, & Gaylard, 2010) (Lanfrit, 2005).

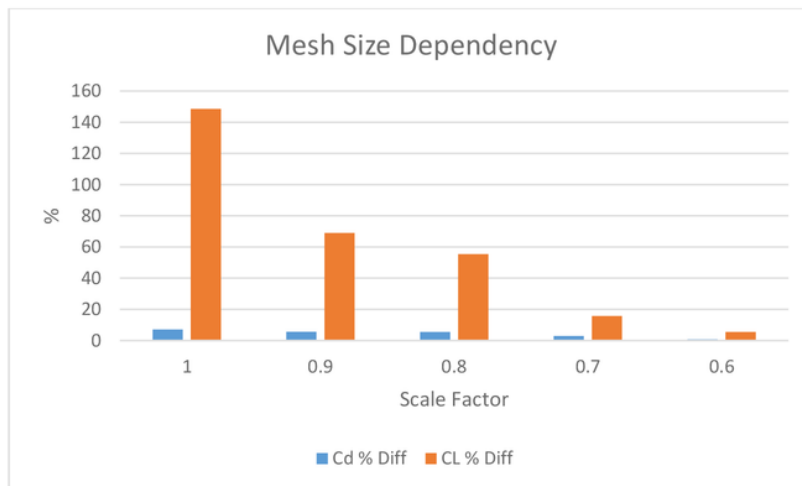


Figure 9 - Graph Shown Mesh Dependency

2.3.2 Prism Layers

Prism layers are used to aid in the prediction of forces within the boundary layer. The boundary layers are formed due to the viscosity of the air acting on the body of an object. The viscosity of air causes it to 'stick' to surfaces, this viscosity causes a resistance to a body moving through it. The air flow around a body can be laminar, where the flow remains stuck to the body or turbulent whereby the viscous forces cause the air flow become separated from the body (Lanfrit, 2005).

Using the same settings as in the previous section a study was carried out adding a prism layer each to see how the result were effected. Figure 9 shows a graph from the results collected. The graph shows that as prism layers are added the Cd and Cl prediction show more accuracy.

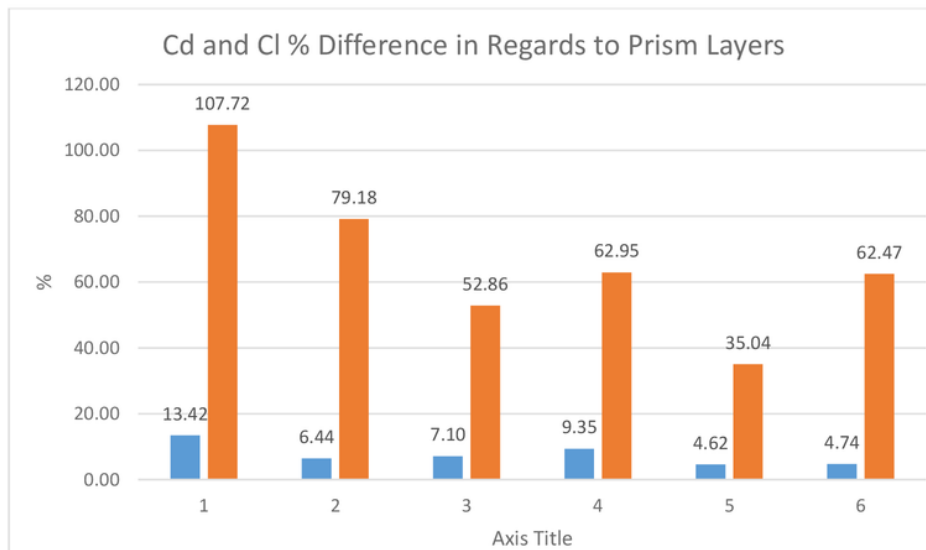


Figure - Cd and Cl % Differences as prism layers are added

2.3.3 Mesh Types

The mesh types were then investigated using the domain size as stated in the previous section. Four different meshing types were investigated, the hex-dominant, the tetra-mixed Delaunay and octree and also the polyhedral. The results are shown in the graph below in figure 11

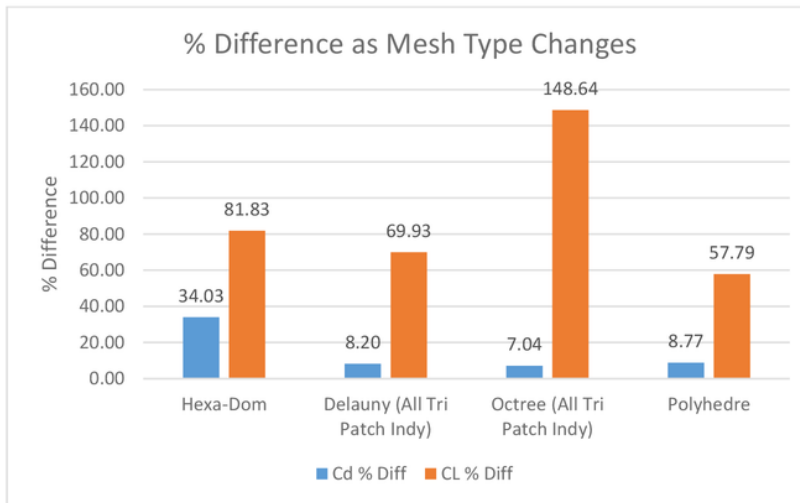


Figure 10 - Cd and Cl % Difference as Mesh Type Changes

2.4 Turbulence Models

As stated in section 2.1 turbulence models are used to bolster the equation governing CFD there a range of different turbulence models availed. In this study the Standard K-epsilon No Equilibrium wall function, the realisable K-epsilon Non Equilibrium wall functions and the Reynolds stress equation model. The K-epsilon model is uses two extra equation to represent talent flow and is one of the most common turbulence models available. The first of the two equations represents kinetic energy (k) and the second represents turbulent dissipation. There are two major of the K-epsilon models the standard and Realisable, both this types have been tested in this study. The Reynold stress model use a series of seven transport equation to model a 3D fluid problem and is the one of the most elaborate models available. Figure 11 shows a graph displaying the finds of the study (Bardina, Huang, & Coakley, 1997).

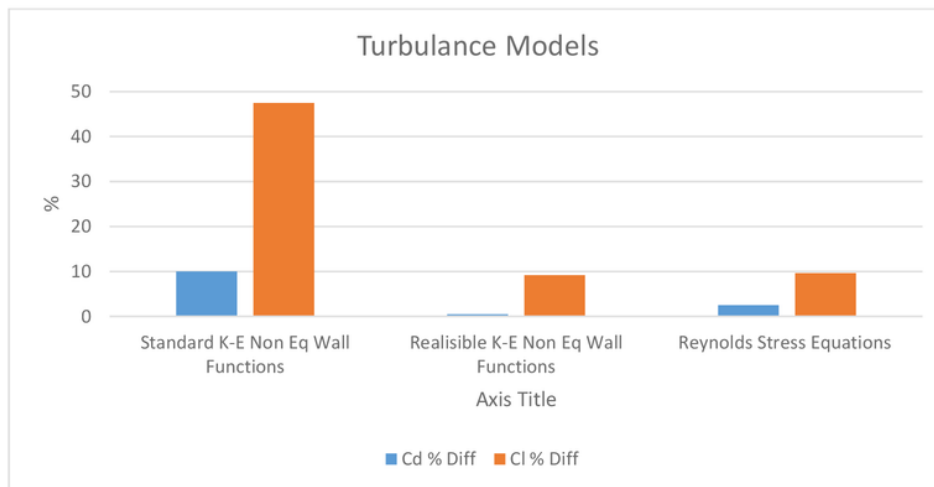


Figure 11 - Graph showing Difference in Cd and Cl for different Turbulence Models

3 Solving and analysing optimum Settings

3.1 First Run

Following on from the study in section 3 it was decided to use the following settings used to solve the aerodynamic flow round the MIRA reference car in the MIRA wind tunnel.

<u>Global Mesh Parameters</u>		
<u>Max Mesh Size</u>	<u>Scale Factor</u>	<u>Shell Mesh Para</u>
		Type
1m	0.7	All Tri
<u>Prism Mesh Para</u>		
Growth Law	Initial Height	Height Ratio
Exponential	0	1.2
Number of Layers	Total Height	
5	0	
<u>Volume Mesh Parameters</u>		
-		-
Method	Type	Tetra Mesh
Patch independent	Tetra/Mixed	Delaunay
-	-	
<u>Parts Mesh Parameters</u>		
MIRA Bottom		
Prism	Max Size	Num Layers
Y	0.05	5
MIRA FW		

Prism	Max Size	Num Layers
Y	0.02	5
MIRA TOP		
Prism	Max Size	Num Layers
Y	0.05	5
MIRA RW		
Prism	Max Size	Num Layers
Y	0.02	5
<u>Fluent Settings</u>		
Viscous	Materials	
	Fluid	Solid
Realizable K-e, Non-EQ		
Wall Fn	air	Aluminium
<u>Cell Zone Condition</u>		
Zone	Type	
Body	Fluid	
-	-	-
<u>Boundary Conditions</u>		
Floor	Inlet	INT_Body

Wall	Velocity inlet (20m/s in X)	Interior
MIRA_Top	Outlet	Symmetry
Wall	Pressure Outlet (0)	Symmetry
MIRA_Bottom	MIRA_FW	MIRA_RW
Wall	Wall	Wall
Topside		
Wall		
<u>Reference Values</u>		<u>Solution Methods</u>
Compute From	Area	Scheme
Inlet	0.9164265m ²	Coupled
<u>Solution Controls</u>	<u>Explicit Relaxation Factors</u>	
Flow Courant Number	Momentum	Pressure
50	0.35	0.65
-	-	
<u>Under Relaxation Factors</u>		
Density	Body Forces	Turbulent Kinetic Energy
1	1	0.5
Turbulent Dissipation Rate	Turbulent Velocity	
0.5	1	

Table 2 - Showing Setting derived from the dependency study

Table 3 shows the coefficient of drag and lift predicted by the model and the difference from the MIRA reference data

Model		MIRA DATA		% Difference	
Cd	Cl	MIRA Cd	MIRA Cl	Cd	Cl
0.29572	0.07707	0.2963	0.0845	0.195748	-8.7929

Table 3 - Coefficient of Drag and Lift

The mesh was built uses the setting shown in table 2, figure 12 show a volume cut plane of the mesh develop following using the setting from the study.

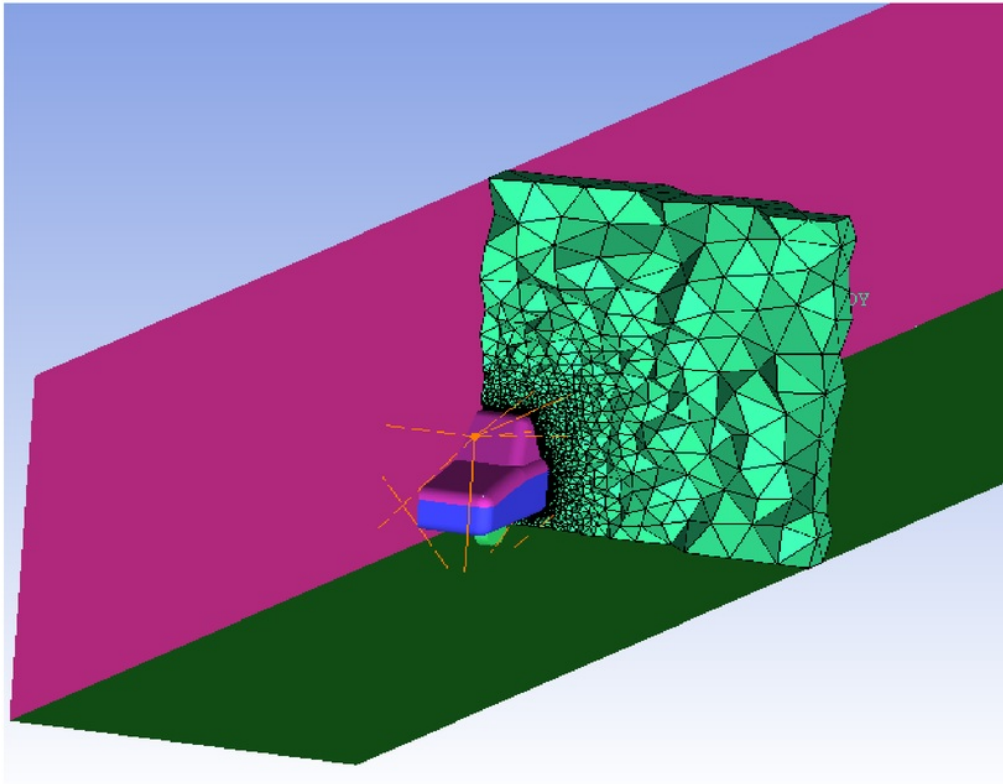


Figure 12 - Showing Optimum Mesh

The modelling was then run through the solver and figure 13 was produce. This a shown the pressure contours on the surface of the car and the velocity contours on the symmetry plane. The area of stagnation in front of the vehicle can be seen where there is high pressure and low velocity as the air moves over the bonnets it speeds up again until it hit the bottom of the windscreen where it slow slightly again before accelerating up the screen and over the roof of the until the end of the roof. Then an area of separation and turbulent flow can be seen in rear wake of the vehicle.

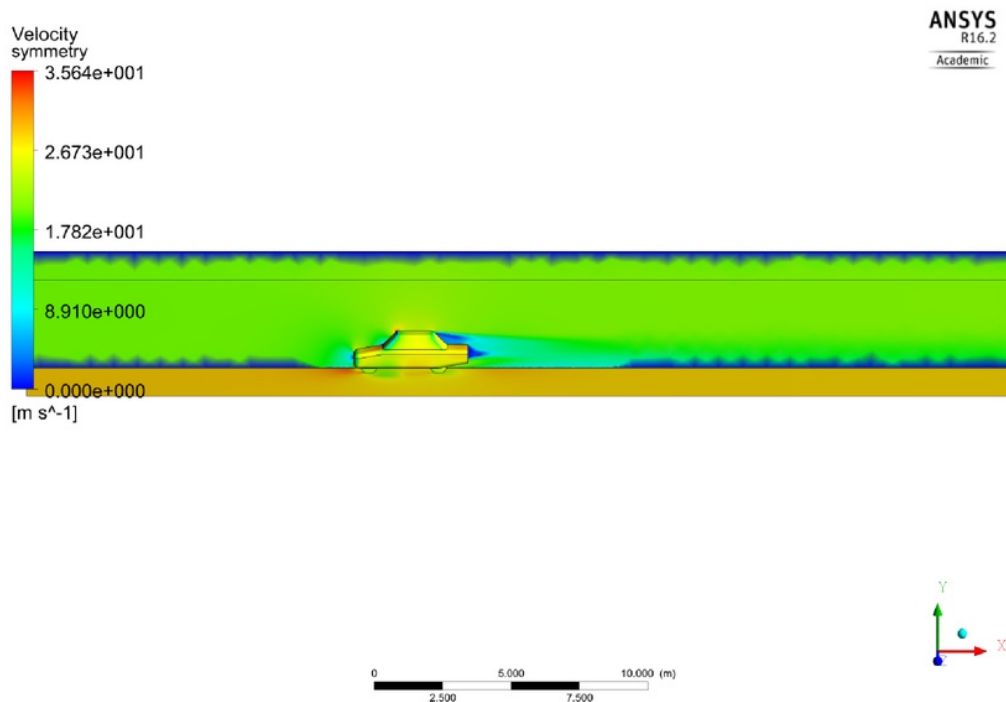


Figure 13 - Results from Optimum Settings

Following on the visualisation of pressure and velocity contours, coefficients pressure charts were made from the data collected for the upper and lower surfaces

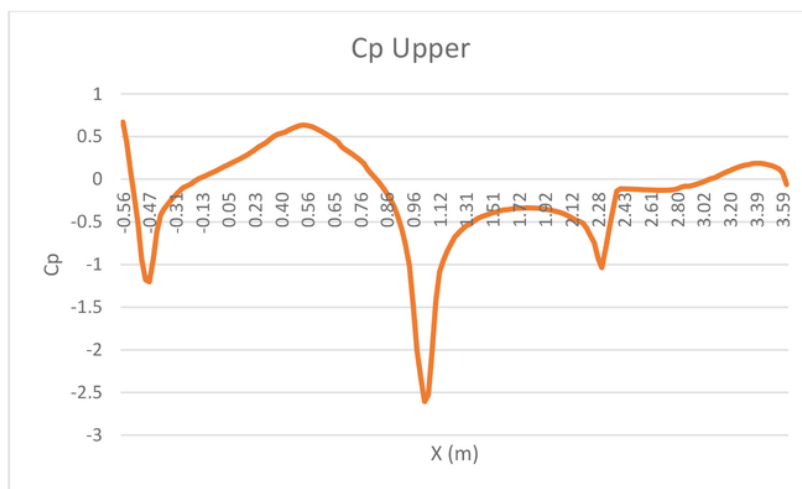


Figure 14 - Coefficient of Pressure Chart for the upper surface

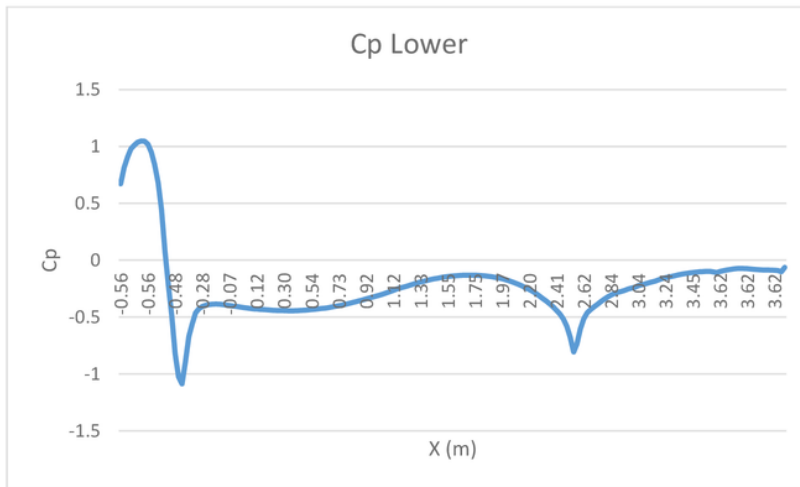


Figure 15 - Coefficient of Pressure for the lower Surface

3.2 Refinement

3.2.1 Additional named faces

An attempt was made to try and refine the mesh using different named faces in areas of interest such as the front of the vehicle where stagnation occurs, the edges of the wind screen and bottom where there are velocity contours and the rear region where there is turbulent flow in the wake region. The face were divided into leading and trailing faces so different mesh setting could be applied in the hope of more accurate lift and drag predictions. The named faces used can be seen in figure 16 and the parameters for theses face in figure 17.

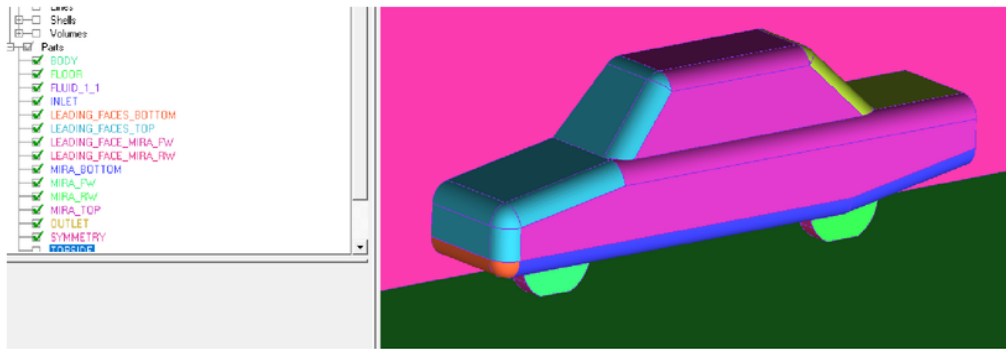


Figure 16 - New Named Face Selection

Part	Pism	Hexa-core	Maximum size	Height	Height ratio	Num layers	Tetra size ratio	Tetra width	Min size
BODY	☐	☐							
FLOOR	☐		0	0	0	0	0	0	
FLUID_1_1	☐		0					0	
INLET	☐		0	0	0	0	0	0	
LEADING_FACE_MIRA_FW	☒		0.01	0	0	5	0	0	
LEADING_FACE_MIRA_RW	☒		0.01	0	0	5	0	0	
LEADING_FACES_BOTTOM	☒		0.01	0	0	5	0	0	
LEADING_FACES_TOP	☒		0.01	0	0	5	0	0	
MIRA_BOTTOM	☒		0.05	0	0	5	0	0	
MIRA_FW	☒		0.02	0	0	5	0	0	
MIRA_RW	☒		0.02	0	0	5	0	0	
MIRA_TOP	☒		0.05	0	0	5	0	0	
OUTLET	☐		0	0	0	0	0	0	
SYMMETRY	☐		0	0	0	0	0	0	
TOPSIDE	☐		0	0	0	0	0	0	
TRAILING_FACE_MIRA_FW	☒		0.01	0	0	5	0	0	
TRAILING_FACE_MIRA_RW	☒		0.01	0	0	5	0	0	
TRAILING_FACES	☒		0.01	0	0	5	0	0	

Figure 17 - Named faces Part Mesh Parameter's

The data produce from this model is shown table 4, this meshing strategy actually produced date that with a greater percentage difference than the 1st run. Again a visualisation of the pressure contours on the car surface and velocity contours in the symmetry plane were produce and can be seen in figure 18. The velocity contour in the wake region using this meshing strategy is better represented.

Model	MIRA DATA	% Difference
-------	-----------	--------------

	Cd	Cl	MIRA Cd	MIRA Cl		
Named				-		
Faces	0.2694	-0.20192	0.2963	0.0845	9.078637	138.959

Table 4 - Results from New Named Faces Run

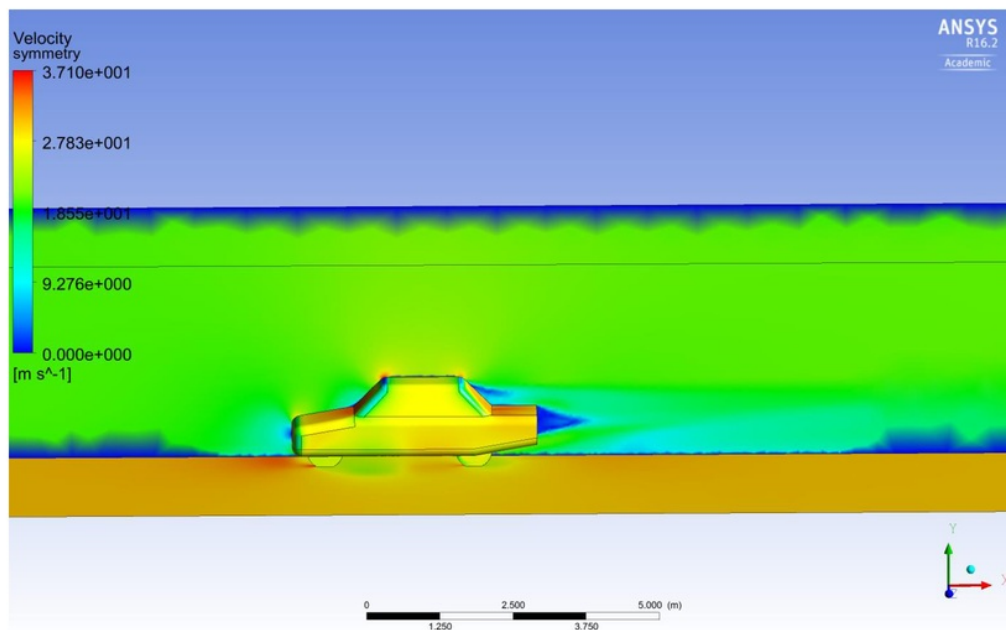


Figure 18 - Visualisation of pressure and velocity contours from named faces meshing strategy

3.2.2 Additional Density boxes

Another attempt was made at refining the mesh using density boxes with different mesh parameter settings getting larger as the density boxes got bigger. This was expected to improve the data on lift and drag but also improve deflection with the visitation post process.

Figure 19 shows the mesh produced, table 5 shows the data collected and figure 20 shows the visualisation produced.

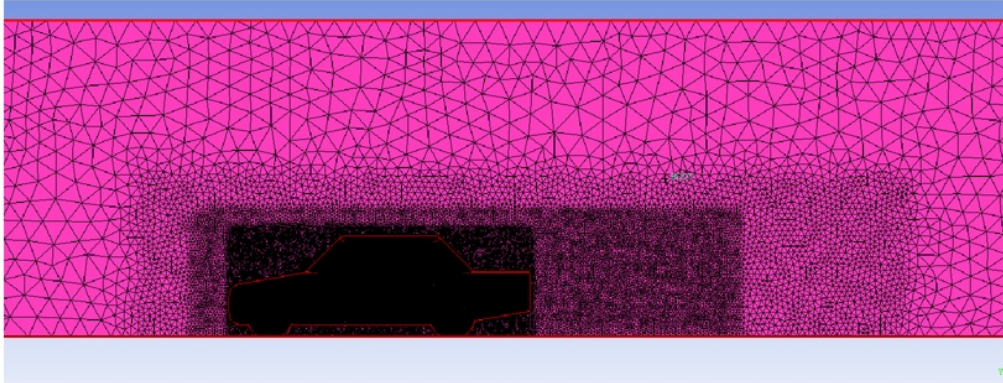


Figure 19 - Mesh created using Density Boxes

	Model		MIRA DATA		% Difference	
	Cd	Cl	MIRA	MIRA		
	Cd	Cl	Cd	Cl	Cd	Cl
Density		-		-		-
boxes	0.26958	0.19044	0.2963	0.0845	9.017887	125.373

Table 5 - Data Collected from the Density Box Meshing strategy

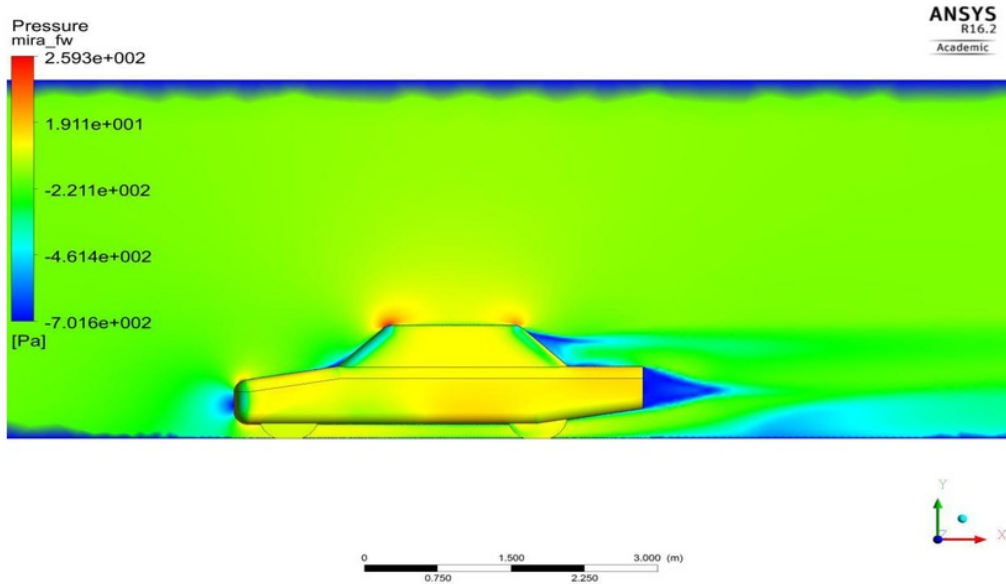


Figure 20 - Visualisation Produced From Density Box Meshing Strategy

4 Geometry Change

The geometry was then changed to represent an estate version of the MIRA reference car.

The model used can be seen in figure 21. A mesh was then built using the same meshing strategy as the first run and solutions were converged.

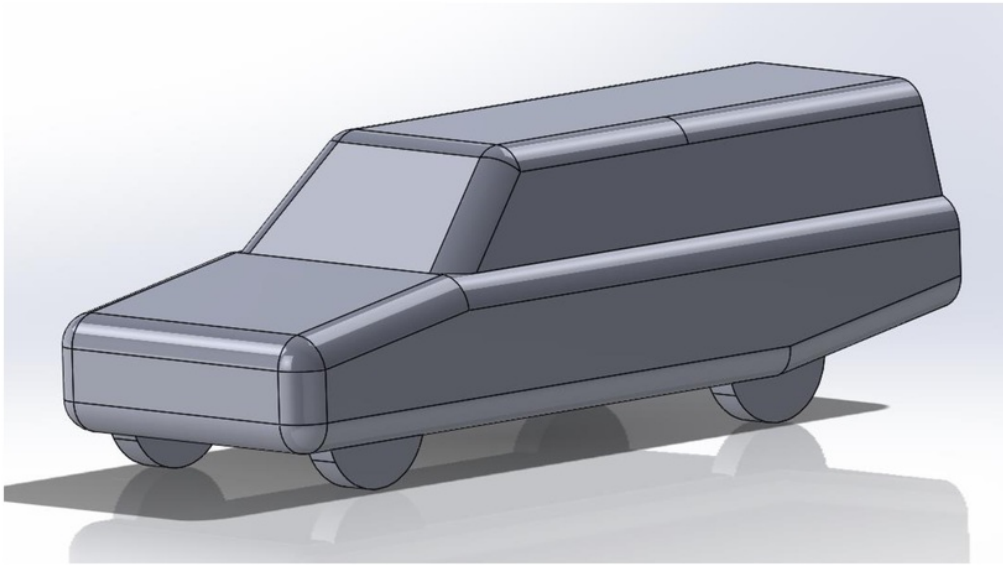


Figure 21 - Estate Model

The data collect for lift and drag is shown table 6, the estate car produce a higher coefficient of drag and lift, this was expected.

	Cd	Cl
Estate Car	0.34466	0.43585

Table 6 - Coefficient of lift and drag data for estate car

A visualisation of pressure contour and velocity contour was then produce figure and the large wake from the estate can be seen in figure 22.

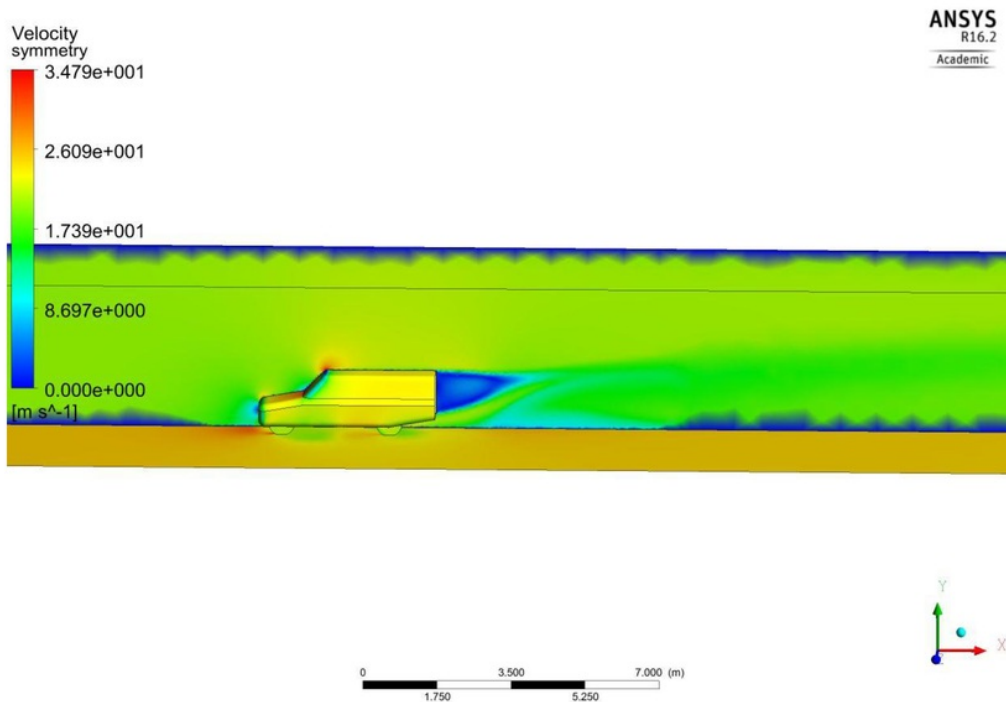


Figure 22 - Estate Car Visualisation

5 Conclusions

The dependency study helped in the development of a suitable meshing strategy for modelling the MIRA reference car. By studying how the different mesh parameters, domain size and solution settings altered the result a pretty robust strategy was formulated. This resulted in solutions close to the MIRA reference data. The refinements made to the mesh however did result in more accurate data but did result in higher definition visualisations.

When the geometry was changed to represent the estate model a higher lift and drag coefficient was found.

6 Bibliography

- Ahmad, N. E., Abo-Serie, E., & Gaylard, A. (2010). *Mesh Optimization for Ground Vehicle Aerodynamics*. Coventry: CFD Letters and ISSR Journals.
- Bardina, J. E., Huang, P. G., & Coakley, T. J. (1997). *Turbulence Modelling Validation, Testing, and Development*. California: Ames Research Center.
- Bertin, J. J. (2002). *Aerodynamics For Engineers*. Upper Saddle River: Prentice Hall.
- Lanfrut, M. (2005). *Best Practice Guidelines for Handling Automotive External Aerodynamics with Fluent*. Darmstadt: Fluent Deutschland GmbH.
- McBeath, S. (2015). *Competition Car Aerodynamics*. Dorchester: Veloce.
- Versteeg, H. K., & Malalasekera, W. (1995). *Computational Fluid Dynamics*. New York: Longman.