

P150652_ACM_CFD_Report

by Krishna Katwa

FILE	5367_KRISHNA_KATWA_P150652_ACM_CFD_REPORT_128450_854504116.DOCX		
TIME SUBMITTED	09-JAN-2019 07:48PM (UTC+0000)	WORD COUNT	5806
SUBMISSION ID	98619547	CHARACTER COUNT	34235

University of Wales Trinity Saint David
School of Engineering
Aerodynamic Vehicle Testing & Analysis



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

Krishna Katwa
P150652
09/01/19
Motorsport Engineering

Contents

1	Introduction	1
1.1	Aims	1
1.2	Objectives.....	1
2	Background Research	2
3	Computational Fluid Dynamic Settings Study.....	4
3.1	Domain	5
3.1.1	Domain Study Results	6
3.2	Mesh.....	7
3.2.1	Mesh Study Results	8
3.3	Inflation layers	10
3.3.1	Inflation Layer Study Results.....	10
3.4	Turbulence Model.....	13
3.4.1	Turbulence Model Study Results	14
3.5	Mesh Quality	15
3.5.1	Skewness	15
3.5.2	Orthogonal Quality	15
4	Results.....	16
4.1	Aerodynamic Modification	23
5	Conclusion	29
6	Recommended Further Work	30
7	References	31
	Figure 1-1, Mira Reference Car	1
	Figure 2-1, Boundary Layer. (www.grc.nasa.gov, 2015)	3
	Figure 3-1, Example Domain. (Rao, et al., 2007)	5
	Figure 3-2, Example of a Delaunay Mesh.....	9
	Figure 3-3, Example of a Robust Mesh	9
	Figure 3-4, No Boundary Layer.....	11
	Figure 3-5, Boundary Layer Applied	11
	Figure 3-6, Near Wall Region. (Ansys Fluent, 2006)	13

Figure 4-1, Velocity Contours	19
Figure 4-2, Rear Deck Velocity Contours	19
Figure 4-3, Pressure Contours	20
Figure 4-4, Velocity Contours	20
Figure 4-5, Vortex Core Region	21
Figure 4-6, Front Streamlines	21
Figure 4-7, Rear Streamlines	22
Figure 4-8, Modified Mira Reference Car	23
Figure 4-9, Wing Profile	23
Figure 4-10, Modified Velocity Vectors	24
Figure 4-11, Wing Velocity Vectors	24
Figure 4-12, Modified Pressure Contours	25
Figure 4-13, Wing Pressure Contours	25
Figure 4-14, Velocity Contours	26
Figure 4-15, Wing Velocity Contours	26
Figure 4-16, Vortex Core	27
Figure 4-17, Wing Vortex Core	27
Figure 4-18, Modified Front Streamlines	28
Figure 4-19, Modified Rear Streamlines	28
Graph 1, Domain Volume Against Lift & Drag	6
Graph 2, Mesh Part Size Vs Cl & Cd	8
Graph 3, Number of Inflation Layers vs Cd & Cl	12
Graph 4, Comparison of Turbulence Models	14
Graph 5, Upper Pressure Tap Data	16
Graph 6, Lower Pressure Tap Data	17
Graph 7, Upper & Lower Pressure Tap Data	18
Table 1, Mesh Part Size Table	8
Table 2, Inflation Layer Results Table	10
Table 3, Turbulence Model Results Table	14
Table 4, Skewness Table	15
Table 5, Settings Used	16
Table 6, Standard vs Modified Comparison Table	24

1 Introduction

1.1 Aims

This report aims to cover the effect on aerodynamic performance of a common aerodynamic device often seen on motorsport vehicles through the use of an advanced computational method such as Computational Fluid Dynamics (CFD). The base vehicle used for this will be the Mira wind tunnel reference car geometry, specifically the notchback car model. This is due to the similarity of the model to a real cars geometry. (Zhang, et al., 2014)

1.2 Objectives

A validated CFD methodology will be created using the Mira reference car, *figure 1-1*, the simulated figures will be compared to the real world known values that were produced for lift and drag from the real car. If these are found to be within an acceptable tolerance, then it can be said that the method and setup of the CFD is valid as the computational results match the real-world experimental data.

Once the methodology has been validated, a common aerodynamic device will be added to the Mira reference car to study the difference it makes not only to lift and drag, but to the flow structure around the car.

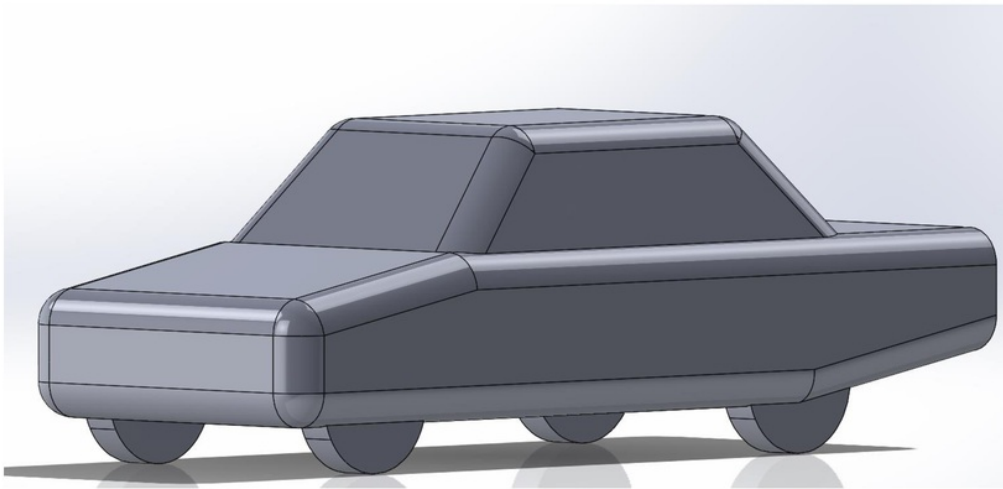


Figure 1-1, Mira Reference Car

2 Background Research

Every time a car moves it needs to displace the air around it, as the car forces the air around it, the air very quickly attempts to fill the void that has been left behind. The movement of the air in this manner produces forces on the cars bodywork that can try and lift it up off the ground, push it down into the ground, and slow it down. These are known as lift, downforce and drag.

Naturally, for racing, lift and drag are not attractive, the aim is to stop the clock with as little time elapsed as possible, so drag, which slows the car along the straights and lift, which prevents the car cornering quickly by limiting the grip the tyres have are minimised as much as possible. (Machin, 2015)

Since as early as 1916, race car manufactures have been trying to reduce the drag of their cars, indicating that there has always been a need to increase the ease with which a car can move through the air, after all, the less drag, the faster the car can go. (Katz, 1995)

As (Katz, 1995) also notes, the biggest lap time improvement can often be found in how quickly a car can travel around a corner. If the car can corner quicker, the average lap speed will be higher and therefore the lap time lower. One way to allow this to happen is increasing the load on the tyres without the tyre slipping. Increasing the mass of the race car would work but the offset is that the car is now heavier so will take longer to accelerate and more force will be needed to turn the car to begin with. This is where aerodynamics is useful as downforce can increase the tyre load without increasing the vehicle mass. This allows the car to corner quicker with no weight penalty.

This phenomenon was only observed towards the late 1960's, what followed was a period of rapid and sometimes dangerous aerodynamic development. Below, the 1916 Indy 500 winner can be seen along with a 2008 Formula 1 car, showing just how much aerodynamics have developed.



Figure 2-1, 1916 Indy 500 Car. (Peugeot, 2018)



Figure 2-2, 2008 Formula1 Car, (FavCars, 2008)

Horiba Mira possess the UK's only full-scale wind tunnel, with a maximum wind speed of 36m/s and the ability to evaluate pitch yaw and roll, it is easy to see why wind tunnels can not only compliment but be more beneficial than on track testing. When in use, data is acquirable through a 62-channel pressure measurement system. The test section of the closed test section is 15*7.9*4.4m (length*width*height) (Mira, 2018)



Figure 2-3, Mira Full Scale Wind Tunnel. (Mira, 2018)

When air flows over a surface, it interacts with the surface and as it is a viscous fluid, a boundary layer is formed. This occurs due to the frictional forces. It is because of this that when looking at a surface directly, the fluid will have a velocity of 0m/s.

There are 3 types of fluid flow, laminar, transitional and turbulent. As the name would suggest, transitional flow is the state between laminar and turbulent.

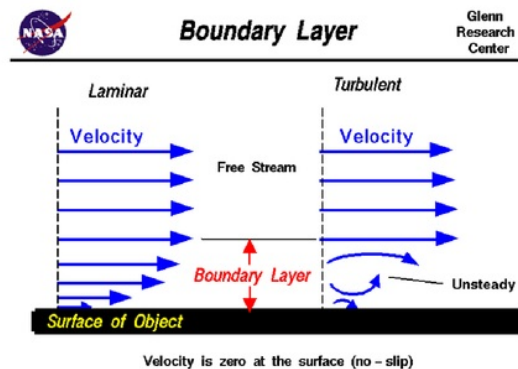


Figure 2-1, Boundary Layer. (www.grc.nasa.gov, 2015)

3 Computational Fluid Dynamic Settings Study

A study was carried out in order to determine the best settings to use for replicating the real-world wind tunnel results. Due to the nature of the sheer number and potential combination of settings this was not an exhaustive study, the time taken to complete such a study and the computing power needed to necessitate it would be excessive and expensive in time and money.

That being said, the study that was carried out has been done so diligently and has been formed from the writer's previous experience with CFD and background research into each area. A half model is used to try and reduce the computing time and power needed. This is achieved by the half model needing less mesh elements to accurately represent the geometry. However, this is only possible with symmetrical geometry.

The following areas were studied to examine the effect on the results and to allow an 'ideal' CFD model to be created and used on the Mira geometry hopefully replicating real-world data, for each area of study, the same parameters were used unless it was the parameter being studied, then this was changed.

The results for c_l and c_d (lift and drag) from the wind tunnel testing are as follows;

- $c_l = -0.0845$
- $c_d = 0.2963$

3.1 Domain

When using CFD, it is often easier to understand this term if it is considered to be the wind tunnel, within which the model to be tested will be placed. As a result, the domain should be of a size that is great enough as to not impact on the results. One popular domain size is inspired by 'Good Practice CFD' (Lanfrut, 2005), in which it is stated that the sizing of the domain should be more than 3 car lengths ahead of the car and 5 behind to allow for minimal interference. However, (Rao, et al., 2007) recommend a domain of 8 times the length of the model with the model placed such that the domain is 2 lengths in front and 5 behind assuming a model length of 1. The width and height are recommended to be 2 times the length.

The blockage effect should also be considered, that is, where the walls of the domain are close enough to the model to accelerate the air between them and alter the results in an unrealistic manner.

When the domain is close to the model, the flow towards the vehicle, 'onset flow', may not be fully developed, this can cause issues in post processing when trying to analyze the flow structure around the model. Especially at the rear of the model where one would expect to find the wake region and any turbulence or separation.

Despite the Mira wind tunnel having set dimensions, a domain study can still be useful to investigate and demonstrate the effect that the domain size can have on the lift and drag. this would provide knowledge that would allow appropriate sizing of a domain for any future work regarding the Mira car without needing to use the wind tunnel or for validation of the CFD technique.

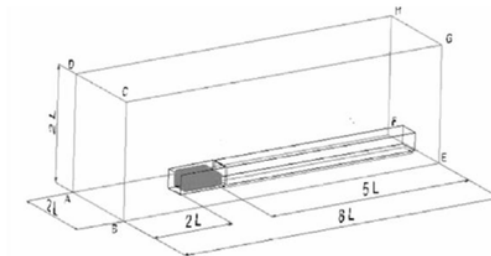


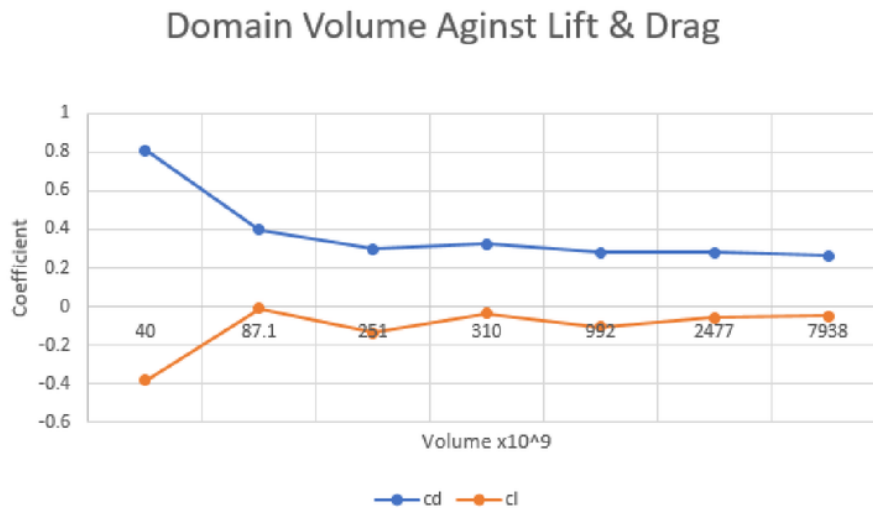
Figure 3-1, Example Domain. (Rao, et al., 2007)

3.1.1 Domain Study Results

A number of different domain sizes were chosen to be tested. These were as follows;

- Lanfrit recommended, (Lanfrit, 2005)
- Half Lanfrit recommended
- Double Lanfrit recommended
- Geometry dimensions to the nearest thousand
- CFD Ahmed body tutorial scaled, (Eoescipy, 2012)
- CFD Ahmed body tutorial scaled halved
- Mira tunnel dimensions, (Mira, 2018)

The volume of each domain was calculated, and this was plotted on a graph against the corresponding results for c_l and c_d , the graph can be found below.



Graph 1, Domain Volume Against Lift & Drag

As can be seen from the *graph 1* above, as the domain volume increases the results for c_l and c_d get more accurate compared to the wind tunnel results. This could be explained by the blockage effect, with a domain barely bigger than the geometry, $40 \times 10^9 \text{m}^3$, the results for c_l and c_d are -0.03834 and 0.8093 respectively compared to -0.0845 and 0.2963 of the wind tunnel. It can also be noted that there is not much difference once the domain size has increased beyond $2477 \times 10^9 \text{m}^3$. Suggesting that at this point the domain size has very little bearing on the results and the additional time needed to run the CFD due to increased mesh size is not practical for the improvement in accuracy.

3.2 Mesh

The mesh is a key element towards obtaining accurate results whenever carrying out CFD. Usually for a generic car type body, known as a bluff body, skin friction is overcome by form drag, therefore when the resulting lift and drag are predicted by the software, the surface mesh needs to accurately represent the geometry. If the mesh doesn't, then as the pressure distributions over the body are calculated by the solver to predict the drag and lift, they will not be accurately predicted leading to figures that are irrelevant. (Lanfrut, 2005)

Due to this, the more accurately the mesh represents the model, the longer it will take the solver to solve and therefore more computational power will be required. That being said, if a simple geometry is being represented then the mesh will not be as dense as less cells will be needed to represent the geometry leading to a shorter run time. The time taken can be a limiting factor as a project will usually only have a certain time period in which to be completed. With no limits on time or computational power then an extremely small mesh size could be used to represent the geometry and gain even more accurate results. However, a point of diminishing returns will eventually be reached where the results obtained are no more accurate for the increased computational power and time.

For this study, the part mesh size was changed in order to study the effect on the results of lift and drag. The global mesh is the mesh that represents the majority of the domain, where the free stream flow is found, therefore a bigger size would be expected to be used here as there is no intricate geometry to capture and model, it is just slab-sided walls. This is not the case for the part mesh, this captures the actual model and can be defined in the meshing programme. For this study, the car was selected and made its own part, this can allow for a much smaller element size on the car to better capture the model.

Once the car has been split into its own separate part in the mesher, a mesh density box can be added. The benefit of doing this is that it allows for a more detailed mesh around the car where the flow structure is at its most intricate and interesting from an aerodynamicist's point of view, at the front of the car, and at the rear to capture the wake region. For this study the density box was sized 1.5 times wide and high and 1.7 times long through the middle of the car, this allowed the density box to be far enough ahead and behind the car to allow a finer mesh, without sacrificing computational time. (Pachpund, et al., 2012)

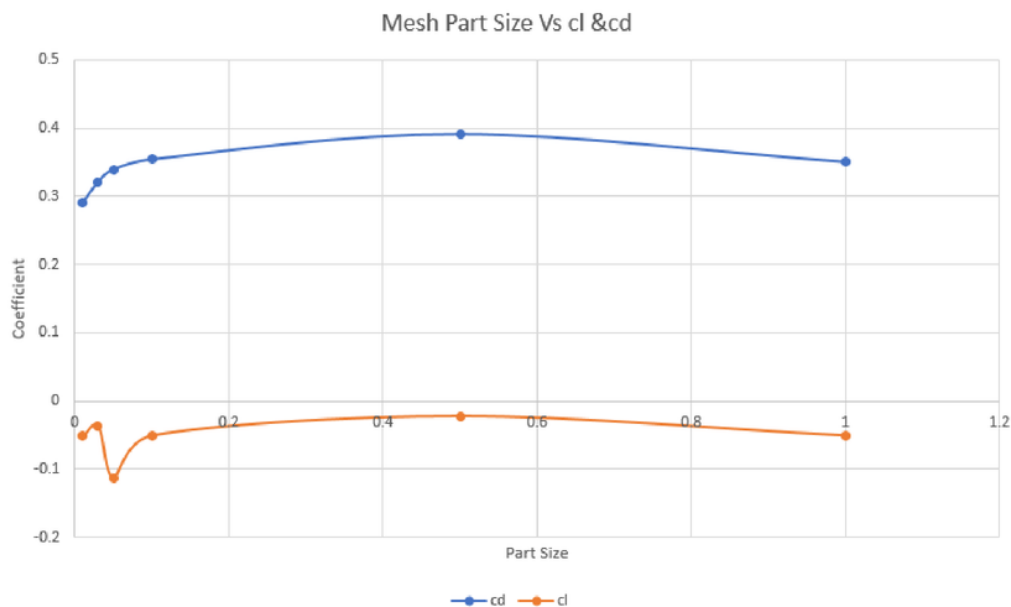
3.2.1 Mesh Study Results

A number of different mesh part sizes were tested. The part size is the size of the mesh that represents the geometry. A global mesh study was also attempted however the computational time and power needed to mesh a global size lower than 0.5 was unavailable. Therefore, only a part size study was carried out, the results of which can be seen below, and the number of elements in each mesh was also observed.

Table 1, Mesh Part Size Table

Part size	cd	cl	Elements
1	0.3508	-0.0511	1406918
0.5	0.3921	-0.0225	1407271
0.1	0.3547	-0.0511	1406890
0.05	0.3400	-0.1138	2652333
0.03	0.3216	-0.0378	1838016
0.01	0.2911	-0.0518	7922599

A graph was also plotted to better show the results of the study.



Graph 2, Mesh Part Size Vs Cl & Cd

With regards to cd, as can be clearly seen, the finer the part mesh the more accurate the result. As to be expected, with roughly an extra 6 million elements, the solution is more accurate as the solver can more accurately predict the drag. The cl however does not respond in the same way, this could be due to the lower number of elements present in the 0.03 part size compared to the 0.01 part size, a difference of 6,084,583, just over 6 million.

It is also important to consider the type of mesh that will be applied to the geometry, there are 3 main types;

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

Each meshing method has its advantages and disadvantages, for example, advancing front needs a high-quality geometry, unfortunately the quality of the geometry provided was not good enough for an advancing front mesh to be generated.

The robust method uses a top down approach to meshing, it is more forgiving with the quality of the geometry so will still mesh a lower quality model. It does not need an existing surface mesh unlike quick Delaunay or advancing front as it creates one in the process of meshing.

The quick Delaunay method is known as a bottom up approach, it is robust and fast, this can be seen in the more organic like structure when viewing the mesh on a cut plane. An example of this is provided below in *figure 3-2*. As can be seen, the mesh is almost randomly structured and free flowing, this is in contrast to a robust mesh, *figure 3-3* which is much more structured and laid out in an almost grid like manor. (Ansys Fluent, n.d.)



Figure 3-2, Example of a Delaunay Mesh

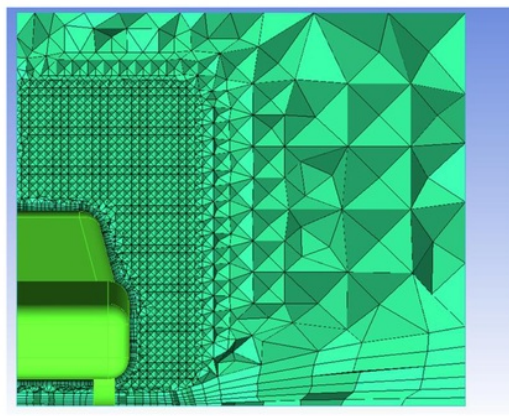


Figure 3-3, Example of a Robust Mesh

3.3 Inflation layers

One key area that needs to be replicated when running CFD, is the boundary layer flow that occurs in real life. An idea that has been discussed previously within this report under section 2.0 Background Research. Within CFD, this is known as prism or inflation layers.

These are added on to the car and to the road (bottom of the domain) within the meshing software, they could also be added to the symmetry plane and opposite side of the domain to the car, to represent the wind tunnel testing conditions but in reality on the open road, the boundary layer would only form on the car and road. The study was concerned with the number of prism layers, initially the model would be run with no prism layers and then a new layer would be added each time to investigate if there was a noticeable difference in drag and lift values.

3.3.1 Inflation Layer Study Results

To begin with, the model was run with no inflation layers, the number of layers was increased until it would not be able to add elements between the car and the road. The number of layers where this occurred was 9. The results were tabulated, and these can be seen below in table 2.

Table 2, Inflation Layer Results Table

Number of Layers	cd	cl
0	0.2983	-0.1959
1	0.3340	-0.0923
2	0.3527	-0.0881
3	0.3297	-0.0788
4	0.3238	-0.0747
5	0.3197	-0.0603
6	0.3216	-0.0378
7	0.3233	-0.0563
8	0.3216	-0.0619

With no inflation layer applied, the drag was predicted closest to the real-world results, however this result would have to be taken with some caution as in real life a boundary layer would form on the car as it moves through the air so this result is not very representative of the real setup.

This is also shown in the below pictures, *figure 3-4* shows no boundary layer and *figure 3-5* shows a boundary layer applied.

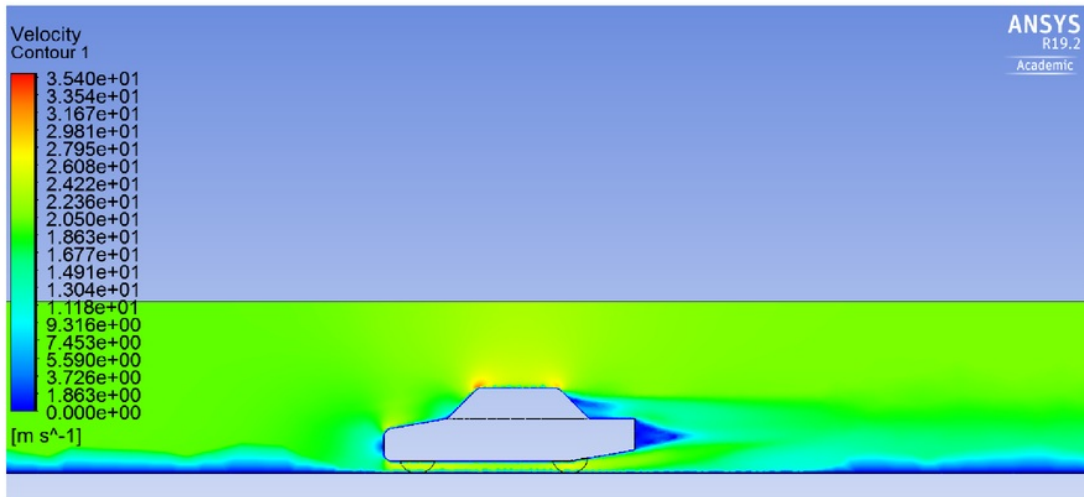


Figure 3-4, No Boundary Layer

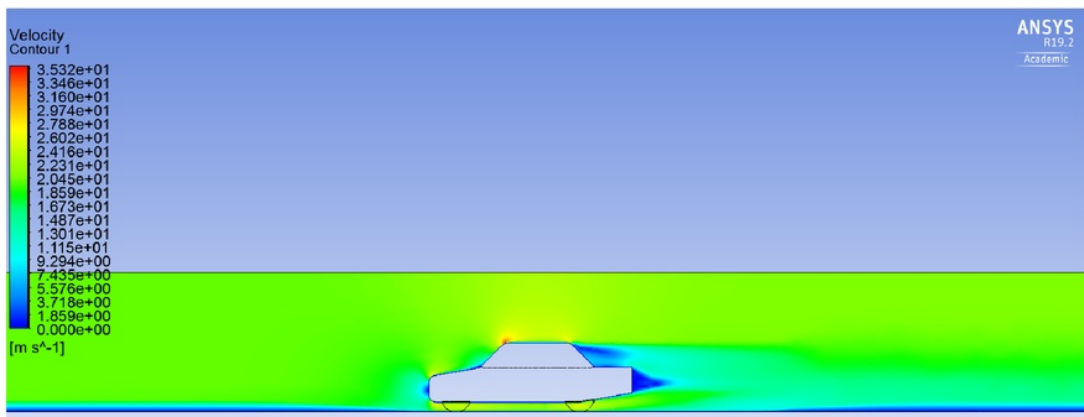
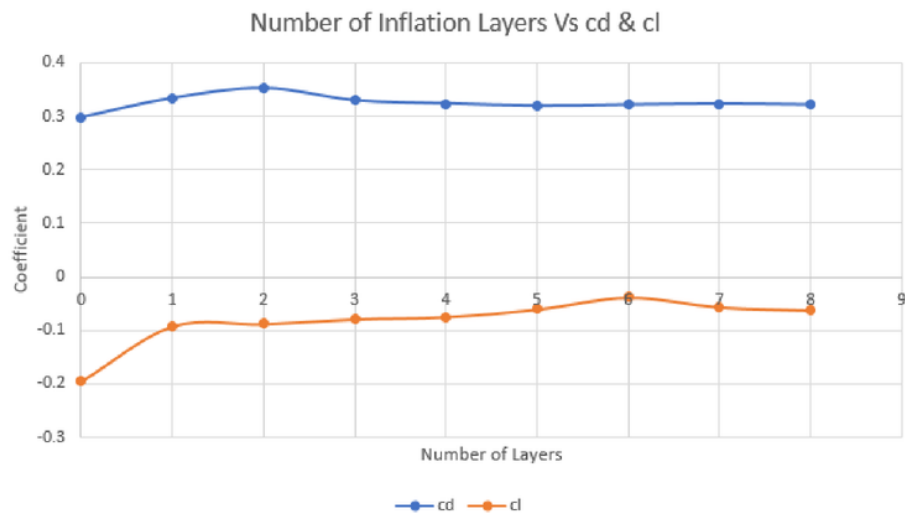


Figure 3-5, Boundary Layer Applied

As can be observed, the flow along the road of the domain is unstructured and random whereas with a boundary layer applied it is much more controlled and smoother ahead and behind the car. There is also a slight difference in the wake region, this is because the boundary layer will influence the flow over the car.

A graph was used to help visualize the results of the inflation layer study and this can be seen in graph 3 below.



Graph 3, Number of Inflation Layers vs Cd & Cl

The graph clearly shows there is not much deviation in the results once 3 layers or more have been added although the result for cl with 6 layers could be an anomaly. Based on the results of this study, it was decided that 4 layers would be used going forward due to the short computational time needed to apply them in the mesher and the small deviation thereafter in the results.

3.4 Turbulence Model

As part of the setup within the solver, (Ansys Fluent), a turbulence model needs to be selected, there are many turbulence models that can be chosen, each with sub options that can be chosen. Some of the main turbulence models available to the used are shown below:

- Laminar
- K-epsilon (2 eqn)
- K-omega (2 eqn)
- Transition k-kl-omega (3 eqn)
- Transition SST (4 eqn)
- Reynolds Stress (7 eqn)
- Large Eddy Simulation (LES)

There is also the option to choose what k-epsilon model and near wall treatments are used within the calculations, as each model will predict the flow in-between the nodes on the mesh, it is of high importance to select an appropriate model. The difference in k-epsilon and k-omega is explored below.

The k-epsilon model will solve for 2 different variables, turbulent kinetic energy and rate of dissipation of the turbulent kinetic energy. This model is traditionally very popular in industry as it has a high convergence rate and doesn't use a huge amount of computer memory. The model is useful for solving external flow around geometries that are complex such as a bluff body. However, despite having advantages, there are also some disadvantages, if there is an adverse pressure gradient present or strong curvature then this model will not very accurately compute the resulting flow fields. (Frei, 2017)

Wall functions also need to be considered when setting up the solver, this is because the presence of a wall can have a significant effect on the turbulent flow around it. Therefore, the near-wall modelling used will have a noticeable bearing on the results. This is because the near wall region can be sub divided into 3 layers (Ansys Fluent, 2006);

- Viscous sublayer
- Interim region
- Fully turbulent layer

This is shown in figure 3-6 opposite.

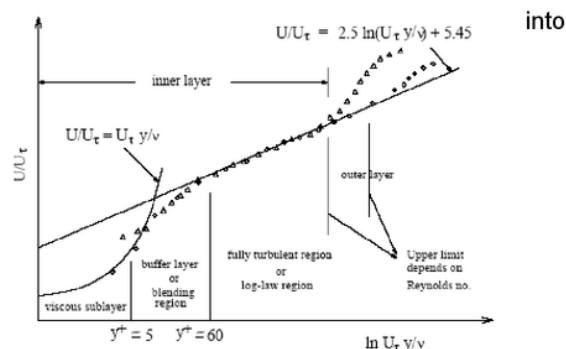


Figure 3-6, Near Wall Region. (Ansys Fluent, 2006)

3.4.1 Turbulence Model Study Results

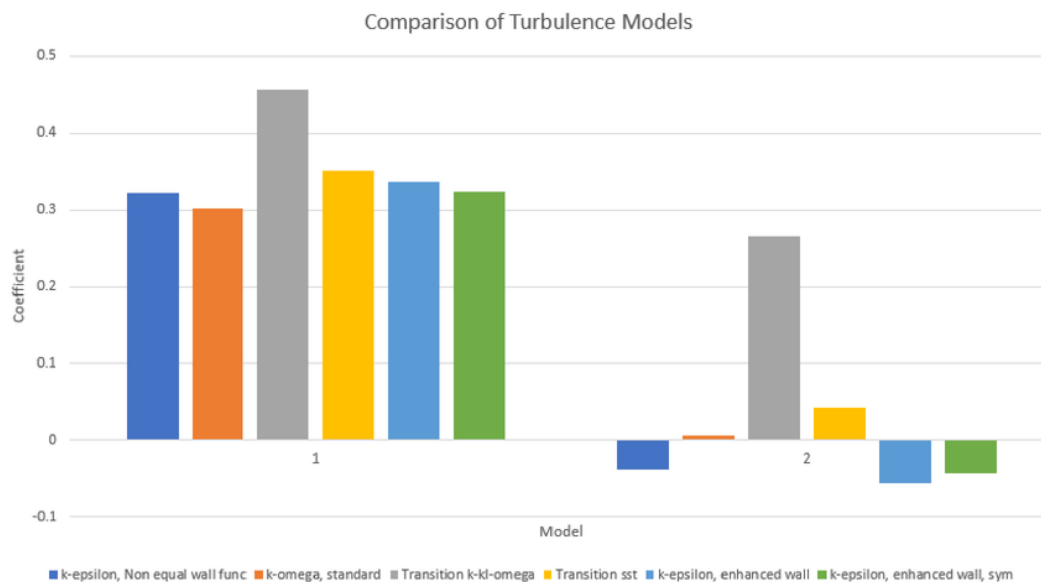
Due to the number of different turbulence models available, not all were tested, however out of the tested the models, the predicted results closest to the wind tunnel data would be used going forward for the study on the Mira reference car.

4 models were tested with differing wall functions selected. Standard wall functions are the default option provided in the solver but can be changed if the user sees fit. Non-equilibrium (two-layer based) wall functions are recommend wherever complex flow is expected to be seen. There is also the option for enhanced wall treatment, this is where the two-layer model is used with enhanced wall functions. (Ansys Fluent, 2006)

Below in table 3 and graph 4 the results of the study can be found.

Table 3, Turbulence Model Results Table

Model	Cd	Cl
k-epsilon, Non equal wall func	0.3216	-0.0378
k-omega, standard	0.3018	0.0056
Transition k-kl-omega	0.4568	0.2655
Transition sst	0.3505	0.0420
k-epsilon, enhanced wall	0.3372	-0.0558
k-epsilon, enhanced wall, sym	0.3237	-0.0440



Graph 4, Comparison of Turbulence Models

As can be noted from the above results, transition k-kl-omega was the least accurate with regards to cd and cl whereas k-epsilon with enhanced wall function was the most accurate.

3.5 Mesh Quality

The quality of the mesh produced is important as a poor quality mesh will lead to poor results and will mean the run has been a waste of time, in many cases the solver will not be able to run with a poor mesh. Skewness and orthogonal quality are two important metrics for a quality mesh.

3.5.1 Skewness

This is a measure of how good the mesh of the geometry is. The mesh elements need to accurately represent the geometry, and this shows how accurate the mesh elements are. Towards the edges of geometry the mesh can become skewed to ensure it fits the geometry, some skewness is to be expected as the elements have to fit in small tight areas such as around the wheels. The quality of the skewness can be represented by tables similar to that seen in table 4 below. (Baker, 2011)

Table 4, Skewness Table



0-0.25	0.25-0.50	0.50-0.60	0.60-0.95	0.95-0.98	0.98-1.00
Excellent	very good	good	acceptable	bad	Unacceptable

3.5.2 Orthogonal Quality

Orthogonal quality should have a minimum value of higher than 0.01 for all cell types. The worst cells will possess a value close to 0 whereas the better cells will have a value closer to 1. The quality is computed for the cells by using a number of different vectors. (Ansys, n.d.)

- Vector from the cell centroid to each of its faces
- The corresponding face area vector
- The vector from the cell centroid to the centroids of each of the adjacent cells

4 Results

Once the initial studies had been carried out, the settings that had shown to give the most accurate results were used to run the Mira geometry to allow comparison to the real world pressure tap data that was acquired from wind tunnel testing, this would allow validation of the settings and model used to allow an investigation into the effect of a motorsport derived geometry change.

The settings used can be found in the table below:

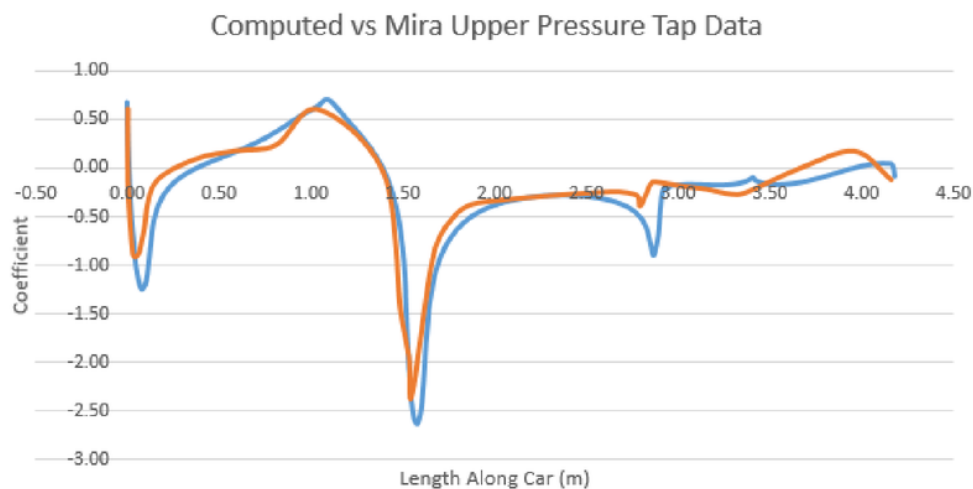
Table 5, Settings Used

Domain	992 x10 ⁹ m ³	
Mesh	0.01	Robust Octree
Inflation Layers	4	
Turbulence Model	k-epsilon enhanced wall	

The model returned results of drag and lift of, 0.2546 and -0.0731 respectively. This compares closely to the Mira figures of 0.2963 for C_d and -0.0845 for C_l .

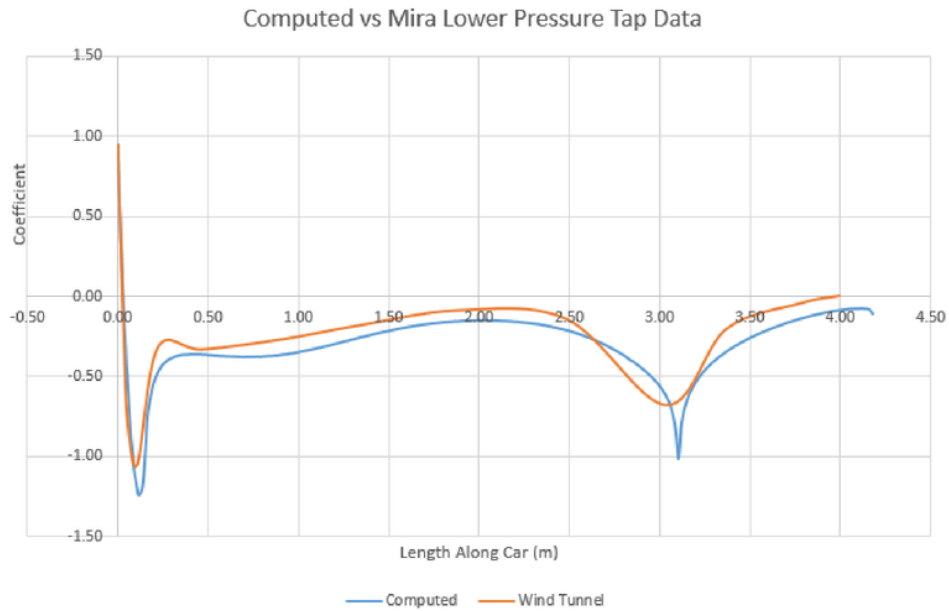
Next, pressure data along the top and bottom surfaces of the car were obtained from the post processing application available within Ansys, this was exported to excel and compared to the wind tunnel pressure tap data. In total 3 graphs were generated to help display the pressure data. One for the upper surface of the car, one for the lower surface of the car and lastly they were combined onto one graph overlaid onto the car to show how the pressure varies along the car.

Below, in graph 5 the upper pressure data can be seen.



Graph 5, Upper Pressure Tap Data

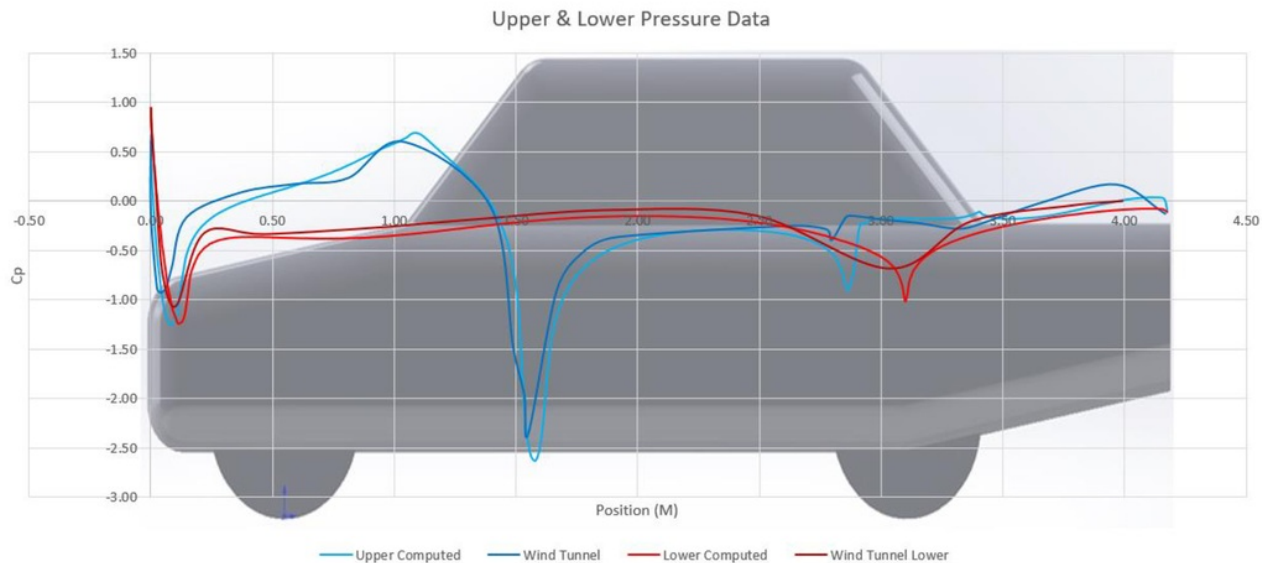
In graph 6 the lower pressure data is displayed.



Graph 6, Lower Pressure Tap Data

As can be seen in both graphs, there is a slight variance between the results and reasons for this are explored later in this chapter.

In graph 7, below, the computed pressure tap data and the wind tunnel data are closely aligned, while the lines are not a total match, this can be attributed to the number of points where the pressure data is being taken on the computed model compared to the wind tunnel model.



Graph 7, Upper & Lower Pressure Tap Data

For example, the upper surface contained 30 pressure taps and the lower surface contained 15 taps when the car was in the wind tunnel, this is much lower than the amount of points of pressure data obtained from the CFD program, the upper surface contained 540 points whilst the lower surface contained 470.

Overlaying the results onto the car allows for a clearer understanding of what is happening to the pressure along both the upper and lower surface of the car. As to be expected, along the upper surface at the base of the windscreen, the pressure is high and at the top of the windscreen the pressure is low. This shows the air flow is following Bernoulli's principle, *'the higher the speed the air molecules are travelling, the lower the pressure'*.

On the underside of the car, the pressure is fairly constant with a slight decrease in pressure roughly where the underside of the body 'kicks' up, this accelerates the air as it is acting as a diffuser allowing the air to increase its velocity.

To further help analyse the air flow around the car, a post processing program was used to create pressure and velocity contour plots around the car, streamlines to help visualize the flow, vectors to show the velocity and direction of air flow and lastly vortex core regions to show where turbulence and separation were occurring.

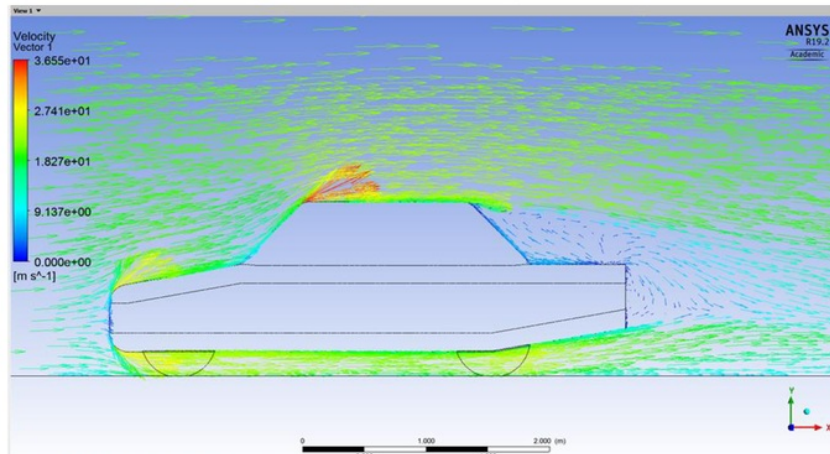


Figure 4-1, Velocity Contours

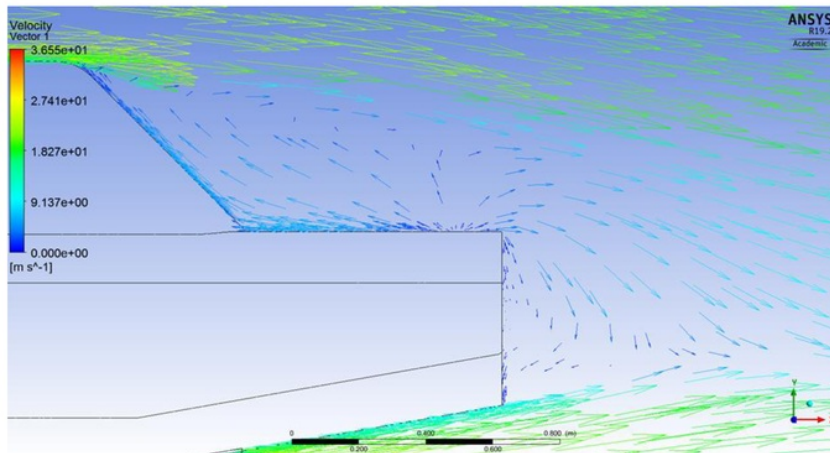


Figure 4-2, Rear Deck Velocity Contours

In figure 4-1 above, velocity contours help to show the high and low pressures at the windscreen as previously discussed. Whilst figure 4-2 shows the turbulent region located on the rear deck of the car and in the void left by the car. It is clear to see that the air is recirculating and has separated.

As shown by the pressure tap data from the wind tunnel for the lower surface, the velocity is fairly constant until the edge that 'kicks' up where a slight increase and thus reduction in pressure is displayed.

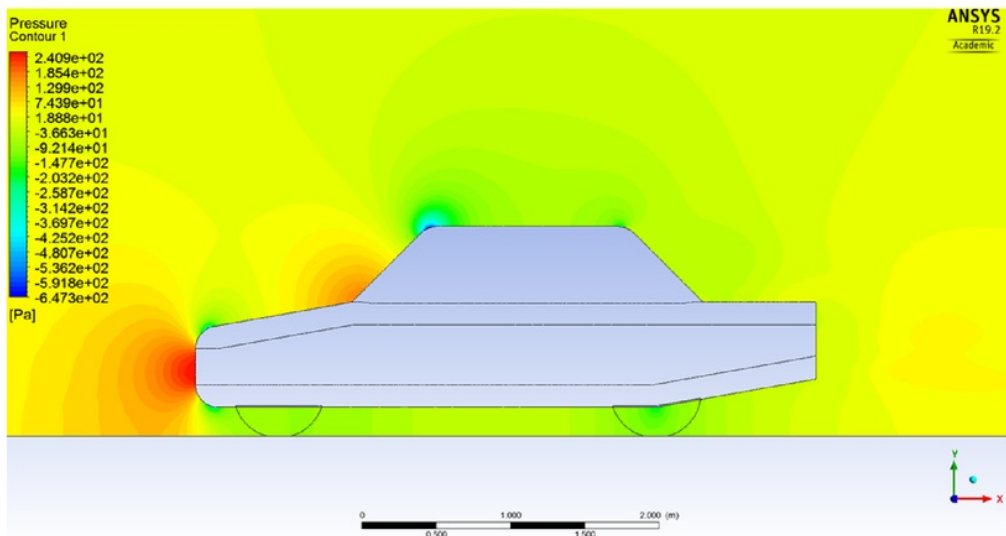


Figure 4-3, Pressure Contours

The pressure contours in *figure 4-3* above demonstrate the correlation between the wind tunnel pressure taps and the computed data from CFD.

Velocity contours in *figure 4-4* show the stagnant air at the front of the vehicle, and the base of the windscreen. The high velocity is also clear around the top of the windscreen. The wake region of the car is also shown with the areas of low speed.

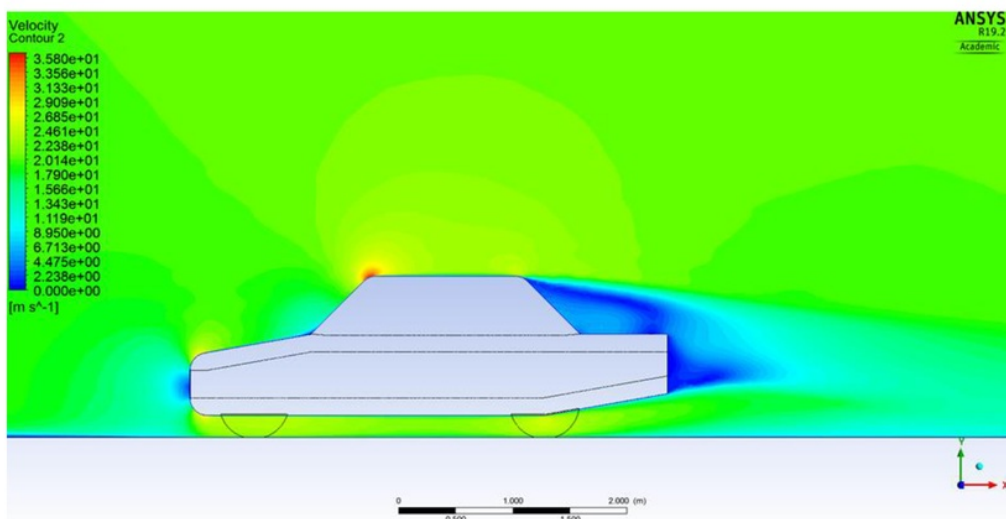


Figure 4-4, Velocity Contours

Below, figure 4-5, shows the vortex core region that is areas where vortices are occurring. As one would expect, the vortices are being generated along the edges of the geometry, this can be useful as the swirling vortex can be used to help improve aerodynamic performance.

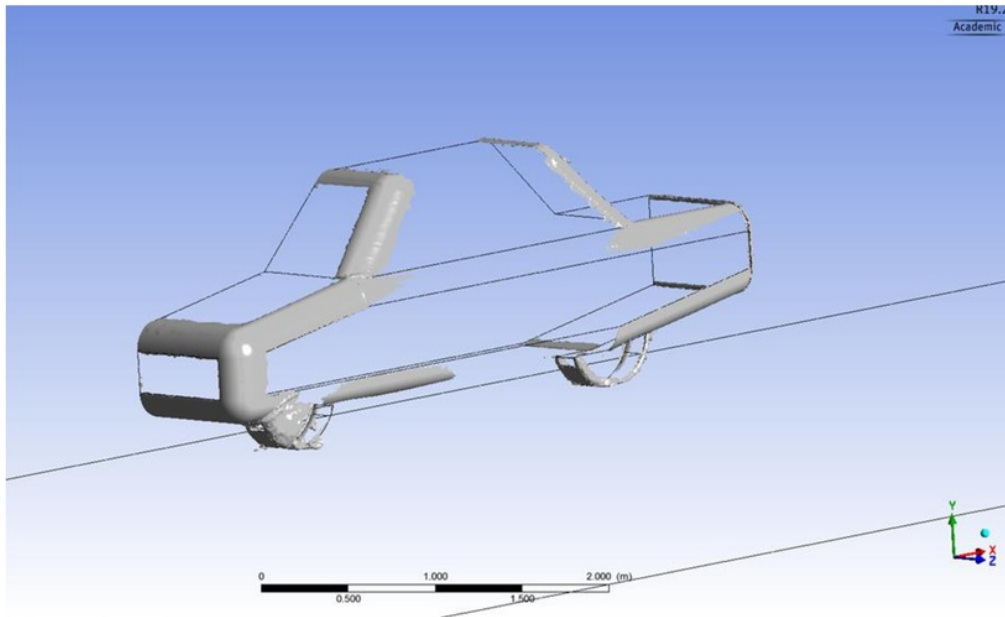


Figure 4-5, Vortex Core Region

The figures below 4-6 & 4-7 give a good visual indication of where the air flows around the car and the structure of the air. From the front of the car, it can be seen that the air decreases where it hits the front bumper then accelerates again up the bonnet. The low speed wake region can also be seen.

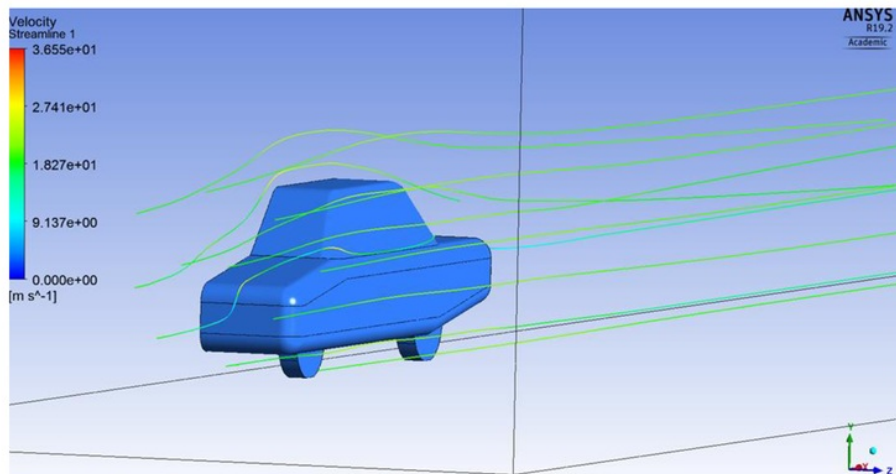


Figure 4-6, Front Streamlines

Looking at the rear of the car, the rotation of the air can be seen in the wake region as it comes of the back deck of the car.

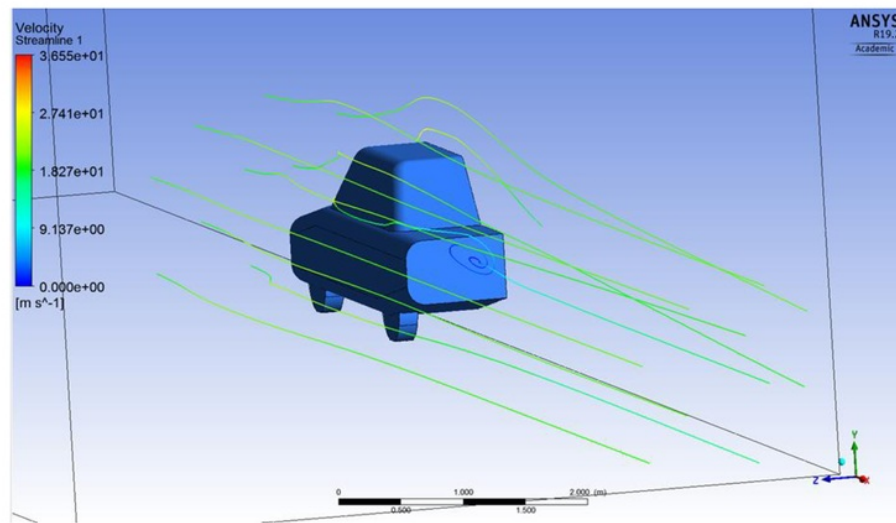


Figure 4-7, Rear Streamlines

As the streamlines were set from a plane in front of the car and limited in number this explains why the streamlines do not extend all the way from the inlet, and why not every area of flow is shown as otherwise it would be difficult to observe what the air flow is doing.

4.1 Aerodynamic Modification

An aerodynamic modification was added to the Mira reference car to observe any difference that it would make, the modification selected was adding a rear wing inspired by motorsport and specifically a Sports2000 MCR racing car. The rear wing profile was added to the model at 7 degrees angle of attack, with endplates used to secure it rather than a central or swan neck mount. The expected results of this modification would be an increase in drag but also an increase in negative lift, or downforce, over the standard car. Below in *figure 4-8* is the modified car geometry.

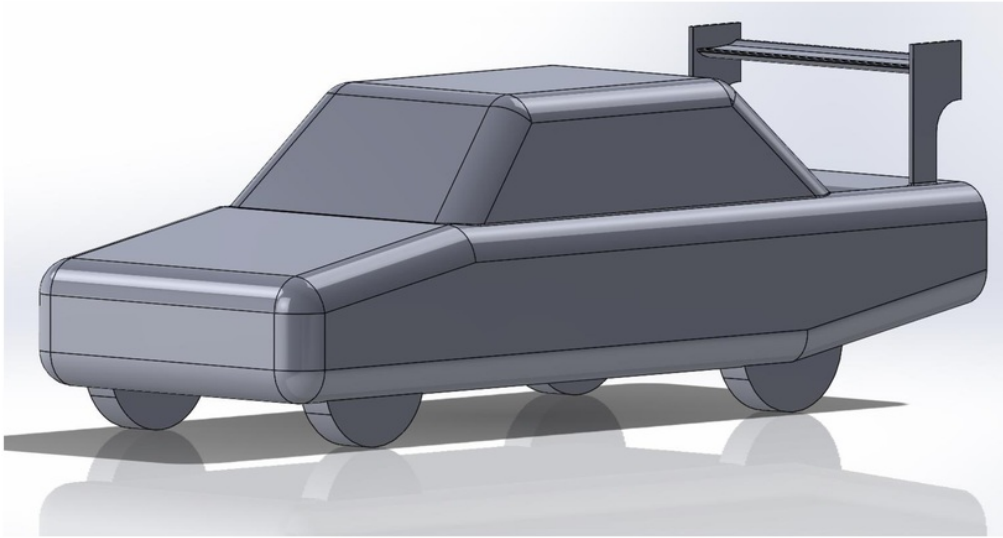


Figure 4-8, Modified Mira Reference Car

In *figure 4-9*, the profile of the wing can be seen close up along with the endplate mount.

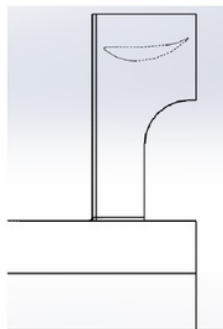


Figure 4-9, Wing Profile

In terms of drag and lift, the wing worked as expected, the drag increased but the negative lift also increased over the standard model. This is displayed in table 6.

Table 6, Standard vs Modified Comparison Table

	cd	cl
Standard	0.2546	-0.0731
Modified	0.3222	-0.3844
Difference	0.0676	0.3113

The same post processing was used to allow a comparison of the difference that the wing made to the standard geometry, this is shown in the figures below.

The velocity contours again show the direction and speed of the air flow over and under the car, whilst this time also showing the influence the wing has had, which has helped to remove some of the turbulence that was occurring on the rear deck of the car.

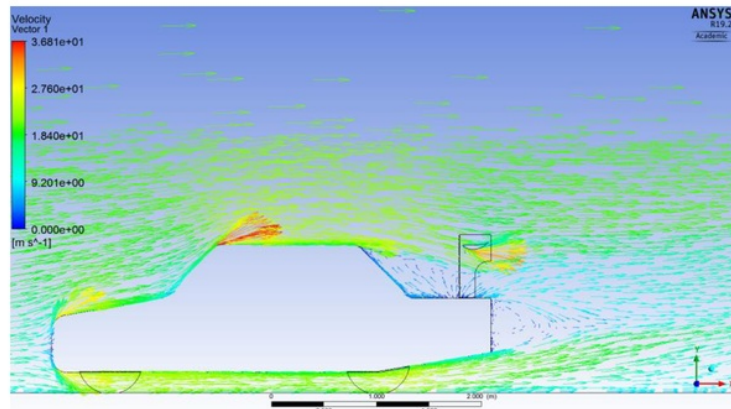


Figure 4-10, Modified Velocity Vectors

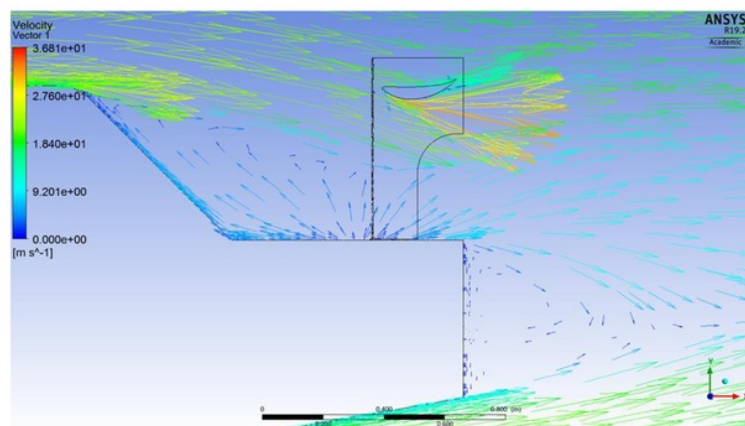


Figure 4-11, Wing Velocity Vectors

The following pressure contours figure 4-12 & 4-13 show that the pressure around the rear deck in general has increased compared to the standard reference car. The contours also show the wing working with high pressure above and low pressure below at the leading edge.

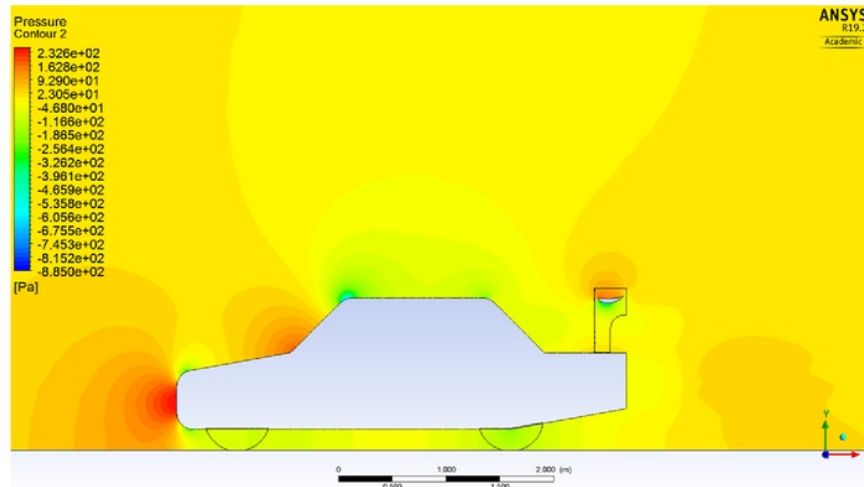


Figure 4-12, Modified Pressure Contours

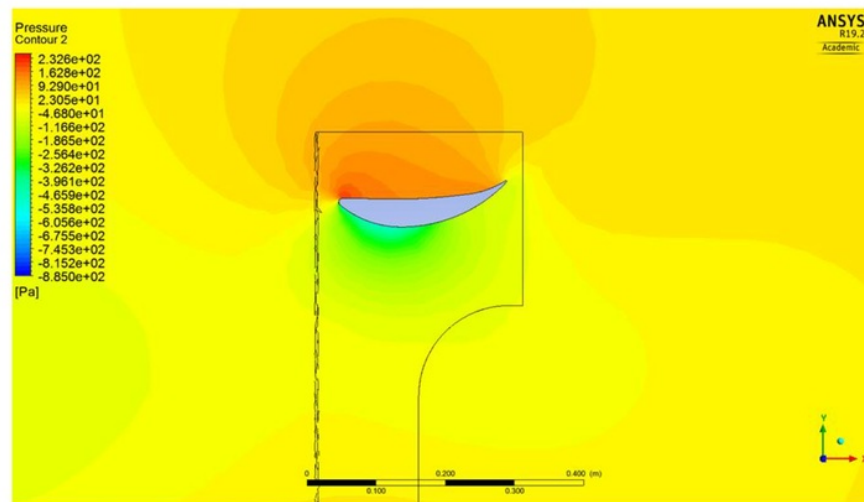


Figure 4-13, Wing Pressure Contours

Figures 4-14 & 4-15 show the wing having an influence on the wake region on the rear deck of the car, the wake region with low velocity is smaller than the standard car, however the wake region behind the car has increased which shows in the increase of drag the modified car has.

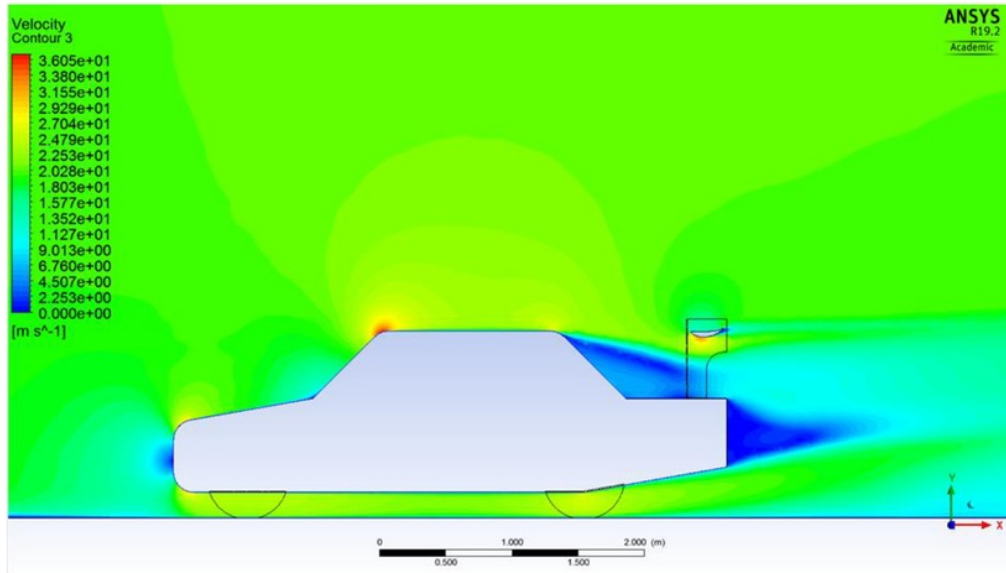


Figure 4-14, Velocity Contours

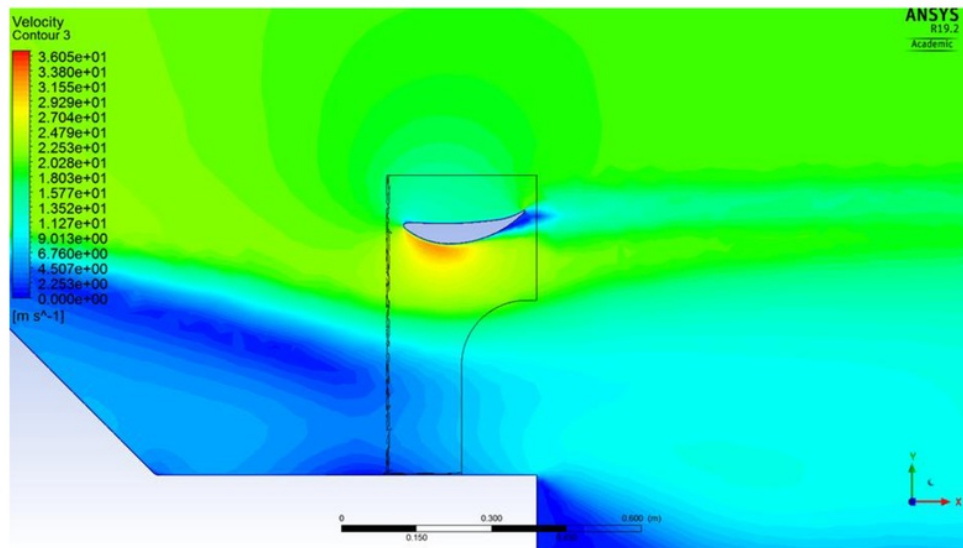


Figure 4-15, Wing Velocity

Figure 4-15 above also shows the wing stalling, stalling is a phenomenon that occurs when separation occurs, this down to a lack of energy in the boundary layer. This means the wing is not working as well as it potentially could do.

The following figures show that the vortices occurring are much the same as the standard car with the exception of the area around the wing, on the edge of the rear deck, the vortex isn't as strong both on the parallel edge and the rear boot lip area. The wing mount however is showing a high number of vortices present on both edges whilst the wing itself is also producing vortices.

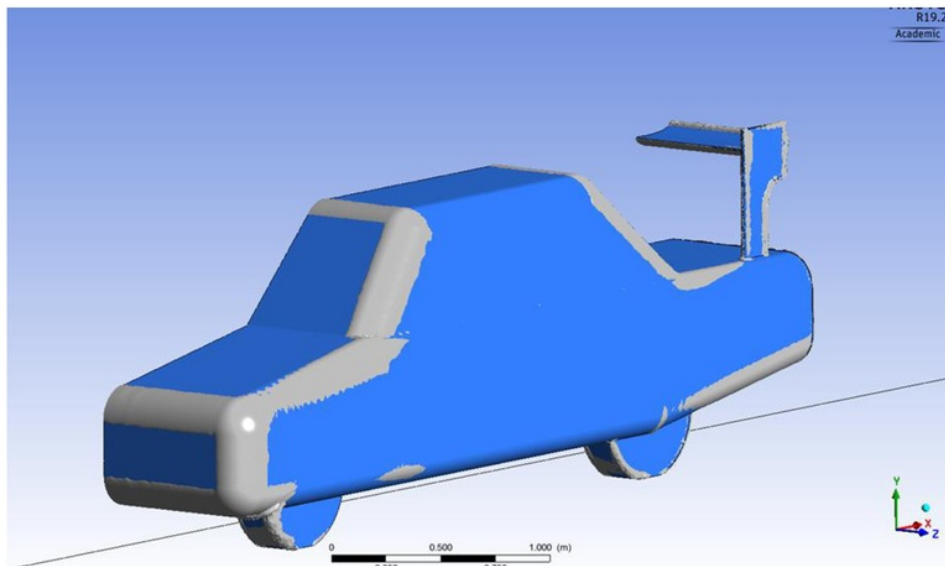


Figure 4-16, Vortex Core Region

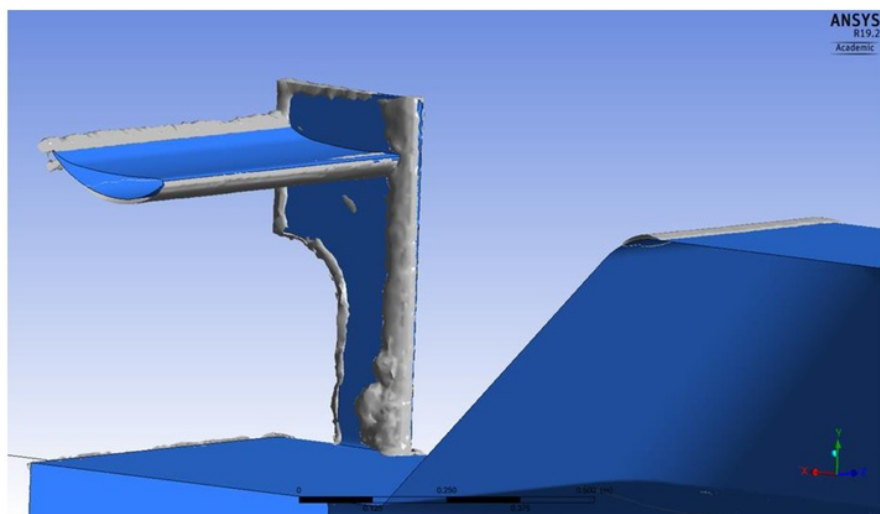


Figure 4-17, Wing Vortex Core Region

Lastly, the streamlines in figures 4-18 & 4-19 show the air interacting around the geometry, this is a good indication of the path the air takes over and under the wing, as well as the wake region produced. The speeding up of the air as it travels under the wing is shown as well as the air accelerating out from the diffuser shape underneath the car. It can also be seen how the recirculating flow from the standard model has been eliminated and is no much straghter and smoother.

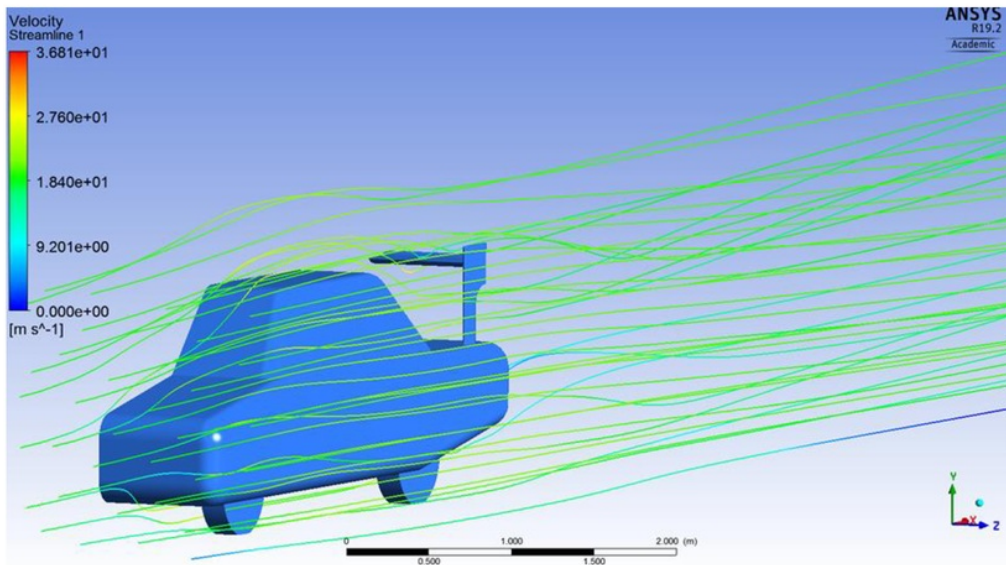


Figure 4-18, Modified Front Streamlines

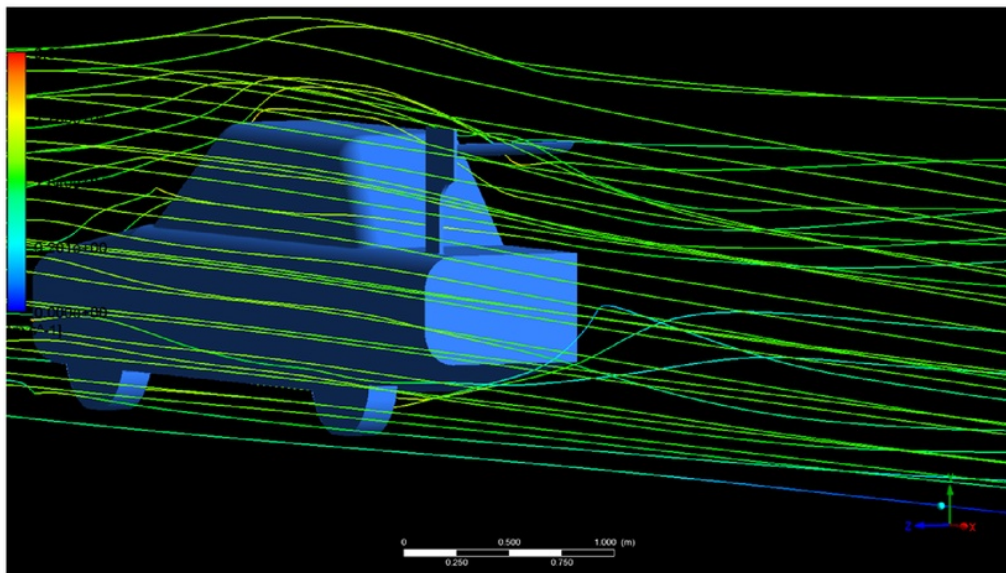


Figure 4-19, Modified Rear Streamlines

5 Conclusion

In total over 100 hours of computing time and over 12GB of data were generated through the course of the studies carried out and the modification to the reference car geometry. The studies have clearly helped to develop a validated base case as shown from the pressure comparison between the computed and wind tunnel pressures over the upper and lower surfaces of the car. From this the modified geometry could be ran and confidence in the results being realistic has been achieved.

The modification carried out to the geometry has been a success as the downforce the base car produced was significantly increased, by as much as over 400% with a drag penalty of only roughly 25%.

This has therefore shown the validity of CFD to complement wind tunnel testing as geometry changes can be carried out quickly and the results can be found out quickly and be reliable as long as a base case has been validated. This saves time and money over having to manufacture the geometry change and then carry out wind tunnel testing.

In an ideal world, both full scale wind tunnel testing and CFD would be used together to optimise the performance of a car, which should then be validated with real on track testing, as there is no substitute for the data gained from that.

6 Recommended Further Work

There are potential areas for improvement within the work that has been carried out as no project is every perfect at the first attempt.

The studies carried out could be more in depth with a wider range of parameters tested, such as density box size and positioning, density mesh size, inflation layer size and growth ratio and localised part meshes on the car in areas of interest.

The wheels of the geometry could be blocked, this helps to eliminate the poor quality of cells around the wheels that has been discussed previously.

More settings within the CFD program could be tested and the results observed to further help the validity of the base reference car case.

Lastly the modification carried out to the car could be further optimised, such as position on the car, location of the wing mount, type of wing mount, angle of attack of the wing and the effect of a gurney flap. The stall that was seen could also be potentially solved with the use of a compound wing.

7 References

- Ansys Fluent, 2006. *Overview*. [Online]
Available at: <https://www.sharcnet.ca/Software/Fluent6/html/ug/node511.htm>
- Ansys Fluent, n.d. *Choosing the Tetrahedral Mesh Method*. [Online]
Available at: https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/icm_user/iuser_tetra_methods.html
[Accessed 18 12 2018].
- Ansys, n.d. *Mesh Quality*. [Online]
Available at: https://www.sharcnet.ca/Software/Ansys/17.0/en-us/help/flu_ug/flu_ug_mesh_quality.html
[Accessed 09 Jan 2019].
- Baker, A., 2011. *Wind turbine blade tip analysis using CFD*. [Online]
Available at: <https://sites.google.com/site/ee535test/andrew-baker>
[Accessed 27 03 2018].
- Eoescipy, 2012. *Youtube*. [Online]
Available at: <https://www.youtube.com/user/eoescipy>
[Accessed 10 October 2017].
- FavCars, 2008. *FavCars*. [Online]
Available at: <https://www.favcars.com/wallpapers-ferrari-f2008-2008-243687>
[Accessed 14 October 2018].
- Frei, W., 2017. *Which Turbulence Model Should I Choose for My CFD Application?*. [Online]
Available at: <https://uk.comsol.com/blogs/which-turbulence-model-should-choose-cfd-application/>
- Katz, J., 1995. *Race Car Aerodynamics*. Massachusetts: Bentley Publishers.
- Lanfrit, M., 2005. *Good Practice CFD*, Southampton: Southampton University .
- Machin, R., 2015. *Introduction to Racecar Aerodynamics*, s.l.: Brook Motorsport .
- Mira, 2018. *Full Scale Aerodynamic Wind Tunnel*. [Online]
Available at: [https://www.horiba-mira.com/our-services/full-scale-wind-tunnel-\(fswt\)](https://www.horiba-mira.com/our-services/full-scale-wind-tunnel-(fswt))
[Accessed 05 December 2018].
- Pachpund, S., Madhavan, J., Pandit, G. & Chimner, T., 2012. *Development of CFD Methodology for Drag Force Prediction on Passenger Car with Rear Mounted Spoiler*, s.l.: The Automotive Research Association of India.

Peugeot, 2018. *Peugeot 1916 Indy 500 Winner*. California : MA Scale Models.

Rao, J., Saravanakumar, M. & Vijay chandar, D., 2007. *External Aerodynamic Flow for High Speed Passenger Car*, Pune: The Automotive Research Association of India.

www.grc.nasa.gov, 2015. *Boundary Layer*. [Online]

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

[Accessed 05 12 2018].

Zhang, Y., Ding, W. & Yu, Z., 2014. *Aerodynamic Shape Optimization Based on the MIRA*, Changchun: SAE.