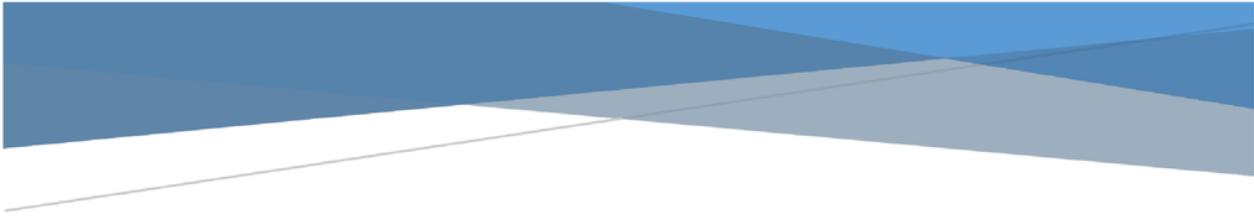


Mira Validation

by Jack Bridgeland

FILE	5368_JACK_BRIDGELAND_MIRA_VALIDATION_128450_1618396327.DOC X (53)		
TIME SUBMITTED	25-JAN-2019 03:19PM (UTC+0000)	WORD COUNT	6099
SUBMISSION ID	99822897	CHARACTER COUNT	38522



MIRA Validation Study

Aerodynamic Vehicle

Testing and analysis

Jack Bridgeland P145335

BEng Motorsport
Engineering

Advanced Computational
Methods

M6X00408

Tim Tudor

Jack Bridgeland (P145335)

Contents

List of figures.....	2
List of Tables	2
List of graphs.....	2
1.0.0 Introduction	4
2.0.0 Background of Computational fluid dynamics and wind tunnels.....	4
2.1.0 Computational fluid dynamics	4
2.2.0 Wind tunnels.....	4
3.0.0 Validation study	5
3.1.0 Domain size	5
3.1.1 Test procedure	5
3.1.2 Results.....	6
3.2.0 Position of geometry in domain.....	8
3.2.1 Test procedure	8
3.3.1 Results.....	9
3.3.0 Domain global mesh size	9
3.3.1 Test procedure	9
3.3.2 Results.....	11
3.4.0 Density box size.....	13
3.4.1 Results.....	13
3.5.0 Density box position	14
3.5.1 Results.....	14
3.6.0 Density box mesh size	15
3.6.1 Results.....	15
3.7.0 Prism layers.....	16
3.7.1 Prism layers on the geometry	16
3.7.2 Prism layers on the road	16
3.8.0 Turbulence model	17
3.9.0 Solid walls.....	19
3.10.0 Mira wind tunnel dimensions	19
4.0.0 Pressure tap comparison	20
5.0.0 Aerodynamic geometry change	23
5.1.0 Modified Mk1.....	23
5.2.0 Modified Mk1 Results	23
5.3.0 Modified Mk1 Analysis.....	24
5.4.0 Modified Mk2.....	27

5.5.0 Modified Mk2 results	27
5.6.0 Modified Mk2 analysis	28
6.0.0 Critical analysis.....	31
7.0.0 References	33

List of figures

Figure 1 Skew Example	4
Figure 2 Domain example	6
Figure 3 Small domain.....	7
Figure 4 Large domain	7
Figure 5 Origin position.....	8
Figure 6 Mesh setting	10
Figure 7 Mesh size comparison.....	10
Figure 8 Detail comparison	12
Figure 9 Density box example.....	13
Figure 10 Density box mesh.....	13
Figure 11 Turbulence model comparison	18
Figure 12 Mira wind tunnel dimensions	20
Figure 13 Pressure tap data	21
Figure 14 vortex core region.....	22
Figure 15 Modified MK1	23
Figure 16 Recirculation on air dam	25
Figure 17 Velocity under Mk1.....	25
Figure 18 Streamline around spoiler.....	26
Figure 19 Separation on fastback	26
Figure 20 Modified Mk2.....	27
Figure 21 Reduced recirculation on Mk2	28
Figure 22 Velocity comparison under Mk1 and Mk2	29
Figure 23 Vortex core region sealing off end-plate	29
Figure 24 Recirculation on rear deck	30
Figure 25 Wake region comparison	30
Figure 26 19m/s Isosurface.....	32

List of Tables

Table 1 Preliminary mesh settings	5
Table 2 Mira notch dimensions.....	5

List of graphs

Graph 1 Domain size results	6
Graph 2 Domain position	9
Graph 3 Domain mesh results	11
Graph 4 Density box size.....	14
Graph 5 Density box position.....	15

Graph 6 Density box mesh size	15
Graph 7 Prism layers on car	16
Graph 8 prism layers on the road	17
Graph 9 Turbulence model comparison	17
Graph 10 Turbulence model comparison 2	18
Graph 11 Wall comparison	19
Graph 12 Wind tunnel dimension comparison.....	20
Graph 13 Upper surface pressure tap comparison.....	21
Graph 14 Lower surface pressure tap comparison.....	22
Graph 15 Mk1 comparison	24
Graph 16Modified Mk2 comparison.....	27

1.0.0 Introduction

This report is aimed at an investigation to find the optimal settings to find the forces acting upon a vehicle using computational fluid dynamics (CFD), validating it against some wind tunnel data of the same geometry from the MIRA wind tunnel. This approach will be an iterative process finding the best results for each parameter and continuing with the optimal settings until the results are representative of the actual wind tunnel data provided.

2.0.0 Background of Computational fluid dynamics and wind tunnels

2.1.0 Computational fluid dynamics

Computational fluid dynamics is predicting what will happen, quantitatively, when fluids flow over or through an object (CFD intro, 2019). External CFD is one of the main applications in the automotive industry (Lanfrit, 2005) which is closely related to the motorsport industry. CFD

There are many different things that can influence the results that can be obtained from CFD some of these are outlined below and how they can affect the results

- CAD or model quality – if the model is very complicated and has many tight areas the software will struggle to mesh it and stay representative of the original geometry, so the mesh size would have to decrease influencing accuracy and computational time
- Domain size – if the test domain is too small there can be an effect on the results in the form of a blockage effect which is seen in actual wind tunnels as well. This can be reduced by using a symmetry wall instead of a solid wall blockage can be controlled in many different ways in a real tunnel and lots of research has been done by many different companies (SAHIN, 2004)
- Mesh quality – if the elements within the mesh are skewed or have a bad orthogonal quality, which means that the elements created are stretched or don't have the correct shape which can influence the results. There is a minimum quality and skew numbers which should be achieved to make the results more accurate which is stated as no more than .001 for orthogonal quality. Skewness is defined as the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution. (sas lp, n.d.). the mesh quality can be checked in Icem, this is shown below in Figure 1

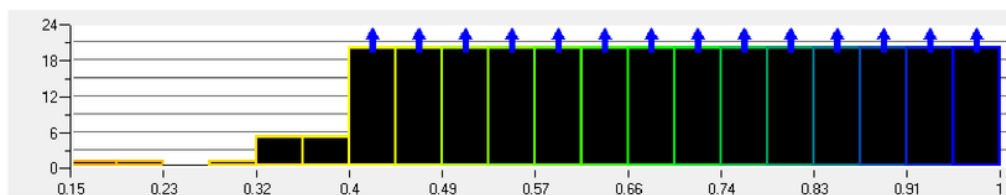


Figure 1 Skew Example

2.2.0 Wind tunnels

Wind tunnels were originally designed and used in the aeronautical sector of engineering to analyse the forces produced by a test body. The first wind tunnel was designed and built by a man called Frank H. Wenham in 1871 in England (Rumerman, 2019) to aid with designing a plane that would be able to fly and look at the forces acting upon it which is the same as CFD but could be done a long time before computers were available.

The Horiba Mira wind tunnel is the only full scale wind tunnel in the UK. (Horiba mira, n.d.) The specification of the wind tunnel can be seen below

- Test section area of 15m length x 7.9m wide x 4.4m high with a cross sectional frontal area of 35m²
- Maximum wind speed is 36m/s (equivalent to 130km/h)
- Maximum weight for the turntable is 4000kg
- Closed test section, open return tunnel
- Balance for measurement of drag, lift and side forces
- Evaluation of rolling, pitching and yawing moments

(Horiba mira, n.d.)

3.0.0 Validation study

This section of the report is to create a representative model within CFD that correlates to the wind tunnel data provided. This is an iterative test process finding the optimal settings within the software provided. For some of the initial tests, for example the domain study a meshing technique had not yet been set, because of this some preliminary meshing method were chosen (sas lp, n.d.). These are outline below in table 1

Meshing parameters	Chosen value
Global volume mesh size	1m
Mira vehicle mesh size	0.01m

Table 1 Preliminary mesh settings

Due to the Mira notch vehicle being symmetric it was possible to do these tests as half vehicle models using a Boolean which reduced the computational time dramatically thus making the whole process quicker.

3.1.0 Domain size

The domain size is a crucial part of simulating fluid flow over an abject. If the domain is too small it could have some adverse effects on the results, for example if the domain is too small it could create a blockage effect (SAHIN, 2004). Blockage effect is where the fluid that is displaced by the test piece must go around it and the faces of the domain can affect how this happens thus changing the values produced. As stated in a paper by Sahin. These blockage effects need to be corrected to comprehend the test results similar to those of the actual road conditions. (SAHIN, 2004) this means that to accurately reduce the blockage effect either a factor needs to be added or the domain needs to be big enough to not have an effect

3.1.1 Test procedure

The domain size for the testing was determined by measuring the dimensions of the Mira Test notch and then multiplying each dimension by a factor to find the height, width and length to input for the domain. The dimensions of the Mira car are shown below in table 2

Mira car dimensions	Size (m)
Length	4.177
Width	1.421
Height	1.625

Table 2 Mira notch dimensions

A range of domain sizes were tested from two times the size of the geometry up to ten times the size. The geometry was positioned in the centre of the total domain for every test to keep the results constant and relatable. Below Figure 2 shows a domain and half of the Mira test vehicle set up ready to mesh.

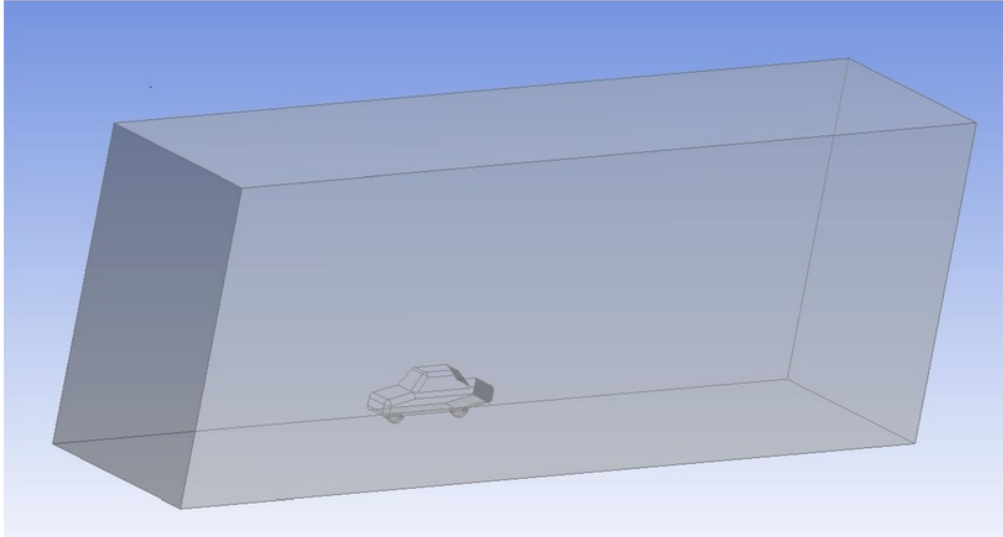
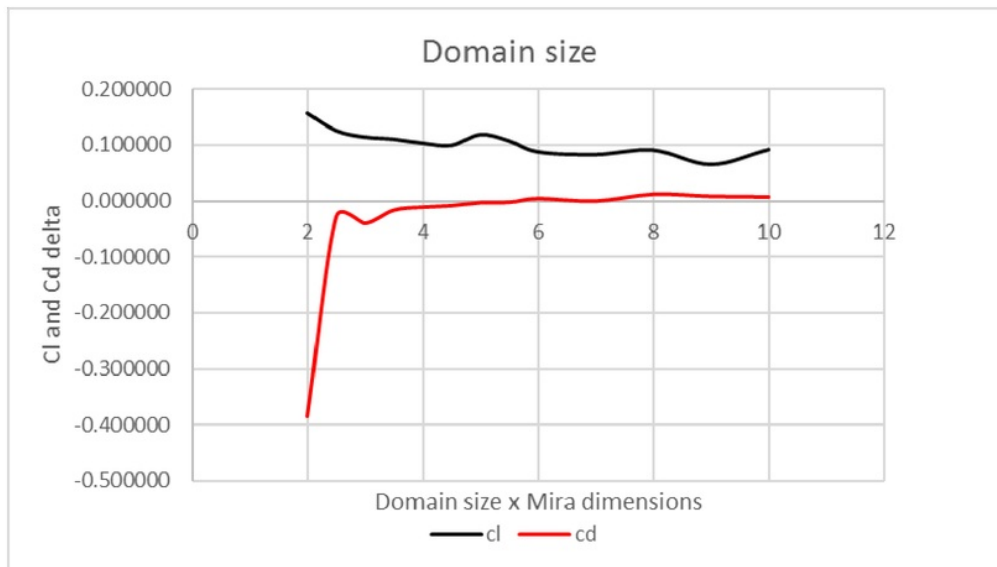


Figure 2 Domain example

3.1.2 Results

Graph 1 below shows the overall results for the domain study parameter sweep



Graph 1 Domain size results

These results are showing Cl and Cd delta from the Mira data provided. From these results it was concluded that the domain needed to be at least six times the size of the Mira notch further

investigation into how the flow was reacting to the sides of the domain within the post processing section of the software were done to see if any blockage effects were effecting the results. Figure 3 below shows some blockage effect.

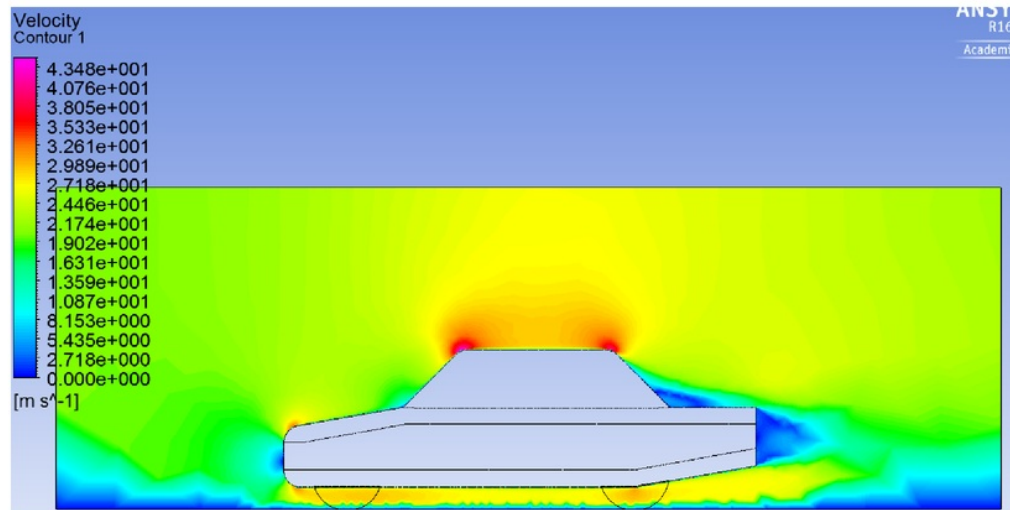


Figure 3 Small domain

From this velocity contour you can see that the fluid within the domain at the boundary on the top it is being effected by the wake protruding from the top of the vehicle. It is also clear that the wake off the rear of the vehicle has not been fully captured as it has not stabilised before the domain ends. The domain below shows the 7X investigation which was decided to be the optimum due to how stable the run seemed to be and the results generated

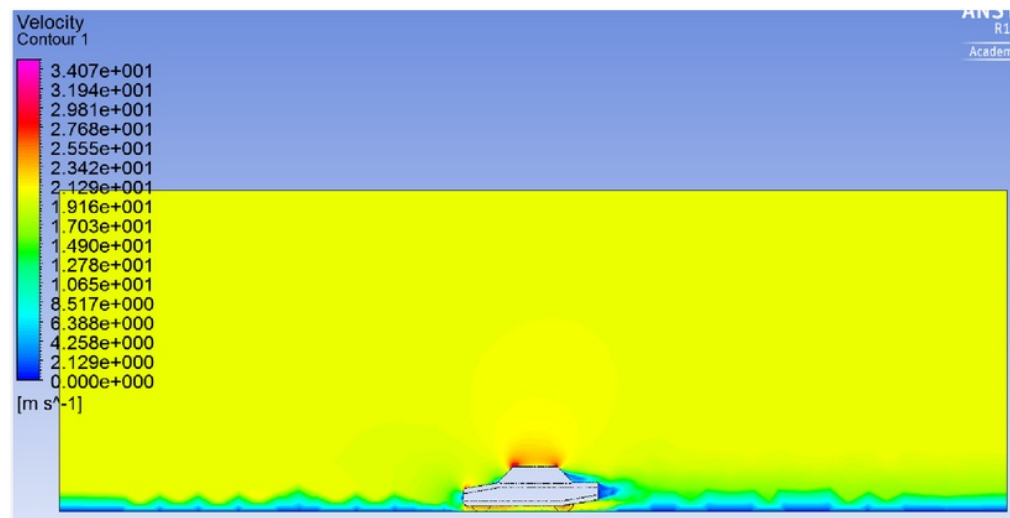


Figure 4 Large domain

From this you can see that the wake behind and on top of the vehicle has been captured and that the sides of the domain are not effecting the results provided. At this stage it is worth mentioning the separation on the road surface. This is due to the speed of the air on the surface is zero this is

not being fully captured because the mesh quality has not been refined yet and there are no prism layers set which help capture separation.

3.2.0 Position of geometry in domain

This section of the process is to investigate the position of the Mira not vehicle within the domain chosen in the first parameter sweep. The position can affect how much of the rear wake of the vehicle is captured due to if it is too close to the pressure outlet some of the turbulence might not be registered inside the domain. There are some guide lines which state for automotive application it should be 3 car lengths in front and 5 behind. (Lanfrit, 2005)

3.2.1 Test procedure

This was done at each meter in the domain starting from the front to the back until the geometry is just off the rear end of the domain. This was measured from the origin of the geometry which was placed at the centre of the front wheel where it will meet the floor to the velocity inlet of the domain. Shown below in Figure 5

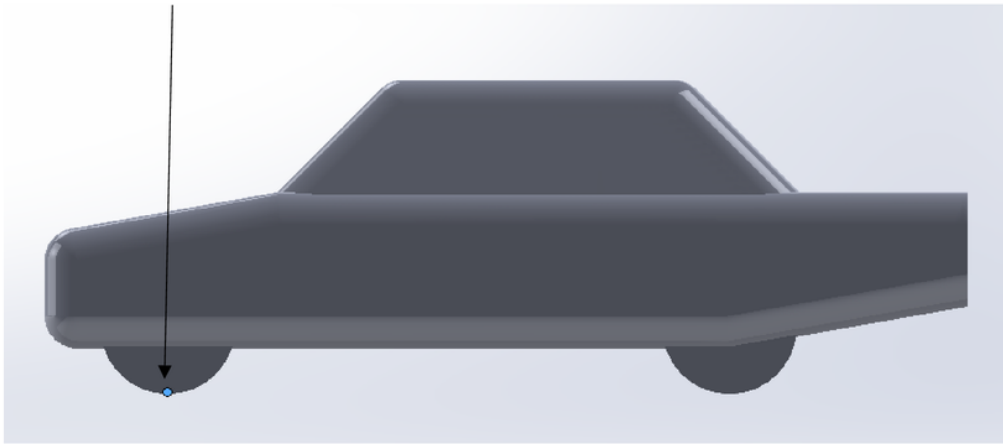
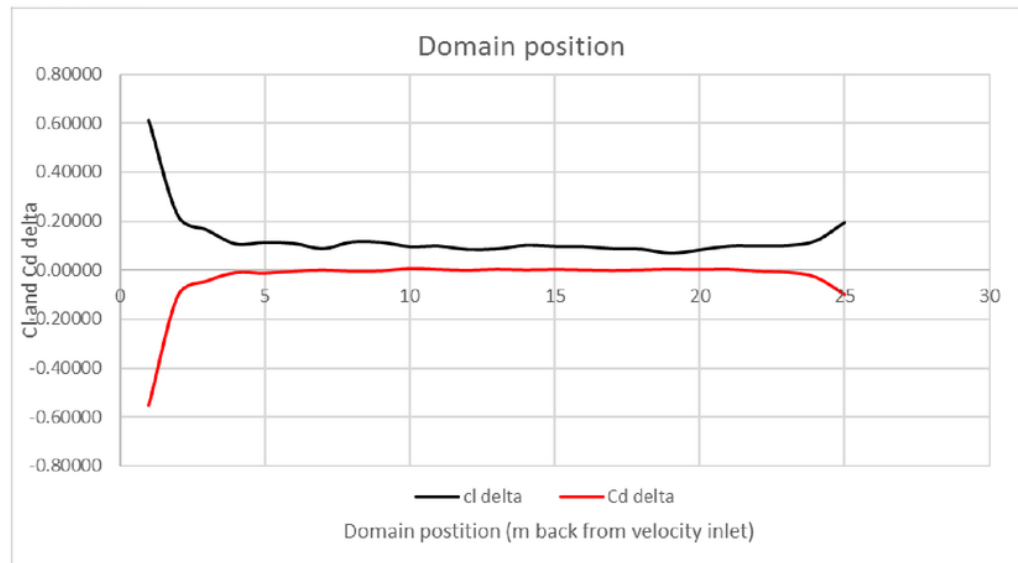


Figure 5 Origin position

Due to the domain overall length being 30 metres long it meant doing 25 runs to investigate this section but due to the simplistic mesh settings used at this point in the investigation computing time was not effected much.

3.3.1 Results



Graph 2 Domain position

Looking at these results the conclusion was that when the geometry is close to both the velocity inlet and the pressure outlet the results are dramatically affected. Between five and twenty metres seems to be the optimum position within the domain. This is a wide window for the geometry to be positioned in so looking at the delta from the given data the closets was at 12m back from the inlet. This means that the best results are produced by having the vehicle slightly farther forward than the centre of the domain. This is supported by a paper by Landfrit that suggests the rough results that I have generated here.

3.3.0 Domain global mesh size

The domain mesh is the size of the mesh in the main body of the test piece. This does not include the surface mesh of the test piece or the mesh within the density box. To be able to capture how the flow reacts to an object the mesh needs to set accordingly. If the mesh is too large it will not capture small areas of detail. A big limiting factor to how small you can set the mesh size is the computing power. The smaller the mesh the more elements there are so more calculations are done, this can dramatically increase a runs time but it can make the calculations a lot more accurate. (W. Seibert, 2002) A compromise in this section can be made by having a larger mesh due to the main body will not have much detailed flow structures compared to close to the vehicle where a lot of turbulence will take place.

3.3.1 Test procedure

The global mesh setting are controlled by the section shown below. The larger the number the larger the mesh will be within the domain.

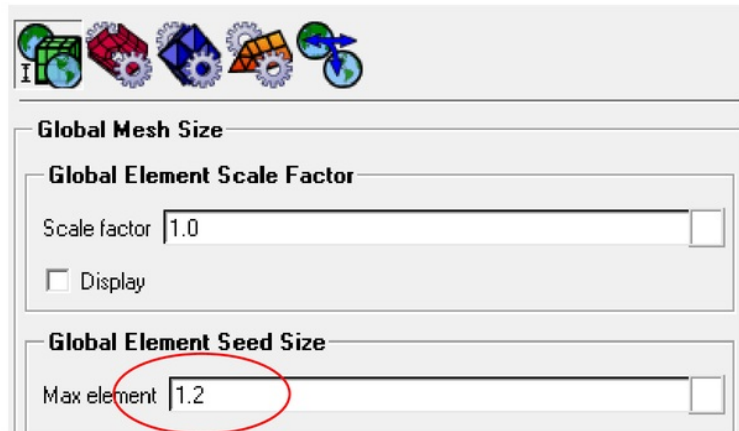


Figure 6 Mesh setting

As stated previously the domain mesh for the previous tests were set at 1m. Test runs were conducted with a larger and smaller element size in the mesh to compare how it changes the results. The maximum size tested was 1.4m and the smallest was 0.3m. Below shows a comparison between a 1.4m mesh size and a 0.3m global mesh size.

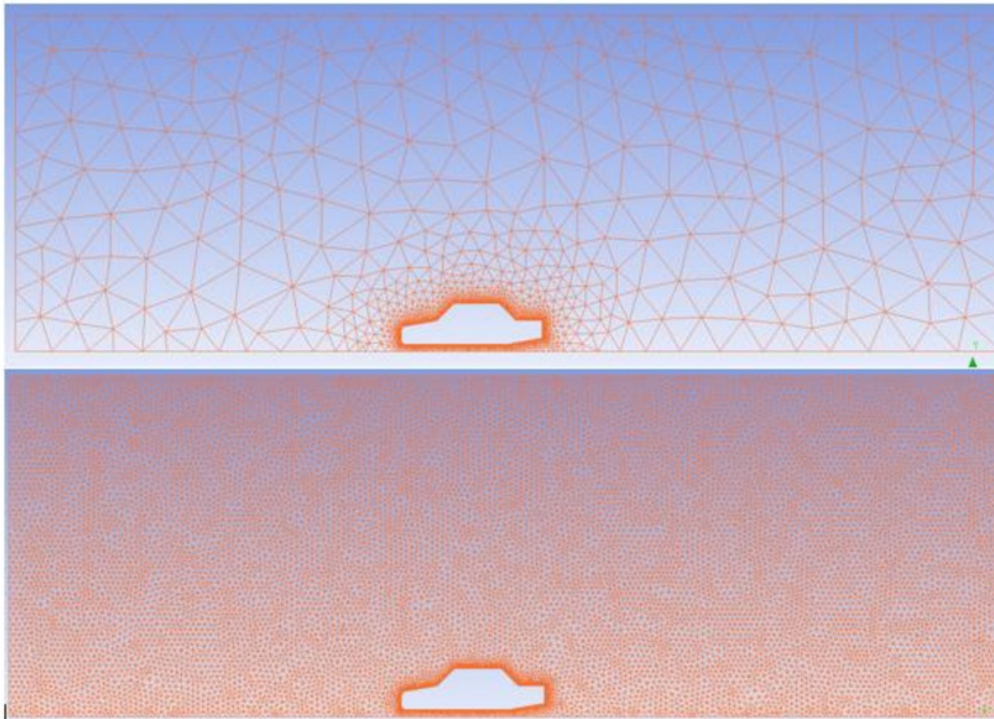
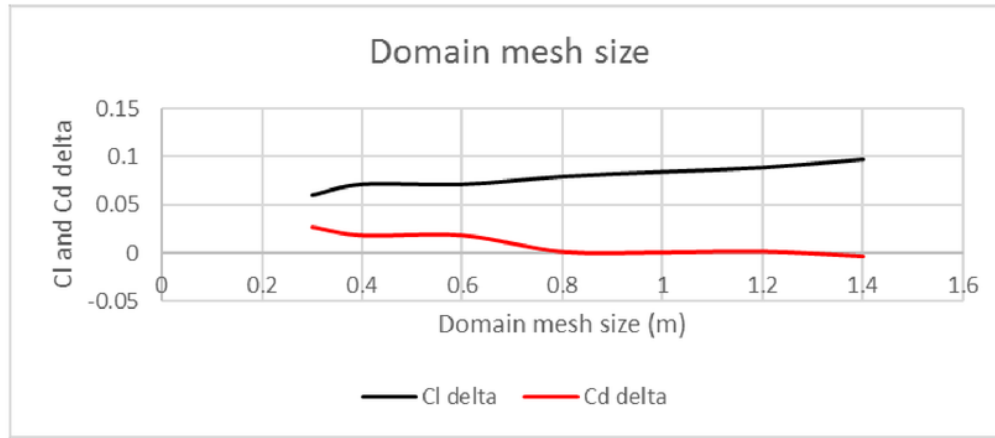


Figure 7 Mesh size comparison

You can see a clear difference between the two different global mesh sizes where the 0.3m mesh has a larger amount of elements. With the 1.4m mesh size you can also clearly see the mesh size

decreasing towards the model. This is some automatic smoothing to bring the elements to the same size as the elements on the piece itself. This is controlled by the scaling factor.

3.3.2 Results



Graph 3 Domain mesh results

From these results it seem that 0.8m and 1m seem to be the best results for the Cd. The smaller the elements go the delta gets larger but it is the opposite for the Cl delta.

Looking at the results in the post processing comparing the runs against each other it is clear that the smaller the domain mesh is the more detail it captures. This is shown below.

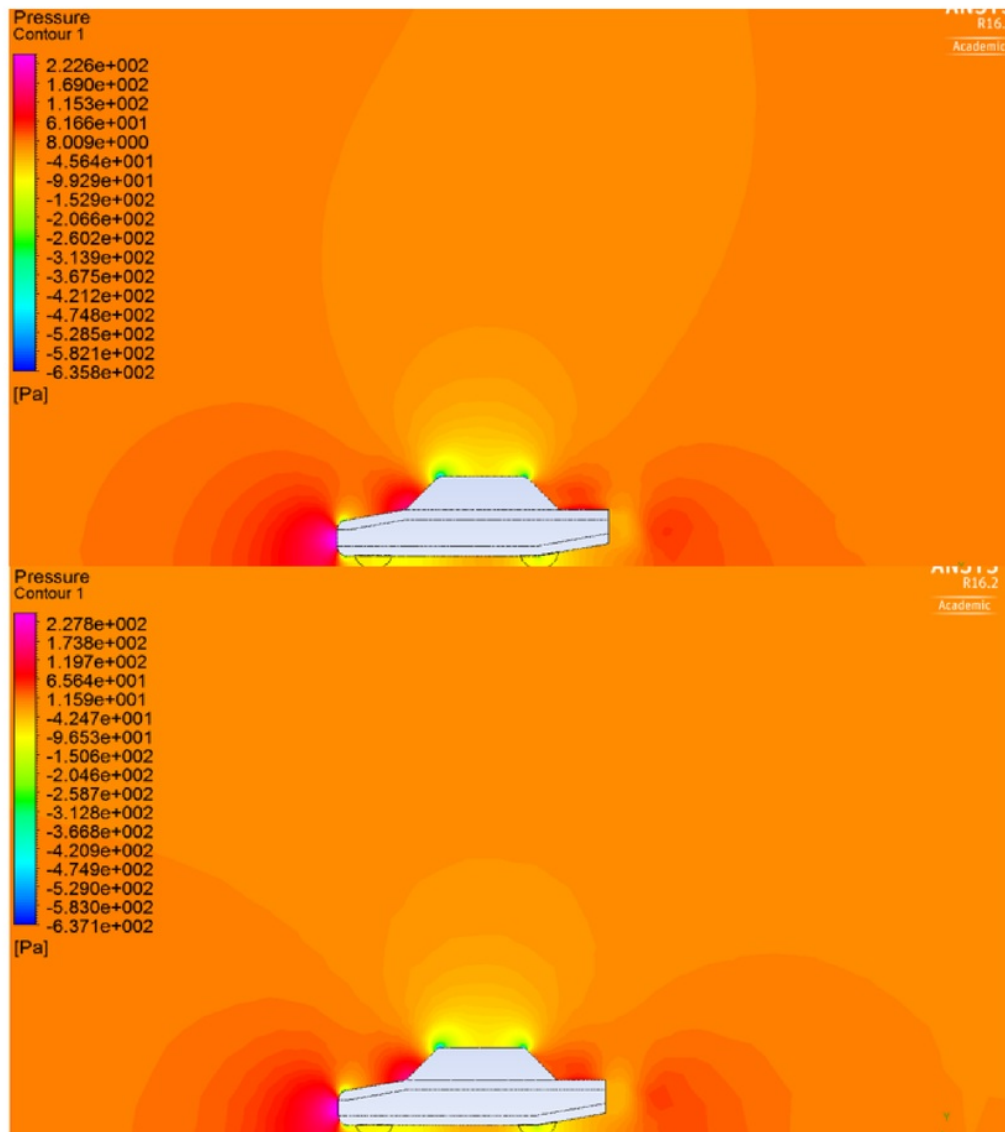


Figure 8 Detail comparison

Each of these results have all the same setting and are showing the same point. You can see that the top picture shows more detail and the curves are much smoother. The reason they are smoother is because the elements are smaller thus not having to estimate a large section of data and also due to the elements being smaller the curves can blend in with each other better.

It was decided to continue with the 0.8m element size due to the closer results than the 1m mesh. When the global mesh was reduced smaller than 0.8m the computing time increased dramatically from around 2 hours to converge up to around 5 hours for the 0.3m. This made this choice not viable because work efficiency is important and due to this only being the global mesh and not the surface or density box mesh the computing time will only increase from here.

3.4.0 Density box size

A density box is an area of denser mesh surrounding the test piece which has set dimensions and mesh size. This section is to look at the size of the density box that surrounds the test piece. This can affect the amount of turbulence that is captured behind the vehicle thus influencing the CI results of the vehicle. Below outlines what a density box looks like before and after meshing.

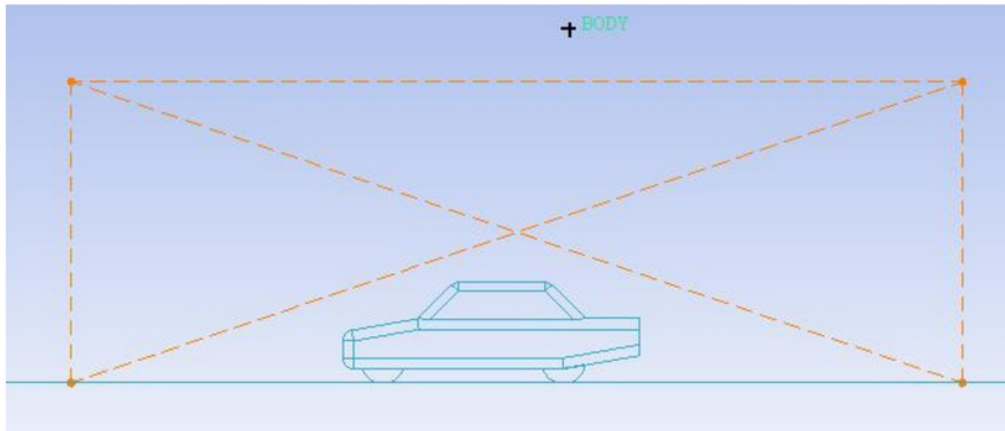


Figure 9 Density box example

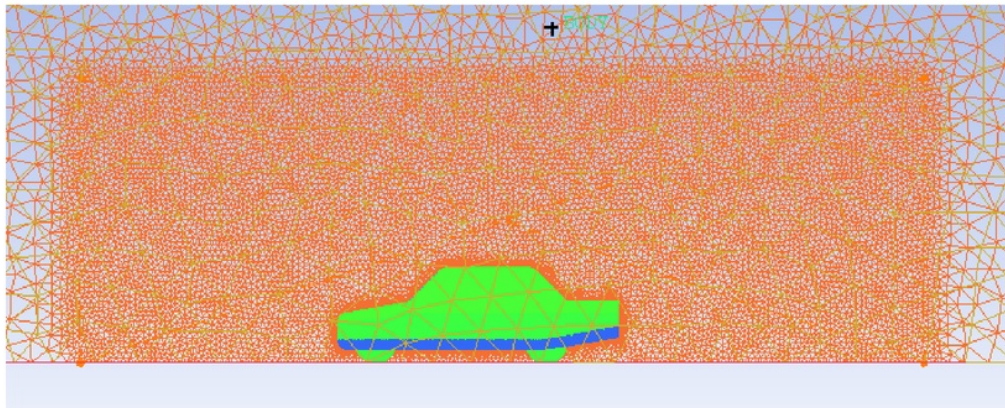
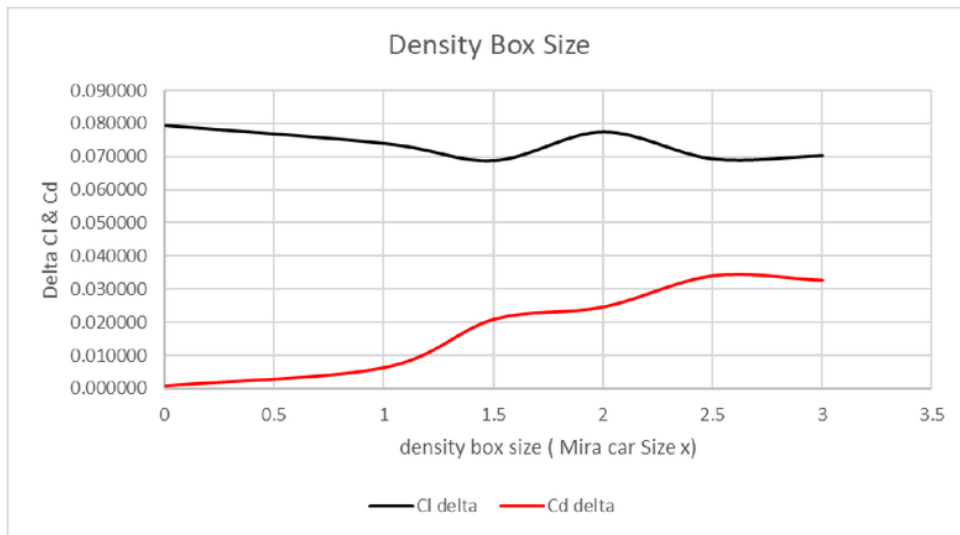


Figure 10 Density box mesh

3.4.1 Results

Below shows the results from multiplying the density box at 0.5 intervals of the Mira notch dimensions



Graph 4 Density box size

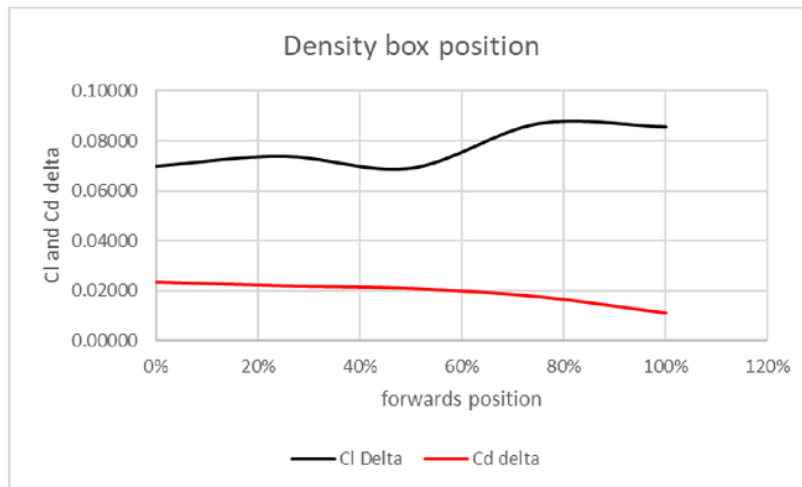
Each density box was set to have a mesh size of 0.1m and to have the test car in the centre for all these runs so only the size was affected. From these results it was concluded that the density box should be at least 1.5 times the vehicles size, this is due to the CI getting closer to the actual wind tunnel data, but the Cd did get further away. Further investigations could be made looking at the height and length of the density box in separate investigations to make sure that the box is long enough and not too high but due to time constraints and run times this seemed to be not viable. But the chosen parameters should be reliable enough to help correlate to the real data.

3.5.0 Density box position

The density box position can also influence the results. If the density box is too far forward it might not catch the full turbulence of the vehicle. The density box that was found to be optimal size from the previous section was used. It was then moved all the way forward and then moved back towards the rear of the vehicle until it lined up with the front of the geometry

3.5.1 Results

The graph below shows the results of the study



Graph 5 Density box position

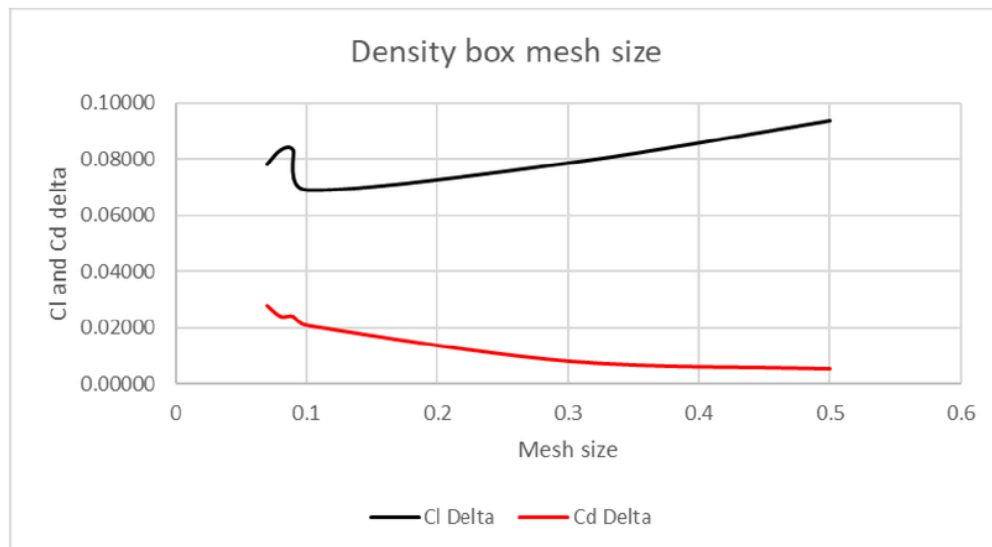
From the results above it shows that when the density box is further forward the drag of the vehicle gets closer to the target from the actual wind tunnel, but the lift gets further away. It seems that the best results seem to be with vehicle in the middle of the density box.

3.6.0 Density box mesh size

The mesh inside the density box was then investigated. this can have the same implications as the domain mesh with there could be a compromise of accuracy against computing time. A study was undertaken to find an optimum mesh size within the density box.

3.6.1 Results

The graph below shows the results from the study. The mesh size ranges from 0.5m down to 0.07m



Graph 6 Density box mesh size

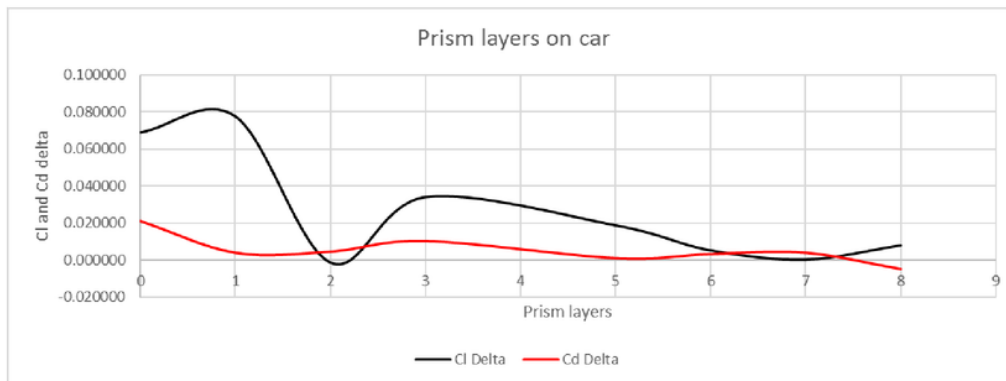
As it seems with a lot of the results that have been found to make either the C_l or the C_d closer to the actual wind tunnel data it has an adverse effect on the other, due to this it was decided to proceed with the 0.3m size elements in the density box. This was chosen because of the better results for the C_l with minimal effect on the C_d .

3.7.0 Prism layers

Prism layers are used over any solid object to capture the separation created by the friction created by the surface. (Aerospace, Mechanical & Mechatronic Engg, 2005). Each solid part can have its own number of prism layers. Each layer will be extruded from the previous. This can have a huge impact on the quality and skew of the mesh. If either of these are too high the results will be effected or not even be able to import into Fluent. Below shows what it looks like when prism layers are added to the mesh.

3.7.1 Prism layers on the geometry

A study was conducted about the number of prism layers on the surface of the Mira Notch. The results can be seen below. There is no maximum amount of prism layers that can be added to a surface but it was decided to only go up to a maximum of eight due to the gap between the base of the Mira notch and the road surface.



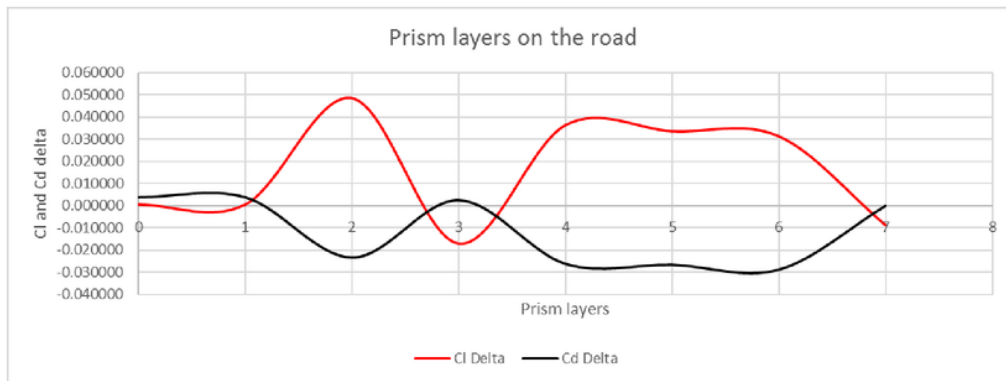
Graph 7 Prism layers on car

From these results it was concluded that above six seems to produce the closest results in both C_l and C_d with a massive improvement on the C_l compared to having none. It was decided to put seven on the vehicle for the next study

3.7.2 Prism layers on the road

While putting prism layers on the road there was some problems with the mesh quality due to a small section of elements between the bottom of the vehicle and the road. When the prisms got too close it created an area with bad orthogonal quality between them. This led to having to change the prism layer max height for a couple of the runs to be able to test them. Anything above two prism layers had to have a slightly less height.

The graph below shows the results from putting prism layer on the road with the optimum amount on the Mira notch of seven.

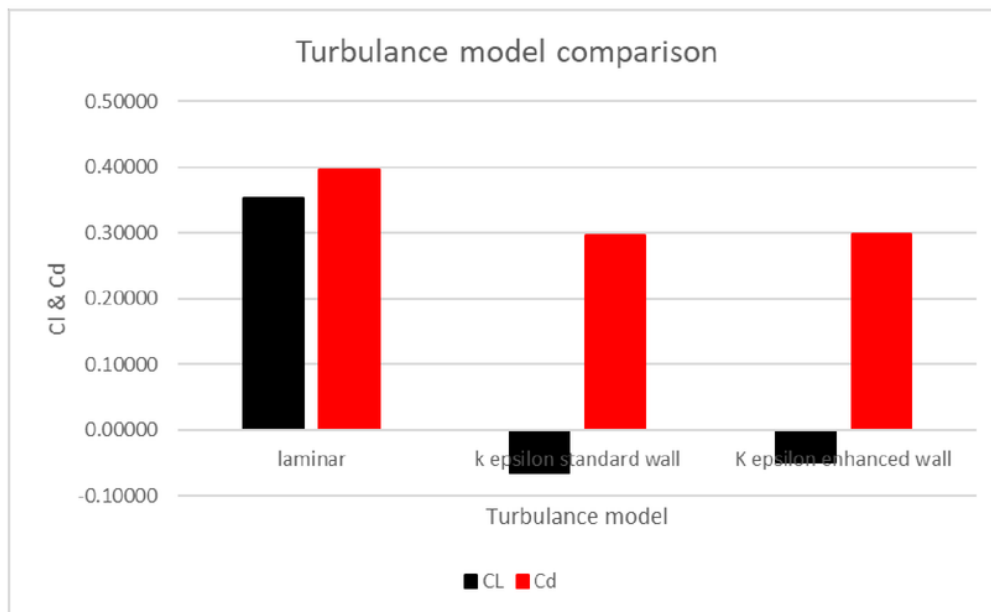


Graph 8 prism layers on the road

Due to the change in the prism layer height stated previously this test was not only changing one variable, this led to only use one prism layer on the road due to how close the results were to the actual wind tunnel data.

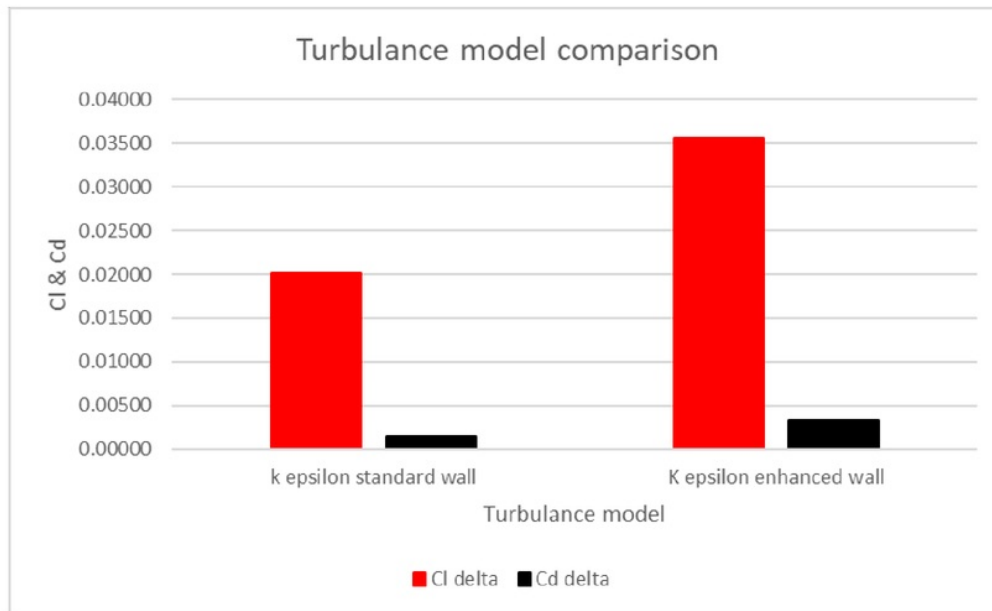
3.8.0 Turbulence model

Within fluent there are many different turbulence models that can be used. Each different turbulence model will predict how the turbulence is acting and the affect it will have on the models results in general. A brief study into a few different turbulence models was undertaken to see t effect of them on the results. These results can be seen below.



Graph 9 Turbulence model comparison

K-Omega was also tested but the results obtained were not accurate. The laminar turbulence model also does not provide any decent results. Due to this factor the two K-epsilon results were compared against the actual wind tunnel data. This can be seen below.



Graph 10 Turbulence model comparison 2

From these results it was concluded out of the turbulence models that were tested the K-epsilon standard wall treatment seemed to give the best results. Below shows how the two different turbulence models predict the turbulence different to each other.

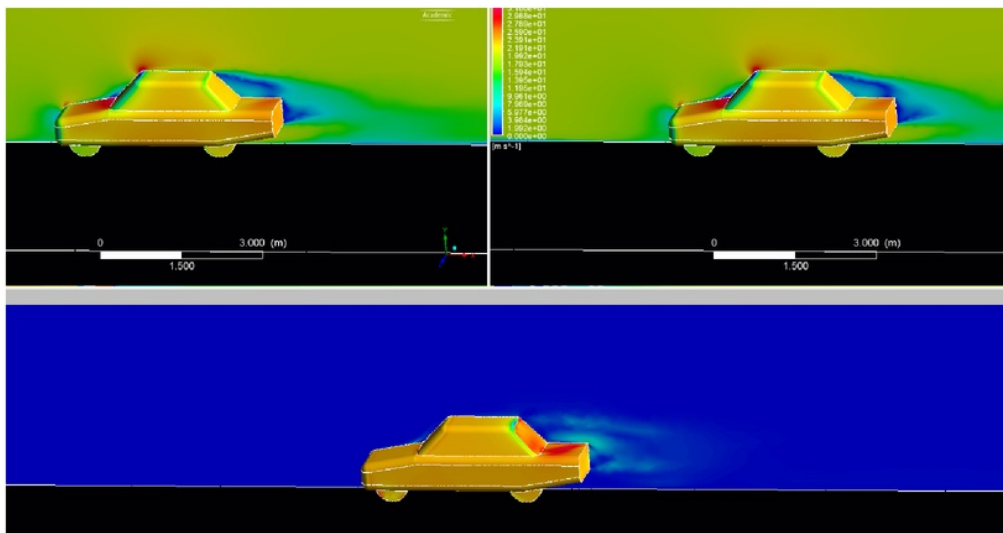


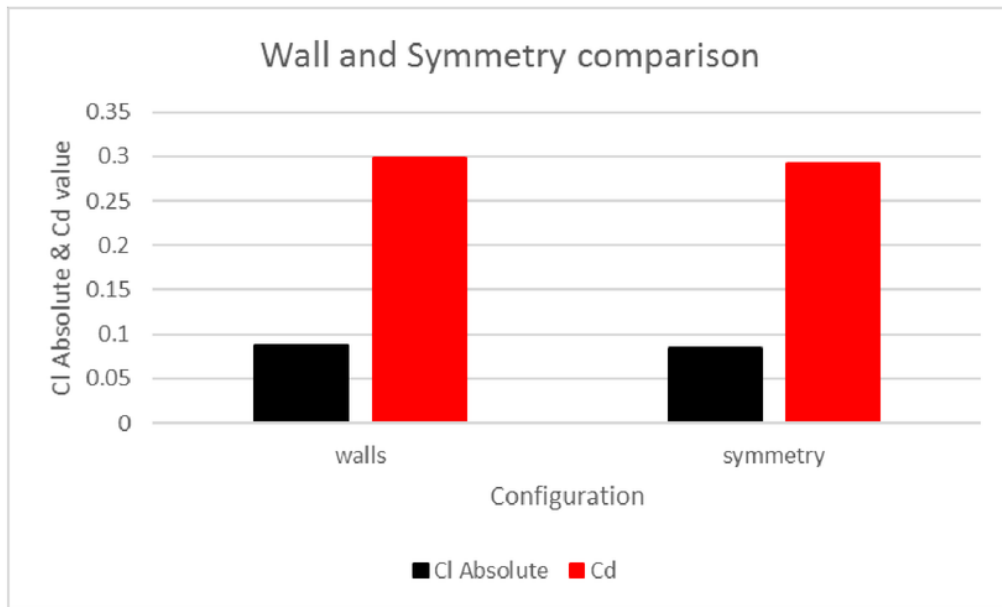
Figure 11 Turbulence model comparison

As you can see there is a clear difference in the speed of the air flow in the wake of the rear screen. Also on the rear deck of the boot there is quite a pressure difference in the two different turbulence models.

Further investigation could be done into other turbulence models but that would require the rest of the setting to be tailored to get decent results.

3.9.0 Solid walls

So far the whole of this investigation the walls have not been modelled as walls but instead as a symmetry. A symmetry wall does not create an adverse effect on the results as it is not modelled as a solid wall. Due to the wind tunnel having physical walls it was necessary to test my settings with the sides and top as an actual wall, this was conducted using the domain settings found to be optimal in the previous sections. Below shows a comparison between using symmetry and a solid wall.



Graph 11 Wall comparison

From these results it is visible that the solid walls made a slight difference to the results compared to the symmetry walls. The wall actually increased the CI and Cd of the geometry. To see why some post processing was done to compare the cases.

3.10.0 Mira wind tunnel dimensions

As my domain is not the same size at the Mira wind tunnel a run was done with my mesh settings to see what the effect of running a domain the same size with sold walls. The dimensions of the Mira wind tunnel are shown below

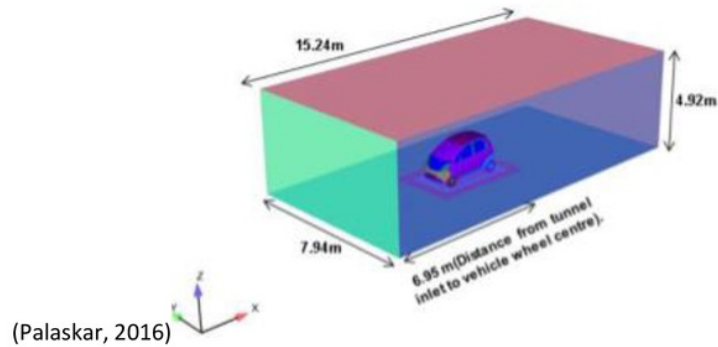
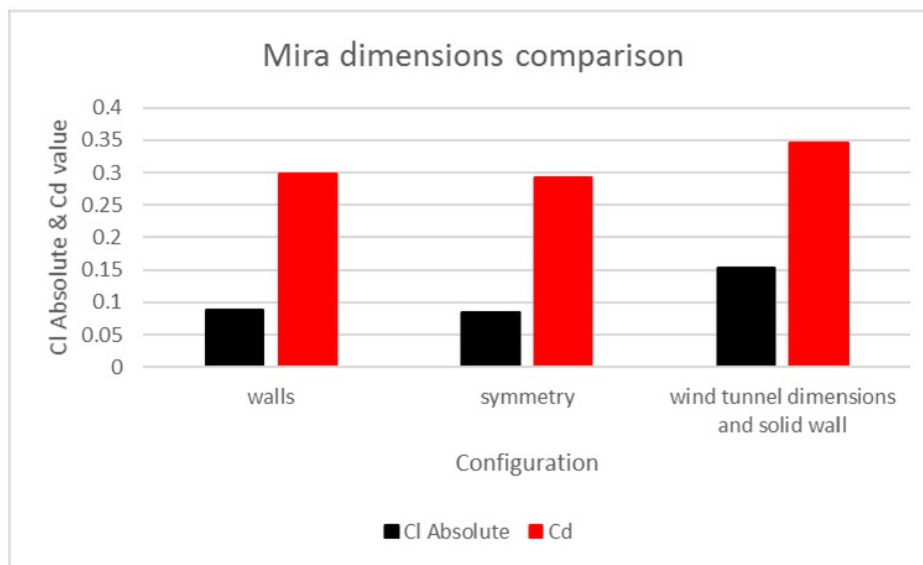


Figure 12 Mira wind tunnel dimensions

The graph below shows a comparison between this run and my final setting with symmetry's and solid walls.



Graph 12 Wind tunnel dimension comparison

With the domain set to the Mira wind tunnel dimensions the CI and Cd increase quite a lot. These results have deviated away from the actual wind tunnel data. This suggests that the setting used were not correct to run in such a small domain and they also seem to be tailored to having a symmetry wall instead of solid walls. The settings could be tailored to suit this small domain and come out with roughly the same final results.

4.0.0 Pressure tap comparison

With the data provided there was data that showed the pressure on the top and bottom surface of the model at different points. This section of the report is to compare the results found from the validation study parameter sweeps with the data provided. Below shows the original pressure tap data that was supplied.

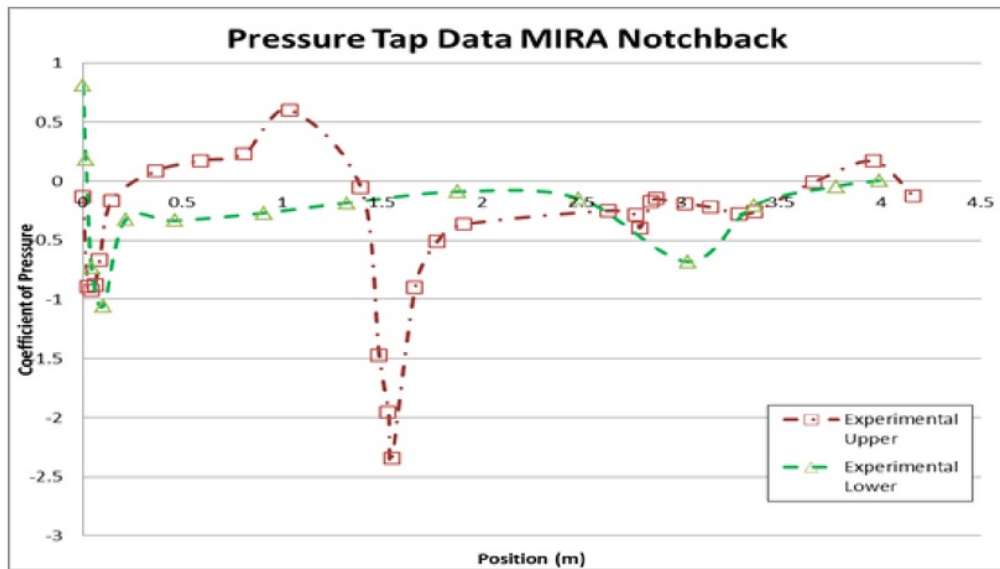
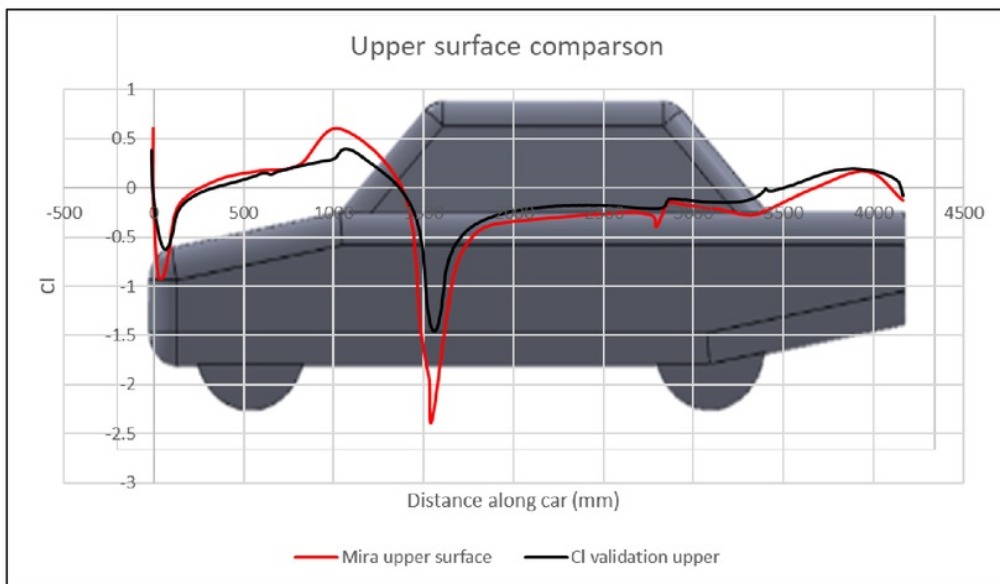


Figure 13 Pressure tap data

To be able to compare these results and export into excel was done from Fluent to overlay my results to compare the differences. Below shows a comparison of the upper surface from my final run



Graph 13 Upper surface pressure tap comparison

As you can see the peak pressures on my validation results are not as high over the top of the test vehicle. Looking at the large negative peak on the top of the windscreen on the Mira test there is more separation around the radius. This could be due to the turbulence model that I have used for my experiment. This could also be due to the surface of the model not being the same as it is in real life, so the boundary conditions will be different. Along the slant of the rear screen my results differ

quite a bit from the wind tunnel data. To make my results closer if I was to do extra runs I would put a density box with a smaller mesh size around the rear screen to fully capture what is happening. As you can see from this vortex core region that there is some separation on the rear deck which could be captured better with the density box in place making the pressure taps be more accurate.

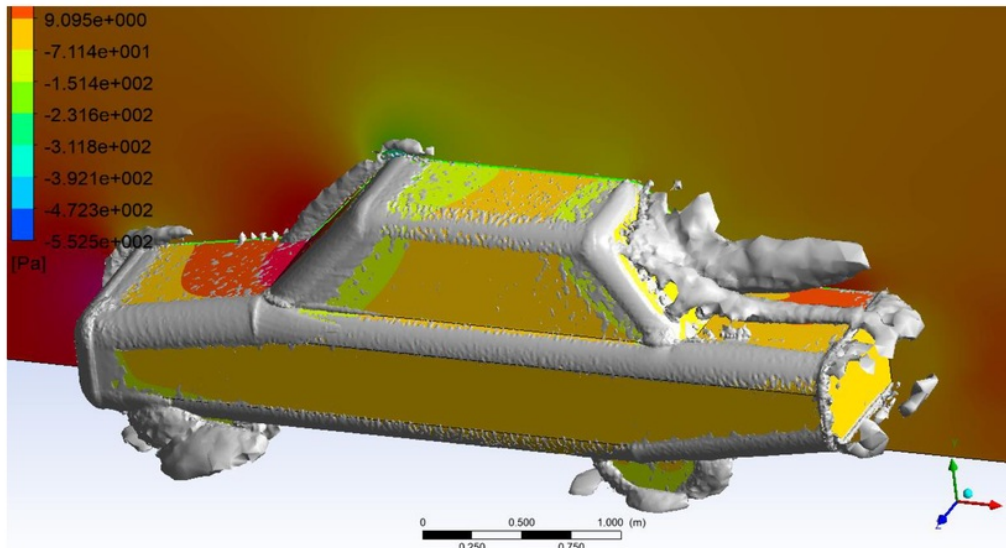
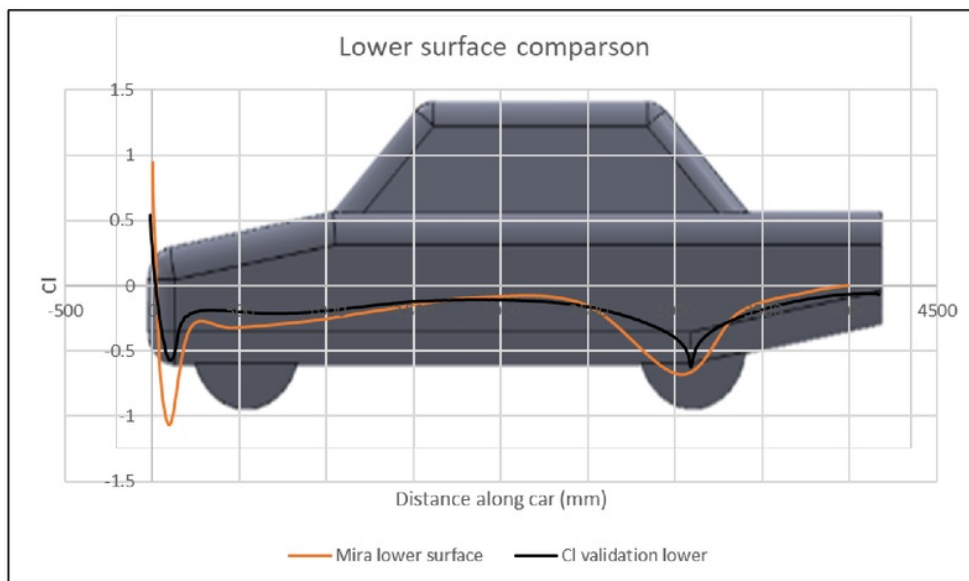


Figure 14 vortex core region

Below shows the comparison of the lower surface.



Graph 14 Lower surface pressure tap comparison

It is clear along the bottom of the vehicle that the peak pressures on the actual wind tunnel data are higher than my results again this could be due to a few factors outlined previously. At the kick point of the rear diffuser area there is a considerable difference with the two results. This could be due to

the amount of pressure taps along the bottom of the real car only being fifteen. The results from the CFD take results from many more points. Due to there being less points on the real car in graphical form there will be a lot more estimating between the points. This is what I believe to be happening here and as a result the graph curves a lot more, but this does not explain the higher peak. Also, the Mira wind tunnel has what's called a boundary layer stop just in front of the test area which stops the build up of the boundary layer. This can also effect the flow under the vehicle thus changing the pressures.

5.0.0 Aerodynamic geometry change

This section of the report is to investigate an aerodynamic modification to the Mira notch vehicle. An air dam, ducktail spoiler and a fast back which has actually been tested in other papers which stated "The "fastback" has a 23° backlight angle (compared to 45° for the "notchback") and a commensurately shorter boot. This leads to the flow remaining attached over the backlight" (A. P. Gaylard, 1998). The ducktail added to this as well should create some good results to produce downforce. The overall C_l , C_d and efficiency of the geometry will be analysed with some explanation as to why these modifications have worked or not worked by examining it in the post processing. Below shows a comparison of the original Mira data plotted with the pressure over the surface of the geometry tested previously.

5.1.0 Modified Mk1

The first set of modifications were just a trail run to find out how the modifications will affect the results. Below shows the CAD of the modified Mk1.

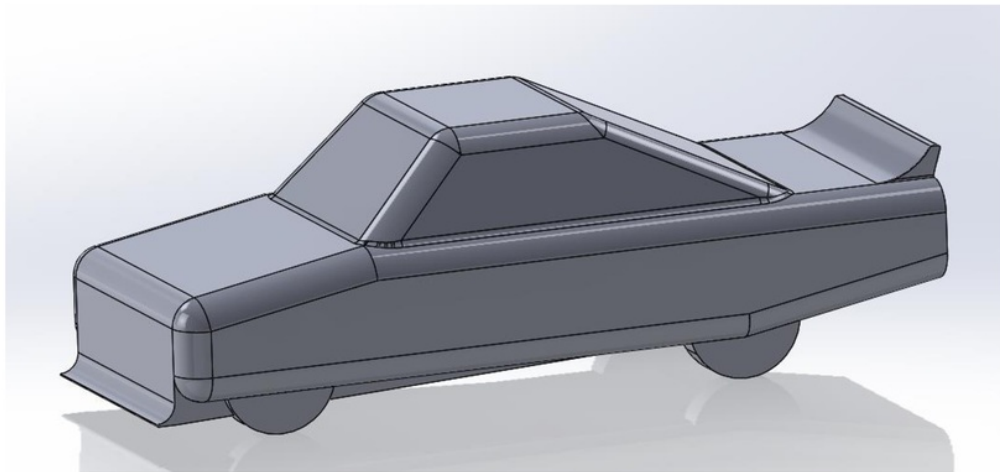
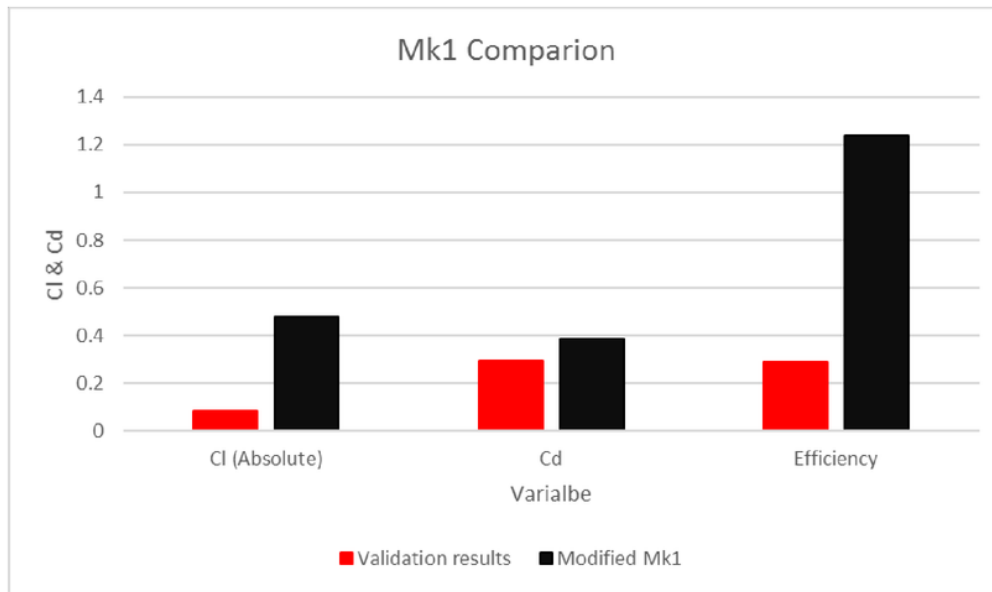


Figure 15 Modified MK1

As you can see a curved air dam was added on the front of the vehicle, the rear slant on the windscreen was decreased and a duck tail spoiler was added to the rear of the vehicle.

5.2.0 Modified Mk1 Results

Below shows a comparison with the standard Mira vehicle and my modified vehicle.



Graph 15 Mk1 comparison

As you can see from the results there is a dramatic effect on the Cl of the vehicle and increased the amount of downforce produced by a lot. Due to this the drag of the vehicle has also increased. Extra drag on a vehicle will change the handling characteristics of a vehicle. An important part of aerodynamics is the aerodynamic efficiency. Aerodynamic efficiency is the Cl of a vehicle over the Cd of a vehicle. As you can see the efficiency of the modified Mk1 has increased a lot.

5.3.0 Modified Mk1 Analysis

Using the post processing in Ansys the Mk1 was analysed in the areas that were modified. Below is the pressure and flow around the front end of the vehicle with the new air dam.

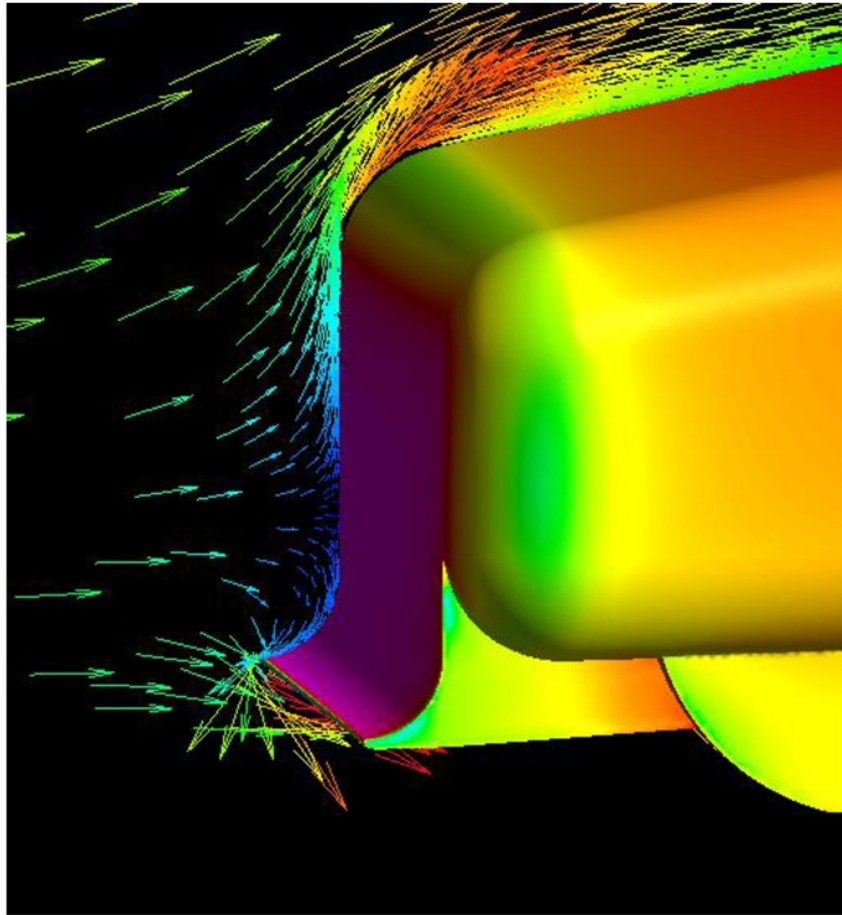


Figure 16 Recirculation on air dam

As you can see there is some recirculation created on the front end of the vehicle. This recirculation is basically stopping the airflow from going over or under the vehicle thus increasing pressure on the front of the car which means the drag would also increase in this area. From the velocity plot shown below it shows that the air dam is restricting the air under the vehicle a lot thus making it “choke”. This is making all the air under the vehicle travel very slow and is stopping the slight diffuser on the rear from working correctly due to the lack of air traveling under the vehicle.

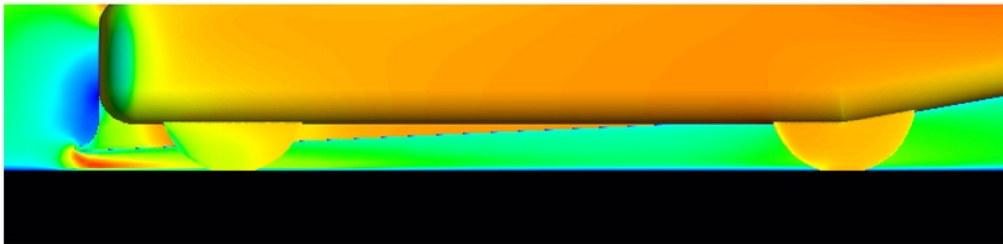


Figure 17 Velocity under Mk1

Below shows that there is some bleed around the sides of the spoiler. This will affect the amount of downforce the spoiler is producing as it is not being directed upwards over it instead it is going around it.

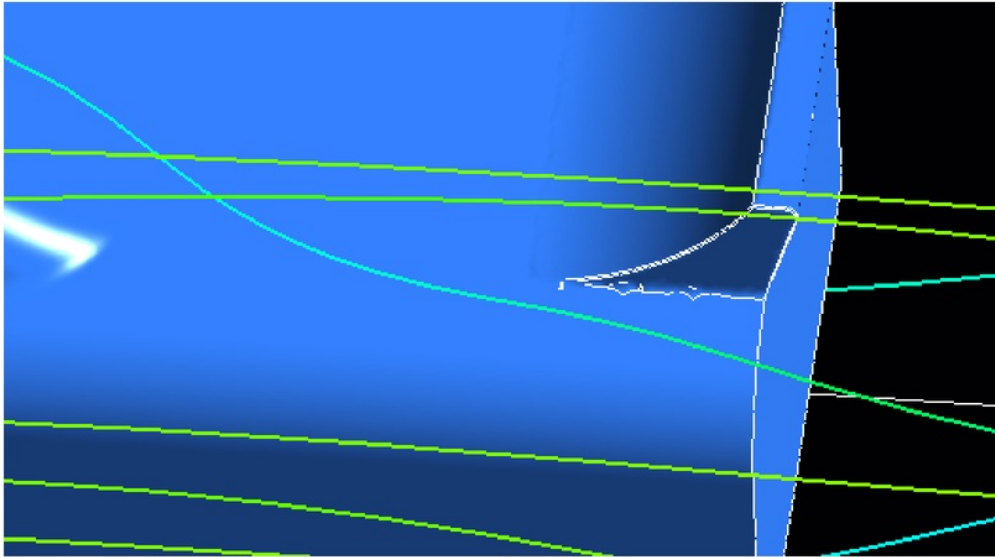


Figure 18 Streamline around spoiler

The fast back seems to be working quite well with not as much separation occurring on the surface compared to when it was standard.

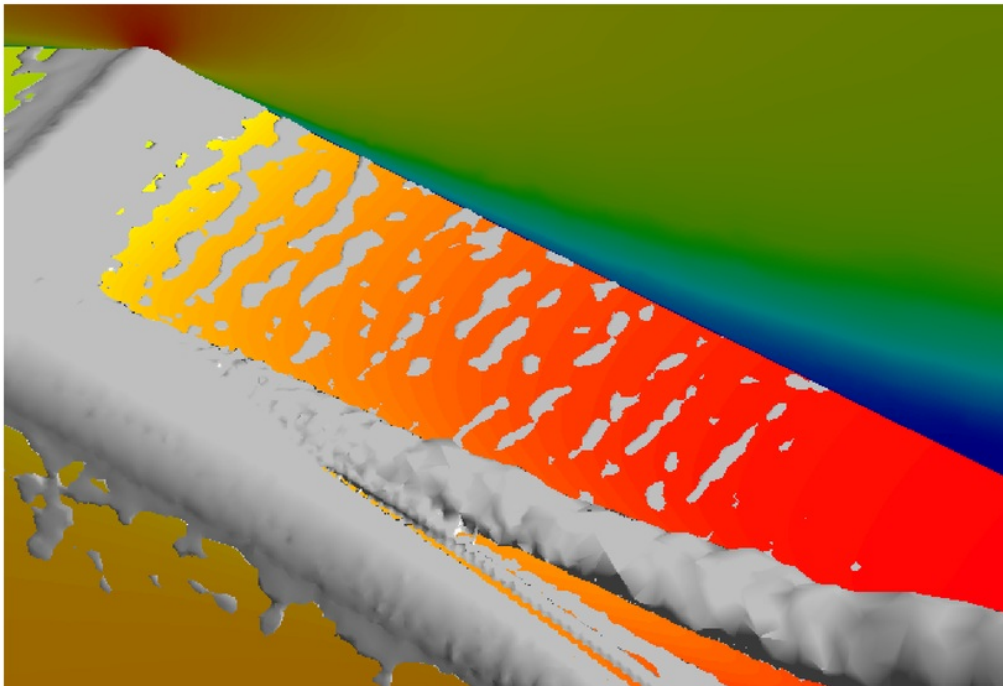


Figure 19 Separation on fastback

5.4.0 Modified Mk2

From looking at the results from modified Mk1 it was clear that some small improvements could be made to give some slightly better results. The height of the air dam was decreased and the radius on it was increased to give a more gradual front to help direct the airflow over the vehicle instead of inducing the recirculation shown in the previous section. Some end plates were also added on the side of the spoiler to stop the pressure bleed going around it instead of going over it which should mean more downforce produced by the vehicle. The slant of the fastback was not changed and kept the same as the previous model.

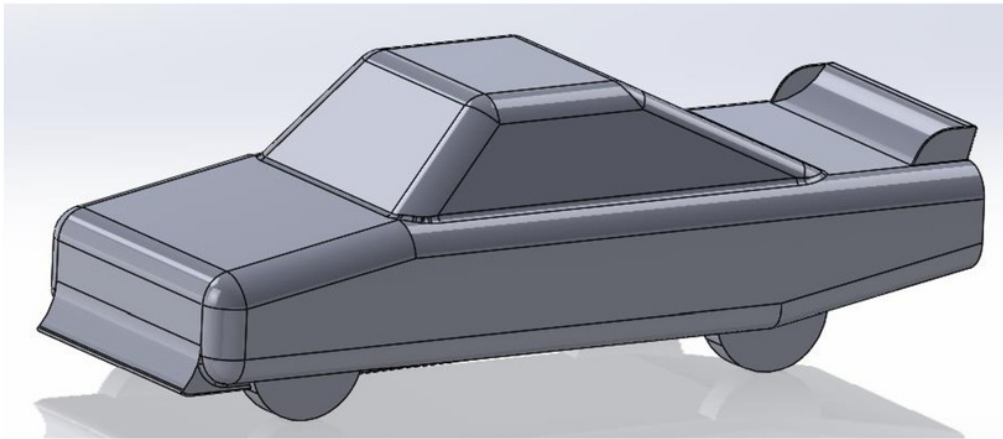
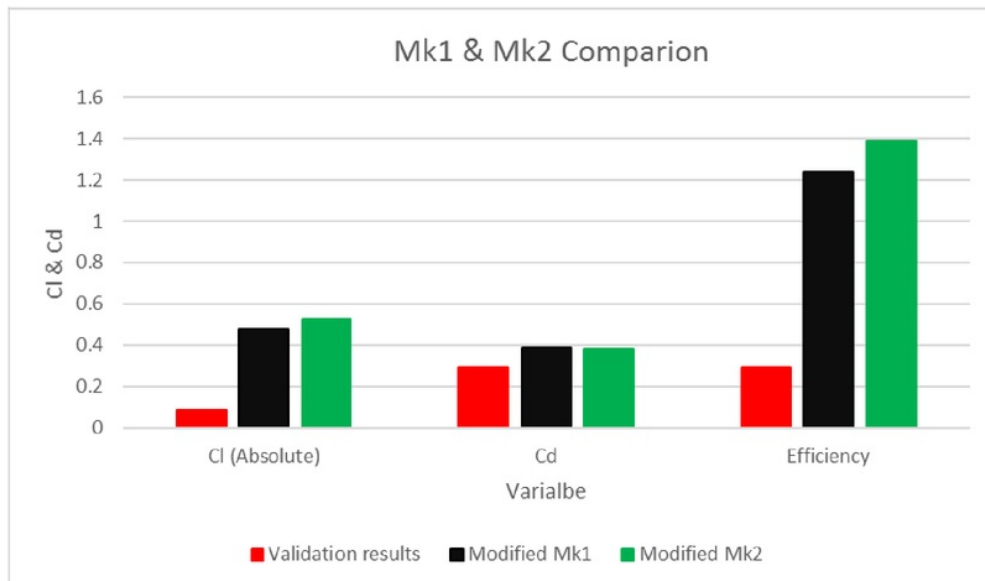


Figure 20 Modified Mk2

5.5.0 Modified Mk2 results

Below shows a comparison of the results of the modified Mk2 against the modified Mk1 and the original model.



Graph 16 Modified Mk2 comparison

These modifications increased the negative CI while also decreasing the Cd of the vehicle which in turn increased the total efficiency compared to the modified Mk1. This proves that the modifications to the Mk1 have worked the way that they were planned. Even though the efficiency has increased, this configuration might not be optimal under certain dynamic conditions.

5.6.0 Modified Mk2 analysis

Looking at where the recirculation was occurring on the modified Mk1 design on the modified Mk2 this recirculation has been reduced, making the whole front of the vehicle more efficient as it is not having to push that recirculating ball forward as the vehicle moves.

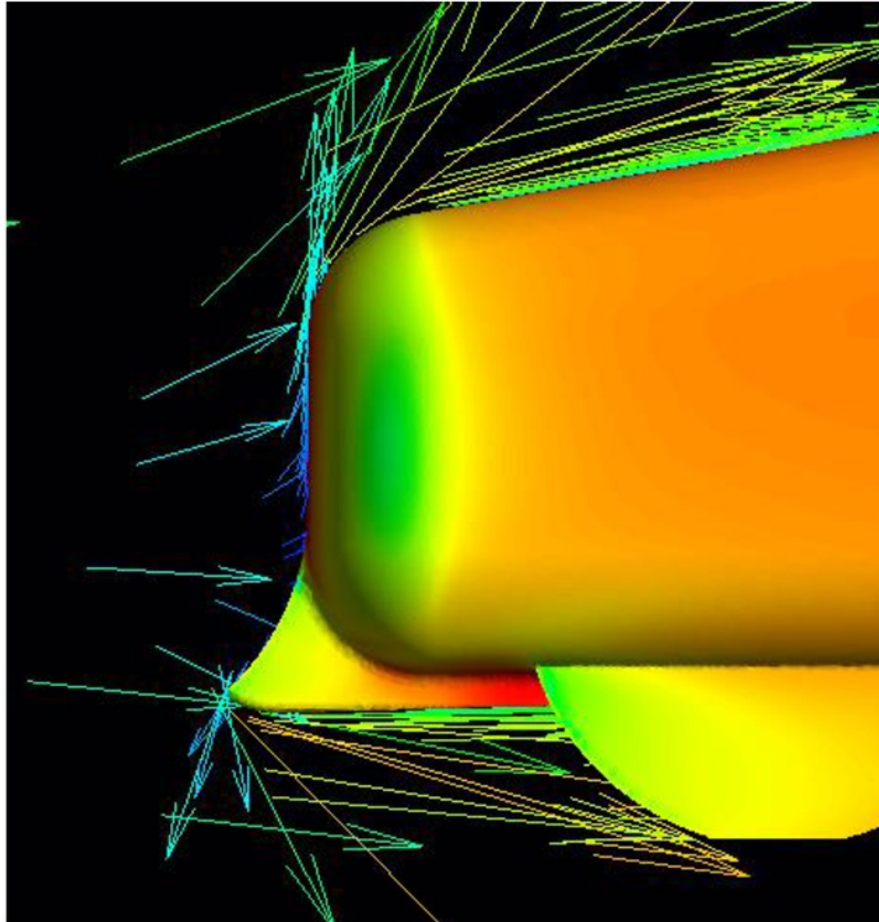


Figure 21 Reduced recirculation on Mk2

Also it is clear that with the increased height of the air dam it had increased the amount of airflow under the vehicle. This has helped the diffuser effect on the bottom of the vehicle. Top left in the comparison is the MK2 geometry.

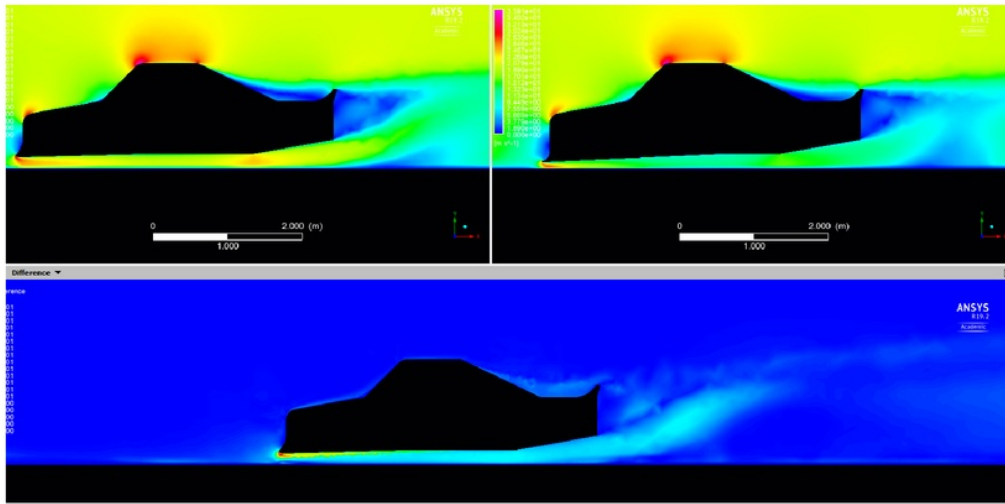


Figure 22 Velocity comparison under Mk1 and Mk2

The end plates on the side of the spoiler have reduced the amount pressure spill around the edges which was helped by the vortex core region that is produced by the top of the roof sealing off the outside of the spoiler which is shown below

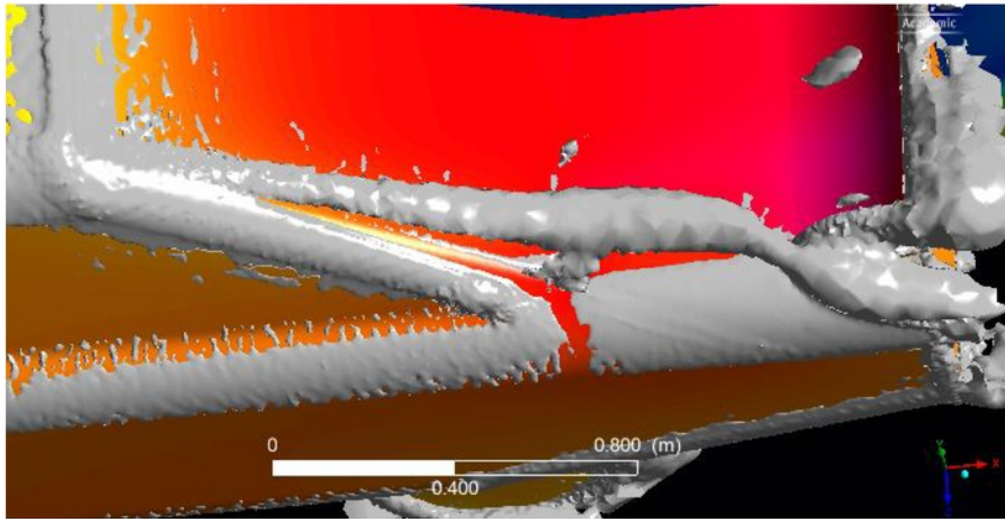


Figure 23 Vortex core region sealing off end-plate

This air that has been stopped going around the vehicle has made an area of slow moving air on the deck of the boot area. This is causing some recirculation and pressure bleed back up the bottom of the rear screen which could be decreasing the downforce levels of the vehicle shown below.

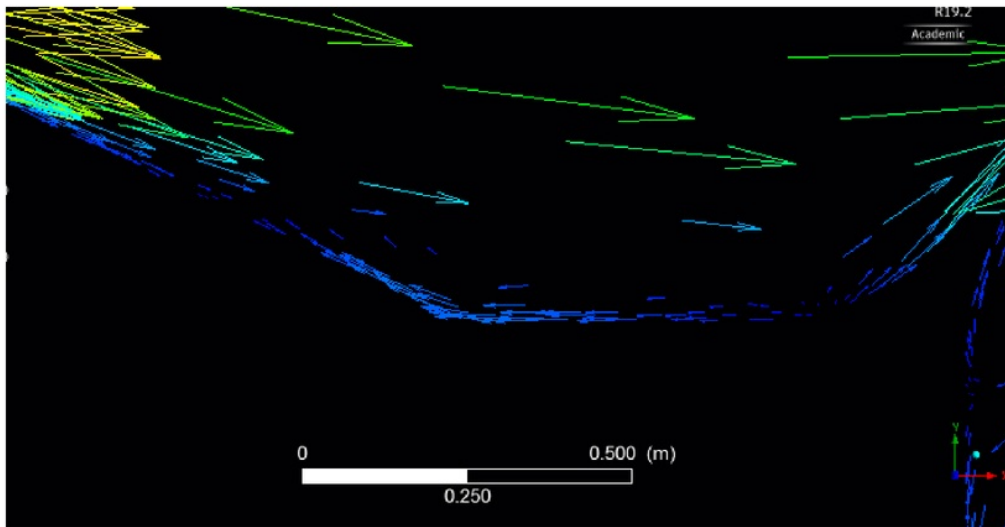


Figure 24 Recirculation on rear deck

This could also be having a positive impact on the effectiveness of the rear diffuser. As you can see below there is a much more of an up wash from this area. This could be caused by the slow moving air going over the top of the vehicle sucking the faster flowing air from under the vehicle upwards thus filling the void that it produces. This is also helped by the faster flowing air under the vehicle produced from the higher air dam on the Mk2

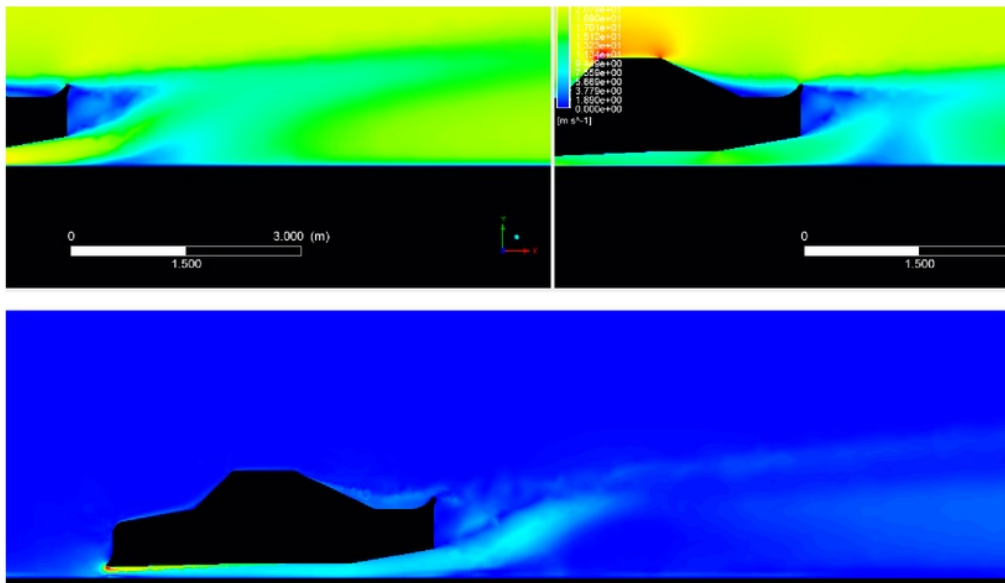
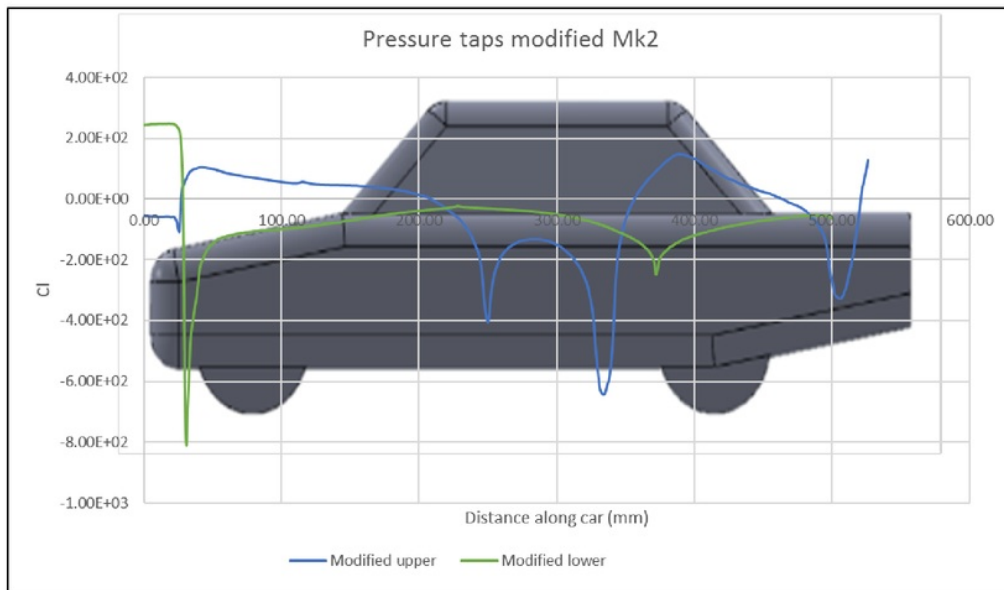


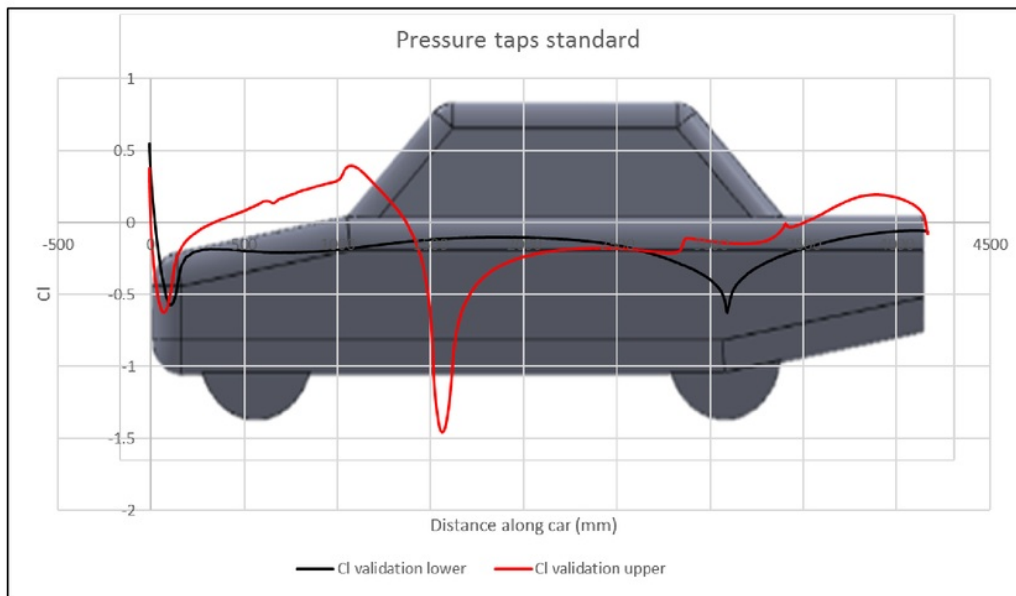
Figure 25 Wake region comparison

This design could be modified further with some slits put into the end plates to bleed some of the pressure off of the deck but still make the rear spoiler work harder and reduce the pressure bleed back up the rear screen of the fastback.

The pressure on the upper and lower surface were compared to the original model to see how the pressure changes the two graphs below show the difference.



Graph 17 Mk2 pressure tap



Graph 18 Pressure taps standard

As you can see the pressure tap data has changed dramatically from the validation study most notably on the top surface. Which was anticipated because most of the modifications are on the top surface of the vehicle.

6.0.0 Critical analysis

This report proved that from real wind tunnel you can create representative results within a CFD program, but it does have its limitations and would require a lot more research into different settings, for example different turbulence models. In conclusion as my whole study was tailored around having the walls as a symmetry it provided the best results with that and when they were treated as a solid object it effected my results so that they were not as representative anymore.

There are some areas which I would go back and continue the parameter sweeps to see if the results would get more accurate, For example, the density box size study. I believe that the density box was not large enough to capture the wake behind the vehicle as in some of my post analysis it seems that not all of the turbulence is captured accurately and you can see where the mesh size increases outside of the density box this is the same with the domain size as shown below there is still some turbulence shown in the Isosurface below

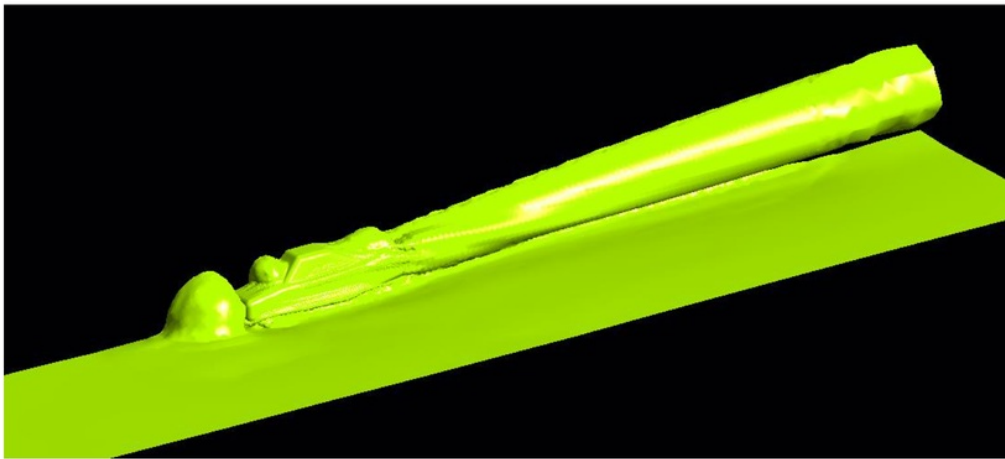


Figure 26 19m/s Isosurface

Figure 26 Above is showing any flow that is below 19m/s this shows that the turbulence behind the vehicle has not equalised back to the original speed of 20m/s that is from the velocity inlet thus not capturing all of the turbulence produced. From this the domain could be slightly larger to get accurate results but as this was to validate the Mira wind tunnel data a larger test section was not needed.

7.0.0 References

- A. P. Gaylard, A. J. (1998). The Use of CFD to Predict the Aerodynamic Characteristics of. *The Use of CFD to Predict the Aerodynamic Characteristics of*.
- Aerospace, Mechanical & Mechatronic Engg. (2005). *aerodynamics seperation of flow*. Retrieved from http://www-mdp.eng.cam.ac.uk/web/library/enginfo/aerothermal_dvd_only/aero/fprops/introvisc/node9.html
- CFD intro. (2019, 1 2). Retrieved from CFD intro: http://www.cham.co.uk/phoenics/d_polis/d_info/cfdintro.htm
- Lanfrit, M. (2005). Best practice guidelines for handling Automotive External Aerodynamics with FLUENT.
- Palaskar, P. K. (2016). "Methodology Development to Accurately Predict Aerodynamic Drag and Lift for Passenger Vehicles Using CFD. " *Methodology Development to Accurately Predict Aerodynamic Drag and Lift for Passenger Vehicles Using CFD*.
- Rumerman, J. (2019). *evolution of technology*. Retrieved from centennial official: https://www.centennialofflight.net/essay/Evolution_of_Technology/first_wind_tunnels/Tech34.htm
- SAHIN, D. (2004). WIND TUNNEL BLOCKAGE CORRECTIONS: . *WIND TUNNEL BLOCKAGE CORRECTIONS: .*
- sas Ip. (n.d.). *mesh quality*. Retrieved from sharcnet: https://www.sharcnet.ca/Software/Ansys/17.0/en-us/help/flu_ug/flu_ug_mesh_quality.html
- W. Seibert, M. L. (2002). A Best- Practice for High Resolution Aerodynamic Simulation around a Production Car Shape . *A Best- Practice for High Resolution Aerodynamic Simulation around a Production Car Shape .*