



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

Aerodynamic Optimisation and Design of the Rear Wing of a Formula 3 Sprint Car

University of Wales Trinity Saint David

School of Engineering

Major Project

M6X00408

Ainsley Martin

P149894

03/05/2020



Abstract

This report will investigate the current design of the SBD formula 3 sprint car. The use of CFD in real world and virtual testing will be explored to see how the results gathered from these tests will impact how CFD is used and how it is setup.

Case studies will be performed on the current design to see if by changing the way it is currently be used more performance will be gained using the current design. Tests will be run both by using the wing in isolation and by using a full car model to see the impact using a full car model has on the results. The current design will be modified to see if more performance can be gained out of it.

Acknowledgements

I would like to thank Tim Tudor for his constant support and advice not only throughout this project but over the past few years. Without him my interest and knowledge in the field of aerodynamics would not be where it is today.

I would like to thank John Hughes for providing addition support and guidance over the past 2 years with both this project and last year's group project.

I would like to thank Matt and Steve at SBD for providing the car and data and allowing me to conduct this project.

I would like to thank my flatmates (Alex, Alf, Dom, Emily, Joe, Lottie, Milo and Tom) for a great final year and for keeping me sane throughout both this project and the final year of university.

Finally, I would like to thank my parents Syd and Dawn for their constant support throughout my education.

Confirmation of Own Work

This thesis is the result of the author's personal research and conducted work. It has been composed by the author with all relevant research cited and has not been previously submitted for examination for the award of an honour's degree.

Confirmation of work:

Signed: A.Martin

Date: 03/05/2020

Contents

1. Nomenclature	6
2. <i>Introduction</i>	10
3. Literature Review	11
3.1. Dallara F397	11
3.2. Basic Equations	11
3.2.1. Bernoulli's Equation	11
3.2.2. Reynolds Number.....	11
3.2.3. Navier-Stokes	11
3.2.4. Forces and Force Coefficients	12
3.3. Boundary Layer	12
3.4. Wing Theory.....	13
3.4.1. 4 Main Theories	13
3.4.2. General Wing Design.....	14
3.4.3. Multi Element Wing	15
3.4.4. Gurney Flaps	16
3.5. CFD	17
3.5.1. Domain	17
3.5.2. Mesh	17
3.5.3. Solver.....	17
3.5.4. Post Processor	17
3.5.5. Y Plus	18
4. Real World Validation	19
4.1. Flow Vis and Straight Line Testing	19
4.2. Wind tunnel Validation	22
4.3. Ahmed Body Validation	25
4.3.1. Introduction	25
4.3.2. Mesh Strategy	25

4.3.3.	Solver.....	33
4.3.4.	35 Degrees	36
4.3.5.	Does Using Various Computers Make a Difference?	37
5.	Full Car CAD Models.....	38
5.1.	Original CAD Model.....	38
5.2.	Scanned CAD Model.....	39
6.	Wing in Isolation	42
6.1.	Wing Angle	47
6.2.	Wing Position	50
6.2.1.	MATLAB Interpolation.....	52
6.2.2.	Results.....	53
6.3.	Wing Overhang	55
6.3.1.	Results.....	55
6.3.2.	Full Car Test.....	56
6.4.	Wing Gap.....	60
6.4.1.	Results.....	62
6.5.	Twisted Wing	63
7.	Extended Diffuser	69
8.	Conclusion.....	72
9.	The Impact of Covid-19 on This Project	73
9.1.	Real World Validation	73
9.2.	3D Printed Wing.....	73
9.3.	CFD Testing	73
10.	References	74

1. Nomenclature

AoA – Angle of Attack

CoP – Centre of Pressure

CL – Coefficient of Lift

CD – Coefficient of Drag

CFD – Computational Fluid Dynamics

CAD – Computer Aided Design

CoG – Centre of Gravity

Re – Reynolds Number

Figure 1 - Boundary Layer Formation	13
Figure 2 - Wing Terminology (Kachare, 2017)	15
Figure 3 - Gurney flap flow vis comparison	19
Figure 4 - Wing mount flow vis comparison	20
Figure 5 - C2 open loop wind tunnel.....	22
Figure 6 - CAD of C2 open loop wind tunnel.....	22
Figure 7 - Comparion of flow attachment vs seperation	23
Figure 8 - Comparion of K Epsilon and K Omega for NACA 0012 profile	23
Figure 9 - Pressure distribution of NACA 0012 at 10 degrees for both K Epsilon and K Omega	24
Figure 10 - Fine vs Coarse mesh for Ahmed body	26
Figure 11 - Skew comparison for different part sizes	28
Figure 12 - Fine vs coarse density box comparison	28
Figure 13 - Velocity contours for fine vs coarse density box	30
Figure 14 - Prism layers on Ahmed body	32
Figure 15 - Comparison of K Epsilon flow structures.....	34
Figure 16 - Comparison of K Omega flow structures.....	35
Figure 17 - K Omega vs K Epsilon flow structure comparison for 35 degree slant.....	36
Figure 18 - Original CAD model.....	38
Figure 19 - Final CAD model.....	39
Figure 20 - Comparison of standard wing models.....	42
Figure 21 - Pressure contour of original wing in isolation	43
Figure 22 - Pressure contour of original wing attached to the car	44
Figure 23 - Flow structure of wing in isolation and attached to the car.....	46
Figure 24 - Wing at minimum angle.....	49
Figure 25 - Wing at maximum angle	49
Figure 26 - Car without the rear wing.....	51
Figure 27 - Wing position placement.....	52
Figure 28 - Original wing vs modified overhang	56
Figure 29 - Vortex core comparison between isolated wing and wing on the car for modified overhang	58
Figure 30 - Pressure contour comparison between isolated wing and wing on the car for modified overhang	59
Figure 31 - Vectors for wing gap comparison	61
Figure 32 - Twisted wing design.....	64

Figure 33 - Twisted wing flow structure	66
Figure 34 - Twisted wing on full car model.....	67
Figure 35 - Spoon wing design REFERENCE.....	68
Figure 36 - Original diffuser both in real life and in CAD	69
Figure 37 - Modified diffuser design.....	70
Figure 38 - Vectors for extended diffuser	70
Graph 1 - Global mesh study.....	Error! Bookmark not defined.
Graph 2 - Part size study	27
Graph 3 - Density box element size study	29
Graph 4 - Density box size study.....	31
Graph 5 - Prism layer study.....	32
Graph 6 - Scanned vs original results.....	40
Graph 7 - Final scanned model vs original results	41
Graph 8 - Pressure distributions of original wing	45
Graph 9 - Wing angle study results.....	48
Graph 10 - Wing position results	53
Graph 11 - Wing overhang results	55
Graph 12 - Original vs modified overhang results	57
Graph 13 - Second element gap results.....	62
Graph 14 - Original vs twisted wing results	65
Graph 15 - Original vs extended diffuser results	71
Table 1 – Table Comparing Straight Line Test Results to CFD Results.....	21
Table 2 - Given values for Ahmed body	25
Table 3 - Laminar results.....	33
Table 4 - Turbulence study results.....	35
Table 5 - 35 degree turbulence study results	36
Table 6 - Multiple computer results	37
Table 7 - Wing on the car vs isolation results	57
Equation 1 – Drag equation	12
Equation 2 - Lift equation	12

Equation 3 - Reynolds Number	12
------------------------------------	----

2. Introduction

This report will look into the optimisation of a rear wing for a formula 3 sprint/Hillclimb car. The car is run by SBD motorsport who use the car in the British sprint championship. The car being used for this report is a modified version of the Dallara F397.

The current rear wing design will be investigated and analysed. This analysis will deduce what the wing currently is doing well and areas that could be improved. The rear wing can be difficult to improve performance on as it is almost always in dirty air from the other aerodynamic devices on the car. Once the current design has been analysed ways to modify or change the design of the wing will be investigated. The current design features three elements, one lower element which will remain as it is for this study. There is currently a multi-element design which will be the focus for this report.

There are two main areas of concern when looking at the aerodynamic performance of these devices, the lift and the drag produced by the overall design. Currently SBD says they are happy with the current amount of downforce being produced overall on the car however they would like to improve the top speed of the car. Thus, decreasing the drag of the wing while maintaining the current levels of downforce will be the objective of the report. The CoP will also remain a concern throughout the project, if the CoP is adjusted too far forwards this could reduce the traction on the drive wheel reducing the performance of the car. If the CoP is adjusted too far rearwards this will reduce the force acting on the front tyres causing the car to understeer in the corners, this means the driver will not be able to make use of any additional downforce being produced by the rear wing.

3. Literature Review

3.1. Dallara F397

The car was originally a Dallara f3 car from 1997. The car competed in the 1997 German f3 championship. Since taking ownership SBD have made many modifications to the aerodynamics of the car. This includes a modified front wing, new floor and diffuser, modified side pods, new front and rear wing endplates and a new multi element rear wing.

The car has a red top 2l Opel engine which produces around 300bhp. Attached to this is a six speed gearbox.

To produce a CAD model of the car it was scanned by Central Scanning using an Artec Leo scanner. This produced a point cloud geometry file. This file was sent off to an external company and converted into a solid CAD model.

3.2. Basic Equations

3.2.1. Bernoulli's Equation

Bernoulli's principle states that as a fluids velocity increases this will result in a decrease in pressure.(Katz, 1995b). When air flows from an area of high pressure to an area of low pressure the air accelerates and vice versa.

Equation 1 - Bernoulli's Equation

$$Bernoulli's = P + \frac{1}{2} * \rho * V^2$$

3.2.2. Reynolds Number

The Reynolds number is a ratio of the inertial forces against the viscous forces of a fluid. The Re will dictate whether fluid flow will be laminar or transient. If the Re is less than 1×10^5 then the flow will be laminar, if the Re is greater than 1×10^5 the flow will be turbulent.(Tientcheu-Nsiewe *et al.*, 2016)

3.2.3. Navier-Stokes

The Navier-Stokes equations are a set of equations that govern fluid dynamics. There are three main parts to these equations. Initially there is a continuity equation for the conservation of mass, secondly there are three equations that relate to the conservation of momentum.(Carnarius *et al.*, 2009). Finally, there is a conservation of energy equation that is related to time.(NASA, 2007c). The first part of these equations was developed by Euler which work for transient solutions, however they struggle with turbulent solutions.(Calinger, 1996). Claude-Louis Navier and Gabriel Stokes

further developed these equations, adapting the Euler equations and making them the Navier-Stokes equations as we know today. This added turbulence into the equation.(White, 2005; Cebeci, 2006).

3.2.4. Forces and Force Coefficients

When looking at the aerodynamics of a car there are 2 main forces that are considered, the drag force acting on the car and the negative lift (downforce).

Equation 2 – Drag equation

$$Drag = cd * \frac{\rho * V^2}{2} * A$$

Equation 3 - Lift equation

$$Lift = cl * \frac{\rho * V^2}{2} * A$$

The drag is the resistance to motion through the air. This is seen as a negative in vehicle aerodynamics as it limits the top speed of a car, reduces the acceleration and can increase fuel consumption. (Batchelor, 1999). When the amount of downforce is increased this will lead to an increase in drag so establishing a balance between the two for the intended purpose is essential.

3.3. Boundary Layer

The boundary layer is a layer that forms on a surface when air passes over it. The thickness of the boundary layer increases from the front of the vehicle and can become a few centimetres towards the rear of the car.(Barnard, 2001).

Depending on the Reynolds number the flow can be laminar, turbulent or a transition between the two. Typically, if the $Re < 1 \times 10^5$ then the flow will be laminar, if the $Re > 5 \times 10^5$ then the flow will be turbulent. If the Re is between these values then the flow will be in a transitional state where it isn't fully turbulent and isn't fully laminar.

$$Re = \frac{\rho V l}{\mu}$$

Equation 4 - Reynolds Number

A boundary layer will start off laminar and can become turbulent further down the body. When the boundary layer is laminar the layer is thin with a low surface friction. When it becomes turbulent the friction increases as the layer is thicker, the velocity close to the surface of the body is also greater. When the flow is turbulent the air can start to separate off the body.(Bertin and Cummings, 2009)

At the leading edge of a body there is no boundary layer as this is the first point the air is in contact with the body, after this point the boundary layer starts to grow. On the surface of the body the velocity of the air is zero if the body has a velocity of zero. (Princeton University, 2014). The boundary layer will start off being laminar. When the boundary layer becomes thicker the shear stresses cause the air to start rotating, this is the transitional phase between laminar and turbulent. The freestream air begins to mix with the boundary layer, this causes the air to become turbulent. This means the fast moving air mixes with the slow moving area, therefore the momentum in the boundary layer is increased. This is the turbulent layer. Inside the turbulent layer a laminar sublayer begins to form, this is due to the shear forces becoming large enough again to all the air again to flow in a laminar state, this occurs very close to the surface of the body and is very thin.

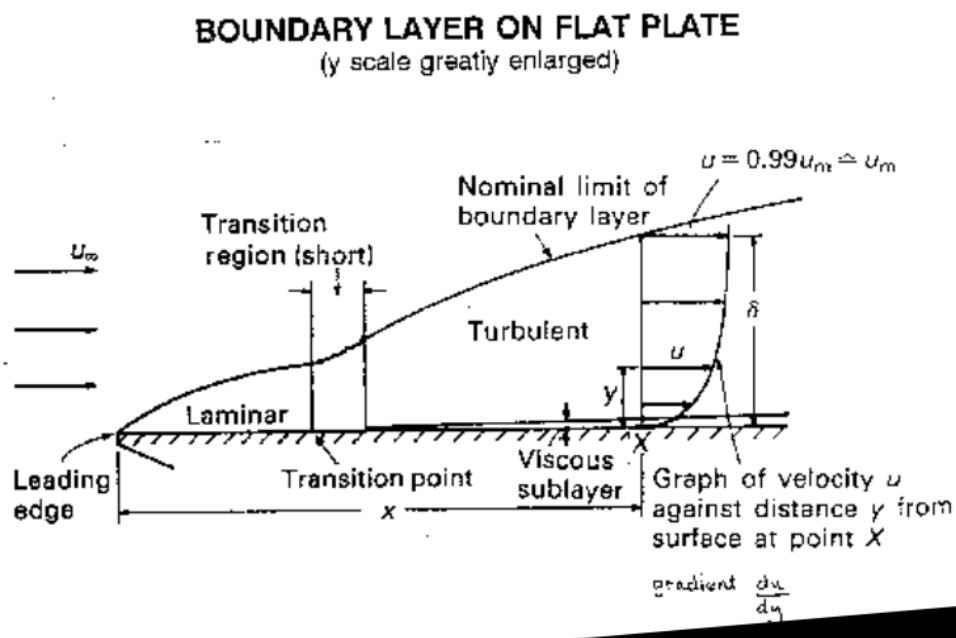


Figure 1 - Boundary Layer Formation (Tudor, 2017).

3.4. Wing Theory

3.4.1. 4 Main Theories

3.4.1.1. Constant Rotating Vortex around Wing

This theory suggests that there is a constant vortex rotating anticlockwise around the wing. When the air flows over the topside of the wing it is traveling in the opposite direction to the vortex therefore, the velocity of the air decreases and therefore increasing the pressure on the topside of the wing. As the vortex on the bottom is travelling in the same direction as the free stream air, it is accelerated.

3.4.1.2. Venturi Effect

This theory suggests that there is free stream above and below the wing. As the free stream air passes over the top of the wing it hits the topside of the aerofoil which causes there to be high pressure on the topside of the wing. As the air flows on the underside of the wing the air is compressed by the free stream air, this causes the air is compressed by the free stream air. This causes a venturi effect to occur under the wing, this accelerates the air.

3.4.1.3. Air Must Accelerate on the underside to match a particle to the topside

While this theory is considered by most to be incorrect it still remains as a potential talked about theory to explain how wings work. This suggests for a wing designed to create lift, if 2 particles start at the leading edge and one goes on the top low pressure side while the other goes on the bottom high pressure side they will both meet up at the trailing edge. Therefore, the low pressure side particle must have a greater velocity as it has to travel a greater distance this causes the pressure to be lower as stated by Bernoulli.

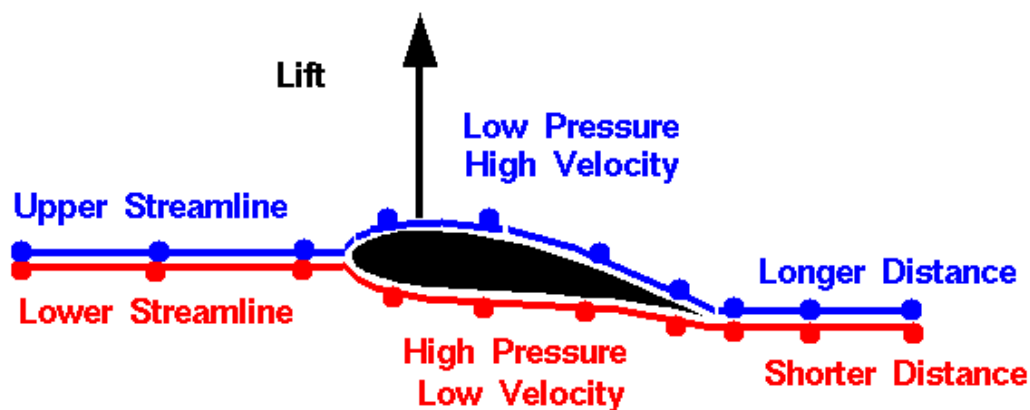


Figure 2 - Particle matching wing theory(NASA, 2007)

3.4.1.4. Every Reaction Must have an equal and opposite reaction. Upwash must equal Downwash

This theory is based from Isaac Newton's third law where every reaction must have an equal and opposite reaction.(Newton, Cohen and Whitman, 1728). As such, when a wing is angled upwards for a car to create downforce, a downwash effect is created. As there must be an equal reaction the reaction force pushing the wing down creating downforce.(Jones, 1979)

3.4.2. General Wing Design

A wing is an aerodynamic device typically used on the front and rear of cars to generate downforce. Wings originated from the aerospace industry and are used to generate lift allowing planes to fly. Wings on cars use similar profiles however they are upside down, allowing them to generate

negative lift (downforce). In simple terms a high pressure acts on the top surface and a low pressure acts on the bottom surface, this pushes the wing down which allows the wheels to have more grip.

Wings are designed using a few main criteria, the chord, camber and span of the wing.

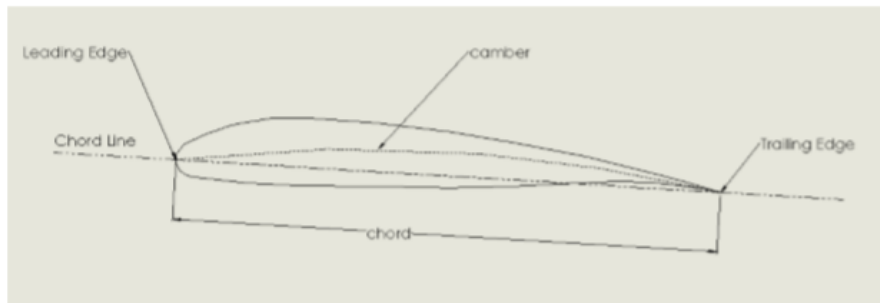


Figure 3 - Wing Terminology (Kachare 2017)

The chord length is the distance from the leading edge of the wing to the trailing edge, the leading edge is the first point the air comes into contact with the wing and the trailing edge is theoretically the last point the air is in contact with the wing however this could occur earlier due to separation.(Houghton and Carpenter, 2003)

The camber of a wing describes the asymmetry of a wings profile. The more cambered a wings profile the more curved it is. A symmetrical wing profile will have zero camber.(Abbott and Von Doenhoff, 1997)

The span of the wing is the distance from one side of the wing to the other. The span of a wing will depend on the application. As the span is changed it will alter the drag and downforce as a larger wing will have a larger surface area which will increase the drag. The span of the wing will have to be tested to find the optimum angle. With a front wing the span of the wing can affect the aerodynamics of the car downstream so this will have to be considered when testing different spans to find the optimal length.(Diasinos, Barber and Doig, 2017)

3.4.3. Multi Element Wing

To try and improve the efficiency of a wing more elements can be added to it. As the AoA of a single element wing is increase it will inevitably reach a point where the flow beings to separate.

Separation occurs when there is a lack of energy in the boundary layer due to surface friction in the shear wall. The flow no longer has enough energy to remain attached and begins to sperate. An

adverse pressure gradient can also cause flow to separate. Air naturally wants to flow from an area of high pressure to an area of low pressure.

To keep the air attached on a high downforce wing the wing may need to be separated into multiple elements, by doing this as the air flows through the gap between the two elements a new high energy boundary layer is created which reenergises the air flow. This reenergised boundary layer enables the flow to stay attached even if there is a larger AoA therefore producing more downforce. More downforce can be created as the overall area of the wing has been increased as governed by the lift equation although, this will also increase the drag produced by the wing.(Merkel, 2013)

By using a multi element wing with the reenergised boundary layer the pressure on the low pressure side of the wing takes longer to return to ambient pressure. As the air is still being accelerated on the low pressure side of the wing when being attached to the second element it still has a low pressure therefore 'sucking' the wing down and producing downforce.(Kachare, 2017)

Careful consideration should be taken with the placement of the second element in relation to the main element, in most cases the leading edge of the second element should slightly overlap the trailing edge of the main element. This is done so the flow separating from the trailing edge of the main element can then be reattached to the second element, this effectively increases the camber of the wing.

3.4.4. Gurney Flaps

A gurney flap is a right angled strip that is attached to the high pressure side of a wing at the trailing edge(McBeath, 2006). The general idea of a gurney flap is to provide an area for the air to hit, this will then produce counter rotating vortices at the trailing edge of a wing.(Schuetz, 2018) These vortices will reduce the effect of each other and should therefore, decrease the drag.

The gurney also slows down the velocity of the air at the trailing edge of the wing which will increase the pressure acting on the topside of the wing, increasing the downforce. A gurney should be able to provide the same effect as increasing the camber on a wing.(McBeath, 2006).

The gurney flap alters the Kutta condition at the trailing edge (Jang, Ross and Cummings, 1998). The Kutta condition occurs when there is a sharp trailing edge, the air wants to bleed around the trailing edge and creates a vortex in this region. By attaching a gurney flap, it tries to prevent this from occurring.(Mohebbi, Evans and Sellier, 2019)

3.5. CFD

For this report Ansys simulation software will be utilised to carry out each run and case studies. Ansys is considered a 'parent software' as it contains individual programs along with its own software. For this report four programs will be used. These are, Design Modeller, which is used to create the domain, ICEM which is used to create the build grid (mesh), Fluent used to solve the fluid investigation and finally CFD Post which is used to analyse the results.

3.5.1. Domain

Design Modeller is where the CAD model is imported. If any holes or lines need fixing on the model or it needs some general refinement it can be done at this stage. A 'virtual Wind tunnel' is created around the model and this is where the simulation is carried out. The dimensions of the domain and the option of using either a full or half car model are decided at this stage. If a half car model is chosen then a symmetry plane will have to be created through the middle of the car.

3.5.2. Mesh

ICEM mesher was used to produce the mesh. There are multiple settings inside the ICEM software that can be used to create and fine tune a mesh. Throughout this report only a select few of the settings will be modified as these have the most impact on the mesh quality and therefore the results produced. These settings were chosen to be modified after reading the user guide of ICEM.(Ansys, 2013)

3.5.3. Solver

Fluent was used to solve the mesh. As with ICEM fluent has many option to adjust the solver to exactly what the user desires, although many of these settings are not relevant for external fluid dynamics. The turbulence model will have to be carefully considered as choosing the wrong setting can have dramatic impacts on the results and the resulting flow structures.

3.5.4. Post Processor

The post processor is used to visualise the results produced from the solver. This is the best place to analyse the results.(Toet, 2019). There are a handful of main ways the results can be analysed. The first is by analysing the flow structures, this can be done in a few ways. Such as using streamlines, vectors or vortex core regions. These will show the flow structures produced by the car, they can also show areas of separation and vortices produced. Graphs and tables can also be produced to display data such as the pressure distribution acting on the top and bottom surface of the car. Finally, contours can be applied to any surface, these can be used to display pressure, velocity among other variables.

3.5.5. Y Plus

Y plus is a dimensionless value used to improve the accuracy of the boundary layer modelling close to the surface.(Schlichting, 1980). The y plus value is used to ensure the first cell is inside the log law region.(SimScale, 2018). Typically the value should be greater than 30 but less than 300, however this is just a guideline and can change depending on the Re being used.(ANSYS, 2013).

4. Real World Validation

In order to prove that the CFD is providing accurate data some real world needed to be gathered and compared against. Three experiments were conducted to ensure the CFD would provide correct results. The first was flow visualisation testing at Brands Hatch on a Sports 2000 MCR, the second was a Wind tunnel validation study conducted using the C2 open loop Wind tunnel located on campus. A NACA 0012 profile was used for this study. Finally, a validation study was done using a bluff body (Ahmed body), comparing the CFD results against Wind tunnel data.

4.1. Flow Vis and Straight Line Testing

From the dates of August 30th to September 1st the universities race team (Team MCR) was at Brands Hatch for the final round of the mini enduro Sports 2000 championship. An agreement was made so the author could spray flow vis on the rear of the car during one of the testing sessions. Flow vis is made up of a fluorescent paint powder mixed with paraffin, this is then sprayed onto the car using a spray bottle, covering the areas that were to be investigated liberally. As the paraffin evaporates the paint dries onto the car and the flow structures that are created on the surface of the car can be seen.(Smedley, 2019).

For this test the flow vis was sprayed onto the rear clam of the bodywork and the rear wing, including the wing mounts, the aerofoil and the endplates. This was done to investigate the flow structures around the rear of the car. This area was chosen as it is the same area being investigated in this project and therefore it would be expected to observe some similarities between the two projects. When the car returned to the pits at the end of the session it was placed away from direct sunlight and photos were taken from all angles of the rear of the car.

These photos were later compared against results from the CFD post processor. The similarities and differences can be observed between the two. As seen in figure 4. By seeing the similarities between the two sets of data it proves that the settings used for the CFD run are providing accurate flow structures. This means in future runs the results from the CFD can be trusted.

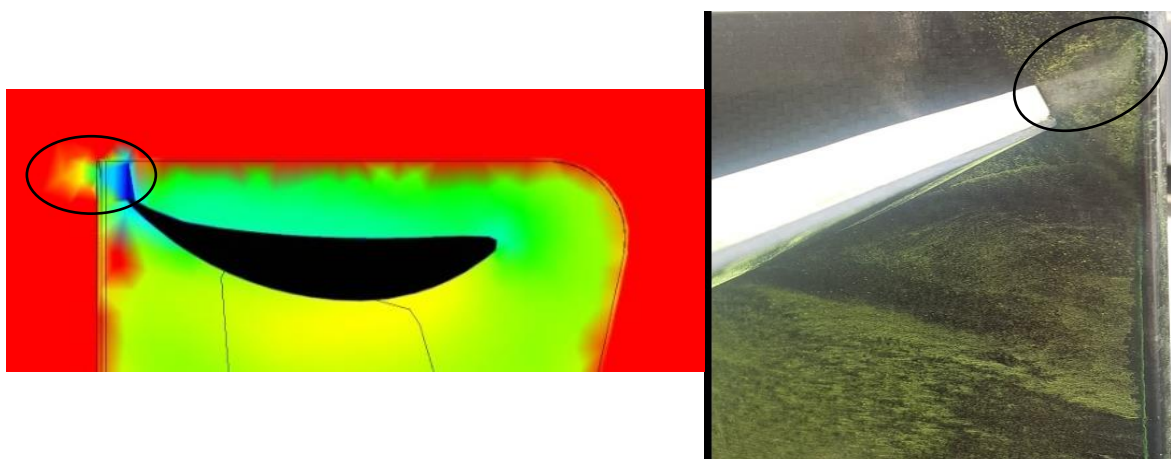


Figure 4 - Gurney flap flow vis comparison

Figure 4 shows the similarities produced by the wings trailing edge and gurney flap. The black circles show the blockage produced by the gurney flap and how it forces the air to flow around it but leaving a void directly behind it.

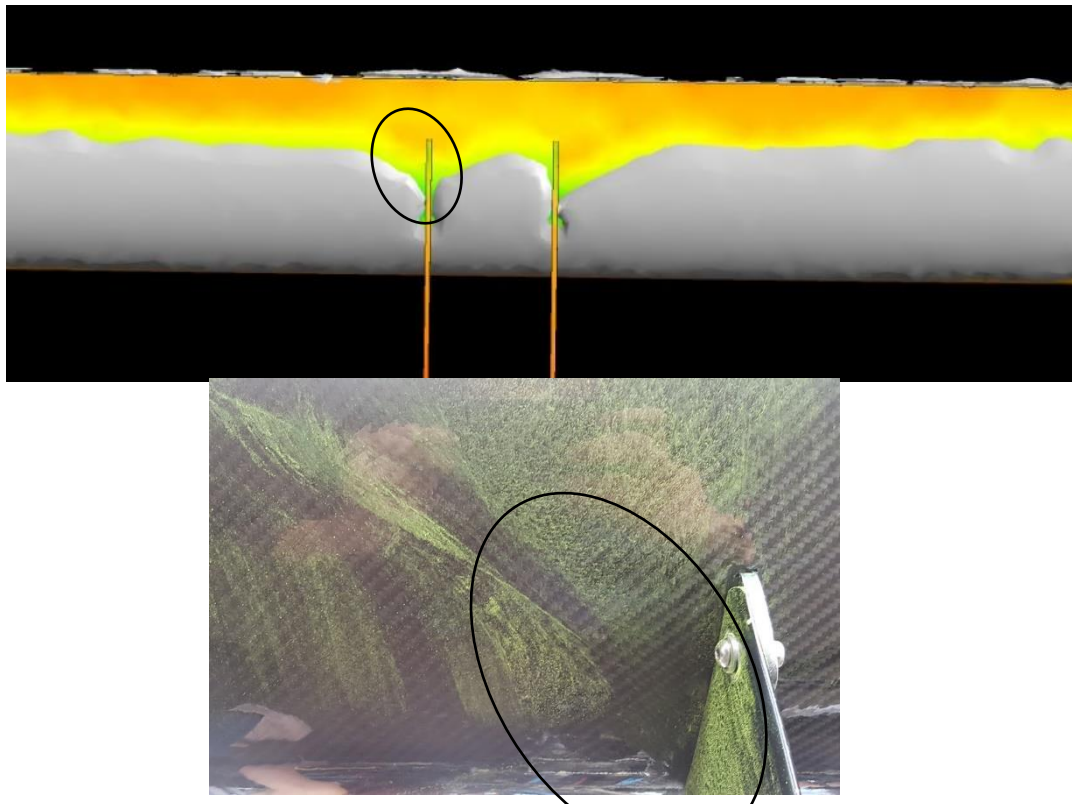


Figure 5 - Wing mount flow vis comparison

Figure 5 shows the flow deflection caused by the wing mount. The black circles show where the flow for the underside of the wing has been deflected by the wing mount. As can be seen the CFD and real world tests produced similar flow structures. This shows some validation with the settings being used in CFD for a full car model with the similarities that can be visually seen.

Again, using the universities Sports 2000 MCR car a straight line test was conducted for other student's dissertation projects. As they were using pushrod sensors this data can be used to find the aerodynamic forces acting on the car. This data can again be used to correlate against the CFD results.

Table 1 – Table Comparing Straight Line Test Results to CFD Results

	cl	cd
Straight Line Test Results	-0.876	0.497
CFD Results	-0.84167	0.491331
% Difference	3.918813	1.140664

From Table 1 it can be seen that CFD produces results to acceptable levels since CFD and wind tunnel data within 5% is seen to be accurate by most people. To get the CL and CD from the straight line test, the aerodynamic forces were taken from the MoTeC data. They were then manipulated using the CL and CD equations.

Looking at both the flow vis and straight line test results it can be seen that the CFD is producing data that produces both accurate force coefficients and accurate flow structures.

4.2. Wind tunnel Validation

Another form of CFD validation is using a Wind tunnel. This is done as each element in the Wind tunnel can be measured and the same settings can be used inside the CFD package.(Favier, 2010). For this experiment the NACA 0012 profile was used with 11 pressure taps located along the chord of the aerofoil. Since the pressure taps are only located on one side of the aerofoil the experiment had to be conducted using both positive and negative AoA, this will show the pressures on the upper and lower surface of the aerofoil. This can be done as the profile is symmetrical. The Wind tunnel used for this experiment was the universities C2 open loop Wind tunnel.

For the test the inlet velocity was set at 6m/s, this velocity could then be set in CFD to get a direct comparison between the data. The pressures were recorded from each pressure tap at each AoA. Results were recorded from -10° to 10° .

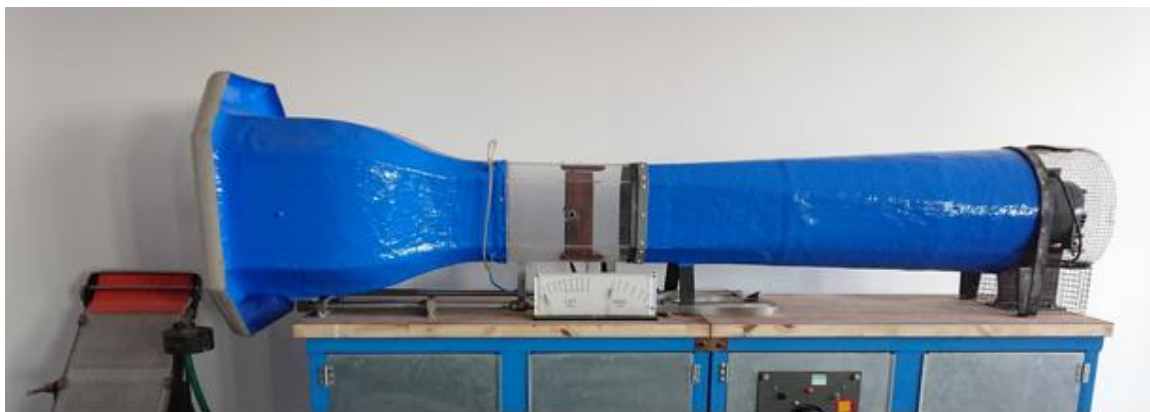


Figure 6 - C2 open loop wind tunnel (Hughes, 2019)

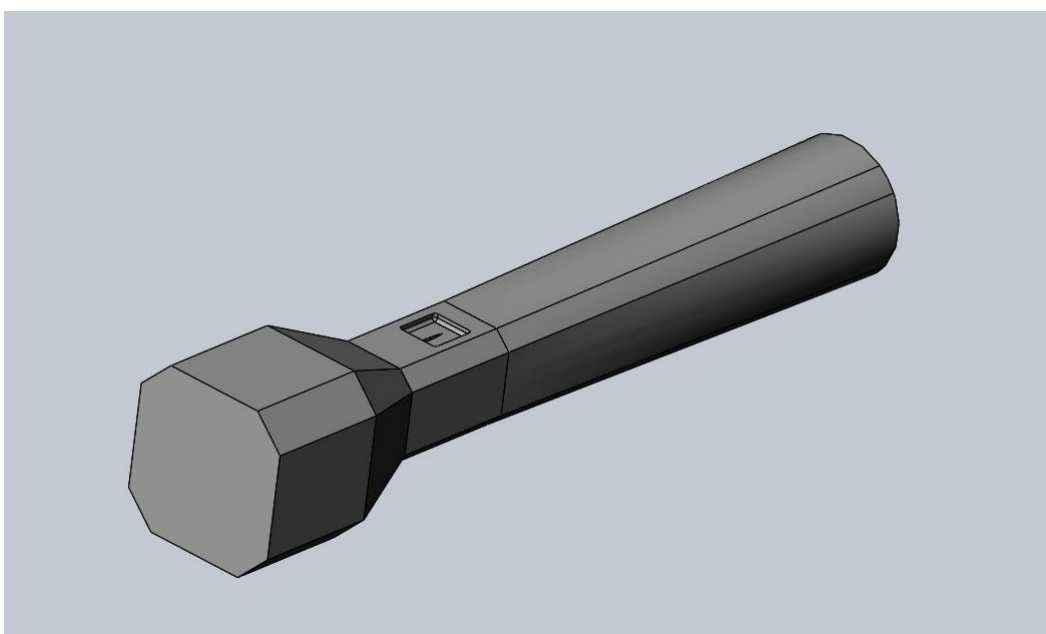


Figure 7 - CAD of C2 open loop wind tunnel

Figure 7 shows the CAD model of the Wind tunnel. A CAD model was created for each angle of the NACA profile.

These tests were replicated in CFD. To do this the Wind tunnel was measured and had to be replicated in CAD. This was also done for the aerofoil and mounts to connect the aerofoil to the Wind tunnel. Once the profile is placed inside the Wind tunnel it can be adjusted inside the CAD package to it at each required AoA. Originally each CFD run was done using the K Epsilon turbulence model as it was found to produce the greatest accuracy with the Ahmed body study. Between 0° and 4° this produced results that showed correlation between both sets of data.



Figure 8 - Comparison of flow attachment vs separation

Figure 8 shows the comparison between 0° AoA (Left) and 20° AoA (Right). At low AoA the flow remains attached across the whole chord of the wing, when the AoA is increased too much the flow begins to separate. This can be visually seen with the string. The string begins to move unpredictably at large AoA and this is due to the turbulence caused by the separation, when this is occurring the downforce produced by the wing will be decreased and the drag will be increased. (IMechE, 2014) An area of extreme low pressure is formed where the separation occurs. This causes the air around this low pressure section to be sucked into this area and recirculate, which increases the drag to increase. This is known as pressure drag. (Juniper, 2012).

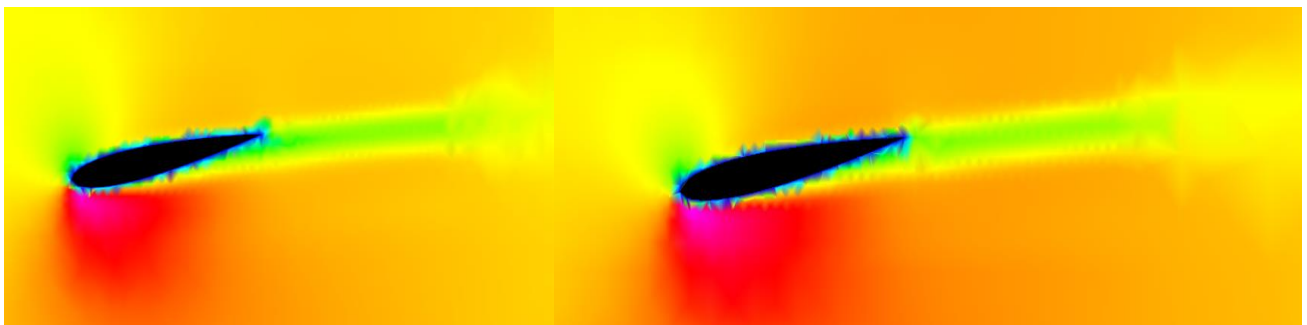


Figure 9 - Comparison of K Epsilon and K Omega for NACA 0012 profile

However, when the AoA was increased past 4° the CFD was not accurately predicting the point of separation. Due to this it was decided to change the turbulence model to try and predict the point of separation more accurately. To do this the K Omega turbulence model was chosen. It was chosen as it has a greater ability to predict near wall interaction, it is also superior at predicting the point where separation will occur. A comparison can be seen in figure 9. Although the flow structures appear to be similar, the pressure distributions between the two are different.

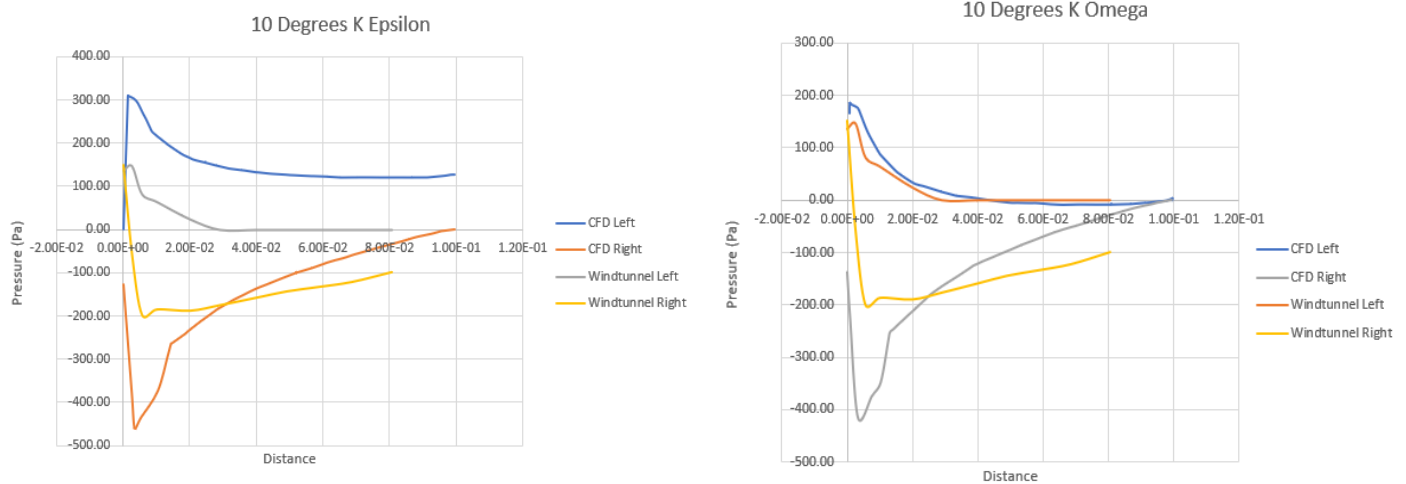


Figure 10 - Pressure distribution of NACA 0012 at 10 degrees for both K Epsilon and K Omega

The pressure distributions can be observed in figure 10. By using the K Omega model, the data especially on the positive pressure side of the wing is able to make matched much closer when compared with the K Epsilon model. The low pressure side was not closely matched with either turbulence model. This was down to a meshing issue where prism layers could be applied to the wing for this AoA. Although, this error did not occur for the other AoA that were being tested. The other tests were able to produce closer results and since this was still a direct comparison as the settings for both tests were equal it was not seen as a problem.

To get accurate results using a K Omega turbulence model the mesh resolution close to the wall needs to be more refined, if it isn't the model can under or overpredict the point of separation.(Menter, 1992)(Bradford, Montomoli and D'Ammaro, 2014).

4.3. Ahmed Body Validation

4.3.1. Introduction

Another way to ensure the CFD results are providing accurate results is to use a bluff body. A bluff body is a shape used to roughly resemble a car shape. The Ahmed body slightly resembles the shape of a hatchback car due to the notch at the rear. Bluff bodies are used to correlate wind tunnels (Kasim and Al Zaidee, 2015). This validation study was conducted to investigate the effects of different mesh strategies and different turbulence models. Wind tunnel results were provided to compare the CFD results against, these can be seen in table 2.

Table 2 - Given values for Ahmed body

Slant Angle (φ)	Lift Coefficient	Drag Coefficient
25	0.345	0.299

4.3.2. Mesh Strategy

When looking at the different mesh strategies two main qualities were looked at when determining the best mesh quality alongside the results these strategies produced. The first being orthogonal quality, this shows how much the tetra has been adapted. The more the tetra is altered the greater the angle becomes between the centre of one node to the next node. The larger this angle becomes the harder it becomes for the software to accurately solve the accurately solve between these nodes. The other quality is the skewness of the cells. Skewness is the 'difference between the shape of the cell and the shape of an equilateral cell of equivalent volume'. (Ansys, 2013). The more skewed a cell the more it's had its shape modified to fit into the build grid. If the skew value is large it can be hard for the solution to solve or it can end up producing inaccurate results as the equations assume the tetras are equilateral.(Adariani, 2019). Therefore, typically the lower the skew value the more accurate the results will be.

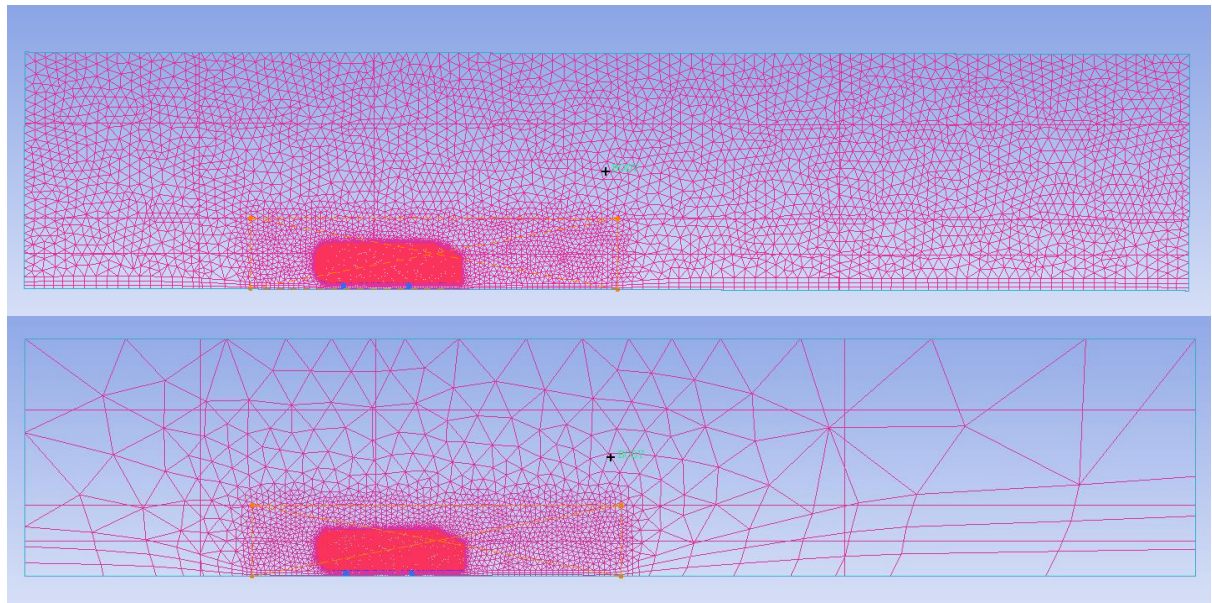
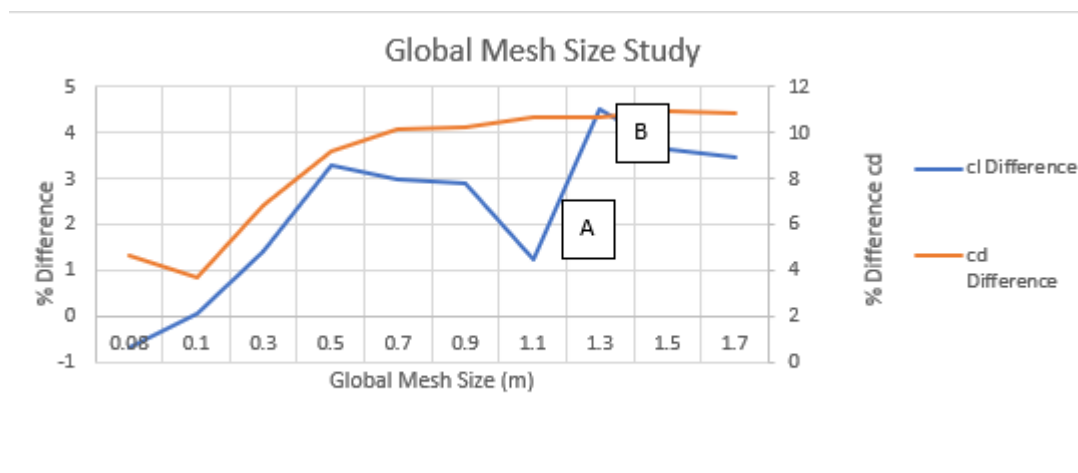


Figure 11 - Fine vs Coarse mesh for Ahmed body

Figure 11 shows the difference between a fine mesh (Top) and a coarse mesh (Bottom). The finer mesh will allow more detail to be captured but will also increase the computing time.(Bakker, 2006)

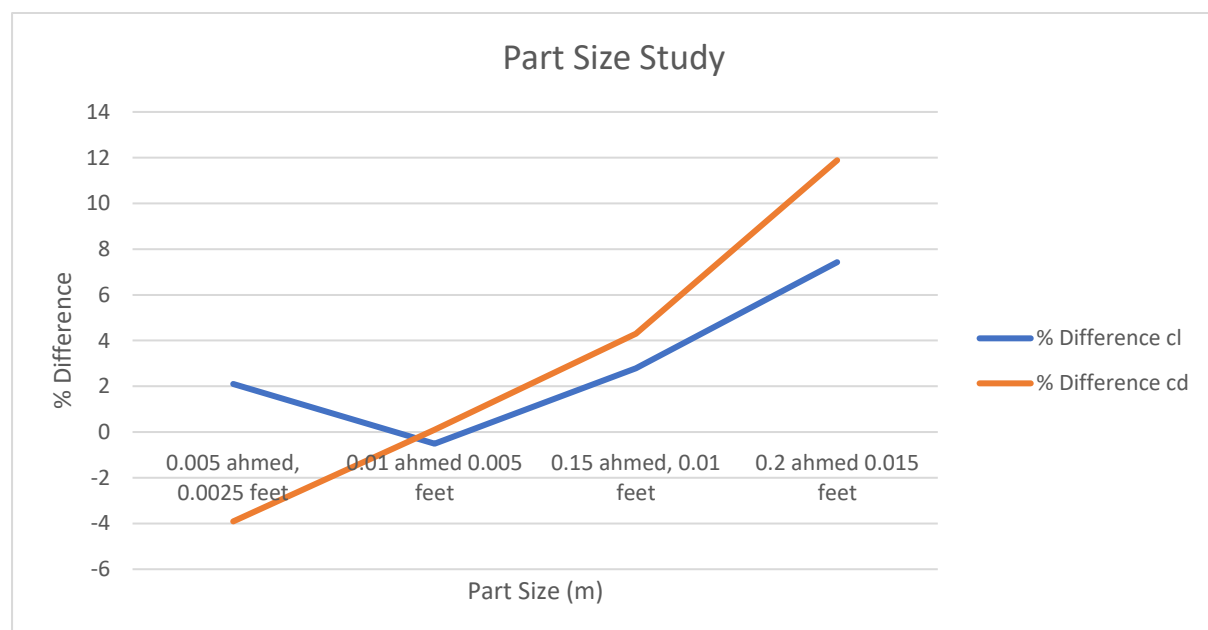
The first case study conducted was the global mesh size, this value determines the elements size in the free stream volume. The smaller the value the more elements there will be in the build grid. The results from this study can be seen in graph 1.



Graph 1 - Global mesh study

Points A and B show two anomalies that occurred during the testing, these were not repeated as they were not expected to produce more accurate results and they don't follow the trend occurring where the larger element size the less accurate the results are. More testing could have been completed using elements smaller than 0.08m however this was not done as this would increase the computing time and meaning the times between runs would become too great and therefore it would not be selected as the optimum global mesh size.

The second case study done was looking at the part size, this is when a separate size can be placed on certain areas of the body to reduce the element size in these specific locations without decreasing the element size in the whole build grid as this was increase the element count and therefore unnecessarily increase the computing time. The model was split into two sections the main body and the feet (stalks). This was done so a finer mesh could be applied to the model and therefore all the details of the model could be captured in the mesh. This is especially important in more complicated geometries with intricate aerodynamic devices where it can be hard to accurately capture the small details. The results of this study can be seen in graph 2.



Graph 2 - Part size study

From graph 2 it can be seen that if the part size is too large it can reduce the accuracy of the results. However, if the part size becomes too small it can skew the elements and therefore produce some inaccuracies in the final results. These can be seen in figure 12 with the 5mm Ahmed on the top and the 10mm Ahmed on the bottom.

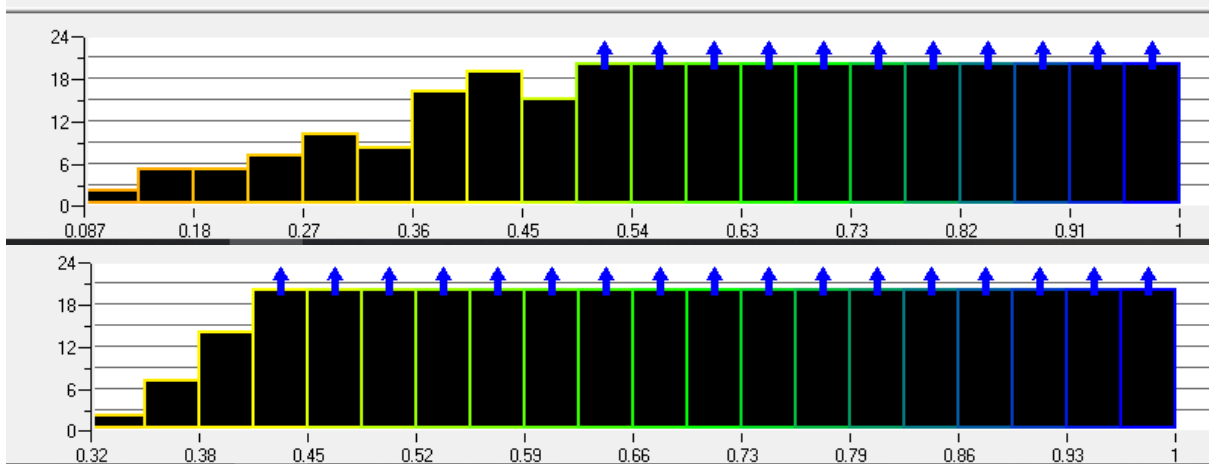


Figure 12 - Skew comparison for different part sizes

The third case study done was investigating the element size of a density box. Density boxes are used to create a finer mesh around certain areas inside the build grid. Normally a density box is placed around the whole model to improve the accuracy both before the air hits the model to accurately predict the effects of the leading edge and also the flow structures produced at the rear of the model. Multiple density boxes can be placed inside the same build grid; therefore, one can be placed around the entire model and another one can be placed around an area of interest such as areas of separation.

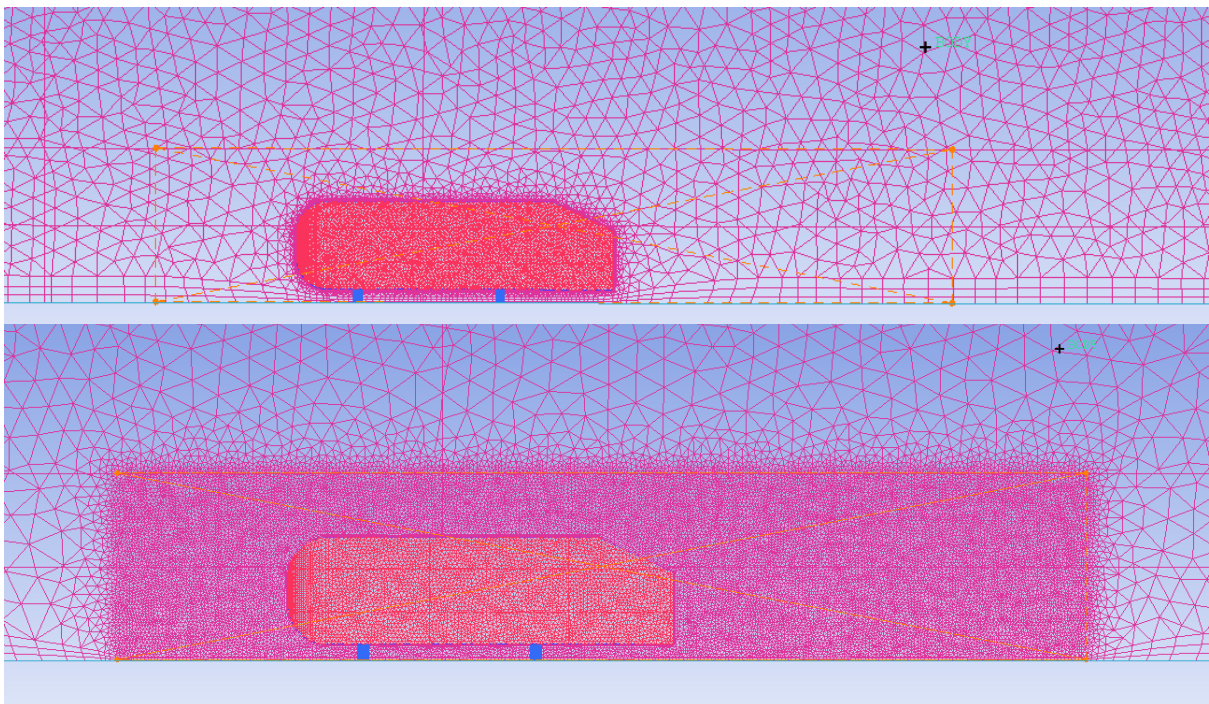
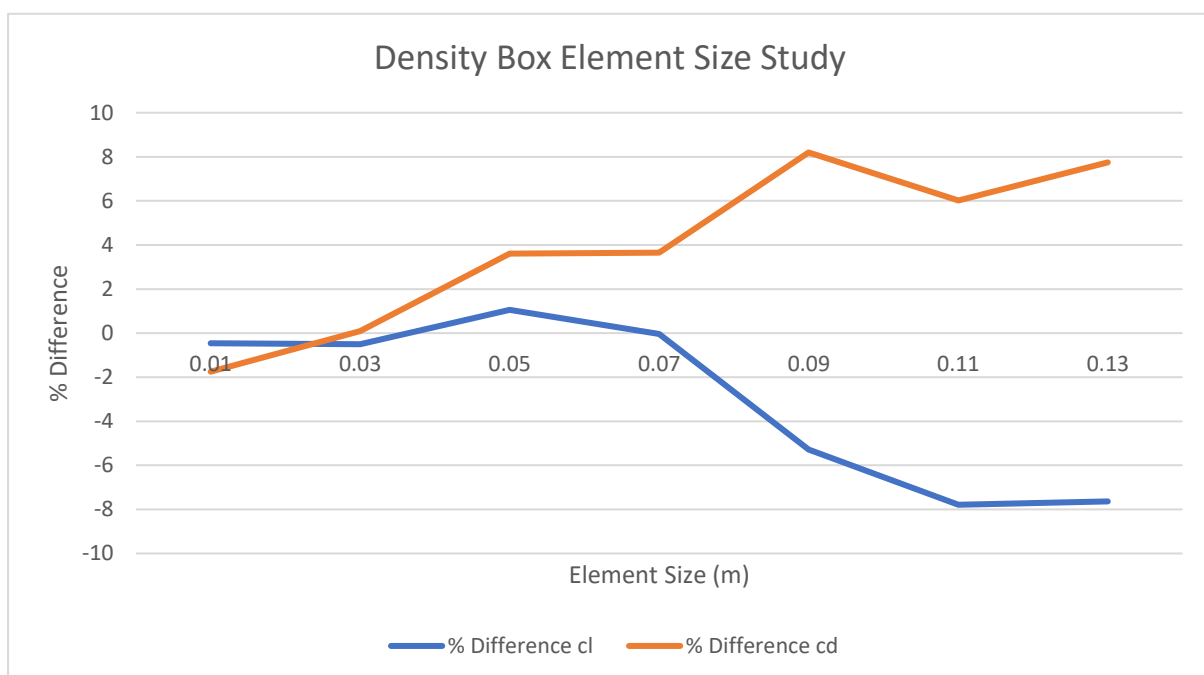


Figure 13 - Fine vs coarse density box comparison

Figure 13 shows the difference between a coarse density box (Top) and a fine density box (Bottom). By having a finer density box, it allows more detail to be captured in the wake region behind the body.

For the Ahmed body a second density box was placed around the notch to see the effects of two density boxes. This didn't have any major impact on the results as the geometry of the Ahmed body is not very complex. Therefore, the rest of the study was done using just one density box.

Density boxes were placed around the Ahmed body with element sizes ranging from 10mm to 130mm in 20mm increments. The results from this study can be seen in graph 3.



Graph 3 - Density box element size study

From graph 3 it can be seen that increasing the element size inside the density box decreases the accuracy of the results. An element size of 30mm was chosen as the most accurate element size as it was the closest results to the datum and the 10mm was slightly further away from the datum and also increased the computing time.

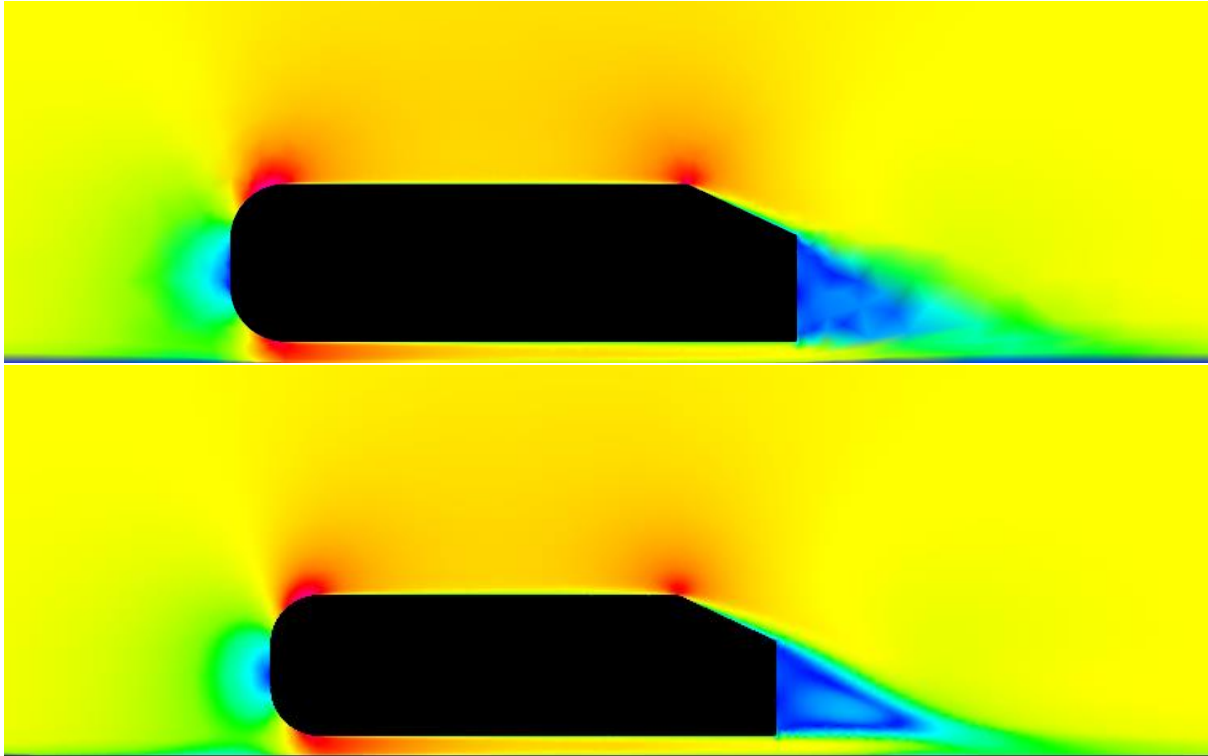
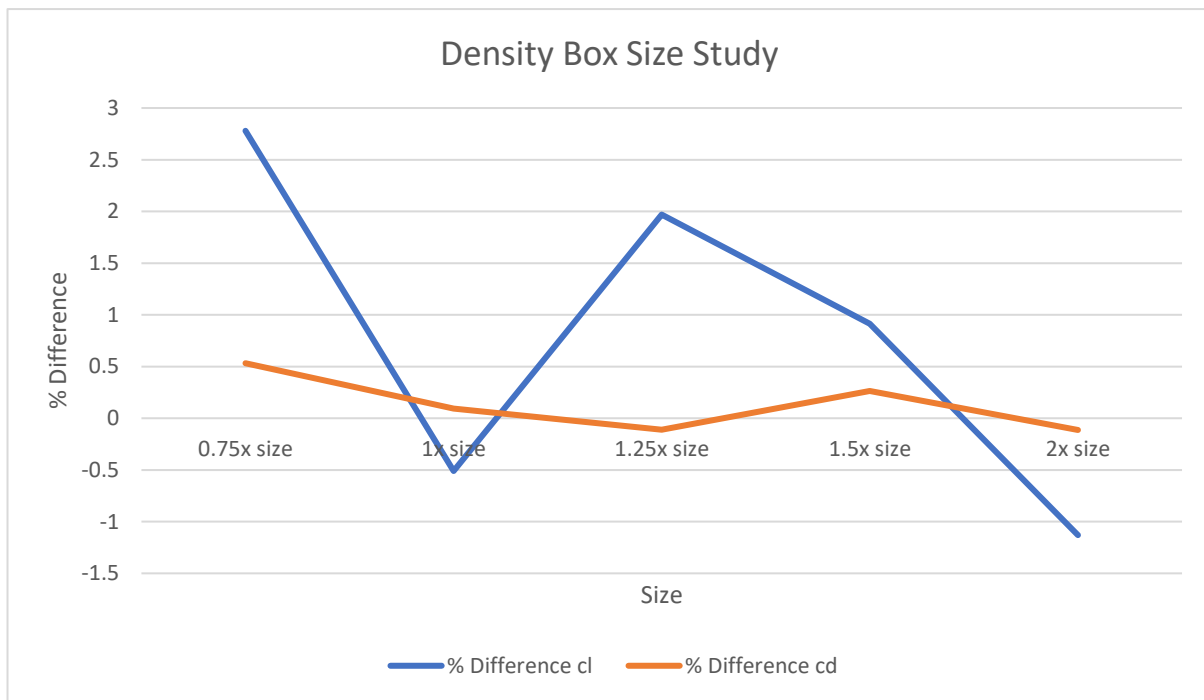


Figure 14 - Velocity contours for fine vs coarse density box

Figure 14 shows the difference with pressure contours when different size elements are used in the density box with the top being a coarse mesh and the bottom being a fine mesh. While both show similar flow structures and the main pressure points can be identified on both the fine mesh makes it easier to analyse the results as they appear more structured since there are more points being calculated.

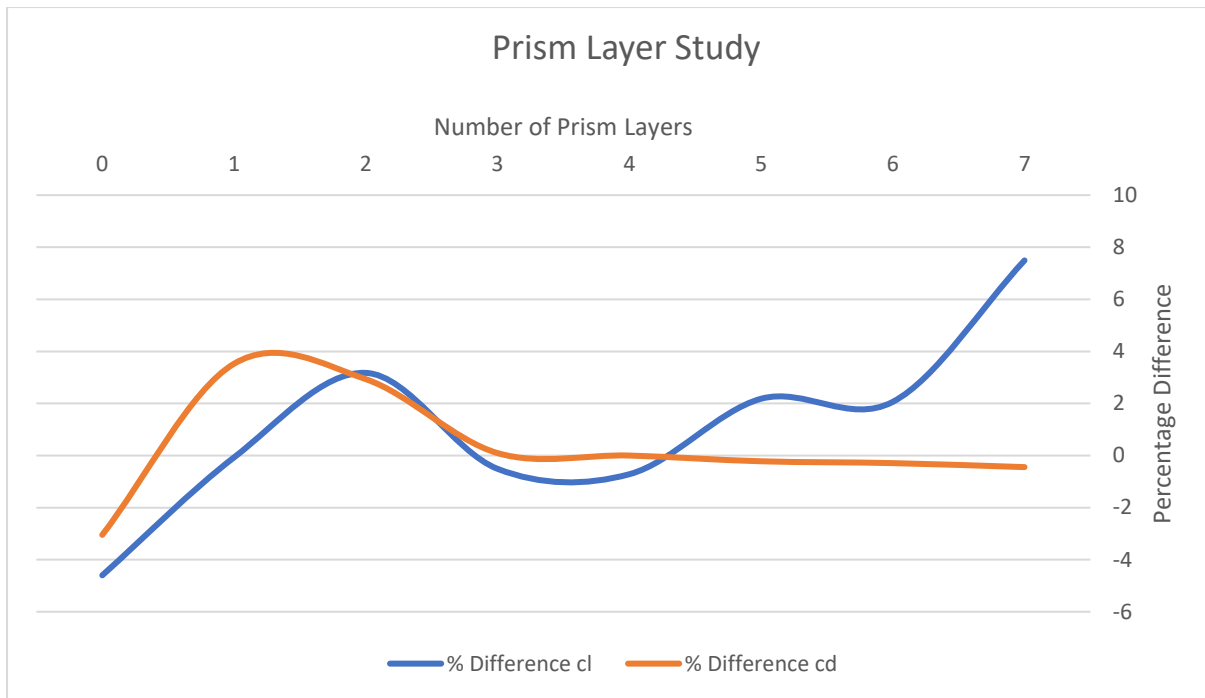
A study was then done to look at the effect of the size of the density box. If the density box is too small it might not capture enough detail, especially in the wake of the model, however if the density box is too large it will increase the computing time while not adding any extra detail. The original density box was 2.5x the length of the Ahmed body and 1.5x the width and height of the Ahmed body. Since the original results were close to the datum it was chosen to scale the modified size density boxes would be scaled up and down from this size. The results can be seen in graph 4.



Graph 4 - Density box size study

From graph 4 it can be seen that the size of the density box has a minor impact on the cd, it does however have a large impact on the cl. The graph does not seem to follow a logical trend which could be due to an anomalous result, although since the questionable results was using the settings that had been used in previous studies it was decided not to repeat the test and continue using the settings used in the previous test.

The final test conducted was looking into how the amount of prism layers placed on both the model and their effects on the final result. Prism layers are placed on the surface of a model to try and simulate a boundary layer acting on the surface. (Dixon *et al.*, 2008). 8 test were done with the amount of prism layers ranging from none to 7. These results can be seen in graph 5.



Graph 5 - Prism layer study

3 prism layers was chosen as the best option as it generated the results closest to the datum. When more layers are added after 3 prism layers the cd levels out, suggesting that any prism layers after 3 has little effect on the cd. The CL does continue to dramatically change however, this could occur as the prisms layers are simulating the boundary layer and by adding more prism layers it is increasing the size of the boundary layer. (Sakamoto and Haniu, 1988). This could be causing the solver to over or under predict the point of separation.

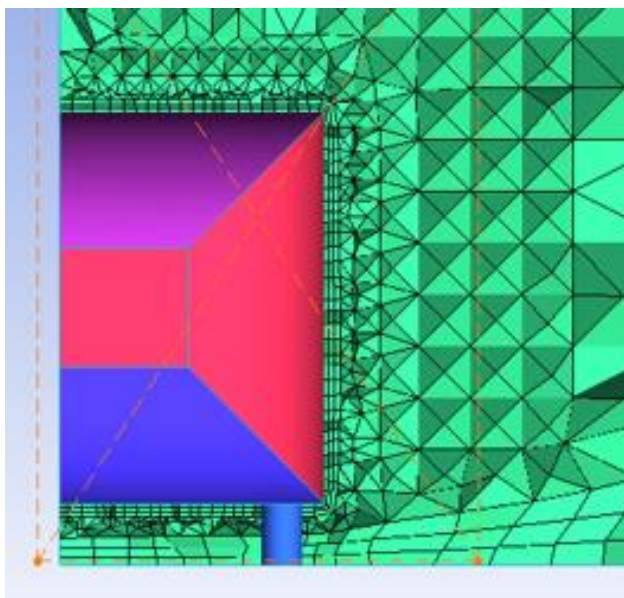


Figure 15 - Prism layers on Ahmed body

Figure 15 shows how the build grid is formed while prism layers have been applied. It can be seen that prism layers have been applied both around the Ahmed body and on the road. There is a gap between the prism layers between these two points, if the prism layers from the road and Ahmed body merged into each other this could cause some solving issues as fluent wouldn't know how to cope with merging prism layers. (Champagne-Ardenne, 2008)

4.3.3. Solver

An investigation was done into the some of the different turbulence models in Fluent. Mainly focusing on the K Epsilon and K Omega models as these are the most common in the automotive industry. However, first and investigation was done to see the impact of using a laminar turbulence model over a 2 stage turbulence model.

The laminar model is only able to model the flow in a laminar state and therefore, can't model turbulent flow or flow separation. This model should only be used when the Reynolds number won't reach a large enough value for the flow to become turbulent. The results can be seen in table 3.

Table 3 - Laminar results

Test	cl	cd	Efficiency	% Difference cl	% Difference cd	%Difference efficiency
Laminar	0.08972	0.19988	0.448879328	-126.0063768	-33.15050167	-138.9028751
Given	0.345	0.299	1.153846154			

As the Ahmed body has a slant on it, some separation will occur and due to this the linear model can't model the flow structures correctly which produces inaccurate results.

K Epsilon was looked at due to it being one of the standard model used in vehicle aerodynamics. This is because it is good at modelling flow where there are small adverse pressure gradients(Singh *et al.*, 1983), this means it is good at modelling flow structures behind the model and predicting flow far away from walls. There are two main differences between the standard and realisable models, the first is the viscosity equation is constant with the standard model and is a variable in the realisable model. The second is the realisable model has a more accurate equation for the rate of dissipation of kinetic energy. This means the realisable model should be better at predicting separation and recirculation.(ANSYS FLUENT, 2012).

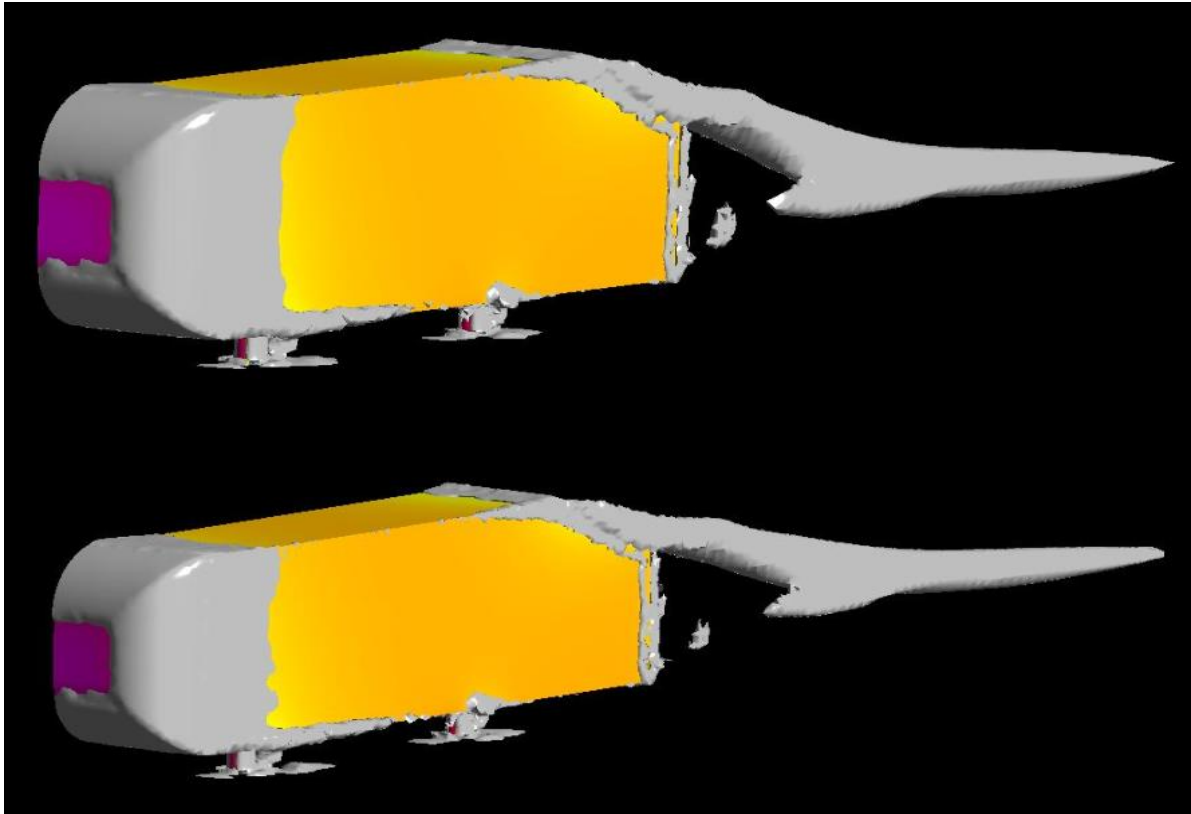


Figure 16 - Comparison of K Epsilon flow structures

Figure 16 shows a comparison of the vortices created by using the K Epsilon with enhanced wall treatment (Top) and K Epsilon non equilibrium wall functions (Bottom). These two flow structures appear to be near identical with the only minor difference being that the enhanced wall treatment produces a larger vortex from the trailing edge.

K Omega is another common model used in vehicle aerodynamics as it is good at modelling near wall conditions. As K Omega is better at predicting near wall interaction this model is greater at predicting the point where separation will occur. Although, this does require the mesh resolution close to the wall to be more refined otherwise the point of separation can be over or under predicted.(Menter, 1993). The SST model has a more advanced turbulent viscosity equation than the standard model. This means the SST model has the close to wall functions of the K Omega model and the away from wall functions of the K Epsilon model while still only having two transport equations. This means the SST model is able to predict a greater range of flow structures and pressure gradients compared to the standard model.

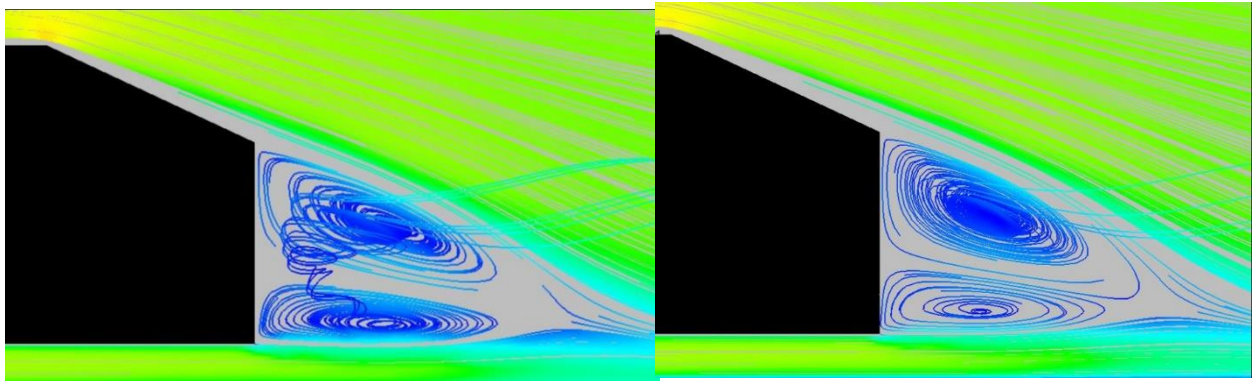


Figure 17 - Comparison of K Omega flow structures

Figure 17 shows how the standard (left) and SST (right) produce different flow structures in the wake region. Both are comparable although, a greater vortex is produced at the trailing edge.

The results from this turbulence study can be seen in table 4.

Table 4 - Turbulence study results

Test	cl	cd	Efficiency	% Difference cl	% Difference cd	%Difference efficiency
K-epsilon Standard	0.35981	0.32494	1.107312119	4.292753623	8.675585284	-4.032949673
K-epsilon RNG	0.33664	0.28875	1.165852814	-2.423188406	-3.428093645	1.040577201
K-epsilon Realizable NEWF	0.34427	0.29861	1.152908476	-0.211594203	-0.130434783	-0.081265419
K-epsilon Realizable EWT	0.3504	0.30776	1.138549519	1.565217391	2.929765886	-1.325708344
K-Omega Standard	0.34226	0.29061	1.177729603	-0.794202899	-2.806020067	2.069898948
K-Omega SST	0.31926	0.29454	1.08392748	-7.460869565	-1.491638796	-6.059618388
Transitional SST	0.3161	0.29486	1.072034186	-8.376811594	-1.384615385	-7.090370571
Reynolds Stress	0.33183	0.29248	1.134539114	-3.817391304	-2.180602007	-1.673276805

From these results it can be seen that all of these turbulence models will produce CL and cd results accurate for most basic cases. Although the choice of turbulence model does have an impact on not only the CL and cd but also on the resulting flow structures. Therefore, the choice of turbulence model will have to be carefully considered depending on the individual case.

4.3.4. 35 Degrees

A small investigation was done to see if the mesh settings used for the 25° model could be directly transferred for the 35° model. If they could this would suggest that there would be an optimal mesh strategy for any model.

Table 5 - 35 degree turbulence study results

Test	CL	CD	Efficiency	% Difference CL	% Difference CD	%Difference efficiency
K-epsilon Standard	0.42431	0.3577	1.186217501	10507.75	28.2078853	8173.867067
K-epsilon RNG	0.062201	0.27312	0.227742384	1455.025	-2.107526882	1488.50313
K-epsilon Realizable NEWF	0.5593	0.44607	1.253839084	13882.5	59.88172043	8645.527608
K-epsilon Realizable EWT	0.26641	0.32095	0.830066989	6560.25	15.03584229	5689.717246
K-Omega Standard	0.13583	0.29133	0.466241033	3295.75	4.419354839	3152.031202
K-Omega SST	0.96079	0.30225	3.17879239	23919.75	8.333333333	22072.07692
Transitional SST	0.127	0.30402	0.417735675	3075	8.967741935	2813.706335
Reynolds Stress	0.15108	0.29895	0.505368791	3677	7.150537634	3424.947316

Table 5 shows the results by using the same mesh strategy. It can be seen that results are completely inaccurate from the test data. This shows that careful consideration needs to be taken with mesh strategy for each individual model.

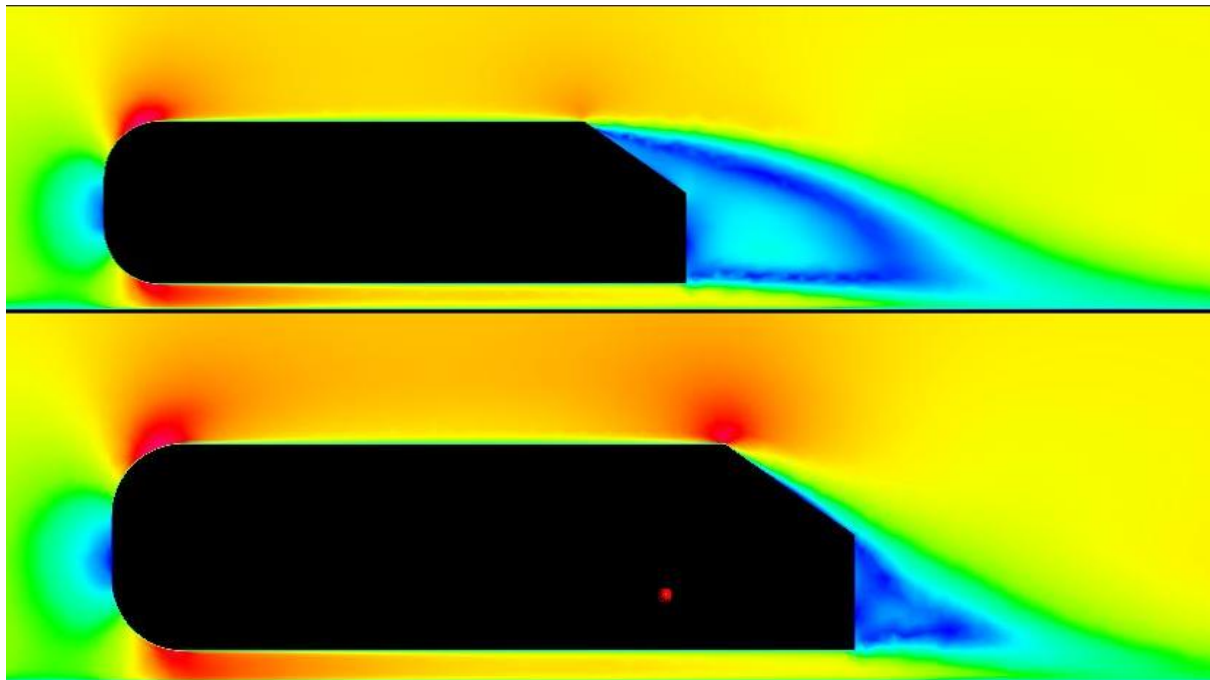


Figure 18 - K Omega vs K Epsilon flow structure comparison for 35 degree slant

Figure 18 shows a comparison in the flow structures with different turbulence models, K Omega (Top) and K Epsilon (Bottom) where there are large areas of separation. This gives a visual example why the choice of turbulence model can have a large impact on the results and the resulting flow structures.

4.3.5. Does Using Various Computers Make a Difference?

Another small investigation was done to see if using various computers can have an impact on the results while running the same case.

Table 6 - Multiple computer results

Test	cl	cd	Efficiency
PC 1	0.35536	0.32937	1.078908219
PC 2	0.35998	0.32965	1.092006674
PC 3	0.3506	0.32898	1.065718281

Table 6 shows the results from this investigation. This shows that there is some variation with the results while run on different computers. There ended up being a 2.41% difference in efficiency across three computers. This proves that to guarantee comparable results they should all be run on the same computer.

For future wing studies they will all be run on the same machine to ensure comparable results.

5. Full Car CAD Models

5.1. Original CAD Model

The original CAD model started off as a formula 4 car. Over the summer this model was adapted to create a representation of the formula 3 car. This was done by John Hughes using both photos and measurements taken by SBD. This model was used for initial runs while an accurate model was being created. While results from this model were able to provide initial guesses for designs there were some problems. The wheelbase of the model was 700mm too long, the wheels were the correct size and many of the intricate aerodynamic devices were not correctly modelled. Due to these inaccuracies it was decided to conduct studies that didn't require the full car model to be completed until an accurate model was created.

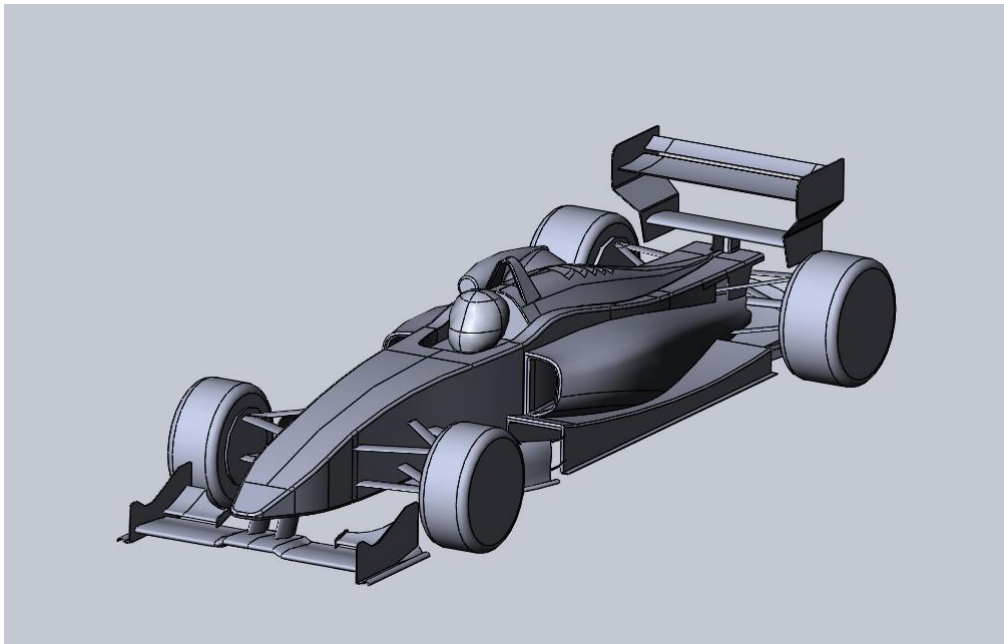


Figure 19 - Original CAD model

During October the car was delivered to the university so it could be scanned, and certain parts could be measured independently of the scan. For this project the rear wing and endplates were removed from the car. The endplates, profiles and measurements of the aerofoils were measured to create an accurate model of the rear wing assembly. Whilst the car was at the university Central Scanning came to scan the car. This scan was sent off to an external company to convert the scan into a solid CAD model.

5.2. Scanned CAD Model

Once the CAD model was completed some changes were made, this included simplifying certain sections of the car to improve the mesh quality and removing the front and rear wing to replace them with those made using measurements taken from the actual car. This was done as it was unsure how accurate the scan of the wings would be and since the CAD models were already created they were added to ensure the accuracy in these areas. A gurney flap was also added to the second element of the rear wing to replicate the gurney flap on the actual car. A helmet was added at the height of the tallest driver, this was done as the tallest driver is assumed to be the worst case scenario as they will disrupt the airflow more than a shorter driver.

When comparing figures 19 and 20 many differences can be seen between the original and scanned CAD model. A few minor improvements were made to the CAD model to try and improve the accuracy of the results produced using this model. Some of these include adding the timing breaker on the front wing, adding the jagged windscreen in front of the driver and adding the camera on top of the roll hoop as this will slightly affect the airflow hitting the rear wing. There is also the section for damper adjustment on the front nose of the car. There were also a few major changes not previously discussed. The suspension positioning has been more accurately modelled and also simplified, the underfloor has also been more accurately modelled. The most significant change is the diffuser and sidepod modelling. This will both effect the overall downforce of the car but will also impact the efficiency of the lower wing.

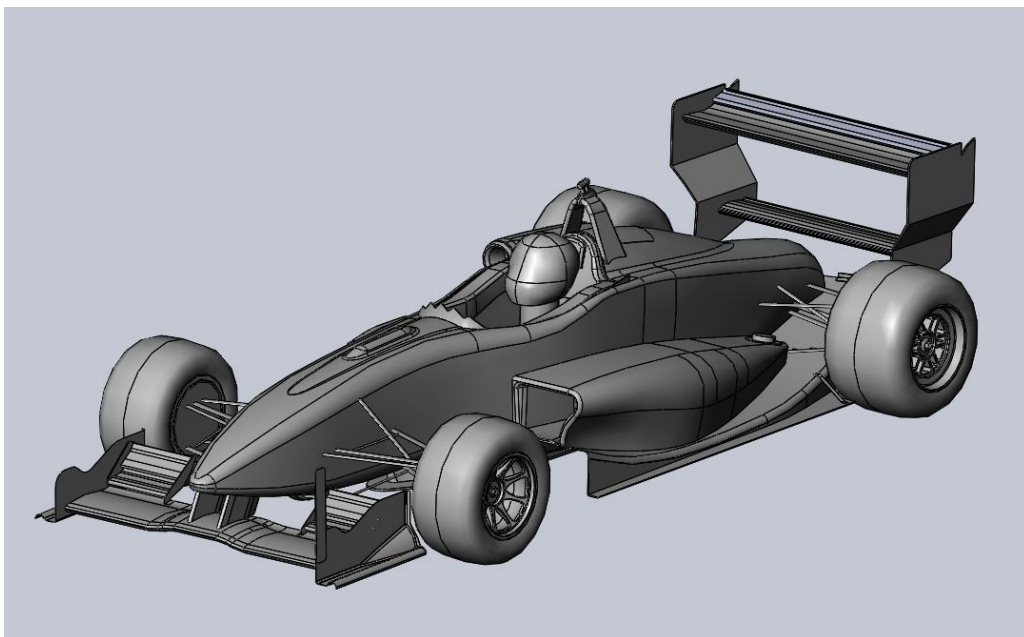
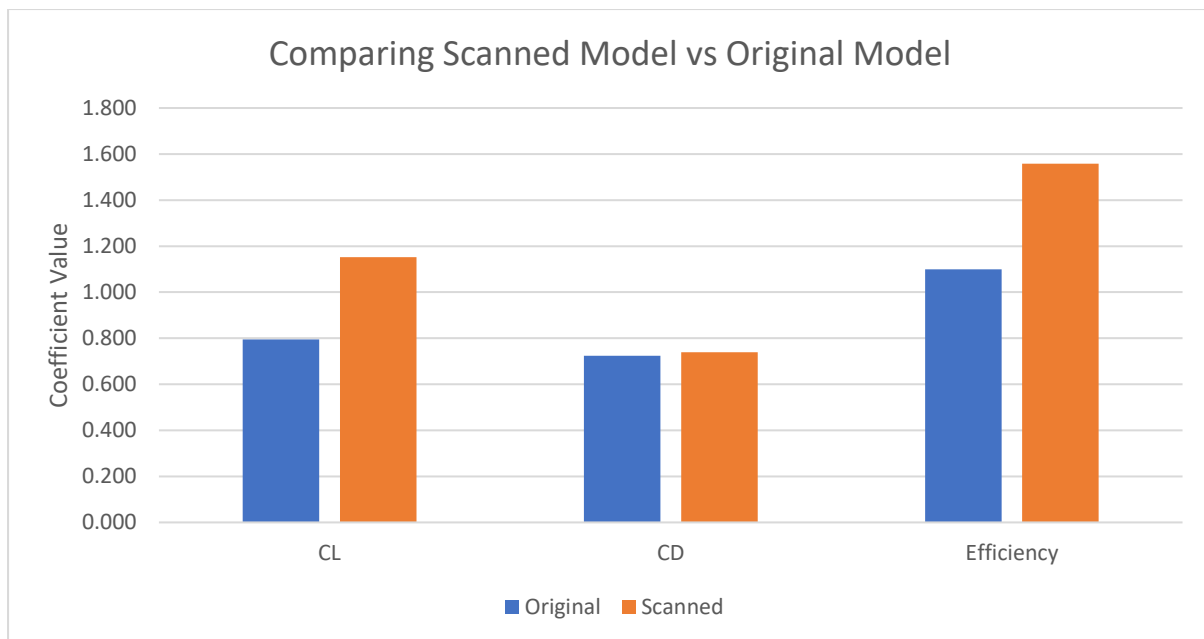


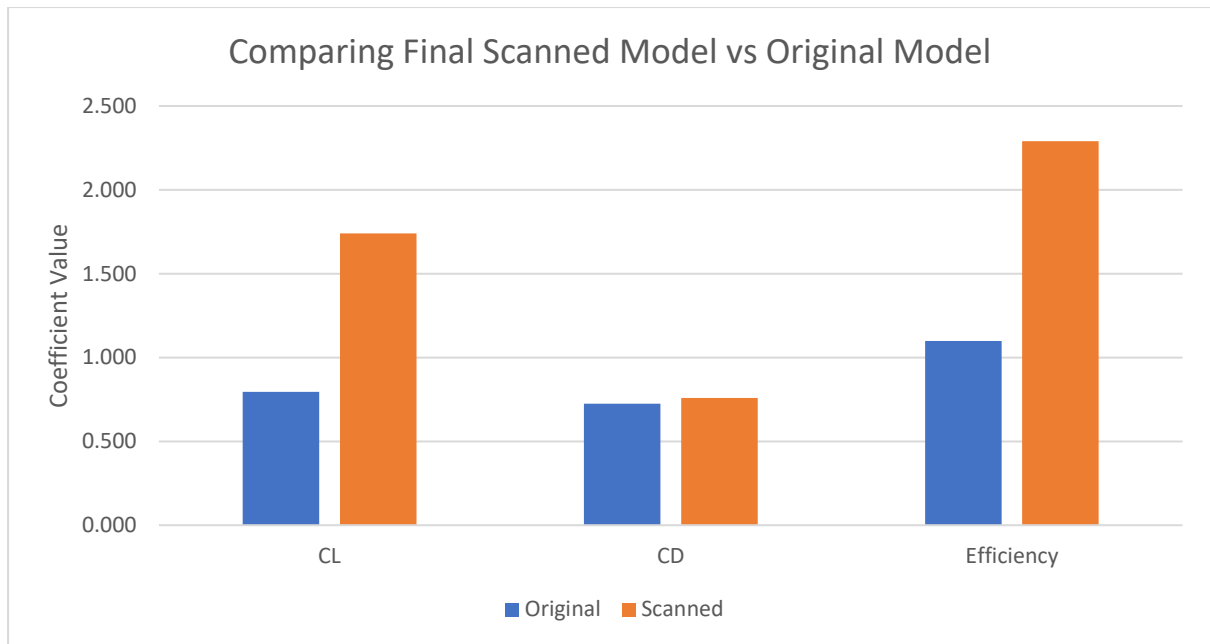
Figure 20 - Final CAD model



Graph 6 - Scanned vs original results

Graph 6 shows the difference between the first scanned and original models. For ease of comparison the CL has been made positive when it really is negative. This graph shows that the original model produces a CD value close to the actual CD value produced by the scanned model and there is only a 2.2% difference. Although there is a large difference in both the CL and overall efficiency. With a 31% difference in the CL and a 29.5% difference in the overall efficiency. This will be due to the differences in the geometry. (Bak, Hwang and Cho, 2017) These results show why the original model couldn't be used as they would end up producing inaccurate results, therefore any changes made possibly wouldn't work for the actual car.

Once the scanned model had been adjusted to improve the accuracy the difference between it and the original model became greater.



Graph 7 - Final scanned model vs original results

From graph 7 it can be seen that the original CAD model produced completely inaccurate results when compared with the final scanned CAD model. This again proves why the original CAD model couldn't be used for aerodynamic development as this model produced results that were not close to the actual model.

6. Wing in Isolation

To try and reduce the computing time while running case studies a test was done to see if the wing could be run in isolation. If the wing is able to be run in isolation it will then allow each of the case studies to be completed by just running the wing without the need to run the whole car in CFD. It was found to reduce the computing time from around 18 hours per run for the full car to just 2 hours for the wing.

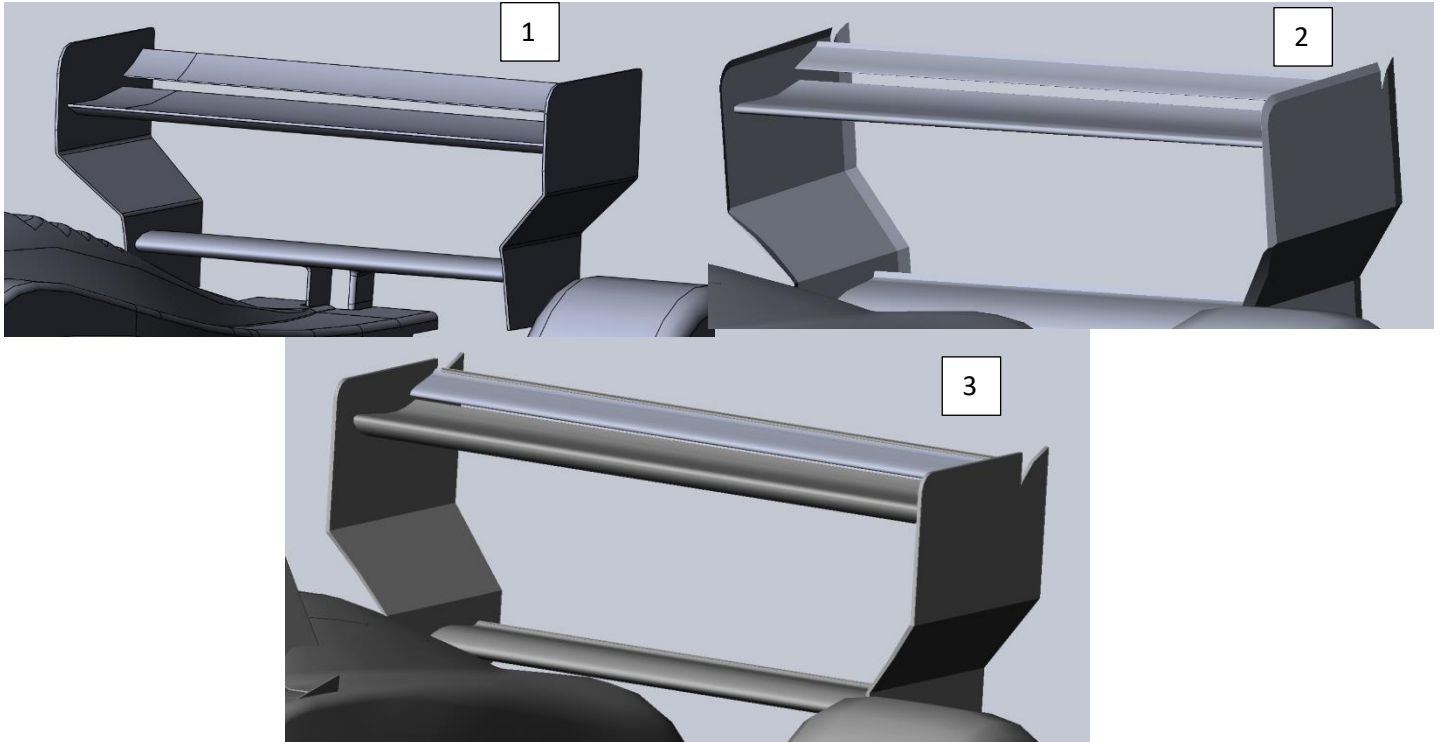


Figure 21 - Comparison of standard wing models

Figure 21 shows three versions of the original wing. Wing '1' is the wing from the original model, wing '2' is the wing from the scanned model and wing '3' is the wing created from measurements. Wing '1' has many inaccuracies including the endplates being the wrong dimensions and not including the notch at the rear of the endplate, the profiles of the wings being too thin and being the wrong shape, the gap between the two elements is too large and the mounting for both the lower element and the lower wing being incorrect. Wing '2' also contains some inaccuracies with the endplates being slightly the wrong design and the wing profiles again being too thin and the wrong profiles. These issues were solved with wing '3', this is why the decision was made to use the wing created by measurements and place it on the scanned model instead of using the scanned rear wing model.

As the air will not hit the rear wing at a vertical angle due to the shape of the car distributing the airflow, the angle of the air needed to be calculated to try and replicate the distributed air that

would hit the wing. To do this first a full car run needed to be completed and this run would be used to calculate the x and y velocity needed. A line was added in the CFD post processor in front of the leading edge of the main plane as this was the first point of contact between the air and the wing. The line stretched across the entire span of the wing to ensure all of the detail was captured including the effects of the endplates which would alter the airflow hitting the edges of the wing. Two graphs were created in the post processor, one for the x velocity and one for the y velocity. These graphs show the velocity on the y axis and the distance. The average of the x velocity and the average of the y velocity were taken. When the wing is run in isolation instead of selecting the x velocity inside fluent as 40 m/s and the y velocity as 0 m/s which is done with the full car model, the x velocity is set as 38.3 m/s and the y velocity is set as -0.87 m/s. This creates a slight downwash effect **ADD REFERENCE** which would be seen on the full car model. This effect is caused as the air flows over the car it becomes attached to the engine cover, as the engine cover is at an angle the air flow approaches the wing also at an angle.

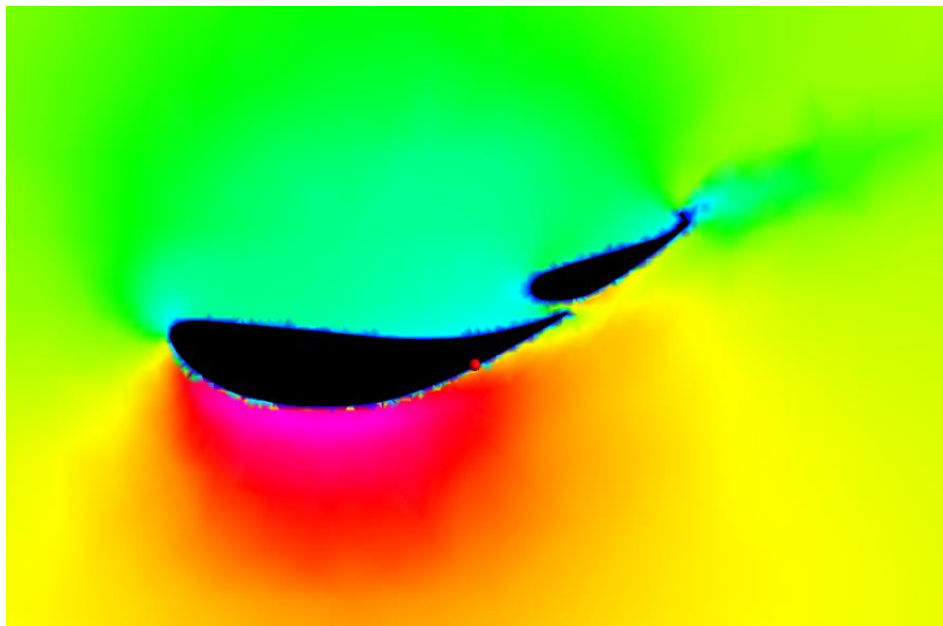


Figure 22 - Pressure contour of original wing in isolation

Figure 22 shows the wing in isolation. The reason the structure on the layer of the wings appears blocky is due to prism layers not being applied on this run. Unfortunately, the prism layers could not be applied to the wing while in isolation, this was due to a meshing issue that couldn't be resolved.

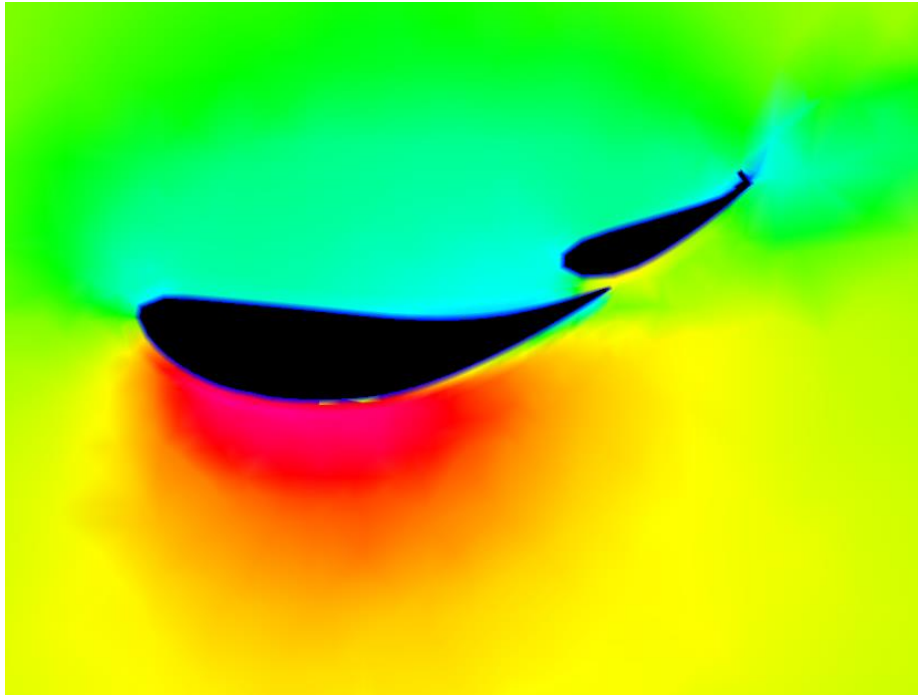
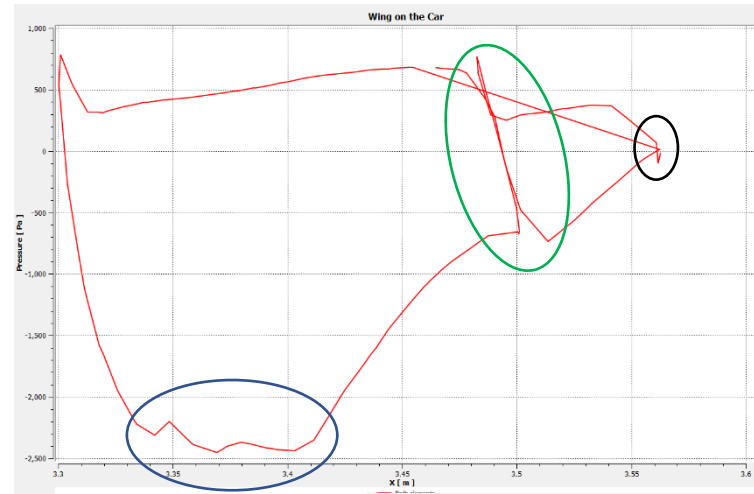


Figure 23 - Pressure contour of original wing attached to the car

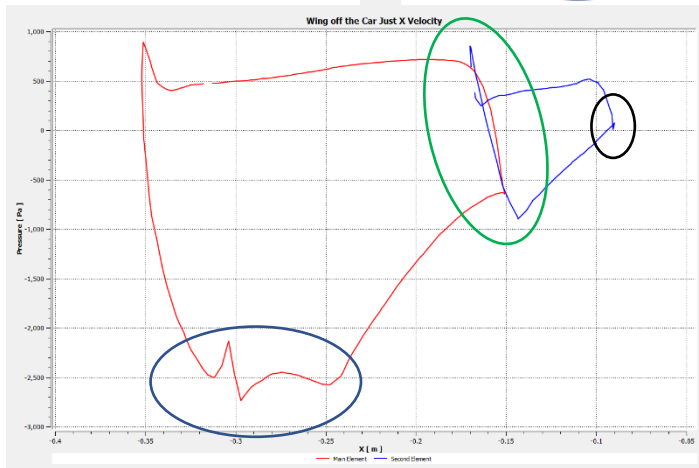
Figure 23 shows the wing while applied to the car. The flow across the wing appears cleaner which is most likely due to the prism layers applied to the wing. Although there are some slight differences the overall flow structure across both wings appears similar.

When looking at the results the wing is isolation looking pressure distributions on the main and second elements of the wing as seen in graph 8. This shows that the x and y velocities are producing similar results as if the car was there. These graphs display the distance from the leading edge of the main element to the trailing edge of the second element on the x axis, it shows the pressure on the y axis. The top graph (1) shows the wing while attached to the full car, the bottom left graph (2) shows the wing in isolation with only an x velocity of 40 m/s coming from the velocity inlet. Finally, the bottom right graph (3) shows the wing in isolation with a resultant x and y velocity applied from the velocity inlet. For the bottom two graphs the main element and second element were separated to make viewing the graph easier, the main element can be seen in red and the second element can be seen in blue.

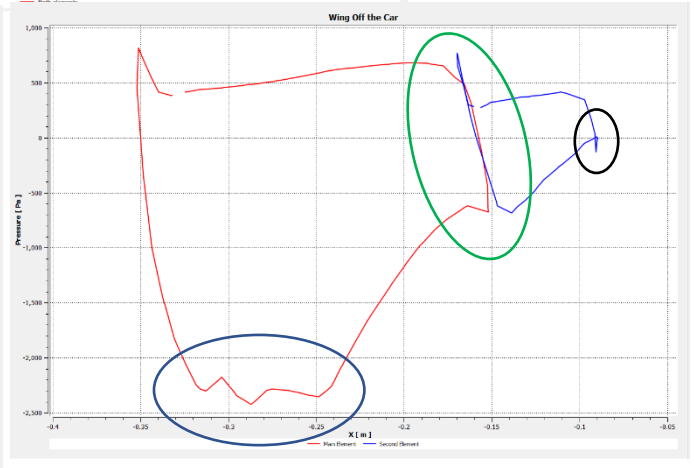
1



2



3



Graph 8 - Pressure distributions of original wing

There were three key areas were identified when comparing the differences between the three graphs. The first is highlighted at by the blue circles, this is occurring on the low pressure side of the main element. The variation occurring with the pressure inside this area is most likely due to a small meshing or CAD issue, this is assumed because it is expected that the pressure will drop to the minimum value before gradually returning to ambient pressure.(Davis and Saltzman, 2000). However, the trends between graphs 1 and 3 closely resemble each other while graph 2 shows a different shaped line. This shows that even if there is a minor error with the mesh or geometry the wing in isolation by applying a resultant velocity the trends will still be replicated. The second highlighted point is the green circle. This point is the transition from the main element to the second element. Graphs 1 and 3 show near identical trends at this transition point, this transition is as expected where the pressure on the main element dramatically drops off at the trailing edge and the pressure on the second element starts at a large high pressure similar to the start of the graph. Graph 2 shows there being a larger overlap between the main and second element, this is another difference by using just an x velocity rather than a resultant velocity for an isolated wing. The third point of consideration is the black circle, this is the trailing edge of the second element. It again can be seen that graphs 1 and 3 show the same trends while there is a difference in graph 2. This again

shows that by adding a resultant velocity it increases the accuracy of the results when compared with just using an x velocity.

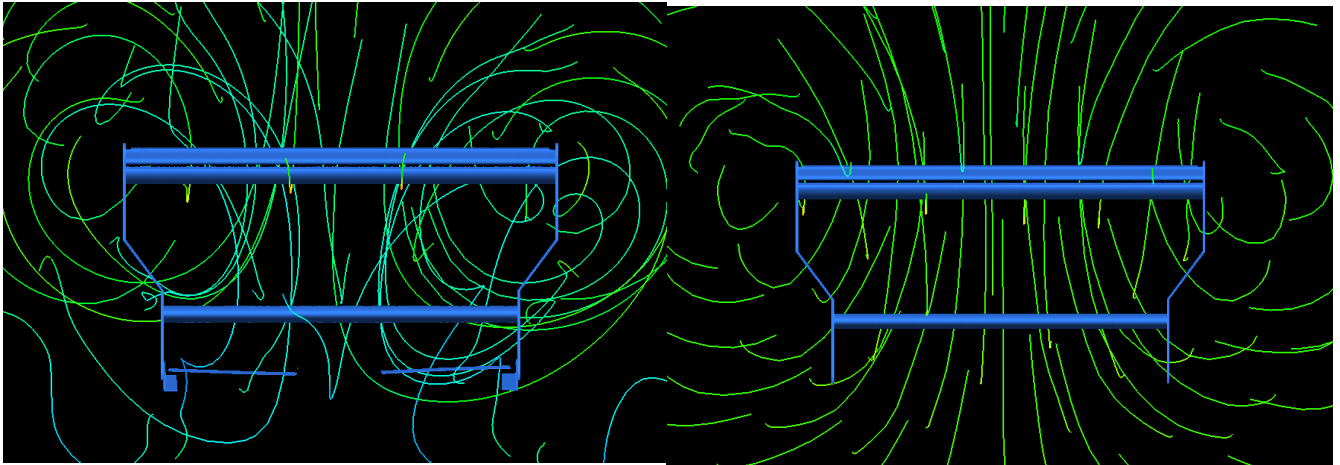


Figure 24 - Flow structure of wing in isolation and attached to the car

However, there are 2 main issues by running the wing in isolation. One is that some of the turbulence being produced by the car won't be replicated with the wing in isolation. This can be seen in figure 24 where the flow structure on the left is taken from the full car and the flow structure on the right is taken from the wing in isolation. It can be seen that the vortices produced near the endplates are similar as is the flow attachment to the main element this is due to the resultant velocity being applied to the wing in isolation. However, the overall flow structure is different, this is due to the resultant velocity not being able to replicate the overall flow disruption caused by the cars body and aerodynamic devices. The second is the new centre of pressure acting on the car won't be able to be calculated until the wing changes are placed onto the full car model. A decision was made that time saved by running the wing in isolation was more beneficial to the project than the accuracy of running the full car.

When looking at the CL for a wing in isolation a different equation is used as seen in equation 5.

Equation 5 - CL equation for wing in isolation

$$CL = \frac{Lift * 2}{\rho * V^2 * Wing Plan Area}$$

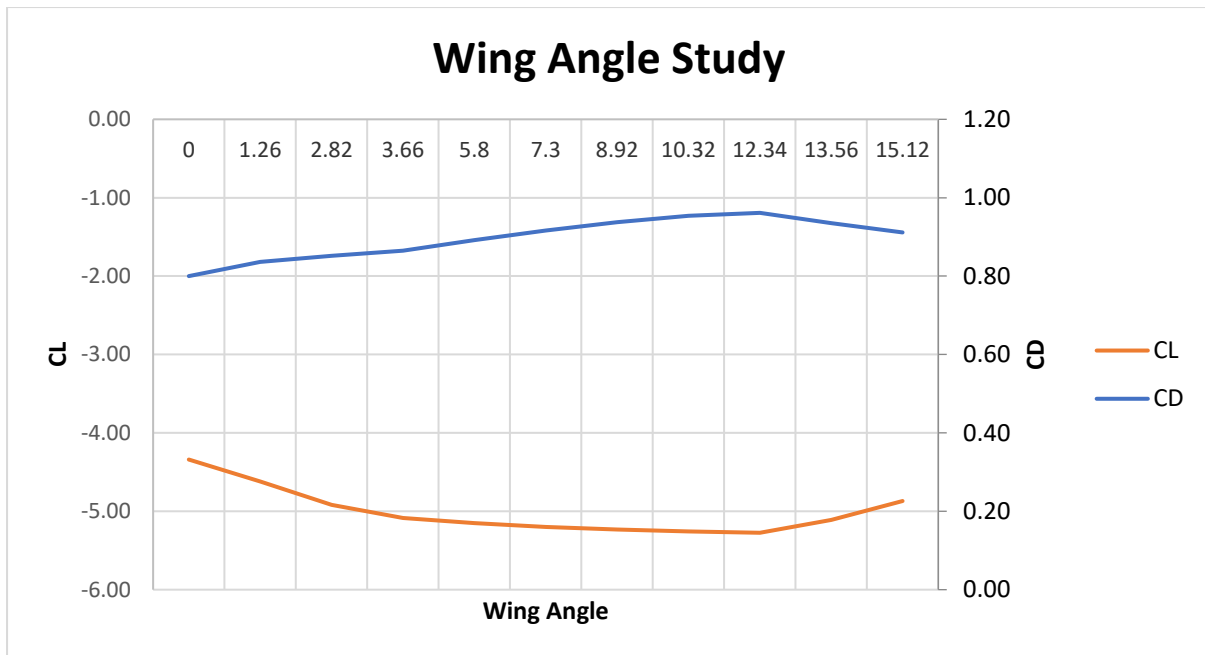
This is the equation that should be used for calculating the CL for a wing in isolation.(Narodytska, no date). However, this was not used for the following case studies, this decision was made as Ansys uses equation 2 to calculate the CL. As the studies were done to compare the CL and CD of the wings

in isolation it was decided to use the CL that Fluent produces as this would still allow a direct comparison to be made between each setting.

6.1. Wing Angle

The first study was done to investigate the impact on the aerodynamics utilising all the wing angles possible on the current rear wing design. The current design has 11 positions that the top two elements can be moved into to adjust the amount of downforce being produced by these elements. To find the angle of the wing at each of these settings the angle was measured on the real car. To do this first the wing was set to its lowest position and use an inclinometer was used to measure both the angle of the main element and the second element. The wing was then raised up step by step until it reached the maximum setting, with the angle of the main and second element being measured at each. The angle at each of these setting could be taken away from the angle at the lowest setting to find the change in angle. The change in angle can then be used to alter the angle of the two elements in CAD, this means 11 different CAD models can be created all with different wing angles to be tested in CFD.

The lowest wing angle would be expected to produce the lowest amount of downforce and the lowest amount of drag, the greatest wing angle would be expected to produce the largest amount of downforce and the largest amount of drag. It is important to look at both the CL and cd in this study as well as the overall efficiency of the wing. This is important as some tracks will have long straights and sweeping corners where a low downforce low drag option on the wing would be preferred, some tracks will be tight and twisty where the additional downforce will be more beneficial even with the additional drag.



Graph 9 - Wing angle study results

Graph 9 shows the CL and CD for all the different wing settings. The angle starts a 0° as this is the minimum setting the wing can be set at, each of the angles shown on the x axis are the angle increases for the main element from the lowest setting. As seen in the graph when the AoA is increased the CD is also increased, this occurs as the frontal area is increased which therefore increases the drag produced.(NASA, 2007). This occurs until the angle passes 12.34 where this is a sudden drop in the CD, this is potentially occurring due to separation. While normally separation will result in an increase of drag.

The potential of this issue being caused being caused by the turbulence model being used was explored, the last wing angle setting (15.12°) was retested using the K Omega turbulence model. As this was found in the Ahmed body study to be better at dealing with separation.(Menter, 1992). However, the K Omega run produced results within 2% of the K Epsilon turbulence model.

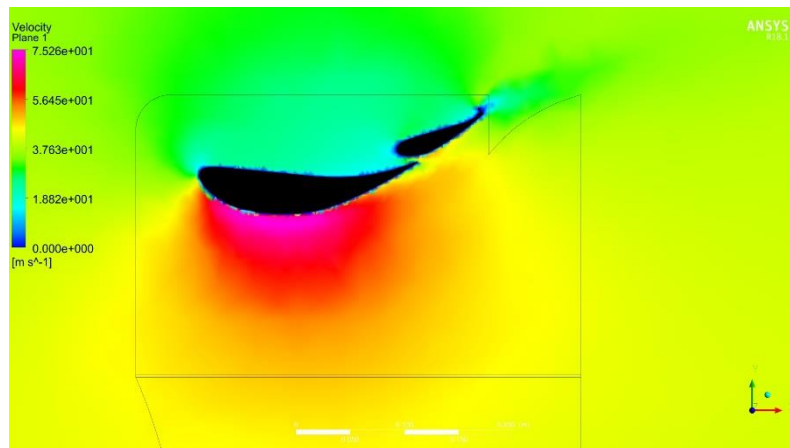


Figure 25 - Wing at minimum angle

Figure 25 shows the wing at the lowest wing setting. At this angle the flow remains attached across both elements.

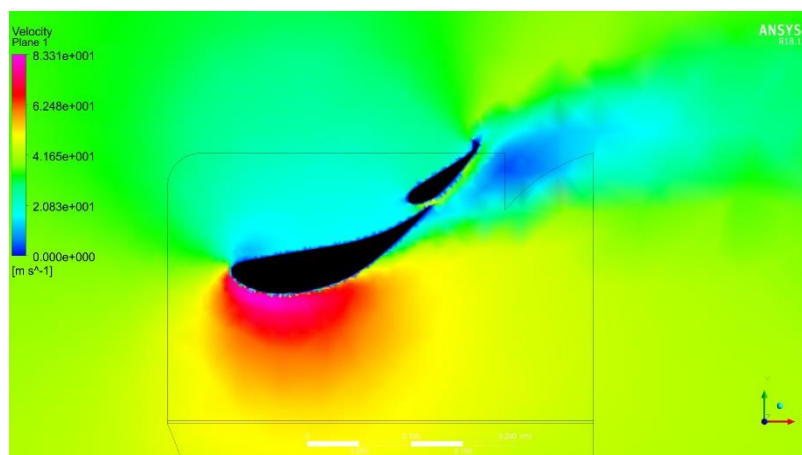


Figure 26 - Wing at maximum angle

Figure 26 shows the wing at the maximum setting. At this setting it can be seen that separation is occurring at this angle. This shows that the wing should not be run at this angle as it is losing performance.

6.2. Wing Position

The second study done was the position of the wing relative to the rear axle. All the variables were done with the wing at its lowest setting (Setting 0) using the original wing and endplate design as a full car run had been done in this configuration which allows another comparison between the full car and isolated wing runs. These variables were kept consistent through the study to allow a direct comparison between the results.

A grid was created as seen in figure 28 using the regulation box from the Motorsport UK blue book. (Motorsport UK, 2019). The regulations for the 2020 sprint championship state that the rear wing can be no more than 90cm off the ground and the rear overhang can be no more than 150cm behind the rear axle as per rule 15.1.1. Therefore, to find the optimal position of the wing a grid was created with the wing being placed in 16 different locations equally spaced apart. The position x13 could not be replicated as this would be placed inside the components of the car but was selected as a point as it technically is inside the regulation box at the lowest position closest to the rear axle.

Consideration will have to be taken to the cars performance by altering the positioning of the wing. It will be expected that by moving the wing further rearwards more downforce can be created as the wing will be in more freestream air, however, with the wing being located further away from the CoG there will be a larger moment acting on the car for the same amount of downforce produced. If there is too much force acting on the rear of the car this cause more force to act on the rear tyres and less force to act on the rear tyres. This will cause the car to understeer more and even though the car is producing more downforce it might not be able to go faster around a track due to it being harder to drive. If the is moved further forward it is possible that it will improve the aerodynamic balance depending, but it is harder to create more downforce. The positioning will depend on the other aerodynamic devices on the car and where the current CoP is acting. Currently on the car the balance is acting slightly too far rearwards according to the drivers. Since there is currently a new front wing being produced that is expected to produce more downforce it is possible that wing might have to be moved further rearwards to maintain an acceptable CoP for the drivers.

The wing can also be moved up and down vertically. By moving the wing up, it will allow the wing to be in more freestream air, with would allow the wing to be run at a greater AoA as the air will be less disturbed and thus have more energy. However, since the wing will be further from the CoG a greater torque will be applied to the wing which will have a larger impact on the CoP of the car. The same problems will start to occur as with moving the wing too far rearwards where there will be too much force acting on the rear tyres causing the car to understeer. By moving the wing down, it will place the wing closer to the cars CoG. This means that more force can be placed on the wing without

dramatically effecting the aerodynamic balance of the car due to torque moments. Since the torque created by the rear wing is the force x distance, if the distance is reduced more force can be applied to create the same amount of torque. However, if the wing is placed lower down it will end up being in more turbulent air due to the body of the car and the other aerodynamic devices placed further forward on the car. This means that the wing won't be able to be run as efficiently so even though in theory more force can be applied to the wing while it is placed lower down it will be harder to generate these levels of downforce and its more likely in reality the wing will become less efficient when placed lower down.

Since the x and y velocity would be different at each of these points a model of the car was created without the rear wing and endplates as seen in figure 27.

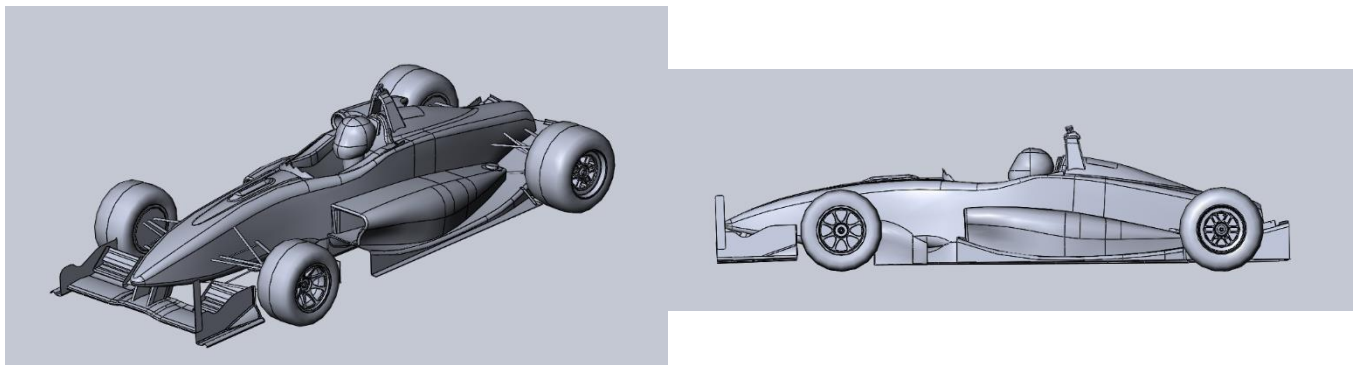


Figure 27 - Car without the rear wing

Lines were created in CFD post at each of the 16 points where the wing could be placed, these were then used to calculate the x and y velocity as these would be different to the original calculated velocities. This then acts as if the wing was in different positions saving the need to create a new CAD model for each position and attaching it to the car. The positioning of the wing can be seen in figure 28. These positions were taken from the leading edge of the main element which was originally located at X1.

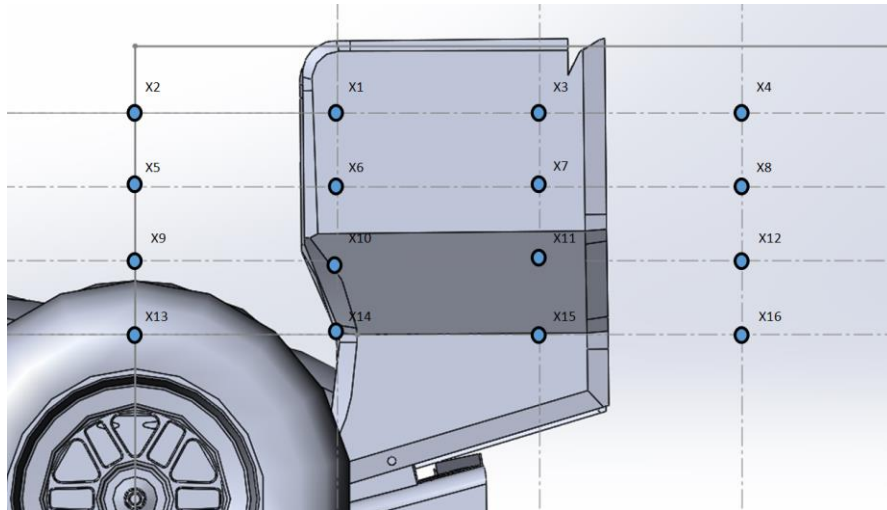


Figure 28 - Wing position placement

6.2.1. MATLAB Interpolation

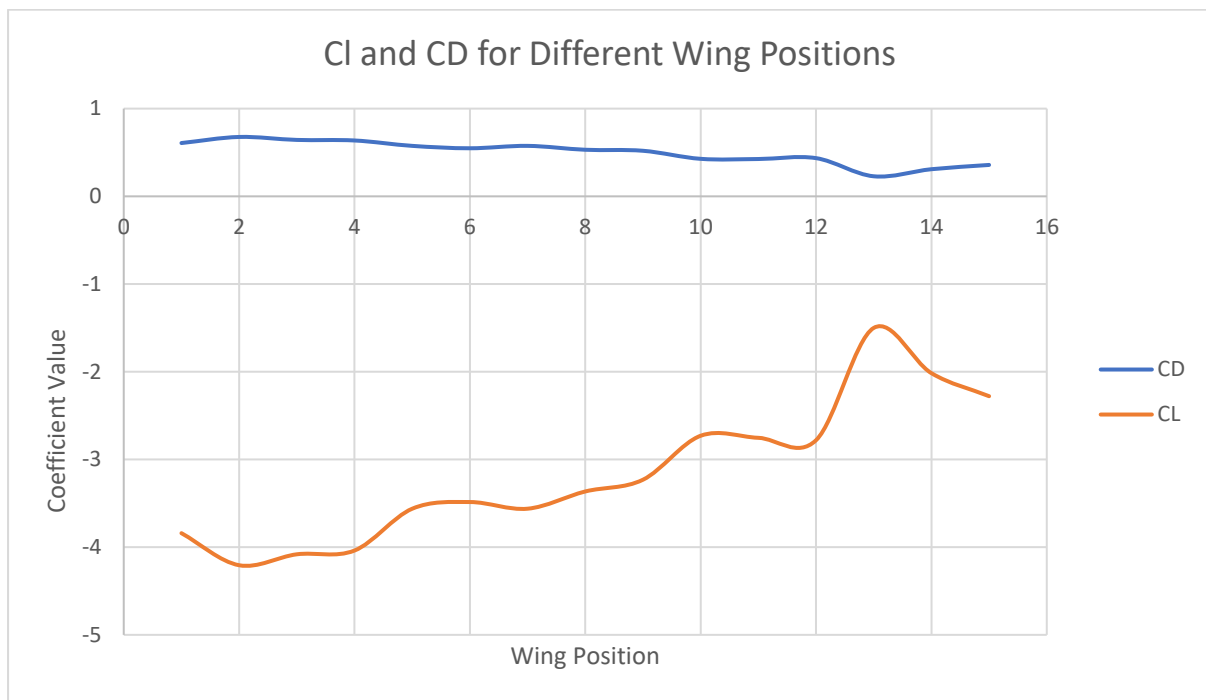
An experiment was done using this study to see if each run needed to be completed or if certain runs could be completed and use MATLAB to interpolate the results to predict the runs that were not completed. If this was successful it would mean only 65% of the runs would need to be completed allowing more runs to be completed before the end of the project. All of the runs have to be completed to allow a comparison between the actual results and the interpolated results. A matrix was created to decide which variables should be run and which should be interpolated, this can be seen in table 7 where all the highlighted variables were to be interpolated.

Table 7 - Wing position matrix

X2	X1	X3	X4
X5	X6	X7	X8
X9	X10	X11	X12
X13	X14	X15	X16

The results produced had mixed success, some of the results provided results within a 5% accuracy which would be within an acceptable range to use as the final result. However, there were some results that were very inaccurate with one value being 43% inaccurate. These results suggest that even though some of the results can produce accurate results there is the potential for large inaccuracies. Due to this MATLAB will not be used to interpolate for the remaining studies as the inaccuracy of the results is too large to justify the time saved by not running each variable individually.

6.2.2. Results



Graph 10 - Wing position results

When looking at the results it can be seen that the positioning on the wing can have a large impact on the C_L , it does have a minor impact on the C_D but it's not as dramatic as the C_L . The positioning of the wings can be seen in figure 28. One point that should be ignored is position 13 as this would not be physically possible to attach to the car, it was only tested to create a full matrix for use with the MATLAB interpolation. Wing position 2 where the wing is moved closer to the rear axle horizontally but is at the same position as the standard wing vertically produces the largest amount of downforce, however, it also produces more drag. This means its overall efficiency is not as good as the original wing. When looking at the greatest efficiency position 3 is the most efficient position. This is located 274 mm behind the standard wing position horizontally but is located on the same axis vertically. This position has a C_L which is 0.241 greater than the standard wing and a C_D that is 0.036 worse than the standard wing. Even though it does produce a greater efficiency by being located further rearward it ends up making the CoP less desirable.

By moving the wing, the endplates would have to be redesigned. This would have a few issues the first being cost, the current endplates would have to be having to be redesigned which would mean either paying someone to redesign it or keeping and modifying the current design. The new design would have to be manufactured whether it has been completely redesigned or not. With it being

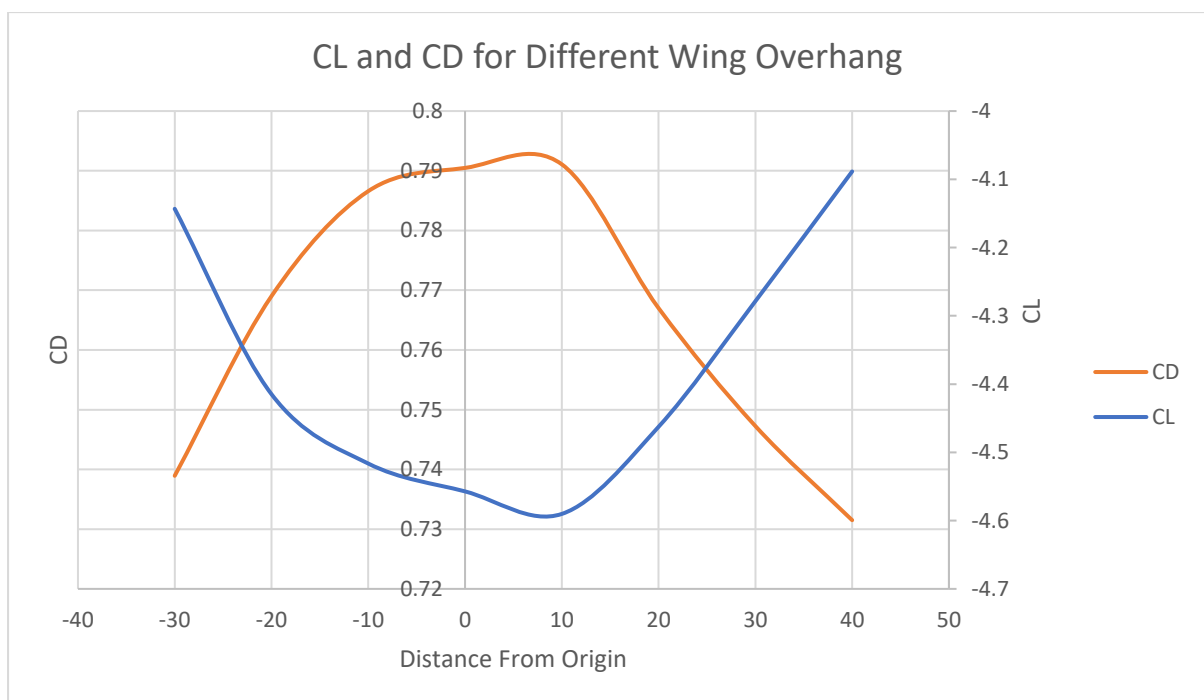
made out of carbon fibre which could end up being expensive as it would be a one off design therefore a new mould would have to be created. The second problem is the new design working. If a new design was created this design would be unproven and could potentially reduce the performance of the overall wing design. If the current design was used and modified then this design would also be unproven and since it has been designed to be that shape it possible won't work as efficiently as it previously did.

6.3. Wing Overhang

The overhang of the wing is how much the second element overlaps the main element. It is important to get the positioning of these two elements in the optimal position. If the second element is placed too far behind the main element it might not be able to drive the main element efficiently and could potentially just increase the drag without producing an extra downforce. If the second element is placed too far forward the main element can end up driving it instead of the other way around. This will reduce the overall efficiency of the wing.

For this study the second element was 50mm towards the front of the car in 10mm increments and 60mm towards the rear of the car in 10mm increments. The wing was run at setting at the angle setting of 0 and the position setting of X1. To start every other variable was run (e.g. -50mm, -30mm etc) to find the range as it would be expected that the extremes at each end would produce results that don't improve the performance of the wing. By running every other setting, the area where the optimal setting would be could be found and the different variables in this range could be tested. This was done to try and reduce the number of runs needed for this study which allows more runs to be completed in other areas.

6.3.1. Results



Graph 11 - Wing overhang results

When looking at the results in graph 11 the results by changing the second element overhang can be seen. When the value is negative the second element is moved further rearwards, so when the value is positive the second element is moved towards the front of the car.

It can be seen that the different overhangs can have a large impact on the CL and the cd of the wing. When more the second element overlaps the main element more than the baseline there is a reduction in both the CL and CD of the wing, this will be due to the second element not acting as efficiently as it previously was. As the overhang returned closer to the baseline the drag began to increase as the downforce began to increase. The maximum downforce is produced is when the second element is placed 10mm further rearwards, there is a 0.05 increase in cl. This does however come with increased drag. The greatest efficiency is produced at when the second element is placed 20mm further rearwards than on the standard wing. The CL is reduced by 0.09 and the cd is reduced by 0.02. This produces an efficiency that is increased by 0.05, even though this only a small increase this test was run at the lowest wing angle possible on the current wing design. This means the reduced downforce could be compensated by increasing the wing angle. Since currently on the car the maximum wing angle is never used this change to the overhang could result with a wing that has more variation with the wing angles that could be used for each track. As one of the requests from SBD was to reduce the drag induced by the rear wing, setting 8 would be a good option to change the wing as it both reduces the drag and increases the overall efficiency of the wing. As the second element keeps getting moved further rearwards both the CL and cd begin to decrease linearly. As this occurs the efficiency drops. This will be due to the second element being placed too far rearward to drive the main element efficiently.

6.3.2. Full Car Test

The rear wing with 20mm of extra overhang was placed on the full car model. This allowed a direct comparison to be made with the original model to ensure the accuracy of the wing isolation tests.

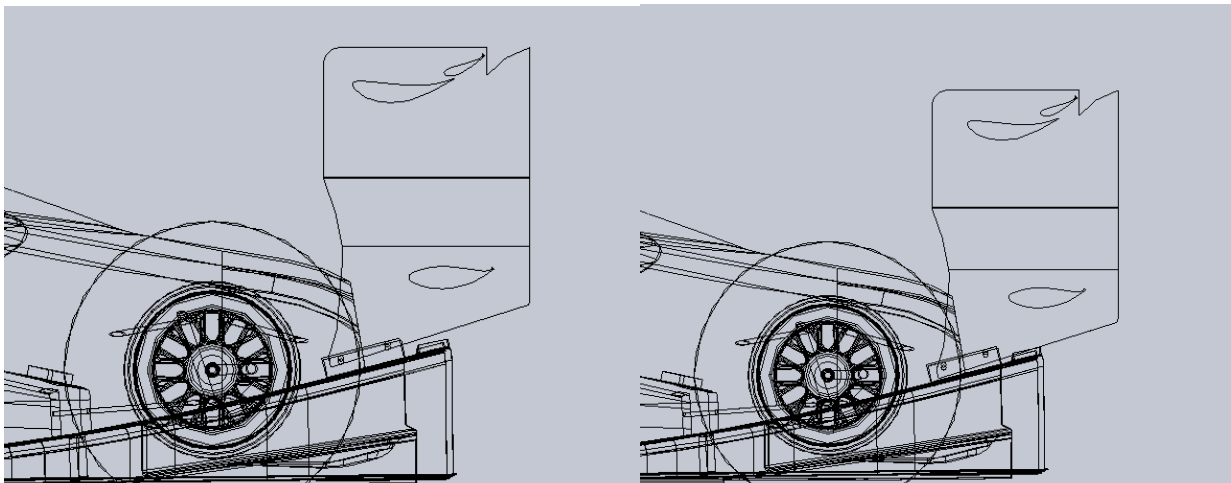
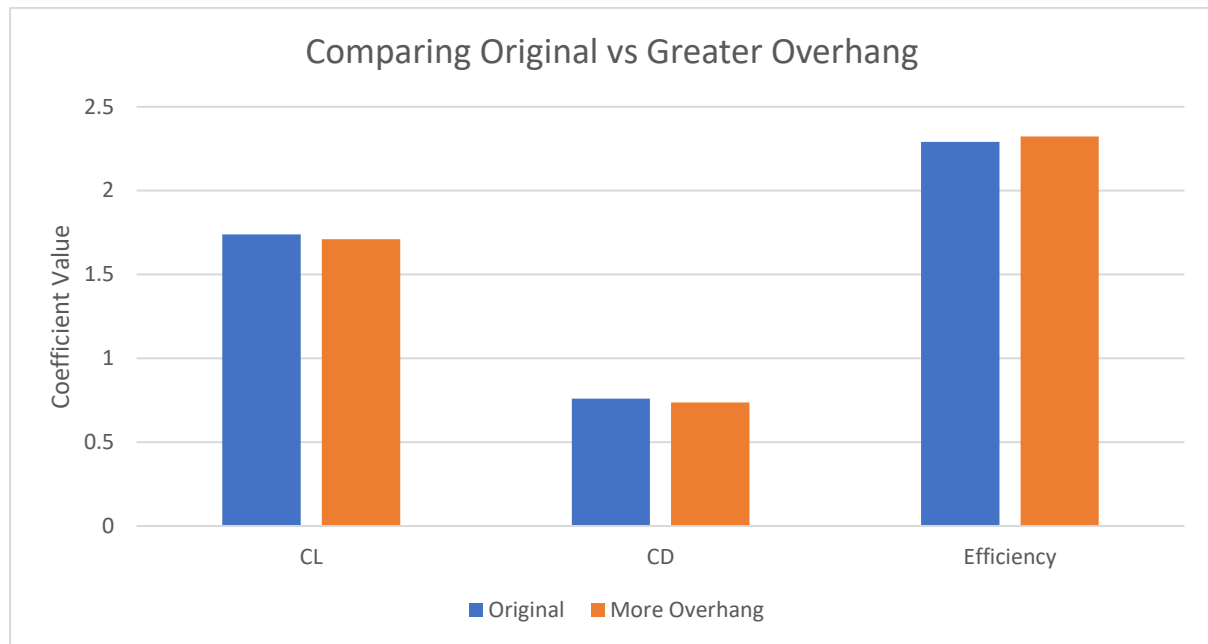


Figure 29 - Original wing vs modified overhang

Figure 29 shows the standard wing on the car (Left) and the wing with more overhang on the car (Right).



Graph 12 - Original vs modified overhang results

Graph 12 shows the comparison between the original and modified overhang when applied to the full car model. From this it can be seen that by changing the overhang the wing will produce less downforce but will also produce less drag. This increases the overall efficiency of the wing. Since these tests were run at the lowest AoA possible on the current wing this should produce a greater working range for the wing. These results again prove that the isolated wing tests is able to produce comparable results with full car tests.

Table 8 - Wing on the car vs isolation results

	CL	CD	Efficiency
Wing on Car	-2.07803	2.970347	0.911253
Wing in Isolation	-1.72414	3.119652	1.440452

Table 8 shows the percentage difference between the current wing overhang and the modified overhang for both the full car tests and the isolated wing tests. These results show that both results show a similar percentage difference with the difference in efficiency between the two test being only 0.37%.

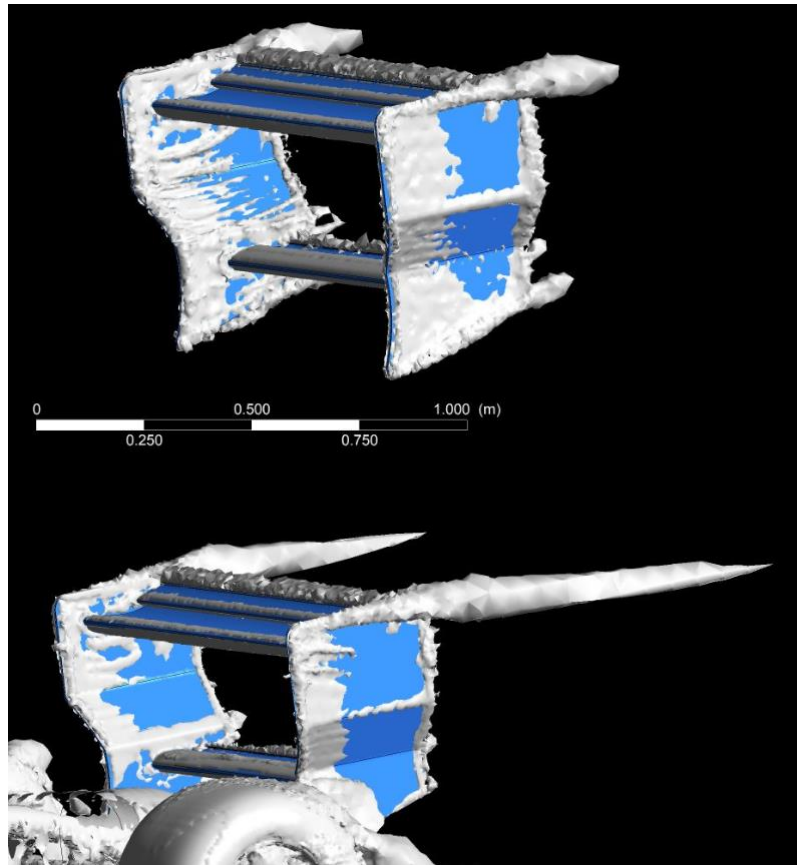


Figure 30 - Vortex core comparison between isolated wing and wing on the car for modified overhang

Figure X shows the vertices created for the modified overhang both in isolation (Top) and on the car (Bottom). It can be seen that the flow structures are similar especially the pressure build up on the leading edges of the main and second element and also the pressure build up on the gurney flap. There are some differences however including the vortex produced at the endplate tips which has more energy on the full car run. The pressure build up on the outside of the endplate is greater with the isolated run.

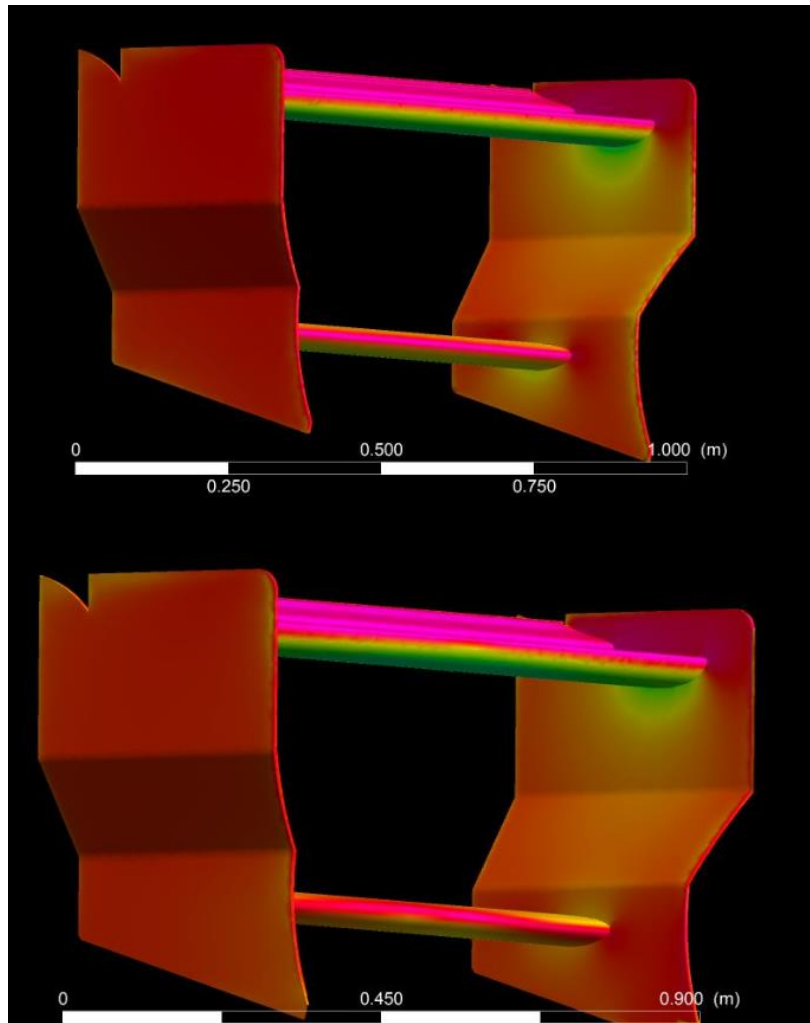


Figure 31 - Pressure contour comparison between isolated wing and wing on the car for modified overhang

Figure X shows a pressure contour for the wing in isolation (Top) and the wing on the car (Bottom). As with the vortices there are some similarities especially with the pressure distribution across the wings. Although there are some differences, the pressure at the endplate tips are slightly lower for the full car model. There is also a greater pressure difference on the low pressure side of the main element against the endplate on the isolated model. These results show that while the isolated tests are acceptable for gathering comparable results a full car model would still need to be run to ensure accurate data is being acquired.

Although the overall efficiency of the wing increased by modifying the overhang it moves the CoP of the overall car from being a 39/61 split to a 34/66 split. It is commonly stated that a good starting point for the CoP of a rear wheel drive car is 40/60.(Newey, 2017). By changing the overhang and therefore moving the CoP further rearwards this could have some negative effects on the cars handling as there will be less force on the front wheels causing more understeer.

6.4. Wing Gap

The wing gap is the gap between the main and second element vertically. As with the overhang the distance between the two elements is very important. If the two elements are too close together it will create a blockage effect (Choi and Kwon, 1998) where the air effectively becomes trapped in the gap between the two elements. This trapped air forms a wall and therefore no air can pass through this gap, this dramatically reduces the effect of the second element. Therefore, there was be a large increase in drag as the frontal area will be increased since the blocked air is acting as a wall, also the downforce will be decreased as the second element won't work as efficiently as it should. If the gap between the two elements is too large there will be very little benefit to having the second element as the venturi effect created by having multiple elements will become less effective.

This venturi effect occurs when a fluid flows through a constrained section. As air at low velocities is an incompressible fluid when it moves through a constricted section the velocity must increase due to Bernoulli's principle (Darrigol and Frisch, 2008), since the velocity has increased the kinetic energy of the airflow at this point will also be increased. This will allow the airflow to remain attached when it moves from the main element to the second element.

For this test the gap between the elements was decreased by 10mm in 5mm increments, the gap was also raised to 15mm again in 5mm increments. The gap couldn't be reduced by any more than 10mm as this would leave no gap between the two elements.

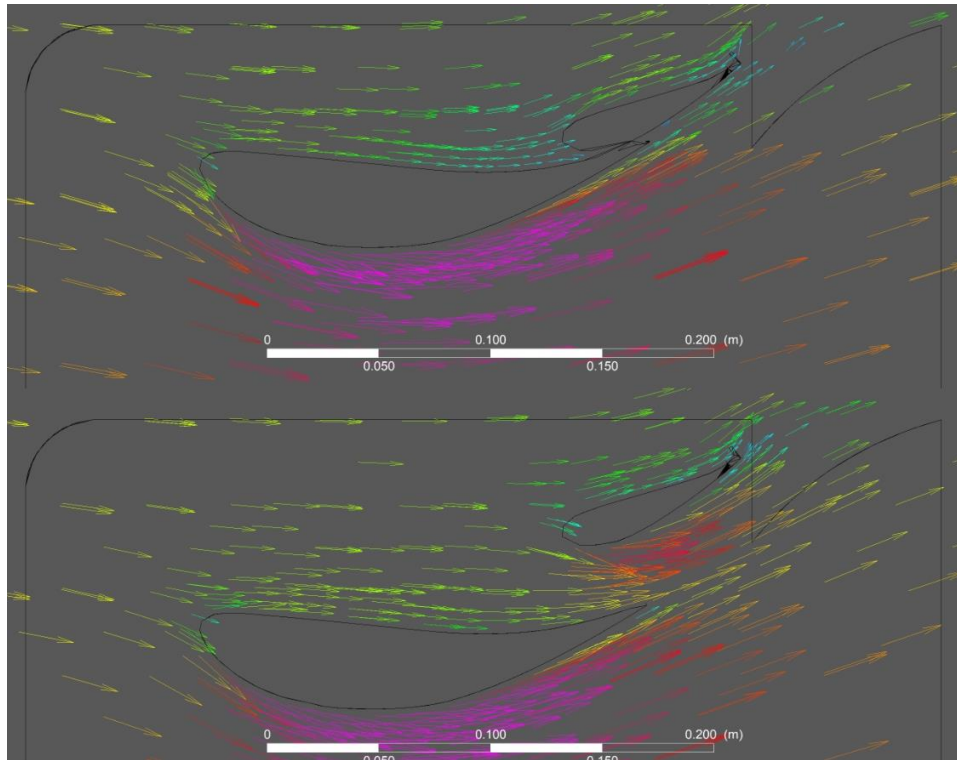
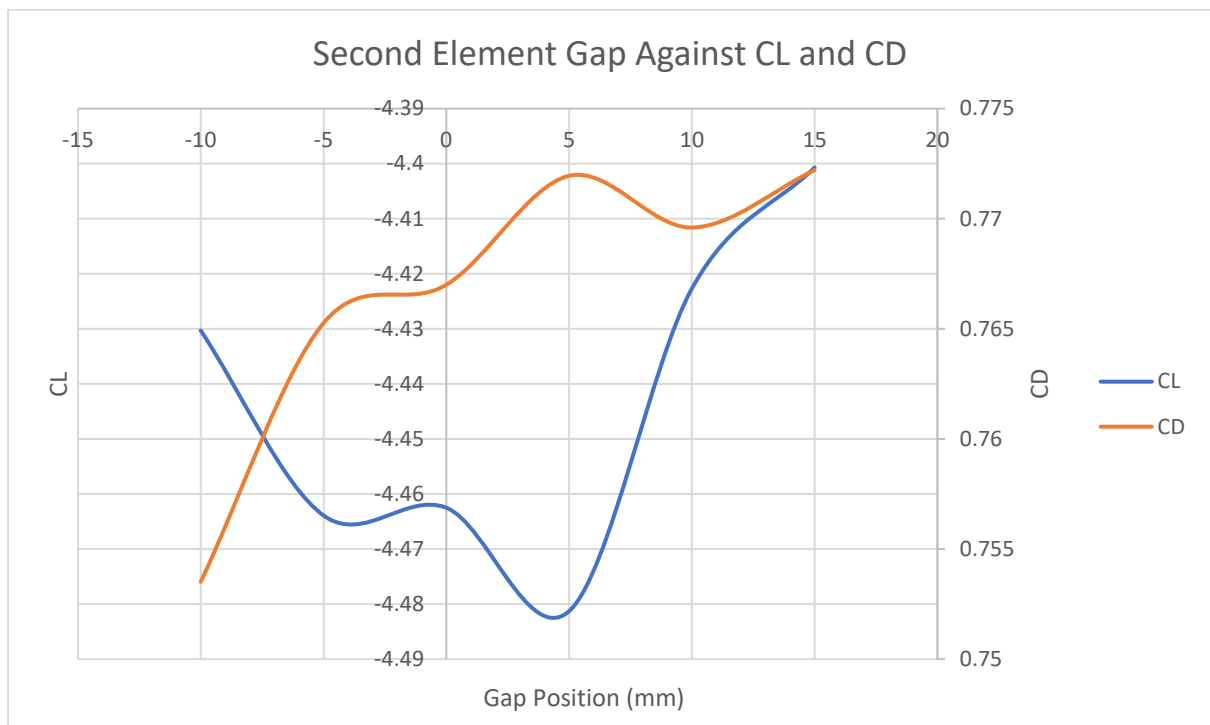


Figure 32 - Vectors for wing gap comparison

Figure 32 shows the difference in the flow structure for the minimum gap (Top) against the maximum gap (Bottom) tested. When the gap is smallest the gap is too small to allow any air to flow through the gap, therefore the second element is not being taken advantage of. The flow remains attached from the main element to the second element, but it would be expected that when the AoA is increased the flow will start to separate quickly and the wing will start to lose performance.

When the gap is at maximum some flow attaches from the main element to the second element although it is not being done in the most efficient way. The venturi effect is not being optimised fully which explains why this gap was not producing the most efficient values.

6.4.1. Results



Graph 13 - Second element gap results

When analysing the results, it can be seen that the lowest cd is produced when the second element is lowered by 10mm, this is also where the greatest efficiency is.

6.5. Twisted Wing

Twisted wings originally derive from aeroplanes where the twist was added to plane wings to stop the outer edge of the wing stalling before the rest of the wing.(Hunsaker, Montgomery and Joo, 2020). The theory behind wing twist on a plane is as the plane pitches up the AoA of the wing increases, the airflow over the outside edge of the wing can separate before the other areas of the wing which would cause a decrease in lift. By twisting the wing, it all the air to stay attached across the whole wing. This allows the pilot to maintain full control of the aircraft especially in the event of a stall.(Anderson, 2009).



Figure 33 - Corvette C7.R twisted wing(Racecar Engineering, 2015)

A twisted wing is used for a slightly different reason on a car however the theory is still the same. The wing is still twisted to maintain attachment across the whole aerofoil however this is done as the main body of the car is placed in front of the wing, therefore the edges of the wing can have a larger AoA than the centre of the wing as it is located in the freestream air. This is if the wing is being designed as a high downforce wing. Since cars don't pitch to the same extremes as a plane the wing is designed to keep flow attached at 0° of pitch. A twisted wing is designed to create high levels of downforce with low induced drag.(Marqués-Bruna, 2013)

To create the twisted wing the angle of each section of the wing needed to be calculated. To do this a full car run was used with the wing angle at its minimum setting. A line was placed in front of the leading edge of the main element to find the X and Y velocity acting at this point. These could then be used to calculate the angle at each section along the wing to find the theoretical optimal amount of twist. From this line 10 points were measured. Using the X and Y velocity of the air at each of these points, trigonometry was used to calculate the theoretical ideal wing twist. A right angle

triangle was created with the X velocity being the opposite side, the Y velocity being the adjacent side and the angle being unknown. By using equation 6.

Equation 6 - Angle of twist

$$\text{Angle of twist needed} = \tan^{-1}\left(\frac{X \text{ Velocity}}{Y \text{ Velocity}}\right)$$

The angle of twist needed can be found and the table used to find the angle can be seen in table 9.

Table 9 - Angle of twist

Distance (m)	-0.57	-0.44	-0.32	-0.19	-0.06	0.06	0.19	0.32	0.44	0.57
Angle (°)	19.62	17.88	19.59	22.49	30.36	29.14	23.79	20.41	21.01	26.03

When looking at the angles it can be seen that that angle of twist needed is not symmetrical. This is due to the airbox interfering with the airflow and therefore not allowing the AoA to be increased as the air has less energy and thus won't remain attached at a higher AoA. This is why multiple points had to be taken across the entire span of the wing as the car is not symmetrical different points equal distance away from the centre will have different resultant velocities and thus will require different angles of twist.

After finding these angles they could be used to create the twisted wing design in CAD. To do this the wing profile was copied using the convert entities feature to ensure both the profiles of the main and second element remained the same. 10 planes equal distances apart to represent each point measured in CFD post. The wing profile was copied onto each of the planes with a point on the

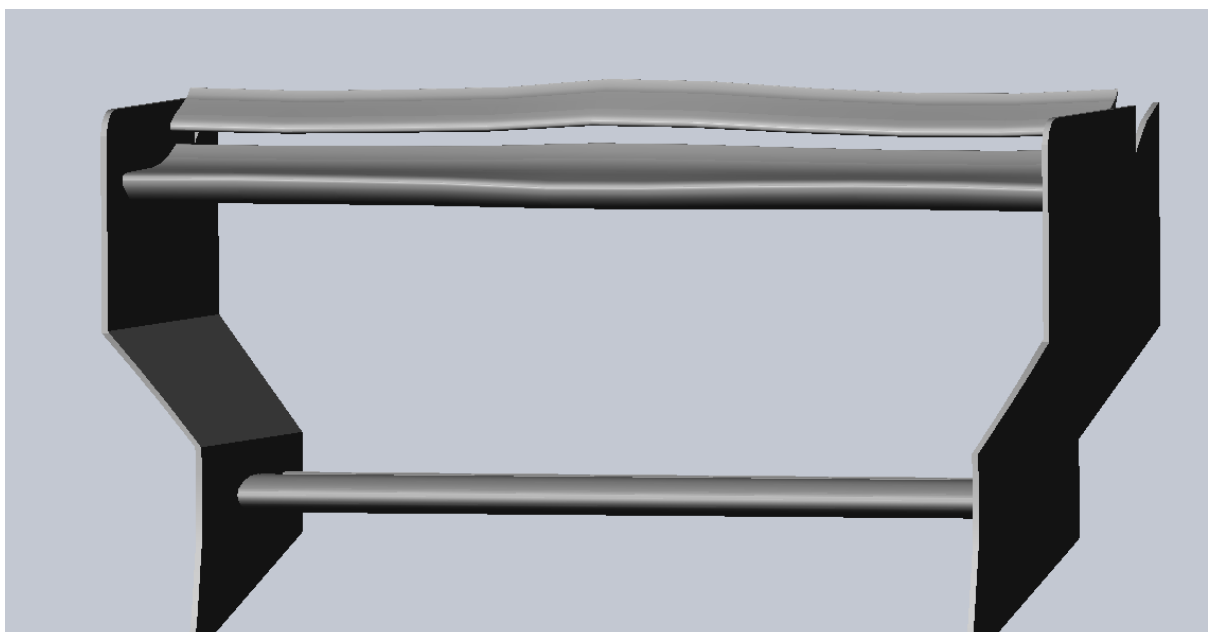
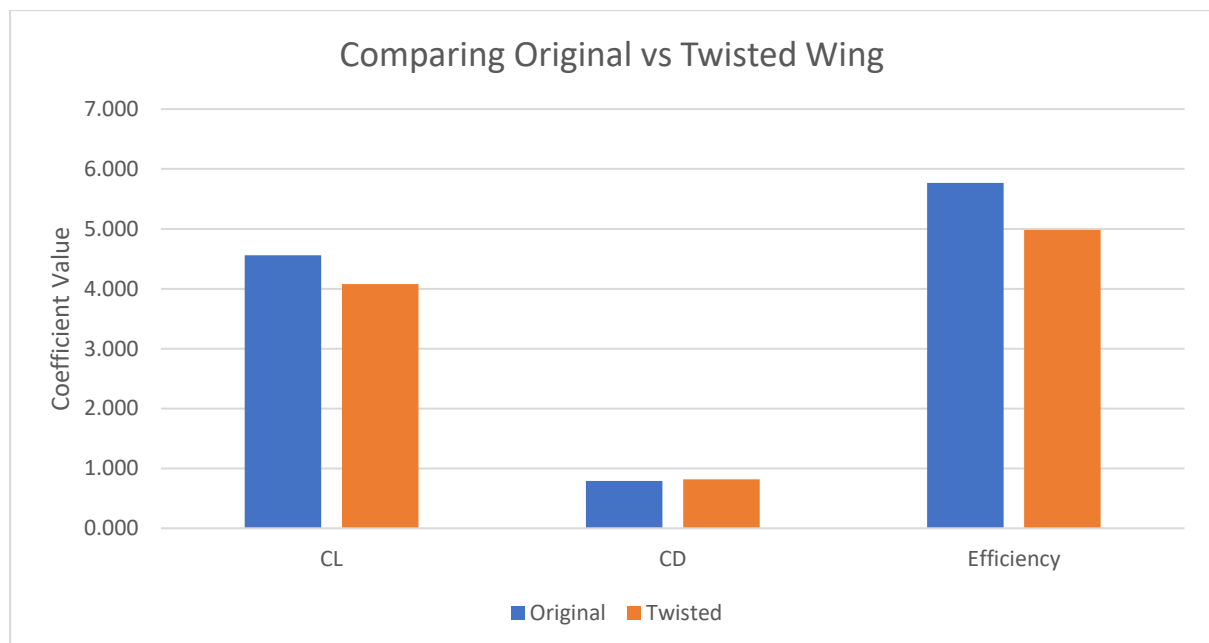


Figure 34 - Twisted wing design

leading edge of the main plane which will be used to rotate the profile to get the twist in the wing profile. Each section was twisted to the correct angle and a loft was created between each sketch to create the twisted wing profile. The second element was also twisted to the same angle as the main element.

Once the new profile had been created it was initially tested in isolation before being tested on the full car. This was done for two reasons, the first was to see how it compared to the standard wing profiles while in isolation and the second was to ensure that the profile has been correctly made in CAD and would run in CFD. To run the wing in isolation a resultant velocity was again used to try to replicate the air flow over the car. The wing was also run in the standard position with the minimum level of AoA. This allowed a direct comparison between the previous tests.



Graph 14 - Original vs twisted wing results

After looking at the results from graph 14 it can be seen that the twisted wing while run in isolation produced results with a lower CL and a higher cd. Therefore, the efficiency produced by this wing is lower than the standard wing. These results could explain why it is uncommon to see twisted wings on formula cars, the reason for them being common on GT cars is because of their sloping back and their less complicated aerodynamics which disturbs the air less. With GT cars the wings are twisted at the end more than the middle because they are in free stream air at the ends, while the air flow closer to the centre has less energy so the angle of twist is less.

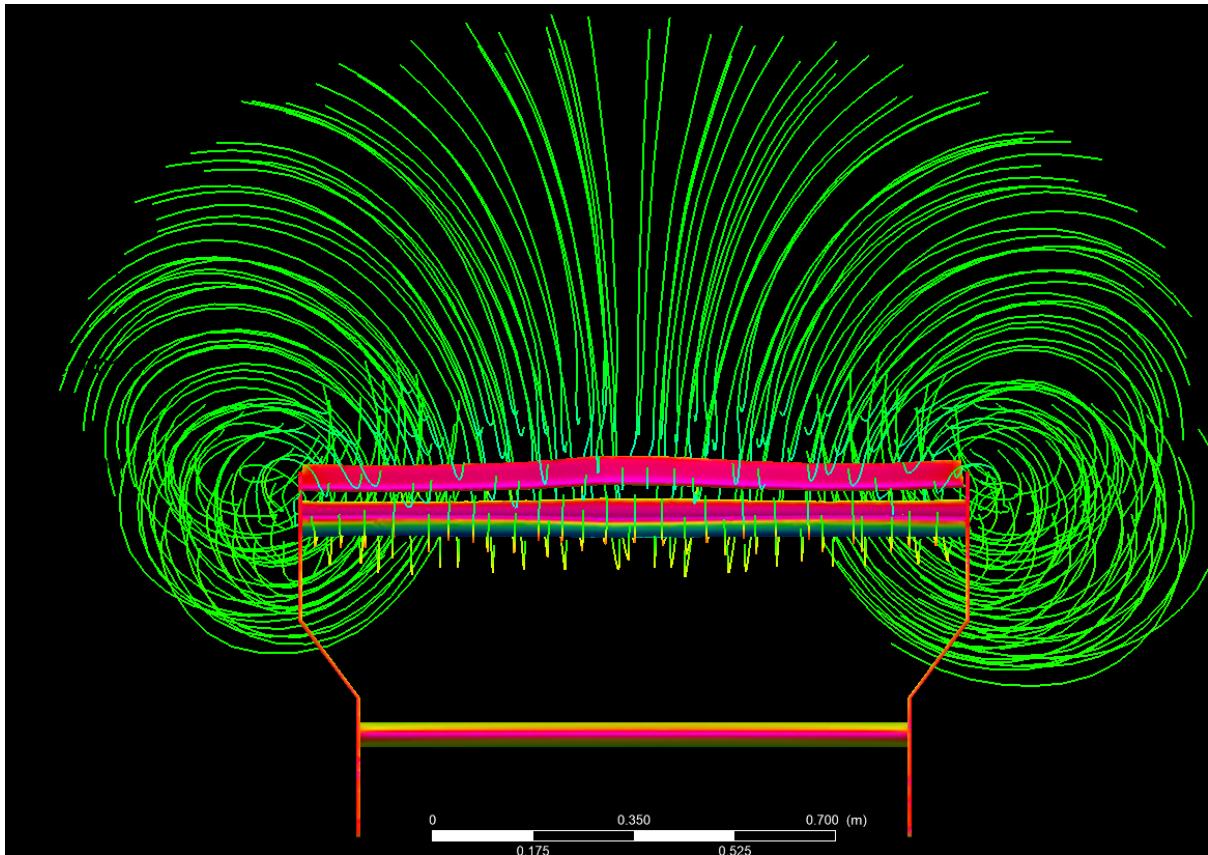


Figure 35 - Twisted wing flow structure

Figure 35 shows that most of the flow is staying attached to the wing. However, when looking at the pressure contours of the two elements the second element is not working as efficiently as it was with the original design. With some optimisation of the second element could be redesigned to improve this design further. Since this design would be costly to manufacture and would require further work to optimise full it has not been chosen to develop further although, it could be investigated in the future.

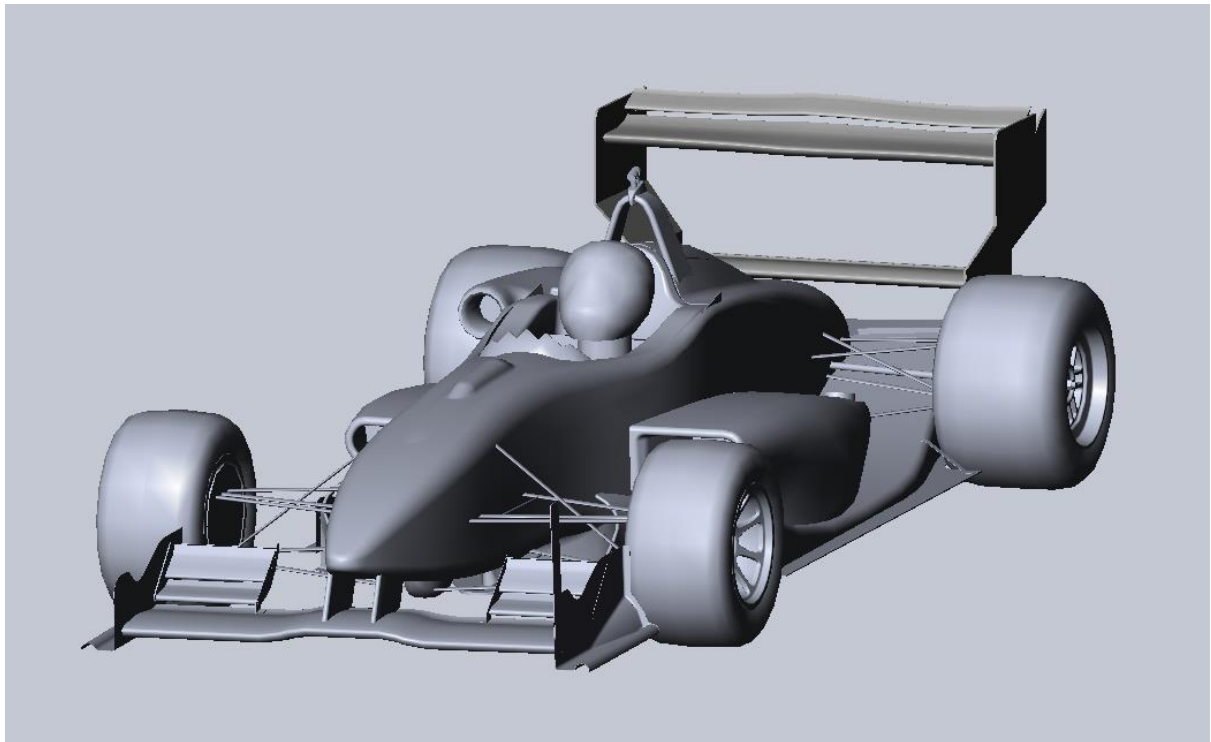


Figure 36 - Twisted wing on full car model

When the twisted wing is used on the full car model the CoP moves slightly forward. It moves from being a 39/61 split to be a 40/60 split since the CoP moves 4.2mm forwards. Since this change is so marginal it is unlikely it would have a major impact on performance.

Another variation of a twisted wing is known as a 'spoon wing' which was introduced by Mercedes F1 in 2016. While the theory behind the two wings is similar their reasoning for implementation is different. The spoon wing is designed to reduce drag and increase straight line speed compared with the standard wing design. The spoon design reduces the overall induced drag of the wing by decreasing the pressure differential towards the edges of the wing, this reduces the vortices created at the edge of the wing which reduces the induced drag. This design also reduces the overall chord of the wing, this increases the aspect ratio and a larger aspect ratio means a lower induced drag.(Motorsport Technology, 2018; Smedley, 2019)(Formula1, 2018). An example of the spoon wing can be seen in figure 37.

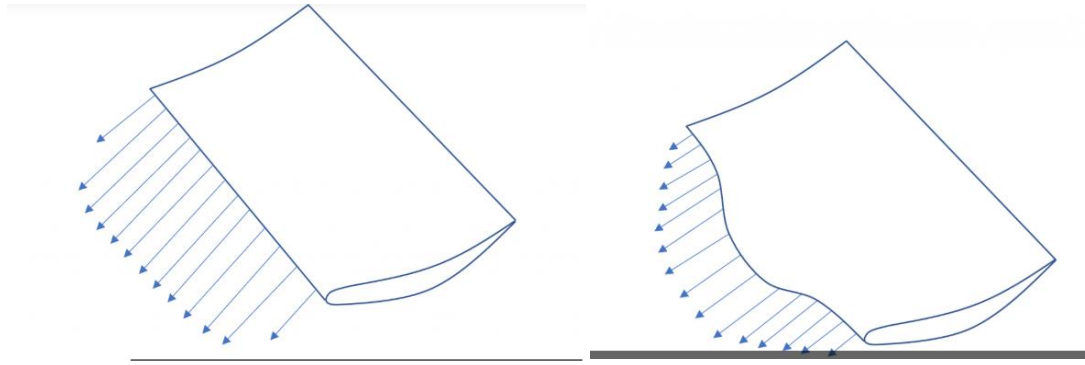


Figure 37 - Spoon wing design (Motorsport Technology, 2018)

The reason for this style of wing not being considered for this project is due to the top speed seen in the sprint championship being half that seen in F1 where these wings are used. In the future this type of wing could be investigated to see its impact with a sprint car, but since there is little scientific research behind this style of wing it was deemed better to spend the limited amount of time investigating other alternatives.

7. Extended Diffuser

During this project SBD wanted to know if there would any benefit to extending the diffuser. A diffuser works by accelerating the airflow under the body of the car, with the F3 car a venturi tunnel is used to do this. This accelerated air is then slowly and smoothly returned to ambient pressure.(Katz, 1995a; Mohankumar, Hudson and Daniel, 2015). The theory being extending the diffuser is that it takes the air longer to longer to return to ambient this creates an adverse pressure gradient under the car which sucks it to the ground producing downforce. A diffuser has a continuous curve from the kick point to the trailing edge, this curve means that the air is constantly changing direction which therefore accelerates the air keeping it attached to the diffuser. If the angle of the diffuser becomes too large the air will start to separate due to the adverse pressure gradient.(Cooper *et al.*, 1998)

The current diffuser design can be seen in figure 39.

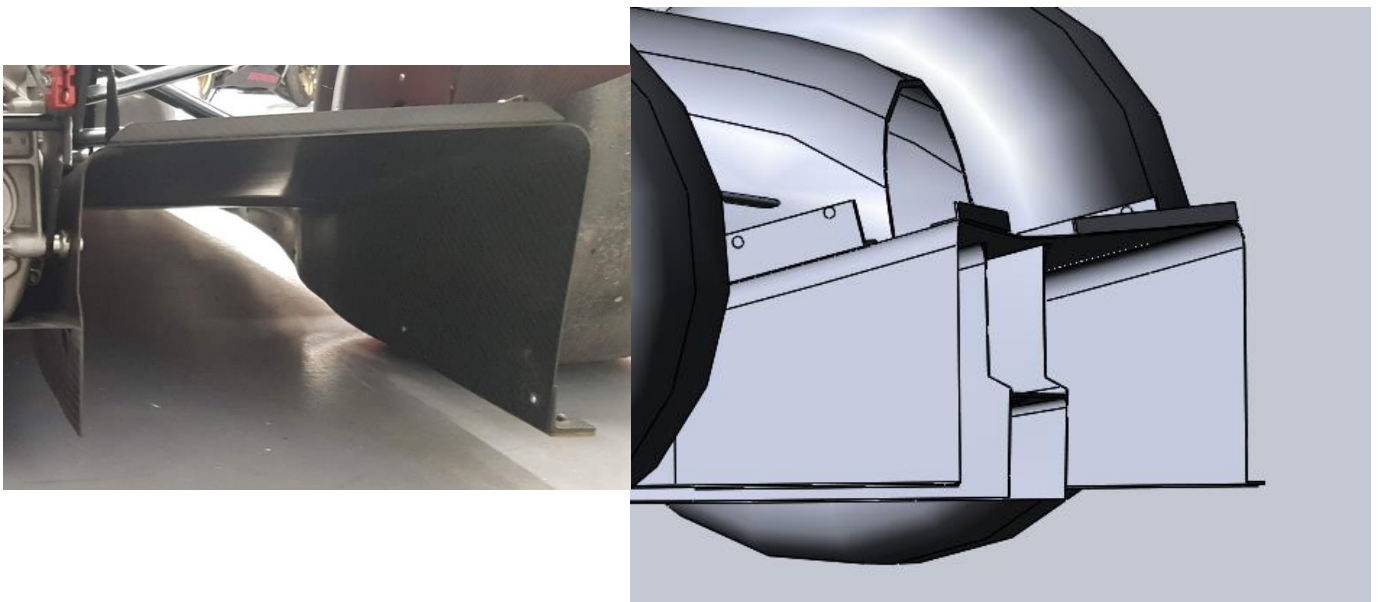


Figure 38 - Original diffuser both in real life and in CAD

To extend the diffuser the current design was copied, a plane was created 125mm further rearwards. The new design continues to flare outwards to try and keep the airflow attached the whole way through the diffuser. The new design can be seen in figure 40. The highlighted section shows where the diffuser has been extended.

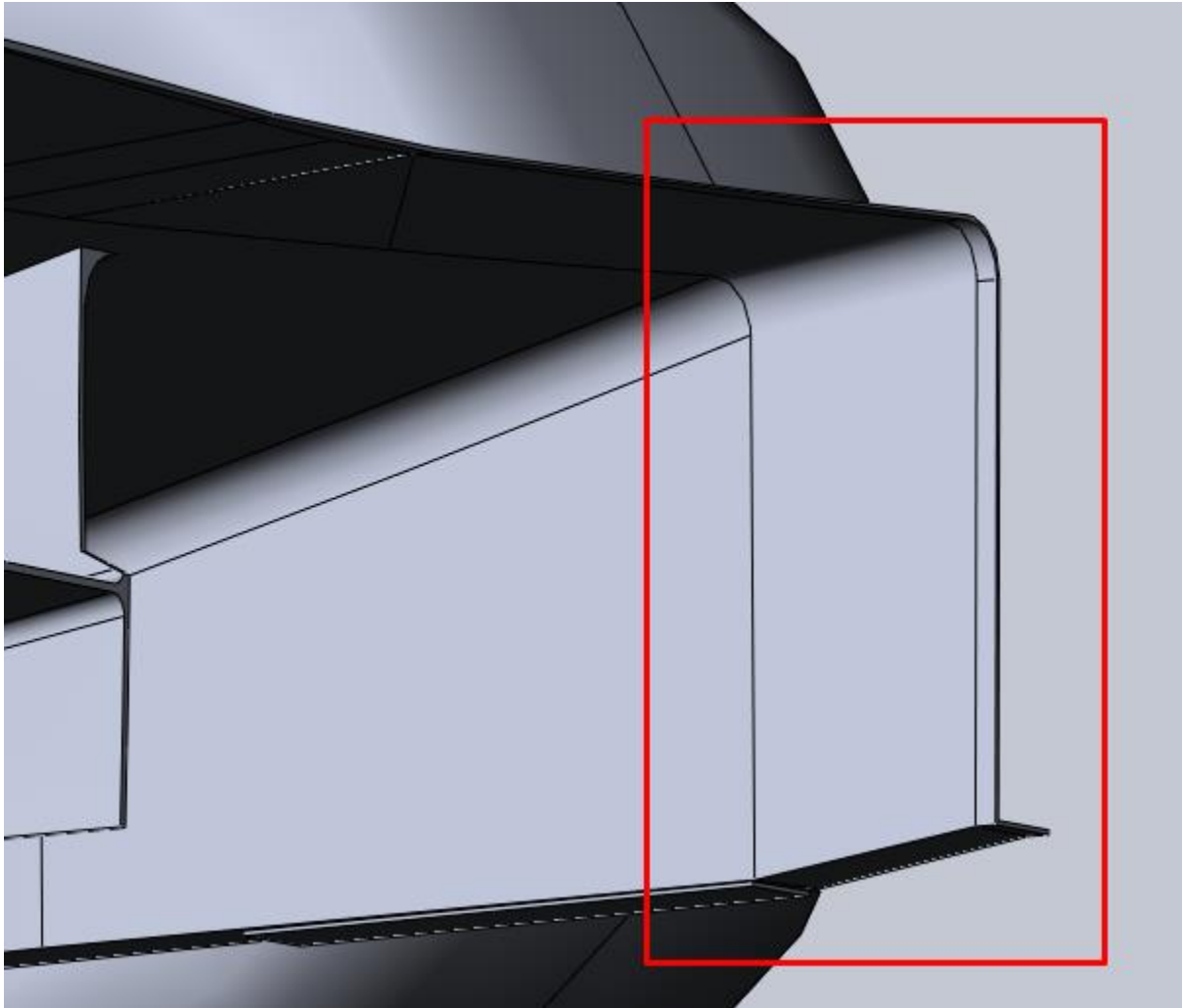


Figure 39 - Modified diffuser design

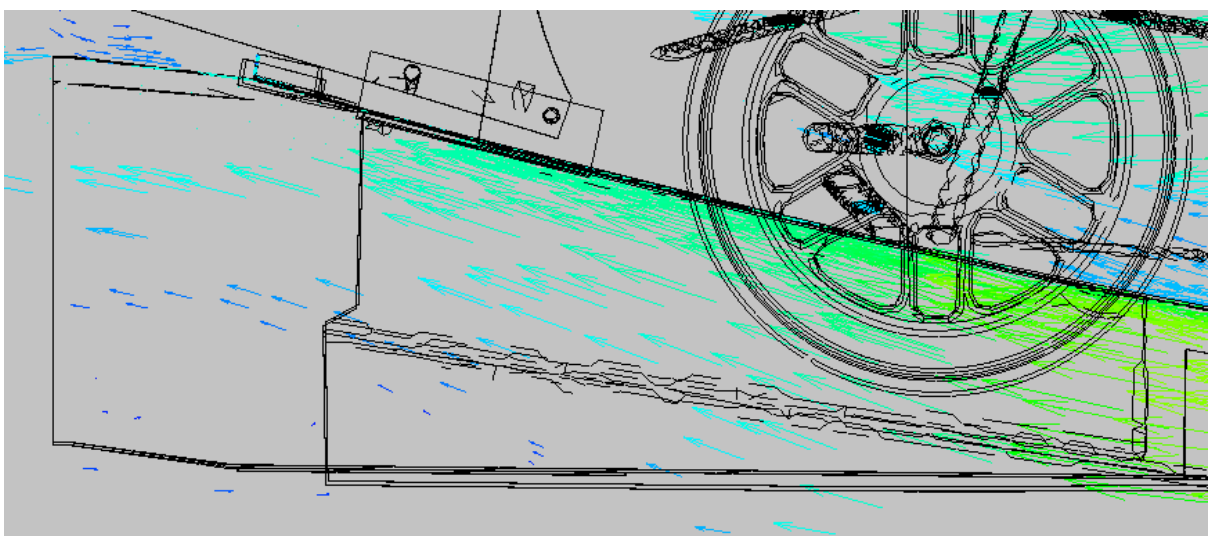
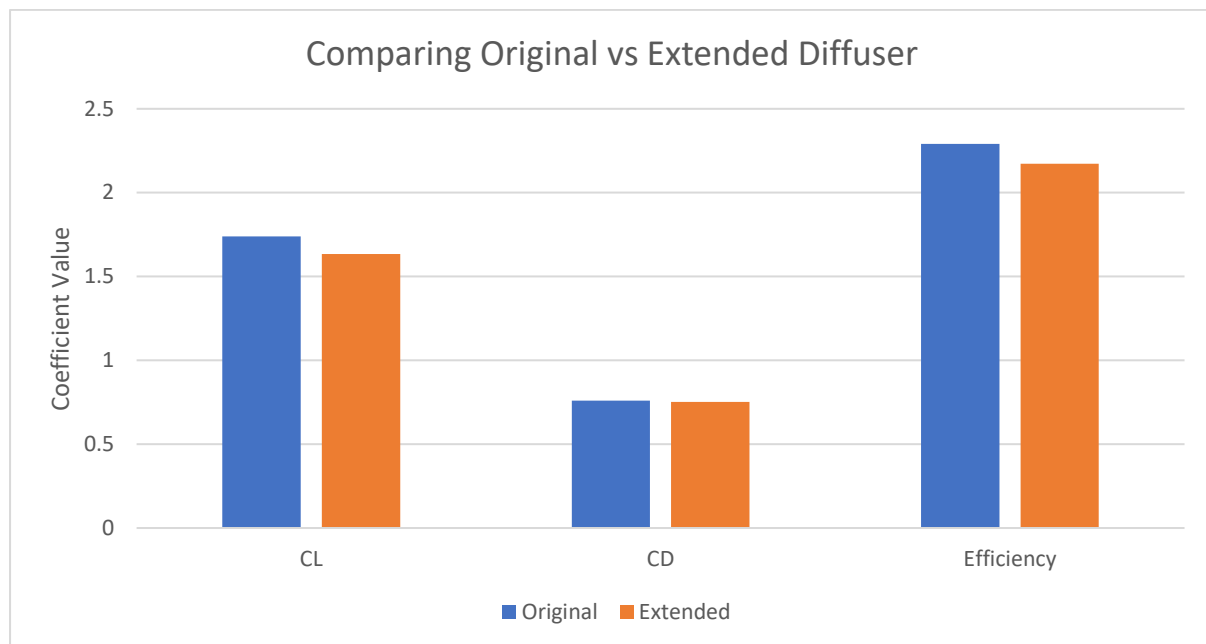


Figure 40 - Vectors for extended diffuser

Figure 41 shows that at the point where the original diffuser ends the flow begins to separate. This would explain why there is decreased performance with this design



Graph 15 - Original vs extended diffuser results

By looking at graph 15 it can be seen that the original diffuser produces a greater CL value with a similar CD value. This means that by extending the diffuser this way less downforce would be produced, and the overall efficiency of the vehicle is reduced.

It is possible that if more time was spent on developing this extended diffuser design then it could produce more downforce and improve the overall aerodynamic efficiency of the car. Although the extended diffuser did move the CoP further rearwards causing the aerodynamic balance to become a 36/64 split which as previously discussed is less desirable. Since this was not part of the project and was just a small investigation for SBD it was not investigated further.

8. Conclusion

From this report it can be seen that there are many factors that can affect the performance of the rear wing. Through all the initial CFD testing the importance in mesh strategy and turbulence model was found, this showed how by using the wrong settings how the results would be impacted. If these incorrect results were used and a physical prototype was created both time and money could be wasted as the prototype might not perform as expected from the CFD results.

The use of real world testing was also explored to show how real world data could be used to correlate CFD results. By comparing the CFD results to those shown in the real world tests it can be seen how there are many forms of real world testing that can be used for validation. It also showed the importance of using these tests if possible to validate any settings used.

By looking at the difference in results for the different full car CAD models the need for an accurate CAD model can be seen. If the geometry being used is inaccurate it is impossible to get any reliable CFD results from it. This means any results gathered from the inaccurate geometry are essentially useless so having a good baseline to start with is key to producing any aerodynamic improvements.

By running the wing in isolation more results were able to be produced compared with full car models in the same time. This allowed for more testing to be completed. The impact of changing where the wing is positioned and how the two elements are set up with each other determines the performance of the wing. If any of the tests were positioned incorrectly this could lead to decreased performance.

9. The Impact of Covid-19 on This Project

9.1. Real World Validation

A test was planned at Llandow Circuit to be used as both a shakedown for the car before the season starts but also for aerodynamic testing and validation. The use of both wool tufts and flow vis was planned as this would allow the flow structures being produced by the car to be visualised. This could be used to compare against the flow structure seen in the CFD results, if the structure seen are comparable then it would add more credibility to the CFD results.

A straight line test was also going to take place during this test. This would allow a CD value to be calculated using the data logger. This could also be used to compare against the CFD results.

9.2. 3D Printed Wing

A 3D printed model of the wing design was going to be created. This were going to be a scale model that could be placed into the Wind tunnel. This could be used to get an idea of the flow structures produced by the wing in isolation by using string and possibly smoke pellets. It could also be used to get the CL and cd produced by the wings. These tests could be used to compare against the CFD results of the wings in isolation. These wings would also be used as a visual aid for the viva to show the assessors the progress from the initial wing design to the final wing design.

9.3. CFD Testing

Since students don't have access to the full Ansys software package at home some of the CFD testing was compromised. This put a stop to any further development of the wing design as they could no longer be tested. Even though a design could be created as Solidworks was available outside of the university the new design couldn't be tested and therefore it was not seen as the best use of time to develop any designs further.

10. References

- Abbott, I. H. A. and Von Doenhoff, A. E. (1997) *Theory of wing sections, including a summary of airfoil data*.
- Adariani, M. R. (2019) *How can I understand mesh quality and simulation results in CFD?* Available at: https://www.researchgate.net/post/How_can_I_understand_mesh_quality_and_simulation_results_in_CFD (Accessed: 20 March 2020).
- Anderson, J. D. (2009) *Fundamentals of Aerodynamics (Mcgraw Hill Series in Aeronautical and Aerospace Engineering)*.
- Ansys (2013) *ICEM User Manual*.
- ANSYS (2013) *ANSYS FLUENT 12.0 Theory Guide - 4.7.1 Overview*. Available at: <https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node73.htm> (Accessed: 19 January 2020).
- ANSYS FLUENT (no date) *ANSYS FLUENT 12.0 Theory Guide - 4.4.3 Realizable - Model*. Available at: <http://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node60.htm> (Accessed: 17 January 2020).
- Bak, J.-G., Hwang, W. and Cho, J. (2017) 'Geometry Defeaturing Effects in CFD Model-Based Assessment of an Open-Channel-Type UV Wastewater Disinfection System', *Water*. MDPI AG, 9(9), p. 641. doi: 10.3390/w9090641.
- Bakker, A. (2006) 'Lecture 7-Meshing Applied Computational Fluid Dynamics Instructor: André Bakker'. Available at: <http://www.bakker.org> (Accessed: 28 April 2020).
- Barnard, R. . (2001) 'Road Vehicle Aerodynamic Design SECOND EDITION'.
- Batchelor, G. K. (George K. (1999) *An introduction to fluid dynamics*. 2nd edn. Cambridge University Press (Cambridge Mathematical Library).
- Bertin, J. J. and Cummings, R. M. (Russell M. (2009) *Aerodynamics for engineers*. Pearson.
- Bradford, J., Montomoli, F. and D'Ammaro, A. (2014) 'Uncertainty quantification and race car aerodynamics', *Proceedings of the Institution of Mechanical Engineers, Part D: Journal of Automobile Engineering*, 228(4), pp. 403–411. doi: 10.1177/0954407013514396.
- Calinger, R. (1996) 'Leonhard Euler: The first St. Petersburg years (1727-1741)', *Historia*

Mathematica. Academic Press Inc., 23(2), pp. 121–166. doi: 10.1006/hmat.1996.0015.

Carnarius, A. *et al.* (2009) 'Numerical study of the controlled flow around a tandem Ahmed body configuration', in *47th AIAA Aerospace Sciences Meeting including the New Horizons Forum and Aerospace Exposition*. doi: 10.2514/6.2009-1079.

Cebeci, T. (2006) *Computational fluid dynamics for engineers: from panel to Navier-Stokes methods with computer programs*, *Choice Reviews Online*. Horizons Pub. doi: 10.5860/choice.43-3431.

Champagne-Ardenne, U. of R. (2008) *Generating Prisms*.

Choi, C. K. and Kwon, D. K. (1998) 'Wind tunnel blockage effects on aerodynamic behavior of bluff body', *Wind and Structures, An International Journal*. Techno Press, 1(4), pp. 351–364. doi: 10.12989/was.1998.1.4.351.

Cooper, K. R. *et al.* (1998) 'The aerodynamic performance of automotive underbody diffusers', in *SAE Technical Papers*. SAE International. doi: 10.4271/980030.

Darrigol, O. and Frisch, U. (2008) 'From Newton's mechanics to Euler's equations', *Physica D: Nonlinear Phenomena*. Elsevier, 237(14–17), pp. 1855–1869. doi: 10.1016/j.physd.2007.08.003.

Davis, M. C. and Saltzman, J. A. (2000) *In-Flight Wing Pressure Distributions for the NASA F/A-18A High Alpha Research Vehicle*. Available at: <http://www.sti.nasa.gov> (Accessed: 27 April 2020).

Diasinos, S., Barber, T. J. and Doig, G. (2017) 'The Influence of Wing Span and Angle of Attack on Racing Car Wing/Wheel Interaction Aerodynamics', *Journal of Fluids Engineering*, 139(6). doi: 10.1115/1.4035877.

Dixon, A. G. *et al.* (2008) 'Wall-to-particle heat transfer in steam reformer tubes: CFD comparison of catalyst particles', *Chemical Engineering Science*, 63(8), pp. 2219–2224. doi: 10.1016/j.ces.2008.01.017.

Favier, D. (2010) 'The Role of Wind Tunnel Experiments in CFD Validation', in *Encyclopedia of Aerospace Engineering*. Chichester, UK: John Wiley & Sons, Ltd. doi: 10.1002/9780470686652.eae034.

Formula1 (2018) *TECH INSIGHT: How Ferrari, Mercedes and Red Bull solved the Baku wing conundrum*. Available at: <https://www.formula1.com/en/latest/article.tech-insight-how-ferrari-mercedes-and-red-bull-solved-the-baku.5D9rBoJK7u4e4lq0o4wgUA.html> (Accessed: 25 April 2020).

Houghton, E. L. and Carpenter, P. W. (2003) *Aerodynamics for {E}ngineerign {S}tudents*. 5th edn. Butterworth Heinemann.

Hughes, J. (2019) *Aerodynamic Development & Optimisation of an MCR Open Sports Series Front Wing*.

Hunsaker, D. F., Montgomery, Z. S. and Joo, J. J. (2020) 'Analytic and computational analysis of wing twist to minimize induced drag during roll', *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*. SAGE Publications Ltd, 234(3), pp. 788–803. doi: 10.1177/0954410019886939.

IMechE (2014) *The International Vehicle Aerodynamics Conference, The International Vehicle Aerodynamics Conference*. Elsevier. doi: 10.1016/c2014-0-03634-9.

Jang, C. S., Ross, J. C. and Cummings, R. M. (1998) 'Numerical investigation of an airfoil with a Gurney flap', *Aircraft Design*. Elsevier Ltd, 1(2), pp. 75–88. doi: 10.1016/S1369-8869(98)00010-X.

Jones, R. T. (1979) *NASA Reference Publication 1050 Classical Aerodynamic Theory*.

Juniper, M. P. (2012) *Fluid Mechanics 8.1 LIFT AND DRAG*.

Kachare, S. C. (2017) 'A CFD Study of a Multi-Element Front Wing for a Formula One Racing Car'.

Kasim, A. B. and Al Zaidee, S. R. (2015) 'Validation of Computational Fluid Dynamics Technique for Turbulent Wind Flow Approach, Bluff Two-Dimensional Body', *International Journal of Science and Research*, 6, pp. 2319–7064. doi: 10.21275/ART20172706.

Katz, J. (1995a) 'Lift, Drag and Side Force', in *Race Car Aerodynamics Designing for Speed*. Bentley Publishers, pp. 45–52. Available at: <http://www.bentleypublishers.com/c/GAER> (Accessed: 29 April 2020).

Katz, J. (1995b) 'Race Car Aerodynamics', *Bentley Publishers*, pp. 271–283. doi: 10.1080/1354983042000219379.

Marqués-Bruna, P. (2013) 'Design of the Grand Touring sports car wing: geometric and aerodynamic twist', *Proceedings of the Institution of Mechanical Engineers, Part P: Journal of Sports Engineering and Technology*. SAGE PublicationsSage UK: London, England, 227(1), pp. 70–80. doi: 10.1177/1754337111429532.

McBeath, S. (2006) *Competition car aerodynamics : a practical handbook*.

Menter, F. (1992) 'Improved two-equation k-omega turbulence models for aerodynamic flows', *NASA Technical Memorandum*, (103978), pp. 1–31.

Menter, F. R. (1993) 'Zonal two equation κ - ω turbulence models for aerodynamic flows', in *AIAA*

23rd Fluid Dynamics, Plasmadynamics, and Lasers Conference, 1993. American Institute of Aeronautics and Astronautics Inc, AIAA. doi: 10.2514/6.1993-2906.

Merkel, J. P. (2013) *DEVELOPMENT OF MULTI-ELEMENT ACTIVE AERODYNAMICS FOR THE FORMULA SAE CAR*.

Mohankumar, M. |, Hudson, E. and Daniel, R. (2015) *Investigating the Effect of Rear Spoiler and Rear Diffuser on Aerodynamic Forces using CFD*. Available at: www.ijert.org (Accessed: 29 April 2020).

Mohebbi, F., Evans, B. and Sellier, M. (2019) 'On the Kutta condition in compressible flow over isolated airfoils', *Fluids*. MDPI AG, 4(2). doi: 10.3390/fluids4020102.

Motorsport Technology (2018) *F1 rear 'spoon wing' explained - Motorsport Technology*. Available at: <https://motorsport.tech/formula-1/f1-rear-wing-explained> (Accessed: 25 April 2020).

Motorsport UK (2019) *Motorsport UK Blue Book 2020*. Available at: www.motorsportuk.org (Accessed: 3 April 2020).

Narodytska, N. (no date) *Chapter 12 Wings of Finite Span*.

NASA (2007a) *Effect of Size on Drag*. Available at: <https://www.grc.nasa.gov/WWW/K-12/airplane/sized.html> (Accessed: 29 April 2020).

NASA (2007b) *Incorrect Lift Theory*. Available at: <https://www.grc.nasa.gov/WWW/K-12/airplane/wrong1.html> (Accessed: 3 May 2020).

NASA (2007c) *Navier-Stokes Equations, Van Nostrand's Scientific Encyclopedia*. doi: 10.1002/0471743984.vse9597.

Newey, A. (2017) *How to build a car*.

Newton, I., Cohen, I. B. and Whitman, A. M. (1728) *The Principia : mathematical principles of natural philosophy*.

Princeton University (2014) *Flows With Friction*. Available at: https://www.princeton.edu/~asmits/Bicycle_web/frictionflows.html (Accessed: 3 May 2020).

Racecar Engineering (2015) *Corvette C7.R - Racecar Engineering*. Available at: <https://www.racecar-engineering.com/cars/corvette-c7-r/> (Accessed: 3 May 2020).

Sakamoto, H. and Haniu, H. (1988) 'Aerodynamic forces acting on two square prisms placed vertically in a turbulent boundary layer', *Journal of Wind Engineering and Industrial Aerodynamics*, 31(1), pp. 41–66. doi: 10.1016/0167-6105(88)90187-0.

Schlichting, H. (1980) *Boundary-Layer Theory*.

Schuetz, T. (2018) *Aerodynamics of Road Vehicles*.

SimScale (2018) *What is y^+ (yplus)? - SimWiki - SimScale CAE Forum*. Available at: <https://www.simscale.com/forum/t/what-is-y-plus/82394> (Accessed: 14 January 2020).

Singh, B. J. et al. (1983) *NASA Technical Memorandum, Nasa Technical Memorandum*. Available at: <https://ntrs.nasa.gov/search.jsp?R=19900019424> (Accessed: 15 January 2020).

Smedley, R. (2019) *Flow-vis in F1: Rob Smedley explains the technology behind flow visualisation paint | Formula 1®*. Available at: <https://www.formula1.com/en/latest/article.testing-explained-rob-smedley-on-flow-vis.7nU2VePGLVrhIGa8wgCoLE.html> (Accessed: 1 March 2020).

Tientcheu-Nsiewe, M. et al. (2016) 'Ahmed body, Wake, Velocity field, Turbulent flow, CFD; Ahmed body, Wake, Velocity field, Turbulent flow, CFD', *American Journal of Environmental Engineering*, 6(6), pp. 157–163. doi: 10.5923/j.ajee.20160606.01.

Toet, W. (2019) *Willem Toet explains...CFD Post Processing. Part 3 - Race Tech Magazine*. Available at: <https://www.racetechnmag.com/2019/02/willem-toet-explains-cfd-post-processing-part-3/> (Accessed: 18 February 2020).

Tudor, T. (2017) *Vehicle Dynamics - Aerodynamics*.

White, F. (2005) *Viscous Fluid Flow (McGraw-Hill Mechanical Engineering)*. McGraw-Hill Science/Engineering/Math. Available at: <http://www.amazon.com/Viscous-Fluid-McGraw-Hill-Mechanical-Engineering/dp/0072402318> (Accessed: 18 February 2020).