



University of Wales Trinity Saint David
School of Engineering

Rear wing design optimisation for MCR Hillclimb car

by

Simon Ellis-Williams

P150613

Supervisor

Tim Tudor

UWTSD

School of engineering

This dissertation is submitted in partial fulfilment

For the requirement of the degree of

Bachelor of Motorsport Engineering

2020

Declaration of authenticity

I, Simon Ellis-Williams, declare that this dissertation is my original work, gathered and utilized to fulfil the purposes and objectives of this study, and has not been previously submitted to any other university for a higher degree. I also declare that the publications cited in this work have been personally consulted.

A big thank you to Ruth, Dai, and Beryl
for supporting me for the past four years (and more).

Also a special thanks to Tim Tudor
for the numerous kicks up the backside to keep me going through this project!

This one is for you, Myrvin x

Abstract

This project aimed to design a new rear wing for an MCR sport2000 car for use in the UK Hillclimb championship. Due to a new and more effective front splitter being fitted, the cars aerodynamic centre of pressure has been shifted too far forward. To rectify this a new wing was needed to bring the centre of pressure back to 50% from the rear axle, whilst also aiming for a 50:1N lift/drag ratio.

models of the SRCC wing, an Eppler e423 profile wing, and a Wortmann FX74 CL5 wing were created using SolidWorks Computer-aided design software and analysed using ANSYS computational fluid dynamics software.

Analysis of the SRCC wing showed it produced a peak of 113N of downforce at 15 degrees angle of attack. Straight-line test data also showed a further 300N needed to be produced by the wing. further analysis of the two new profiles found that they were overly powerful for what was required, each producing over 400N of downforce at 0 angle of attack and 900N+ of downforce at higher angles of attack. The effects of wing position, endplate depth are also discussed, and a design for alternating onset airflow is proposed.

Contents

Declaration of authenticity	ii
Abstract.....	iv
Contents	v
Table of figures	viii
1) Introduction	1
2) Project aims.....	2
4) Literature review	3
4.1 Rules and regulations.....	3
4.1.1 Plan view:	3
4.1.2 Side elevation:.....	4
4.1.3 Rear elevation	4
4.2 Wing theory	5
4.3 The Navier-Stokes equations	8
4.4 Wing geometries and how they affect performance.....	9
4.4.1 Chord line	9
4.4.2 Angle of attack	9
4.4.3 Leading edge radius	9
4.4.4 Camber.....	9
4.4.5 Thickness.....	10
4.5 Race vehicle wing design	10
4.5.1 Multiple elements.....	10
4.5.2 Wing twist	10
4.5.3 Current wing	11
4.6 Computational fluid dynamics	12
4.6.1 Domain.....	12
4.6.2 Meshing.....	13

4.6.3 Checking mesh quality	13
4.6.4 Turbulence modelling	13
4.6.5 Solving.....	14
5) CFD validation cases	15
5.1 Ahmed body.....	15
5.1.1 Domain study	16
5.1.2 Mesh study.....	21
5.1.3 Prism layers.....	24
5.1.4 Turbulence model	25
5.1.5 Final run	26
5.2 Wind tunnel scale wing model.....	29
6) Modelling and analysis of the SRCC wing	31
6.1 CAD model	31
6.2 CFD Analysis	33
7) Straight line test data.....	36
8) Choosing and analysing new wing profiles	38
8.1 Eppler e423	39
8.2 Wortmann FX74 CL5	39
8.3 Angle of attack study	40
Eppler E423	41
Wortmann FX74	42
9) Wing placement study	44
9.1 Floating-point errors	45
10) COVID-19 pandemic impact.....	46
11) Future planned work	47
11.1 Wing placements continued	47
11.2 Designing for onset flow	49

11.3 Endplates.....	51
12) Conclusion.....	53
References	54
Appendix A.....	58
NACA 0012 scale wind tunnel pressure distribution data	58
Appendix B	59
Comparison between SRCC, FX74, and E423 wings.....	59

Table of figures

figure 4.1.1 - plan view of the MCR car with the regulation wing position box shown in blue	3
Figure 4.1.2 - Side view of the MCR car with the regulation wing position box shown in blue	4
Figure 4.1.3 - Rear view of the MCR car with the regulation wing position box shown in blue	4
Figure 4.2.1 - Streamlines showing that air traveling under the low-pressure side of a wing reaches the trailing edge first, thus proving Bernoulli's principle is wrongly applied to wing theory. (Babinsky, 2003)	5
Figure 4.2.2 - wake region behind a circle.	5
Figure 4.2.3 - magnus effect shown on a spinning circle, the air being deflected down over the top results in an upward force. Invalid source specified.	6
Figure 4.2.4 - sketch showing the starting vortex produced as the wing begins to move, and the anti-vortex of circulatory air around the wing.	6
Figure 4.3.1 - The five separate parts that make up the Navier-Stokes equation. The equation details the flow of fluids in three-dimensional space. (NASA, 2015)	8
Figure 4.4.1 - wing profile with the main four dimensions marked out. The red line marks the chord line, the yellow shows the mean camber line, green shows maximum thickness, and blue shows maximum camber thickness.	9
Figure 4.4.2 graph showing how the change in angle of attack for a NACA 0012 aerofoil affects lift, and how too much AOA can lead to stall.Invalid source specified.	9
Figure 4.5.1 a multi element wing design. (Hepperle, 2018)	10
Figure 4.5.2 twisted wing design (Vortex, 2016)	10
Table 4.1 - Dimensions for the SRCC RA1 wing	11
Figure 4.5.3 – The current SRCC RA1 wing and mounting on the MCR chassis, along with a view of the profile.	11
Figure 4.6.1 Difference between structured and unstructured mesh types. as shown the unstructured is more chaotic, with no discernable pattern (Research gate, 2019)	12
Figure 5.1 - Ahmed body dimensions. The angle of the upper rear slope can be changed to see the effect of the lift and drag. (simscale.com/docs/_images/AhmedBody-diag.png)	15
Table 5.1.1 - Domain size settings for run 1	16
Figure 0.2 - Visualisation of the domain for run 1	16
Table 5.1.2 - Mesh settings for the domain runs	16
Table 5.1.3 - Coefficient results for domain run 1, including percentage difference from experimental data, and run time.	17
Table 5.1.4 - Run 2 domain sizes and results	18
Figure 0.3 - Visualisation of the domain for run 2	18
Table 5.1.5 – Run 3 domain sizes and results	19
Figure 0.4 - visualisation of the domain for run 3	19

Table 5.1.6 - Run 4 domain sizes and results	20
Figure 5.5 - Visualisation of the domain for run 4	20
Table 5.1.7 - Mesh settings used for run 1 of the mesh sweeps	21
Figure 5.6 - Visualisation of the mesh for run 1 of the mesh sweeps	21
Table 5.1.8 - Results for run 1 of the mesh sweeps	21
Table 5.1.9 - Settings used for run 2 of the mesh sweeps.	22
Figure 5.7 - Visualisation of the mesh used for run 2 of the mesh sweeps. a finer mesh was used for the surface of the part.	22
Table 5.1.10 - Results from run 2 of the mesh sweeps	22
Table 5. 1. 11 - settings used for run 3 of the mesh sweeps	23
Figure 5.8 – Visualisation of the mesh used in run 3 of the mesh sweeps.	23
Table 5.1.12 - results for run 3 of the mesh sweeps.	23
Table 5.1.13 - Settings and results for run 1 of the prism layer investigation	24
Table 5.1.14 – Settings and results for run 2 of the prism layer investigation with decreased total height.	24
Table 5.1.15 - Settings and results for run 3 of the prism layer investigation with increased number of layers	25
Table 5.1.16 - Turbulence model and results for run 1 of the turbulence model investigation	25
Table 5.1.17 - Model and results for run 2 of the turbulence model investigation	26
Table 5.1.18 - Model and results for run 3 of the turbulence model investigation	26
Table 5.1.19 – settings used during the final run of the Ahmed body, using the settings that provided the most accurate results all consolidated into one set up.	27
Table 5.1.20 – final run results comparison to experimental data from W. Meile. Results were within 4%, proving the settings used to be accurate.	27
Figure 5.9 – Pressure distribution across the surface of the Ahmed body during the final CFD run.	28
Figure 5.2.1 (a b c) – 3D printed scale model of the SRCC wing being tested in the wind tunnel at different angles of attack. In the right picture (c) separation can be seen to occur as the thread violently moves around in the turbulent air.	29
Table 5.2.1 – Averaged results of the five wind tunnel tests, conversions to newtons and coefficient values are also calculated.	29
Figure 5.2.2 – Graph showing the averaged coefficients for lift and drag through angles of attack from -5 to +20 degrees.	30
Figure 5.2.3 - Graph comparing the coefficient values for the averaged wind tunnel data and the CFD runs of the SRCC wing in isolation.	30
Figure 6.1 - a) chord, thickness, and camber dimensions for the wing profile of the SRCC RA1. b) Finished CAD model of the wing to be used in CFD analysis.	31
Table 6.1 - Dimensions for the SRCC RA1 wing	31

Figure 6.2 - initial trace of the SRCC wing profile and measurements for creating the CAD file.	32
Figure 6.3 - Using the measurements from the trace to create the profile in solid works. The mean camber line can be seen in relation to the chord.	32
Figure 6.4 – adding the points for upper and lower surface, and using the curve tool to join them, creating the profile of the SRCC RA1 wing	32
Table 6.2 - Settings used for the angle of attack study of the SRCC wing	33
Table 6.3 - SRCC wing coefficients results from the angle of attack study	34
Figure 6.5 – separation beginning to occur at AoA 15 on the SRCC wing. The blue region shows separated airflow recirculating in a vortex.	34
Figure 6.6 - Graph showing the coefficients of lift and drag for the SRCC wing, using the k-epsilon turbulence model	35
Figure 6.7 - Graph showing the efficiency of the SRCC RA1 wing.	35
Figure 7.1 - Graph showing the front aero loading on the MCR car at different ride heights with different splitters. (Tudor,2019)	36
Figure 7.2 – Graph showing the rear aero loading on the MCR car at different ride heights with different splitters. The data in green shows the (Tudor, 2019)	36
Figure 7.3 – Graph showing centre of pressure on the MCR car at different ride heights for three different splitters. (Tudor, 2019)	37
Figure 8.1 - Eppler 423 airfoil profile (Airfoil Tools, 2020)	38
Figure 8.2 Wortmann FX74 CL5 airfoil profile (Airfoil Tools, 2020)	38
Figure 8.1.1– Graphs taken from airfoil tools website, displaying the coefficients of lift and efficiency of the Eppler e423 profile. The different colour lines represent different Reynolds numbers from 500,000 to 1,000,000. ranging from around 20m/s to 35m/s. (Airfoil tools, 2020)	39
Figure 8.2.1 -Graphs taken from airfoil tools website, displaying the coefficients of lift and efficiency of the Wortmann FX74 CL5 profile. The different colour lines represent different Reynolds numbers from 500,000 to 1,000,000. (Airfoil tools, 2020)	39
Table 8.1 - Table showing the settings used during angle of attack study on the Eppler and Wortmann wings	40
Table 8.2 - table showing the results from the angle of attack study of the Eppler e423 wing	41
Figure 8.3.1 - Graph showing the coefficient of lift and drag for the Eppler e423 wing over a range of angles of attack	41
Table 8.3 -Table showing the results from the angle of attack study of the Wortmann FX74 CL5 wing	42
Figure 8.3.4 - Graph showing the coefficient of lift and drag for the Wortmann FX75 wing over a range of angles of attack	42

Figure 8.3.5 – Post processing images of the Wortmann FX74 starting to show separation between 8 degrees on the left, and 9 degrees on the right	43
Figure 8.3.6 – Post processing images of the Eppler E423 starting to show separation between 9 degrees on the left, and 10 degrees on the right	43
Table 9.1 - Table showing the location coordinates for the position of the rear wing	44
Figure 9.1 - Picture showing the proposed positions for wing placement within the rule box coordinates are taken from the bottom left corner	44
Figure 11.1 - Graph showing the results of Simon McBeath's wing height study at X = 0mm. wing downforce can be seen in green. (McBeath, 2014)	47
Figure 11.2 - Graph showing the effect of moving the wing rearward in 150mm steps and repeating the height study. X = 150mm showed the best downforce even as height was lowered. (McBeath, 2014)	47
Figure 11.3 - Ideal location for wing placement according to McBeath study.	48
Table 11.1 - Table of X and Y air velocities at position 3. Trigonometric calculations were then used to calculate onset flow. Values in red are to be ignored as these extend past the regulation zone.	49
Figure 11.2.1 - Eppler E423 wing designed to suit onset flow. the wing has a spanwise twist that maximises efficiency.	49
Table 11.2 - Table showing the resultant AoA across the span, this was used to design the twist so each part of the wing sits at 4 degrees AoA to the onset flow.	50
Figure 11.3.1 - Graph displaying McBeath's results of endplate depth against downforce. An increase in downforce is seen as depth increases, but a large spike in downforce is seen between 75mm and 125mm. (McBeath, 2008)	51
Figure 11.3.2 - Examples of different shapes or slots cut out of endplates to improve performance under high yaw manoeuvres.	52
Figure 13.1 - pressure distribution on the NACA 0012 at AoA 0 and 2	58
Figure 13.2 - pressure distribution on the NACA 0012 at AoA 4 and 6	58
Figure 13.3 - pressure distribution on the NACA 0012 at AoA 8 and 10	58
Figure 14.1 – Graph showing the coefficients of drag and lift for the SRCC wing, the Wortmann FX74 wing, and the Eppler E423 wing.	59

1) Introduction

An Ornithologist by the name of Roger Tory Peterson once said “Birds have wings; they are free; they can fly where they want when they want. They have the kind of mobility many people envy.” It could be argued that it was this envy that lead the Wright brothers to pioneer flight in 1903, sparking an entire industry of innovation over the next century which continues to this day. Aerodynamics has changed the way in which we see the world around us, it has shaped buildings, improved car design, and helped us conquer the skies. (Woodford, 2012)

During the late sixties, Hap sharp and Jim Hall realised the importance of aerodynamics within the field of motorsport. In the search for more grip and faster lap times, the Chaparral 2E was unveiled - A race car featuring a large inverted wing which shaped the beginning of true aerodynamic performance in motorsports (Chaparral cars, 2005).

The aim of this project is to gain an understanding of the aerodynamic concepts which lead to a wing producing lift, whilst also designing a new wing for use in motorsport which increases the rear downforce seen on an MCR sports2000 car for use in Hillclimb racing. ANSYS computational fluid dynamics software will be used to analyse the performance of several wings created using SolidWorks computer aided design software. The effects of angle of attack, wing position, and endplate depth will be investigated with hope to understand how these factors affect wing performance.

2) Project aims

- 50 to 1 lift/drag ratio

The new wing will aim for an overall efficiency ratio of 50N of downforce to every 1N of drag.

- Bring CoP rearwards Increase rear wing downforce by 40%

Currently, the centre of pressure sits too far forward, making the handling of the car feel twitchy and unpredictable. Balancing the front and rear downforce will improve this problem, and lead to a more predictable handling car, with a more competitive edge.

- Make sure wing works with the diffuser to increase overall rear end downforce by 116%

Working closely with the diffuser project to ensure the correct amount of downforce is created to balance the aerodynamics of the vehicle.

- Design the new wing within current regulations.

To be allowed to compete, the new wing must be developed to comply with the rules and regulations set out by the MSA's blue book 2019.

4) Literature review

4.1 Rules and regulations

The MSA blue book states in rule S-13.5.7 that “Rear wings for rear-engine sports racing cars must not in width exceed the outer faces of rear wheel rims. Maximum permitted height is 900mm from ground. No part of bodywork, including aerofoils must extend more than 900mm behind the rear wheel axis” (Motorsport UK, 2019). An amendment was later made to this rule, allowing the wing to extend to 1100mm behind the axis. A visualisation of these rules around the MCR bodywork has been created in CAD.

A visualisation of these rules around the MCR bodywork has been created in CAD and can be seen in the figures below:

4.1.1 Plan view:

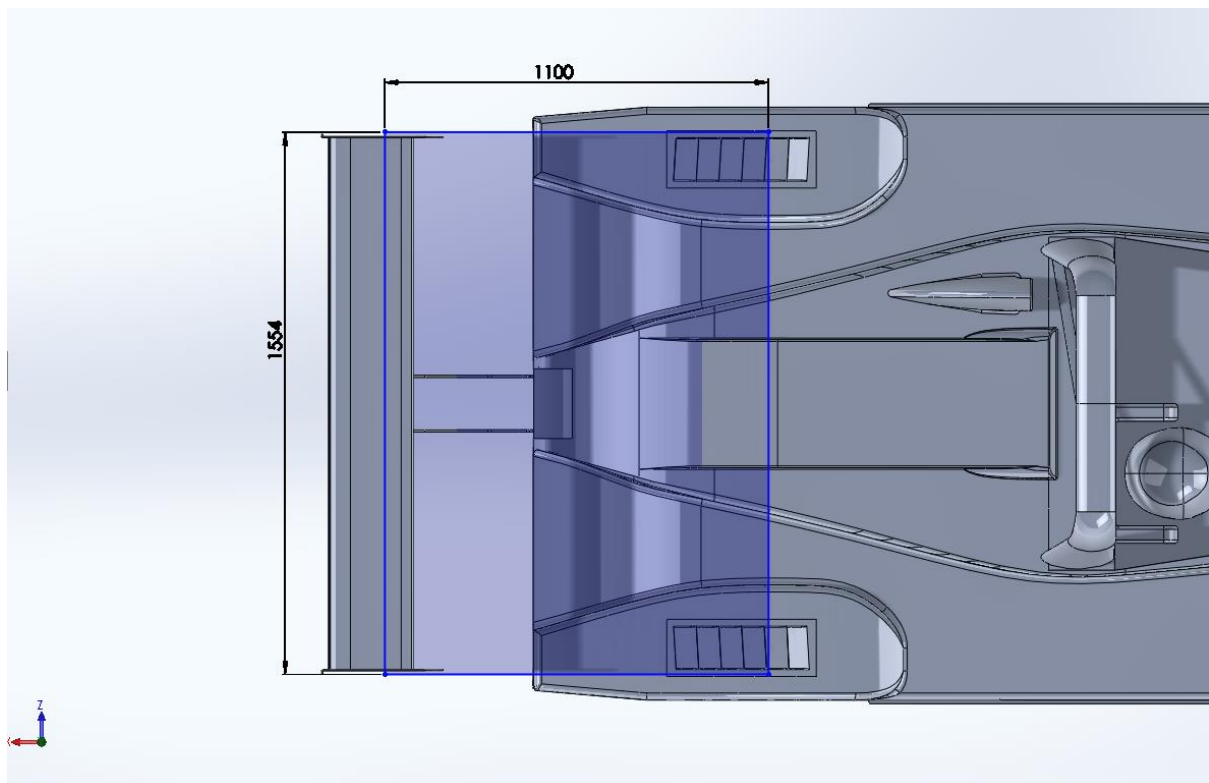


figure 4.1.1 - plan view of the MCR car with the regulation wing position box shown in blue

4.1.2 Side elevation:

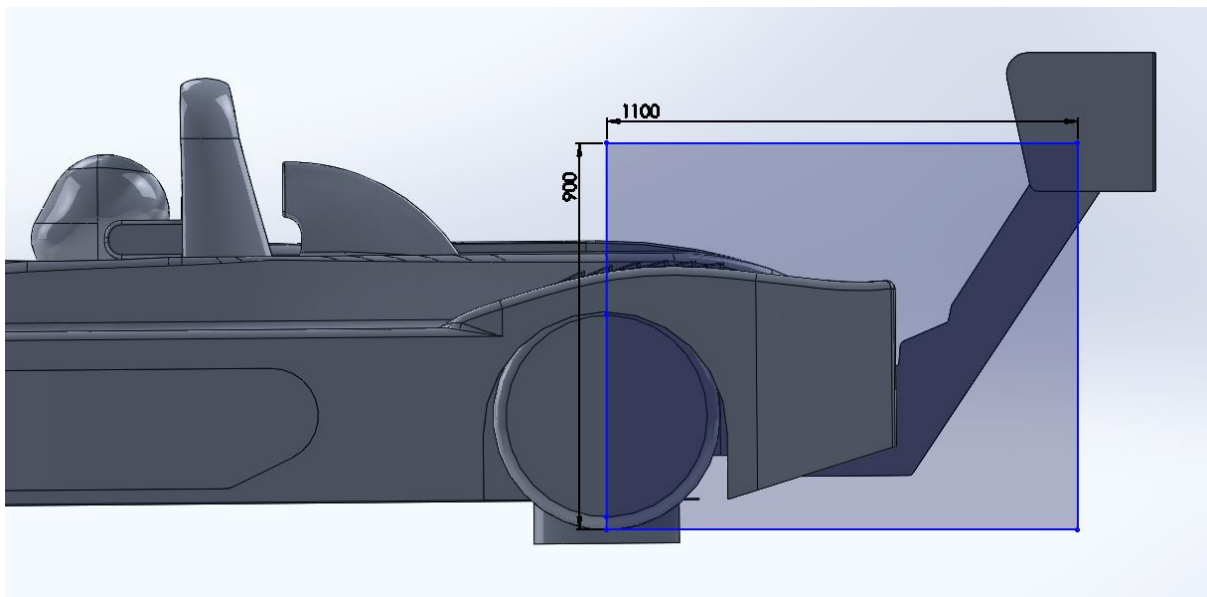


Figure 4.1.2 - Side view of the MCR car with the regulation wing position box shown in blue

4.1.3 Rear elevation

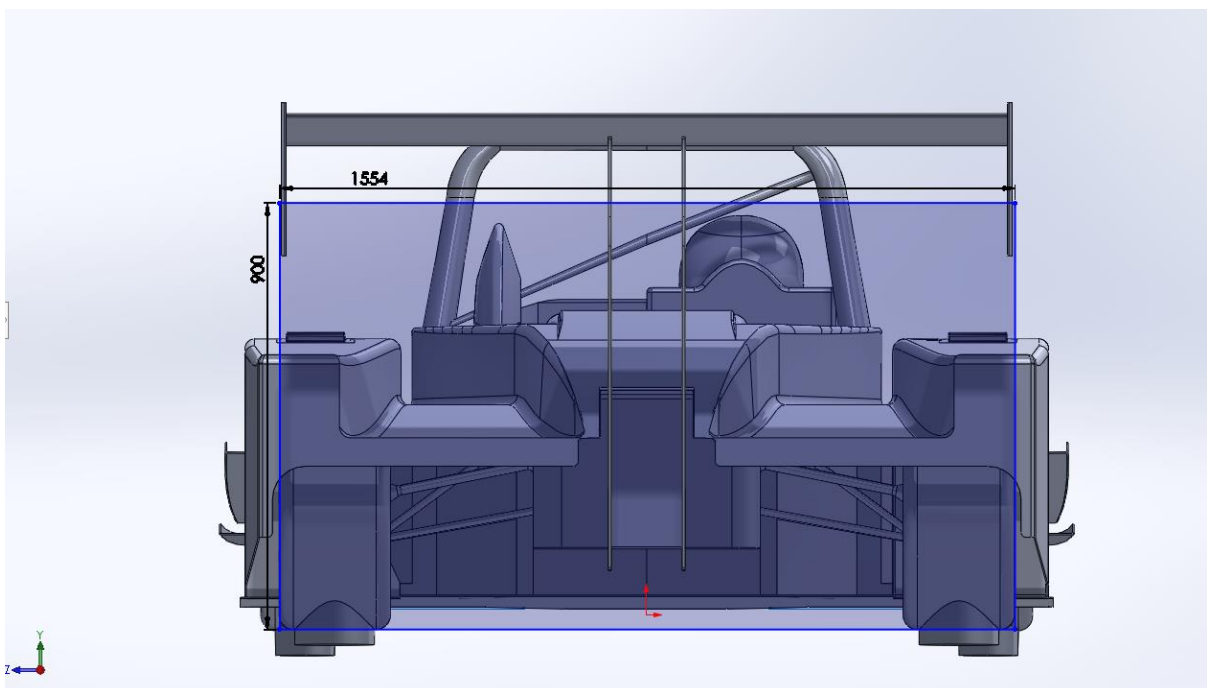


Figure 4.1.3 - Rear view of the MCR car with the regulation wing position box shown in blue

As can be seen, the current wing is out of the 2019 regulations and a new wing and mount system need to be designed to fit the current rulings so the car may enter the series again.

4.2 Wing theory

Wings generate downforce by creating a pressure difference between the two wing surfaces, however, how they do this is not quite as simple. A lot of basic explanations state that Bernoulli's principle can be used to show how wings create downforce, but this is partially wrong. Bernoulli's theory assumes that two particles of air that hit the wing at the same time each separate, one taking a path around the high-pressure side of the wing, and the other via the low-pressure side must meet back up once they reach the trailing edge.

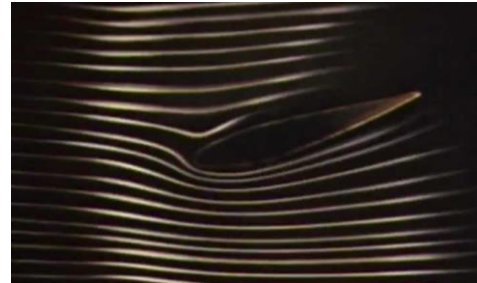


Figure 4.2.1 - Streamlines showing that air traveling under the low-pressure side of a wing reaches the trailing edge first, thus proving Bernoulli's principle is wrongly applied to wing theory. (Babinsky, 2003)

The particle that takes the longer low-pressure side must travel further due to the geometry of the wing and in so doing will have to speed up to meet with the high-pressure side particle at the trailing edge, and Bernoulli's principle

states that an increase in velocity leads to a decrease in pressure. This has been proven to be partially wrong, as the particle travelling over the low-pressure side reaches the trailing edge first and the two particles never meet up again, as shown in figure 4.2.1. (Babinsky, 2003)

Another theory uses parts of Bernoulli's theory along with Newton's third law of motion to try and make sense of how wings produce lift. Newton's third law states every force must have an equal and opposite reaction force. Applying this to an aerofoil, the effect the shape of the wing has on the air travelling underneath it, forcing it to curve upwards (due to the Coanda effect), and the air being deflected around the top surface will result in a reaction force of the wing being forced downwards.

Newton's second law can also be applied, along with Bernoulli's theory, to a particle travelling over a wing to help explain why the low-pressure particle moves faster. If a particle is imagined moving from a high pressure to a low-pressure area, the pressure difference between the front and rear of the particle will force it forward causing it to accelerate, whereas a particle moving from a low pressure to high pressure will decelerate. Although these theories make sense, they are still not applied correctly to the problem (Singh, et al., 2018)

If airflow around a circle is analysed, it is shown to produce a stable wake region behind it as per figure 2.2, but if the circle begins to spin clockwise, the wake region becomes unstable due to the boundary layer at the surface of the circle travelling with the circle and pulling air over the top, even air that is approaching from below the original stagnation point. Due to boundary layer attachment, air passing the bottom of the circle is slowed by

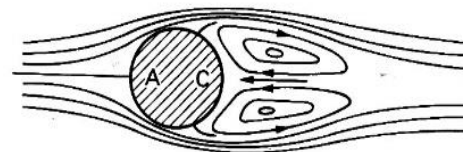


Figure 4.2.2 - wake region behind a circle.

the attached air, whilst air around the top surface is accelerated and deflected downward. Applying Newton's third law, the air being deflected results in an equal and opposite force on the circle forcing it upwards. This effect is known as the Magnus effect and is used to manipulate the travel of objects in a lot of sports such as football or golf. (Swanson, 1961)

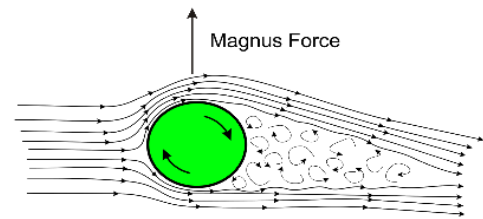


Figure 4.2.3 - magnus effect shown on a spinning circle, the air being deflected down over the top results in an upward force. Invalid source specified.

The Kutta-Joukowski circulation of lift theorem applies the Magnus effect to wings and theorises that airflow circulates around a wing (figure 4.2.4) which creates the speed difference. As a wing begins to move through the air, an anticlockwise vortex is generated after the trailing edge of the wing known as a starting vortex. Helmholtz theorem states that net circulation in a fluid must remain constant, so due to this, the starting vortex needs an anti-vortex of the same strength spinning in the opposite direction, so that the net effect is zero. Due to the circulation of air around the wing, the air travelling underneath the wing is met with resistance from the circulation anti-vortex which slows it down, whereas the air travelling over the top of the wing speeds up as it receives extra energy from the vortex travelling with it. The total flow over the wing can be thought of as a product of both the free stream air and the circulatory air in the anti-vortex. (Kersale, Unknown)

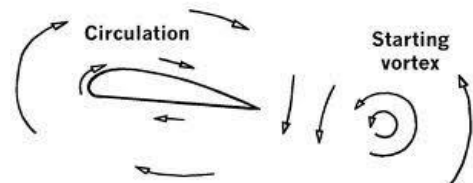


Figure 4.2.4 - sketch showing the starting vortex produced as the wing begins to move, and the anti-vortex of circulatory air around the wing.

Unfortunately, the circulation theory of lift does not model what is physically happening either, as circulation theory is an alternative way of using a mathematic expression to express the consequences of surface pressure distribution, by using formulas that were developed especially for analysing airflow problems. Therefore, it is wrong to say that the circulation around the wing causes lift, and more fitting to say lift is caused by a net imbalance of surface pressure distribution, with circulation being a defined quantity due to the same pressure. Equation (1) shows how circulation and surface pressure distribution are related, and finding the circulation is an easier task than working out pressure distribution. Once circulation is found, it can then be used to calculate the lift per unit span. (Anderson, 1985).

$$L' = \rho_{\infty} V_{\infty} \Gamma \quad (1)$$

Where:

L' = lift per unit span

ρ = viscosity of air

V = velocity of air

Γ = circulation strength

Many more theories that try to explain how wings work have been argued by many fluid dynamists and mathematicians over the years, each with their own inconsistencies and counter-arguments. There seems to be a lack of clarity in the field on how and why wings generate the lift forces they do, or why the airflow behaves in the way it does. This is down to the number of assumptions needing to be made when trying to model air turbulence behaviour and wing theories. (Matthews, 2012)

4.3 The Navier-Stokes equations

The Navier-Stokes (NS) equation was created by Claude-Louis Navier, a French engineer in 1821 to try and model and predict the flow of incompressible fluids and turbulence. The British Physicist Sir George Gabriel Stokes furthered the work previously done by Navier midway through the 19th century.

The coupled partial differential equations (PDEs) are based upon Euler's equation for modelling two-dimensional fluid flow (NASA, 2015). The coupled PDEs are expressed based on the principles of conservation of mass, momentum, and energy in three-dimensional space, while also accounting for the viscosity of the fluid. (Fefferman, 2017). Computational fluid dynamics (CFD) programs such as ANSYS use the formulas to calculate airflow and turbulence around a body.

The equation works well in solving two-dimensional flow problems but cannot solve the complexities of turbulence in a three-dimensional model without using approximations in the numerical analysis method. These assumptions produce potentially inaccurate solutions to the NS equation, so solutions are never perfectly accurate. (SimWiki, 2016) The equations are seen below in figure 4.3.1:

Coordinates: (x,y,z)	Time: t Pressure: p	Heat Flux: q
Velocity Components: (u,v,w)	Density: ρ Stress: τ	Reynolds Number: Re
	Total Energy: Et	Prandtl Number: Pr

Continuity:
$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

X – Momentum:
$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = -\frac{\partial p}{\partial x} + \frac{1}{Re_r} \left[\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} \right]$$

Y – Momentum:
$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} + \frac{\partial(\rho vw)}{\partial z} = -\frac{\partial p}{\partial y} + \frac{1}{Re_r} \left[\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} \right]$$

Z – Momentum:
$$\frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho uw)}{\partial x} + \frac{\partial(\rho vw)}{\partial y} + \frac{\partial(\rho w^2)}{\partial z} = -\frac{\partial p}{\partial z} + \frac{1}{Re_r} \left[\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \right]$$

Energy:
$$\begin{aligned} \frac{\partial(E_T)}{\partial t} + \frac{\partial(uE_T)}{\partial x} + \frac{\partial(vE_T)}{\partial y} + \frac{\partial(wE_T)}{\partial z} = & -\frac{\partial(up)}{\partial x} - \frac{\partial(vp)}{\partial y} - \frac{\partial(wp)}{\partial z} - \frac{1}{Re_r Pr_r} \left[\frac{\partial q_x}{\partial x} + \frac{\partial q_y}{\partial y} + \frac{\partial q_z}{\partial z} \right] \\ & + \frac{1}{Re_r} \left[\frac{\partial}{\partial x}(u \tau_{xx} + v \tau_{xy} + w \tau_{xz}) + \frac{\partial}{\partial y}(u \tau_{xy} + v \tau_{yy} + w \tau_{yz}) + \frac{\partial}{\partial z}(u \tau_{xz} + v \tau_{yz} + w \tau_{zz}) \right] \end{aligned}$$

Figure 4.3.1 - The five separate parts that make up the Navier-Stokes equation. The equation details the flow of fluids in three-dimensional space. (NASA, 2015)

4.4 Wing geometries and how they affect performance

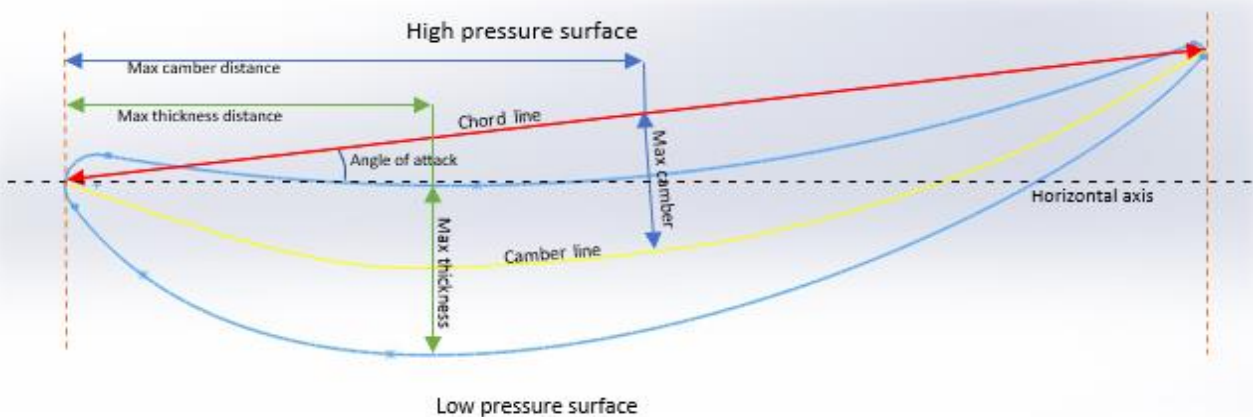


Figure 4.4.1 - wing profile with the main four dimensions marked out. The red line marks the chord line, the yellow shows the mean camber line, green shows maximum thickness, and blue shows maximum camber thickness.

Figure 4.4.1 shows how the main dimensions of a wing profile are measured, each of these dimensions influences how the wing behaves. The book “Theory of Flight” by L.N.V.M. Society Group of Institutes discusses the effects of these dimensions on wing performance:

4.4.1 Chord line – The chord line is a line that joins the tips of the leading edge (LE) and trailing edge (TE) together. The rest of the dimensions are all expressed as percentages of this dimension, and maximum camber is measured from this line.

4.4.2 Angle of attack – The angle of attack (AoA) is the angle of the chord line relative to the oncoming airflow. The angle of attack is important as it can increase the lift of a wing easily without having to change the profile of the wing. But if the angle of attack is increased too far, separation will occur and wing stall will occur leading to a loss in lift, which can be seen in figure 4.4.2 at an angle of around 18 degrees.

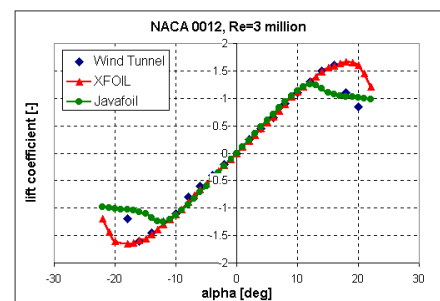


Figure 4.4.2 graph showing how the change in angle of attack for a NACA 0012 aerofoil affects lift, and how too much AOA can lead to stall. Invalid source specified.

4.4.3 Leading edge radius – the radius of the leading edge is measured as a percentage of the chord, and its dimension has a significant effect on the boundary layer and flow separation.

4.4.4 Camber – The mean camber line of an airfoil is a line from the LE to TE that runs exactly through the midpoint of the profiles upper and lower surfaces. So on a symmetrical wing, it would follow the same path as the chord line. Camber relates to how much concave or curve the wing has. Maximum camber is measured as a percentage of the chord length and is measured from the chord line to locate where the maximum camber point is found. Camber is one of the main factors that affect

the performance of a wing, as a wing with camber will still produce lift at zero angle of attack due to the differences in shape between the upper and lower sides. Large camber can increase downforce at lower speeds, although if the camber is too large separation will begin to occur at lower speeds as well.

4.4.5 Thickness – Thickness is another parameter that has a strong influence on how the wing behaves. It is measured as a percentage of the chord, with maximum thickness location being measured from the leading edge as a percentage of the chord also. The thicker the wing, the lower the pressure and higher the velocity of the air travelling around it; therefore the lift coefficient increases with the thickness of the wing. Toward the rear of the wing where there is an adverse pressure gradient, the boundary layer begins to thicken. This leads to separation and form drag if the wing is too thick, so the benefits of the effects of thicker wings tend to be limited. (L.N.V.M. Society Group of Institutes, 2007)

4.5 Race vehicle wing design

Due to rules and safety regulations, rear wings on cars have to work within a confined space compared to those seen on aeroplanes. To maximise efficiency, race car wings make use of design features such as multiple elements, and wing twist.

4.5.1 Multiple elements

A multiple element wing design utilises two or more aerofoils stacked in such a way as to let high-pressure airflow bleed through to the low-pressure side, doing so re-energises the boundary layer, helping to keep flow attached and create higher downforce. In most applications, each element can be adjusted so the overlap between each element, and the angle of attack of each element is adjustable, helping to maximise the potential for a given situation.

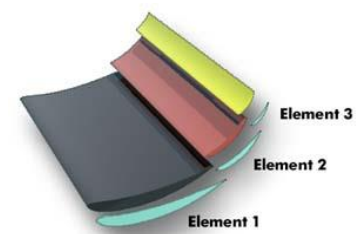


Figure 4.5.1 a multi element wing design. (Hepperle, 2018)

4.5.2 Wing twist

A twist in the span of a wing is to account for changes in the angle of the onset flow approaching the wing. Most GT class cars have a twist in their rear wing design to account for airflow over the top of the car approaching the middle of the wing at a different angle to that coming around the sides of the car. Accounting for different airflow lets a wing fully utilise all its span to its maximum potential.



Figure 4.5.2 twisted wing design (Voltex, 2016)

4.5.3 Current wing

The current wing on the MCR car is a championship regulation wing made by Elite Carbon Fibre, known as the SRCC RA1 wing. (Sports Racing Car Club, 2019)

Dimensions for the wing are shown in table 4.1:

Table 4.1 - Dimensions for the SRCC RA1 wing

Parameter	
Chord	244mm
Camber	12% of chord (30mm)
Camber position	56% of chord (143mm)
Thickness	16% of chord (40mm)
Leading edge radius	4mm
Length	1554mm



Figure 4.5.3 – The current SRCC RA1 wing and mounting on the MCR chassis, along with a view of the profile.

With the measurements taken, the aspect ratio (AR) is given by:

$$AR = \frac{span}{chord} \quad (2)$$

So:

$$AR = \frac{1554}{244} = 6.37 \text{ (2dp)} \quad (2a)$$

4.6 Computational fluid dynamics

Computational fluid dynamics (CFD), is used to predict the turbulence and interactions of air around a body. A domain is created, which represents the area of study around the object being analysed. The computer-aided design (CAD) model is processed and checked for any areas that need smoothing or any gaps filling in. Lanfrit recommends for external studies that any areas on the vehicle such as vents or air intakes should be smoothed over, and sharp angle areas avoided so as not to end up with a large amount of highly skewed cells. These areas can be blunted in CAD before importing into ANSYS. (Lanfrit, 2005)

Once the domain is specified, it is then filled with the mesh. This creates a mesh of either hexahedral, or tetrahedral elements ranging in sizes which can be specified by the user, with higher element counts usually resulting in more detail captured and better results. However, increased detail requires increased computational time. Elements can be set to be a structured (ordered) or unstructured mesh (unorderly), the difference

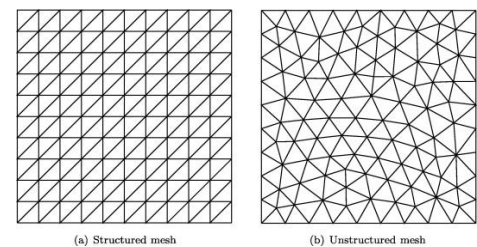


Figure 4.6.1 Difference between structured and unstructured mesh types. as shown the unstructured is more chaotic, with no discernable pattern (Research gate, 2019)

between these can be seen in figure 4.6.1. Hybrid meshes are also available, which use a mixture of both unstructured and structured depending on the geometry. Prismatic layers can also be computed to model the horizontal flow in the boundary layer with more detail.

Once a mesh has been created, the user needs to specify the physical modelling such as the equations of motion for fluid and other factors that apply to the study. Boundary conditions are then stated, specifying bounding surface properties, and fluid behaviours. Once these are set, the simulation is started. The equations are solved in iterations, converging on a solution. Once the calculations are completed and a solution is given, a post-processing program can be used to help the user visualise the solution and conduct any analysis.

4.6.1 Domain

With regards to the domain the simulation will be computed within, Lanfrit suggests that for an open-air simulation, a domain size of at least three body lengths in front of the body, and five lengths behind the body should be used. With the body itself not displacing more than 1.5% of the cross-sectional area. Whereas if the user is trying to replicate a wind tunnel, then the dimensions of said wind tunnel should be applied so the wall effects are accounted for. (Lanfrit, 2005)

4.6.2 Meshing

Different studies will require different meshes. A few types of mesh strategies found in ICEM include:

Robust (octree) – This process is top down, meaning the volume mesh is created before the surface mesh. The Octree strategy uses a patch independent approach to generate the surface mesh during the process, therefore, not needing to have an existing surface mesh. (Baehmann, 1987)

Quick (delaunay) – This method does things in the opposite order to the octree method, generating the surface mesh first, then moving on to the volume mesh. Due to the necessity of a closed surface when using this method, it is patch dependant, and if a body doesn't have a closed volume then the mesh will create a surface using global mesh parameters. (Shewchuk, 1996)

Smooth (advancing front) – A higher quality mesh, the smooth advancing front method is similar to the octree method, in that it creates a surface mesh first, then transitions to create the volume mesh at a smooth growth rate. This method should produce a surface mesh with no sharp changes in element sizes. (Schöberl, 1997)

4.6.3 Checking mesh quality

A quality volume mesh should ensure that the largest cells within the domain are of a smaller size than those connected to the bounding box if a pure tetrahedral mesh is being used. Particles travelling from the inlet to the stagnation point should pass through a minimum of 100 cells before hitting the body. Lanfrit recommends a mesh cell quality of below 0.9 for the domain, and below 0.85 for prism elements on the body surface. The surface mesh should be fine enough so that it can represent the shape of the parts it is meshing accurately enough, especially around stagnation points. Spherical and sharp-edged areas should be meshed to a high quality, whilst flat areas such as doors or roofs can be meshed more coarsely. Lanfrit states that surface mesh skewness should ideally be no less than 0.45, and that surface elements should grow at a maximum rate of 20% even from radius to planar faces. (Lanfrit, 2005)

4.6.4 Turbulence modelling

Once the mesh is complete, the airflow conditions for the problem must be specified. Turbulence models and boundary conditions are set detailing the behaviour of the airflow. There are several different turbulence models that can be used in different situations.

RANS (k-omega) – The Reynolds averaged numerical solution uses the Navier-Stokes equations with approximations made based on knowledge of flow turbulence properties. RANS is usually used for solving turbulent flows. (Reynolds, 1895)

LES – A large eddy simulation disregards the smaller eddies, concentrating on modelling the larger more important eddies in greater detail. LES is used regularly in aerospace engineering for modelling highly turbulent flows, such as those around an aircraft's landing gear.

SST – Developed by Menter, the SST approach to turbulence modelling blends the K-omega and K-epsilon models together using blending equations to better model the boundary layers and wake regions. (ANSYS, 2006)

4.6.5 Solving

The flow velocity and turbulent intensity of wind tunnels, or the scenario being replicated, are already known in the majority of cases, boundary conditions in CFD should replicate these parameters as close as possible. The velocity inlet models the air coming into the domain, specifying the velocity magnitude of air, the direction it comes in at, and the turbulent values. Other effects such as a moving floor to replicate the road, or rotating wheels can be added into the analysis to further replicate the real-life application.

Once the turbulence models and boundary conditions are set, the simulation is initialised and solving can begin. Solving is done iteratively, trying to drive down the residual imbalance of each element to as close to zero as possible. The closer the residual values are to zero shows convergence accuracy, with $1e-3$ showing a loose convergence, $1e-5$ showing good convergence, whilst anything above shows very good convergence. It is worth bearing in mind that the more complex a shape being modelled, the harder it is to achieve the higher residual values. (Engineering.com, 2015)

5) CFD validation cases

To validate the accuracy of the CFD studies being performed during this project, a study on the Ahmed body with a 25-degree slope on the rear was undertaken and compared to experimental data collected by W. Meile, Et al. The experimental results collected by Meile stated that the Ahmed body has a coefficient of lift of 0.345, and a coefficient of drag of 0.299. (Meile, et al., 2011) The study comprised of four investigations into the domain, mesh, prism layers, and turbulence models, and the effects these parameters had on the results.

Another study was also performed using a scale model of the current SRCC wing in the University wind tunnel and comparing the results to CFD runs as the angle of attack changes. If the coefficients calculated show similar trends this will help to validate the accuracy of the CFD.

5.1 Ahmed body

The Ahmed body is a bluff body made to vaguely represent the aerodynamics of a small vehicle, created from a rectangular block with rounded front edges, a slant on the rear upper surface, and four small legs. The slant on the rear may be altered to study the effect on the airflow. Figure 5.1 shows the dimensions for the Ahmed body shape. (Ahmed, 1984)

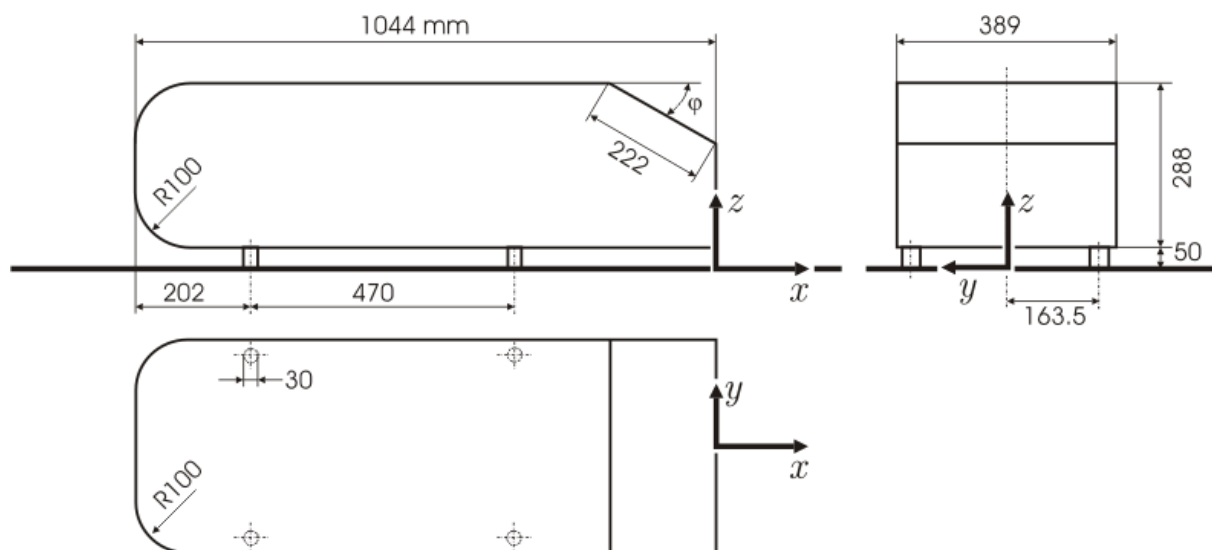


Figure 5.1 - Ahmed body dimensions. The angle of the upper rear slope can be changed to see the effect of the lift and drag. (simscale.com/docs/_images/AhmedBody-diag.png)

5.1.1 Domain study

Run 1

Initially, the effect of the domain size was investigated, the domain represents a virtual wind within which the CFD study takes place. To begin the study, dimensions provided in an instructional video by Tim Tudor were used to set a starting point, these dimensions can be seen in table 5.1.1 (Tudor, 2018)

Table 5.1.1 - Domain size settings for run 1

Geometry	
domain sizes	(m)
front to rear of body	3
rear to rear of body	5
bottom to top	2
floor to bottom	0.02
FD1 depth	1

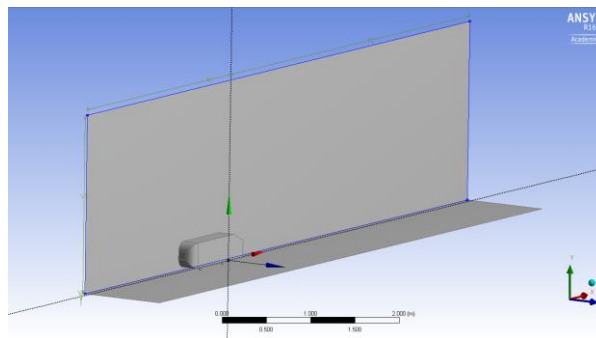


Figure 0.2 - Visualisation of the domain for run 1

Continuing with the setup of this case using the settings from the video, a mesh was also created with the settings shown in table 5.1.2. This mesh was used as a base mesh for all the domain runs.

Table 5.1.2 - Mesh settings for the domain runs

element size	0.5	
shell mesh parameters		
mesh type	all tri	
mesh method	patch independent	
volume mesh parameters		
mesh type	tetra mixed	
mesh method	robust octree	
parts mesh set up	body	feet
	0.01	0.005
density box	0.01	

A K-Epsilon realisable turbulence model was then specified in ANSYS fluent, with non-equilibrium wall functions. A velocity of 20m/s in the X direction was specified. This was also used as a base setting for

all domain runs. The results of the initial run after 1611 iterations gave the following coefficient of lift (C_L) and coefficient of drag (C_d) values:

Table 5.1.3 - Coefficient results for domain run 1, including percentage difference from experimental data, and run time.

results:	C_L	C_d
	0.4018	0.298
% diff	16.50%	1%
runtime	2h10m	

As can be seen, the value for C_d was within 1% accuracy, but the C_L value was not very accurate. Moving forward it was decided to try a larger domain, and see what effect decreasing the size had on the values, runs 2, 3, and 4 domain sizes and their respective results are shown in table 5.1.4 to table 5.1.6

Run 2

Table 5.1.4 - Run 2 domain sizes and results

Geometry		
domain sizes		(m)
front to rear of body		6
rear to rear of body		10
bottom to top		4
floor to bottom		0.001
FD1 depth		2
results:	Cl	Cd
	0.374	0.2918
% diff	8.40%	2.50%
runtime	2h30m	

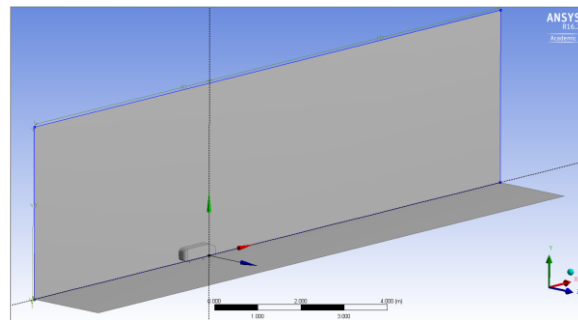


Figure 0.3 - Visualisation of the domain for run 2

Run 3

Table 5.1.5 – Run 3 domain sizes and results

Geometry		
domain sizes		(m)
front to rear of body		2
rear to rear of body		3
bottom to top		1
floor to bottom		0.001
FD1 depth		0.5
results:	Cl	Cd
	0.446	0.389
% diff	29.20%	30.10%
runtime	1h35m	

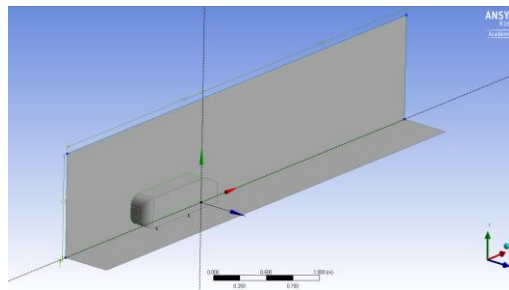


Figure 0.4 - visualisation of the domain for run 3

Run 4

Table 5.1.6 - Run 4 domain sizes and results

Geometry		
domain sizes		(m)
front to rear of body		1.5
rear to rear of body		2
bottom to top		0.5
floor to bottom		0.002
FD1 depth		0.3
results:	Cl	Cd
	-	-
% diff	-	-
runtime	-	

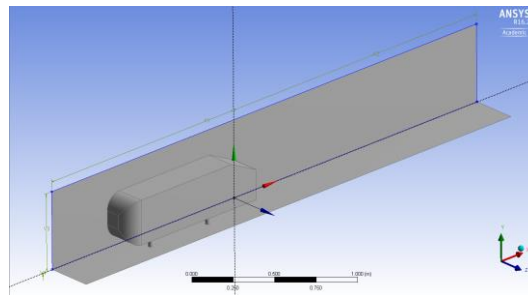


Figure 5.5 - Visualisation of the domain for run 4

The domain study showed that the size of the domain can have a significant effect on the results, with larger domains providing better accuracy, but taking longer to compute. The largest domain from run 2 showed the closest accuracy for both lift and drag coefficients together. Whilst the domains decreased in size accuracy was also shown to decrease. Run 4 was found to be too small to accommodate the mesh element size being used, therefore results could not be recorded for this run.

5.1.2 Mesh study

Run 1 – Finer mesh

The effect of the mesh on the accuracy of the results was then investigated. As a run using the robust octree meshing strategy had already been completed for the baseline, a finer mesh using the same strategy was created by halving the element size in order to see the effect of element size on results.

Table 5.1.7 - Mesh settings used for run 1 of the mesh sweeps

global mesh set up	0
scale factor	0.5
max element size	1
parts mesh setup	
body size	0.05
feet size	0.01

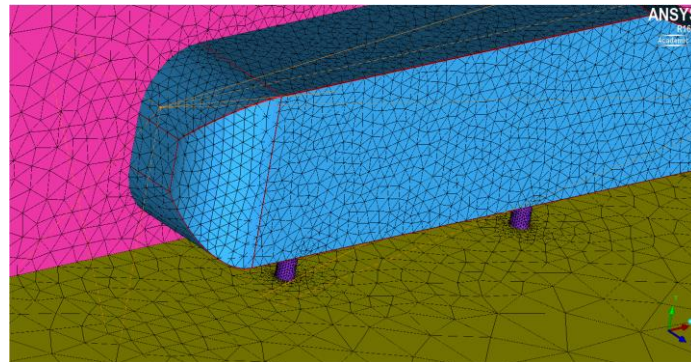


Figure 5.6 - Visualisation of the mesh for run 1 of the mesh sweeps

The run converged after 282 iterations, and as a result of using smaller elements the lift and drag coefficients saw the following results shown in table 5.1.8:

Table 5.1.8 - Results for run 1 of the mesh sweeps

results:	Cl	Cd
	0.33	0.36
% diff	4.54%	20.40%
runtime	5 mins	

The effects of the settings used showed good accuracy for the coefficient of lift but drag coefficient values were still seen to be fairly inaccurate.

Run 2 – Quick Delaunay

Run 6 used a finer mesh for both the volume and surface meshing, whilst also using a quick Delaunay meshing method. The settings and results are shown in table 5.1.9:

Table 5.1.9 - Settings used for run 2 of the mesh sweeps.

global mesh set up	
scale factor	0.5
max element size	0.3
volume mesh paramet	
mesh type	tetra mixed
mesh method	QUICK DELAUNEY
parts mesh setup	
	max size
body	0.03
wheels/feet	0.01

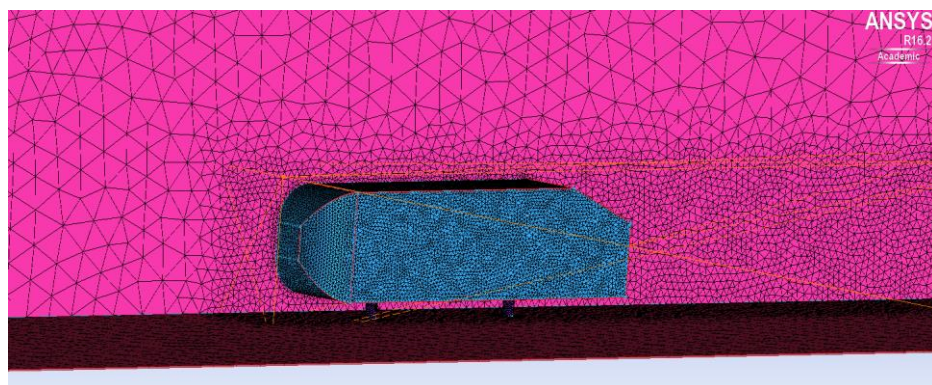


Figure 5.7 - Visualisation of the mesh used for run 2 of the mesh sweeps. a finer mesh was used for the surface of the part.

Run two converged after 349 iterations, with the results shown in table 5.1.10:

Table 5.1.10 - Results from run 2 of the mesh sweeps

results:	Cl	Cd
	0.4176	0.3027
% DIFF	21%	1.24%
runtime	7 mins	

The effect of the settings used in this run saw improved accuracy for drag coefficient but lift coefficient had now become inaccurate.

Run 3 – Advancing front

Run 7 was set up to use an advancing front meshing method, this method obtains an initial triangulation of the tool surfaces, then tetrahedral elements are generated by deleting faces from the generation front. (Löhner & Parikh, 1988). The settings and results are shown in table 5.1.11 below.

Table 5. 1. 11 - settings used for run 3 of the mesh sweeps

global mesh set up	
scale factor	0.5
max element size	0.3
volume mesh parameter	
mesh type	tetra mixed
mesh method	SMOOTH ADVANCING FRONT

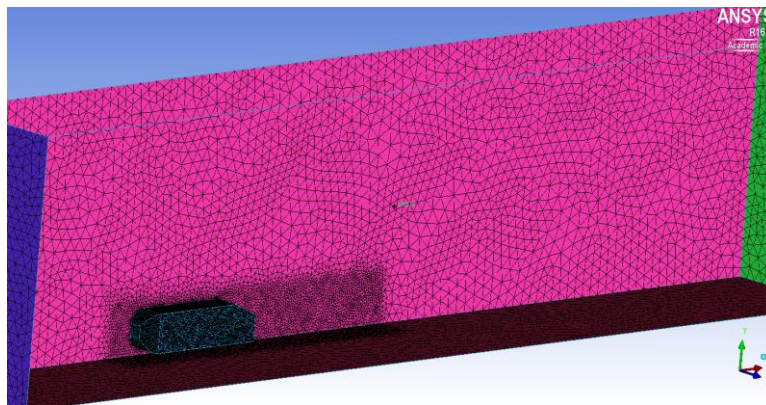


Figure 5.8 – Visualisation of the mesh used in run 3 of the mesh sweeps.

This run converged after 762 iterations with the following results shown in table 5.1.12.

Table 5.1.12 - results for run 3 of the mesh sweeps.

results:	Cl	Cd
	0.3657	0.341
% diff	6%	14%
runtime	10m	

The effect of different meshing strategies was also shown to have significant effects on the results. The advancing front mesh showed reasonable accuracy for lift coefficient, but the drag coefficient was still out by a noticeable amount.

5.1.3 Prism layers

Prism layers are used for modelling the horizontal flow or the boundary layer across the surface of a body in higher detail. The effect of adding prism layers to the model is shown for multiple simulations in tables 5.1.13 – 5.1.15.

Run 1

Table 5.1.13 - Settings and results for run 1 of the prism layer investigation

prism meshing		
growth law	expo	
initial height	0.001	
height ratio	1	
num of layers	3	
total height	0.05	
results:	Cl	Cd
	0.371	0.339
% diff	7.50%	13.30%
runtime	13m	

Run 2

Table 5.1.14 – Settings and results for run 2 of the prism layer investigation with decreased total height.

prism meshing		
growth law	expo	
initial height	0.005	
height ratio	1	
num of layers	3	
total height	0.01	
results:	Cl	Cd
	0.31	0.376
% diff	11.29%	25.76%
runtime	13m	

Run 3

Table 5.1.15 - Settings and results for run 3 of the prism layer investigation with increased number of layers

prism meshing		
growth law	expo	
initial height	0.001	
height ratio	1	
num of layers	5	
total height	0.01	
results:	Cl	Cd
	0.363	0.338
% diff	5.22%	13%
runtime	25m	

Run 3 of the prism layer investigation was found to give the most accurate results to the experimental data. Showing that a higher number of prism layers produced more accurate results. It is worth noting that run 3 also took the longest time to compute due to the extra prism layers.

5.1.4 Turbulence model

Turbulence models are numerical approaches at modelling turbulent airflow. Each model is used for calculating turbulence in different applications. The benefits of the K-Epsilon realisable model are that it can accurately model for rotating flows, boundary layers with strong adverse pressure gradients, recirculation, and separation.

The K-Omega SST models are used for modelling boundary layer flows with pressure gradients, for example, a wing. The sheer stress transport or SST model of the K-Omega model makes use of both the K-Omega model and K-Epsilon models, with the close to wall sections being modelled by the k-omega model, and the outer parts of the boundary layer being modelled by the K-epsilon model. A blending function is used to gradually transition between the models. (ANSYS, 2006)

As a K-Epsilon model had been used throughout the previous runs, three other turbulence models were investigated. The effect on the results can be seen in the tables 5.1.16 - 5.1.18 below.

Table 5.1.16 - Turbulence model and results for run 1 of the turbulence model investigation

model	k-omega standard low-re corrections	
Near wall treatment	non eq wall functions	
results:	Cl	Cd
	0.316	0.363
% diff	9.17%	21.40%

Table 5.1.17 - Model and results for run 2 of the turbulence model investigation

model	K-omega	
	transitional SST	
results:	Cl	Cd
	0.2687	0.358
% diff	28.40%	19.70%

Table 5.1.18 - Model and results for run 3 of the turbulence model investigation

model	large eddy simulation	
subgrid scale model	WALE	
results:	Cl	Cd
	0.1235	0.255
% diff	172.00%	17.20%

As seen above, the effect of turbulence models greatly altered the accuracy of the results, with the large eddy simulation being incredibly inaccurate, and still not showing convergence after 1500 iterations. Considering that a LES usually requires a much higher processing power to run, it was surprising that this simulation would calculate.

5.1.5 Final run

Consolidating knowledge from the previous runs, a final run was set up selecting the best domain size, mesh strategy, prism layers, and turbulence model. It was expected to see that this would achieve results with similar accuracy to those collected by Meile, et al. in their paper - Experiments and numerical simulations on the aerodynamics of the Ahmed body (Meile, et al., 2011). The setting used for the final run are detailed below in table 5.1.19.

Table 5.1.19 – settings used during the final run of the Ahmed body, using the settings that provided the most accurate results all consolidated into one set up.

Geometry		
domain sizes		
front to rear of body		6
rear to rear of body		10
bottom to top		4
wing to bottom		0.001
FD1 depth		2
mesh		
global mesh set up		
scale factor		1
max element size		0.5
shell mesh parameters		
mesh type		all tri
mesh method		patch independent
volume mesh parameter		
mesh type		tetra mixed
mesh method		delauney
scheme		tGlib
parts mesh setup		
	max size	prism layers
body	0.01	5
wheels/feet	0.01	5
road	0.05	3
density mesh/wake box		
size		0.05
fluent		
model		
		viscous laminar
		kepsilon realisable
		non equilibrium wall functions
velocity		
		20m/s
reference values		
compute from		inlet
frontal area		0.05752187
solution controls		
flow courant number		50
momentum		0.25
pressure		0.25
turbulent kinetic		0.5
turbulent dissipation		0.5

The result of consolidating these settings provided results that were within 2% accuracy for coefficient of drag, and within 4% accuracy for coefficient of lift.

Table 5.1.20 – final run results comparison to experimental data from W. Meile. Results were within 4%, proving the settings used to be accurate.

results:	Cl	Cd
experimental data results	0.345	0.299
personal results	0.3570	0.2937
% difference	3.47%	1.80%

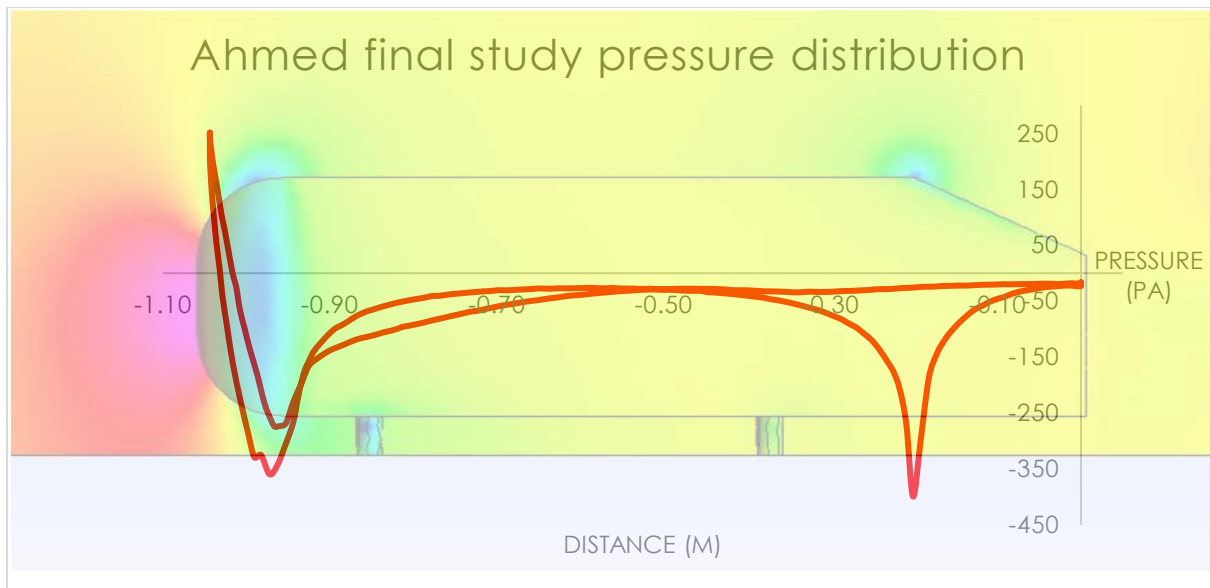


Figure 5.9 – Pressure distribution across the surface of the Ahmed body during the final CFD run.

Looking at the final run pressure distributions across the body, high pressure is seen at the stagnation point at the front, which drops to a low negative pressure as the air travels up and over the curved frontal radius. Pressure along the bottom surface is then seen to gradually return to almost atmospheric pressure before reaching the end of the body.

Air travelling over the top, however, returns to almost atmospheric pressure much faster due to it not being restricted by the floor. As the airflow reaches the kick point of the 25 degree slant, another large drop in pressure is seen.

5.2 Wind tunnel scale wing model

Using a scaled-down CAD model of the current SRCC wing, a 3D model was printed out, painted, and made to fit the university wind tunnel mount. Results were recorded for a sweep of angles of attack from -5 to +20 degrees, at an air velocity of 20m/s. The test was run 5 times so that an average of all runs could be taken to minimise any anomalies.



Figure 5.2.1 (a b c) – 3D printed scale model of the SRCC wing being tested in the wind tunnel at different angles of attack. In the right picture (c) separation can be seen to occur as the thread violently moves around in the turbulent air.

Before the wing was attached, the wing mounts drag was recorded at 20 grams, this was then subtracted for future drag values recorded to gain an accurate reading of the wings drag. The average results were then converted from grams to Newtons, which can be seen in table 5.2.1 below.

Table 5.2.1 – Averaged results of the five wind tunnel tests, conversions to newtons and coefficient values are also calculated.

AOA (deg)	Air speed (m/s)	downforce (g)	drag (g)	lift N	drag N	cl	cd	efficiency
-5	20	-35	94	-0.343238207	0.921839757	-0.08624	0.23161	-0.37234
0	20	2	82	0.019613612	0.804158086	0.004928	0.202043	0.02439
5	20	150	98	1.471020888	0.96106698	0.369591	0.241466	1.530612
10	20	268	124	2.628223987	1.216043934	0.660336	0.305529	2.16129
15	20	363	162	3.55987055	1.58870256	0.89441	0.399158	2.240741
20	20	436	196	4.275767383	1.922133961	1.074278	0.482932	2.22449

The coefficients and efficiency were then calculated using formulas 3, 4, and 5:

$$\text{Coefficient of lift} = \frac{\text{lift}}{0.5 \rho A V^2} \quad (3)$$

$$\text{Coefficient of drag} = \frac{\text{drag}}{0.5 \rho A V^2} \quad (4)$$

$$\text{Efficiency} = \frac{\text{lift}}{\text{drag}} \quad (5)$$

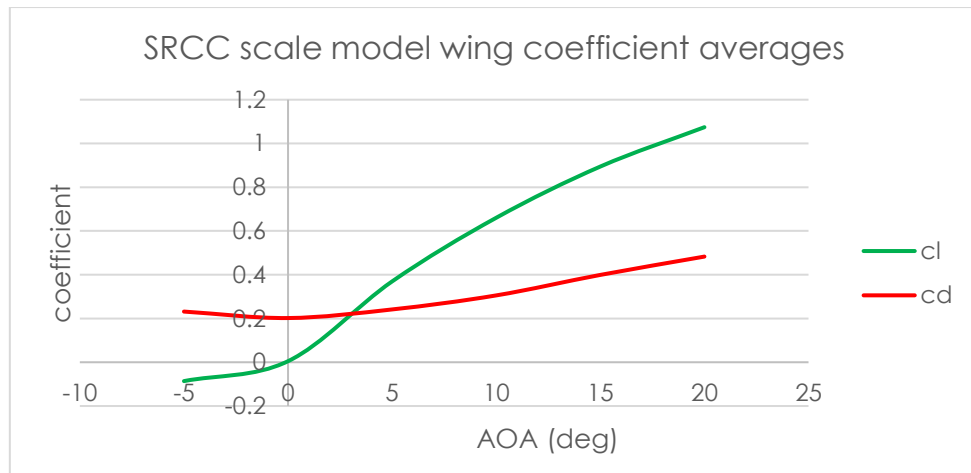


Figure 5.2.2 – Graph showing the averaged coefficients for lift and drag through angles of attack from -5 to +20 degrees.

The wind tunnel data was then compared to data collected running the SRCC in Ansys to see how similar the results were. As can be seen in figure 5.2.3 similar trends were seen between the two cases, but not a perfect correlation is shown. This was due to not recording the atmospheric pressure in the wind tunnel when conducting the experiment or replicating this in the CFD runs. It is also noted that the accuracy of the wind tunnel data is only partially accurate, as the lift and drag gauge the results are taken from increase in 50-gram steps, so trying to accurately record data comes with some estimations. Also, the mounting post of the wind tunnel may have been slightly rubbing on the surround which will have had an effect of the accuracy of the results.

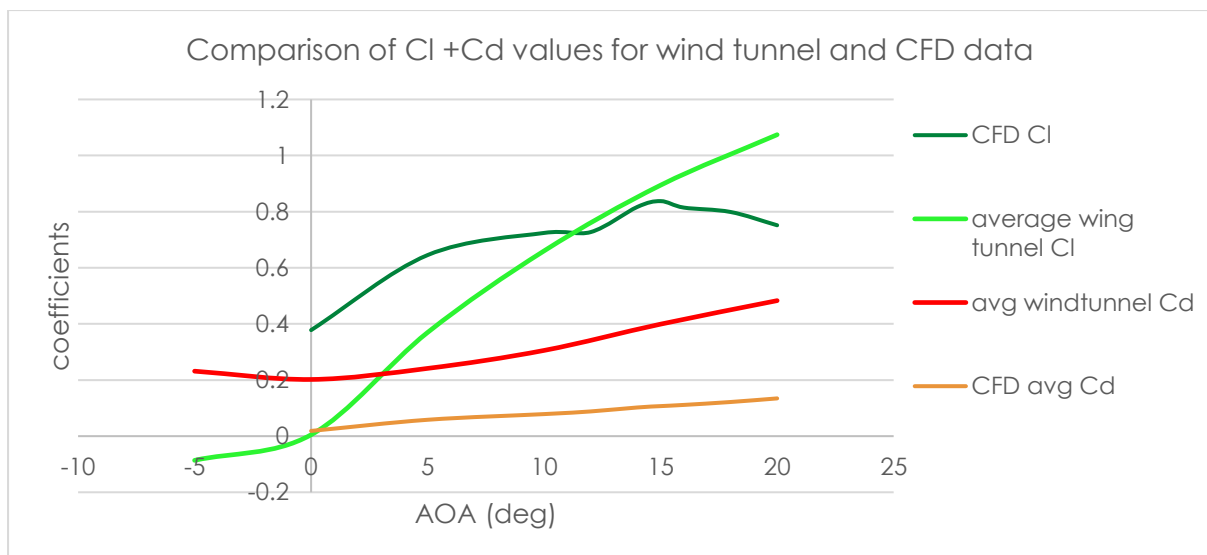


Figure 5.2.3 - Graph comparing the coefficient values for the averaged wind tunnel data and the CFD runs of the SRCC wing in isolation.

6) Modelling and analysis of the SRCC wing

6.1 CAD model

As stated previously, the current wing on the vehicle is an MCR sports2000 championship specific SRCC RA1 wing designed by Elite Carbon Fibre. The profile of the wing was measured and traced to paper, with measurements for chord, camber, and thickness noted so a CAD file of the wing could be created for CFD analysis.

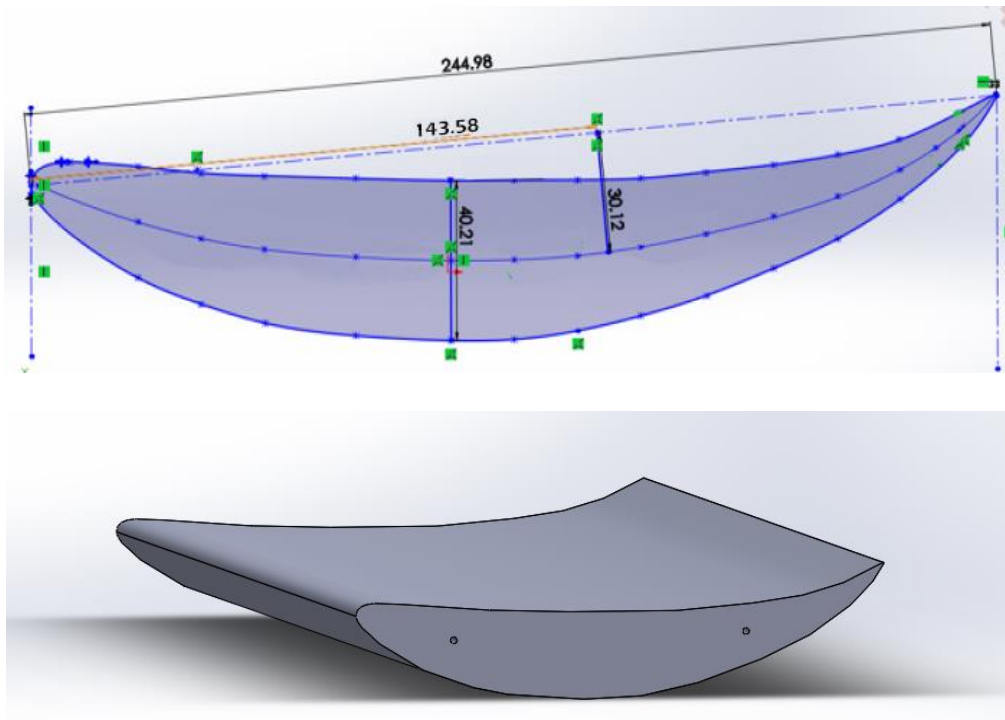


Figure 6.1 - a) chord, thickness, and camber dimensions for the wing profile of the SRCC RA1. b) Finished CAD model of the wing to be used in CFD analysis.

Table 6.1 - Dimensions for the SRCC RA1 wing

Parameter	
Chord	244mm
Camber	12% of chord (30mm)
Camber position	56% of chord (143mm)
Thickness	16% of chord (40mm)
Leading edge radius	4mm
Span length	1554mm

[illegible]

32

6.2 CFD Analysis

The wing was then imported into ANSYS so an angle of attack study could be conducted with the wing in isolation. From this study it was hoped to discover when separation begins to occur on the SRCC wing, as well as how efficient it was. Several different angles of attack from 0 to 20 were studied using the settings shown in table 6.2 in ANSYS:

Table 6.2 - Settings used for the angle of attack study of the SRCC wing

Geometry		
domain sizes		
front to origin		3
front to back dom		7
bottom to top		7
axis to bottom		3.5
FD1 depth		4
max element size		0.3
shell mesh parameter		
mesh type		all tri
mesh method		patch independant
volume mesh parameter		
mesh type		tetra mixed
mesh method		octree
edge criterion		0.2
prism meshing		
growth law		expo
initial height		0
height ratio		1.2
num of layers		3
total height		0
parts mesh setup	max size	prism layers
wing	0.0075	3
density box		
size		0.05
fluent		
model		
		viscous laminar
		k-epsilon
		non equilibrium wall functions
velocity		35m/s
reference value		
compute from		inlet
planar area		projection in y direction
solution control		
flow courant number		50
momentum		0.15
pressure		0.15
turbulent kinetic		0.5
turbulent dissipation		0.5
turb viscosity		0.9

From the study, the results in table 6.3 were gained for the coefficients, and further calculations were made to work out the lift and drag values using the lift and drag equation. (it is worth noting that these coefficient values have been corrected at a later point in the project, and were originally miscalculated to be much higher, leading the author to believe the SRCC wing provided around 400N of downforce more than it does, thus leading the author into thinking the new wing would need to produce 600N to balance the vehicle)

Table 6.3 - SRCC wing coefficients results from the angle of attack study

AOA	Cl	Cd	efficiency	lift N	drag N
0	0.377916072	0.018626214	20.289474	52.51360417	2.588219142
5	0.645293117	0.058395347	11.05042	89.79285603	8.125933814
10	0.724202752	0.078580607	9.2160494	99.61489769	10.80885025
12	0.727313643	0.088144634	8.2513661	99.34140092	12.03938832
14	0.816946733	0.102237778	7.9906542	110.6764268	13.85073411
15	0.837547055	0.107072777	7.8222222	113.0237008	14.44905266
16	0.814490835	0.111282178	7.3191489	109.3723935	14.94332121
18	0.798902279	0.121715112	6.5637066	106.4638478	16.22008035
20	0.751507887	0.13452919	5.5862069	98.85891291	17.69696589

Figure 6.5 shows that the wing was starting to show separation at an angle of attack of 7 degrees and stalling after 15 degrees. Drag was shown to make a steady linear increase as the angle of attack increases. From the calculated lift and drag values in table 6.3, it is seen that the most downforce created by the SRCC wing in isolation is 113N at an angle of attack of 15 degrees.

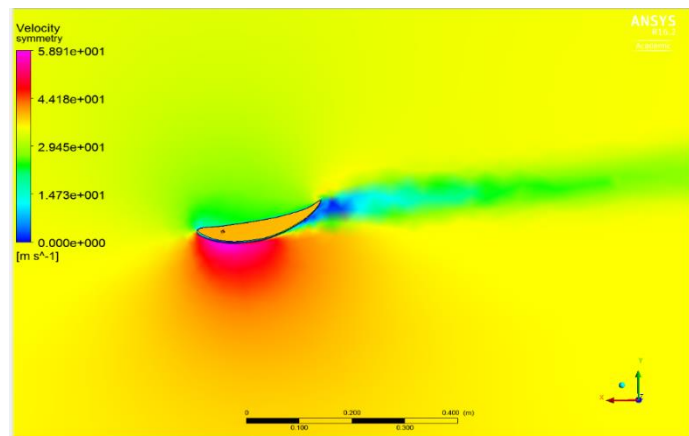


Figure 6.5 – separation beginning to occur at AoA 15 on the SRCC wing. The blue region shows separated airflow recirculating in a vortex.

Feeling some disagreement with the results after AOA 10-12, further research into turbulence models was made and it was discovered that for working on cases with strong pressure gradients in the boundary layer (ANSYS, 2006), which is what would have been seen on the underside of the wing, it would have been more beneficial to use a K-Omega SST turbulence model, especially for the higher angles of attack where a larger amount of turbulence would be expected.

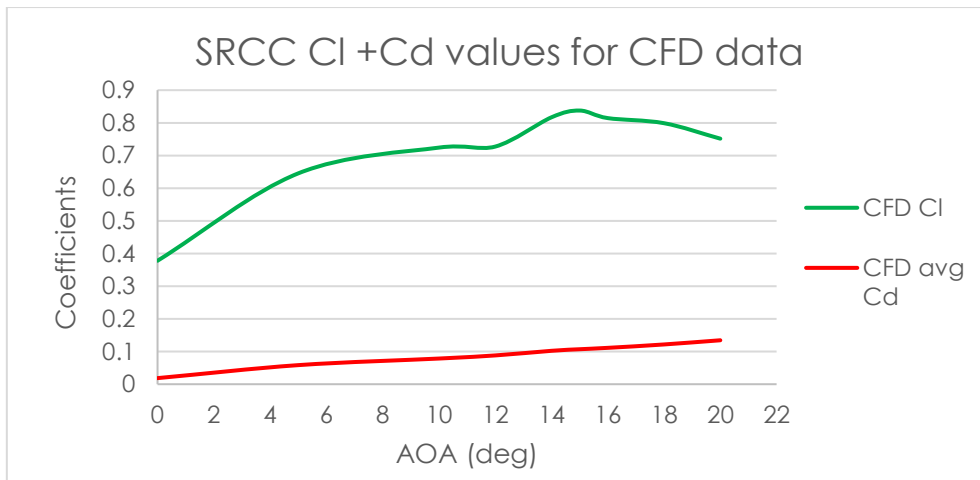


Figure 6.6 - Graph showing the coefficients of lift and drag for the SRCC wing, using the k-epsilon turbulence model

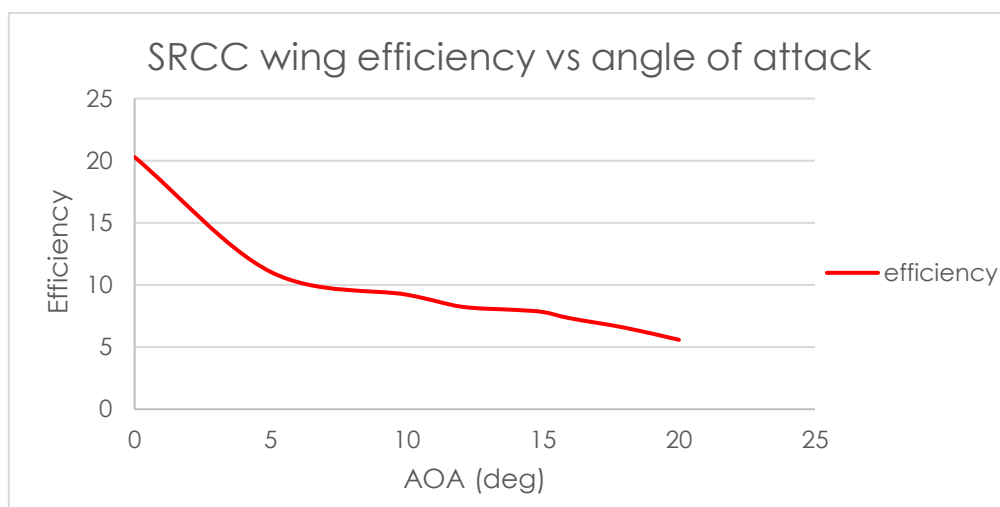


Figure 6.7 - Graph showing the efficiency of the SRCC RA1 wing.

Figure 6.6 shows the efficiency of the wing as the Angle of attack increases. The wing is shown to decrease in efficiency by around 50% between 0 and 5-degrees angle of attack, then dropping almost another 25% as drag increases and lift begins to tail off from 5 to 20 degrees. A comparison of the performance of the SRCC wing against the two new wings can be found in appendix B.

Due to the SRCC RA1 wing being a championship specific wing, there is a distinct lack of research into its performance to compare results with. Even looking for similar profile wings to compare returns little to no results as the shape of the SRCC profile is rather unique.

7) Straight line test data

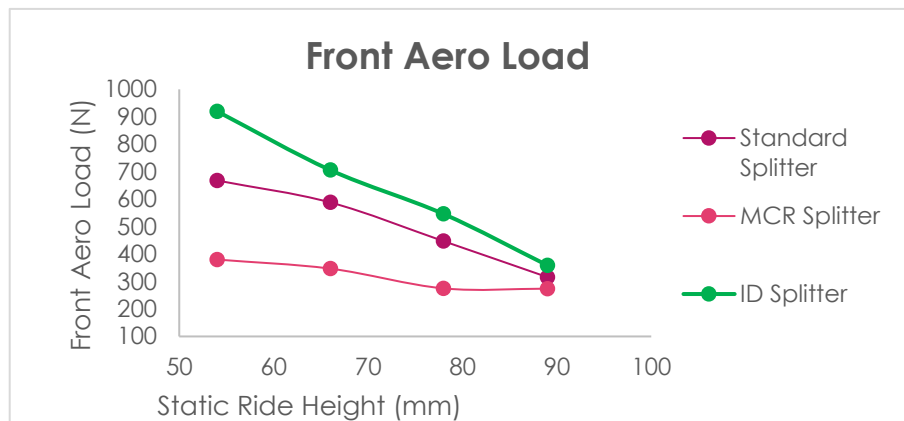


Figure 7.1 - Graph showing the front aero loading on the MCR car at different ride heights with different splitters. (Tudor, 2019)

Aerodynamic load data was collected during a straight-line test at Llandow circuit with three different splitters being tested on the MCR s2000 car at different ride heights. The data in figure 7.1 shows that at a ride height of 54mm with the new Illey Design splitter attached, the car produces 920N of downforce at the front of the car.

While the new splitter shows a major increase in front end downforce, the rear end sees a significant loss due to the shift in centre of pressure (CoP). Total rear downforce with the Illey Design splitter at 54mm ride height is 425N. In order to give the car a neutral aerodynamic balance again, another 500N of rear end downforce is needed.

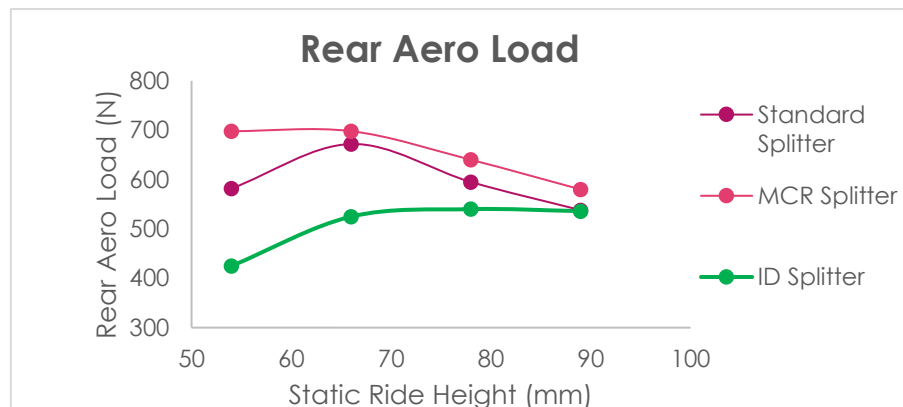


Figure 7.2 – Graph showing the rear aero loading on the MCR car at different ride heights with different splitters. The data in green shows the (Tudor, 2019)

Conferring with Nicholas Thomas, who is in charge designing the new rear diffuser for the car, it was decided to split the rear end downforce so that the wing produced an extra 200N, whilst the diffuser project was to produce 300N making up the total 500N of extra rear end downforce needed to balance the car. (Thomas, 2020). Due to the earlier miscalculation of coefficients that wasn't discovered until near the end of the project, it was thought the new wing would need to produce a total of around 600N of downforce, when in fact it would only need to produce around 300-400N.

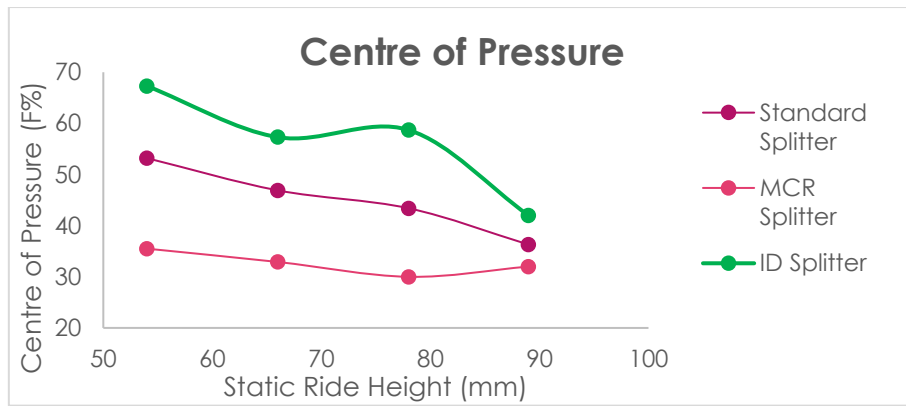


Figure 7.3 – Graph showing centre of pressure on the MCR car at different ride heights for three different splitters. (Tudor, 2019)

Figure 7.3 shows the CoP on the vehicle with the new Illey Design splitter at 67% toward the front of the car. Katz states that rectifying this by increasing rear downforce so that the centre of pressure is at a more central location on the car, ideally just behind the centre of gravity of the vehicle, will give a more stable handling characteristic when cornering and braking, and also increase rear-end grip giving the car a more competitive edge. (Katz, 1995). A new centre of pressure at around 50-45% would be the new target value.

8) Choosing and analysing new wing profiles

Knowing that an extra 200N of extra downforce was needed to be produced, alternate wing profiles were investigated that would be able to produce the required amount of downforce.

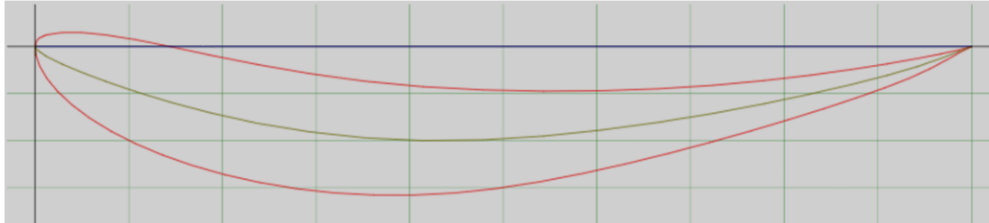


Figure 8.1 - Eppler 423 airfoil profile (Airfoil Tools, 2020)

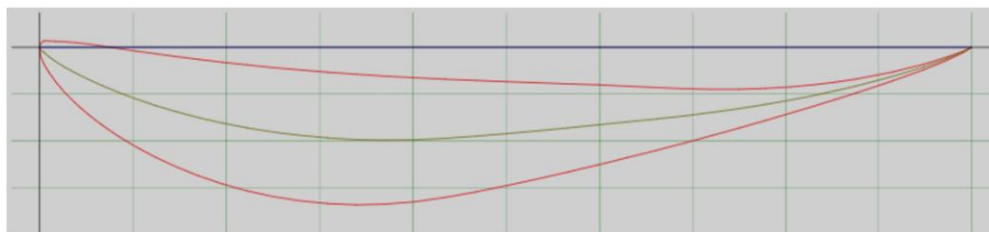


Figure 8.2 Wortmann FX74 CL5 airfoil profile (Airfoil Tools, 2020)

The project aimed to produce a new wing design that would have a high lift to drag ratio. Two wing profiles were selected for this purpose, the Wortmann fx74 CL5, and Eppler e324. These profiles were chosen due to motorsport wings having to work with a low aspect ratio, therefore needing to maximize wing efficiency. Both wing profiles are stated to produce high levels of lift at relatively low Reynolds numbers, and at low angles of attack. The combination of these features makes them ideal for use in motorsport. (Fozan Ur Rab, 2018)

Initially thinking the wing needed to produce around 600N of total downforce, an estimate for the coefficient of lift was calculated using the same span length of 1554mm and three different chord lengths: 300mm, 350mm, and 400mm, giving three different planform areas. The lift equation was then used to calculate an estimate Cl value. It was decided the chord length would be 350mm as an increased chord length would increase the aspect ratio, producing more downforce.

$$300mm \text{ chord } Cl = \frac{2 \times 600}{1.22 \times 35^2 \times 0.4662} = 1.72 \quad (6)$$

$$350mm \text{ chord } Cl = \frac{2 \times 600}{1.22 \times 35^2 \times 0.5439} = 1.476 \quad (7)$$

$$400mm \text{ chord } Cl = \frac{2 \times 600}{1.22 \times 35^2 \times 0.6216} = 1.29 \quad (8)$$

8.1 Eppler e423

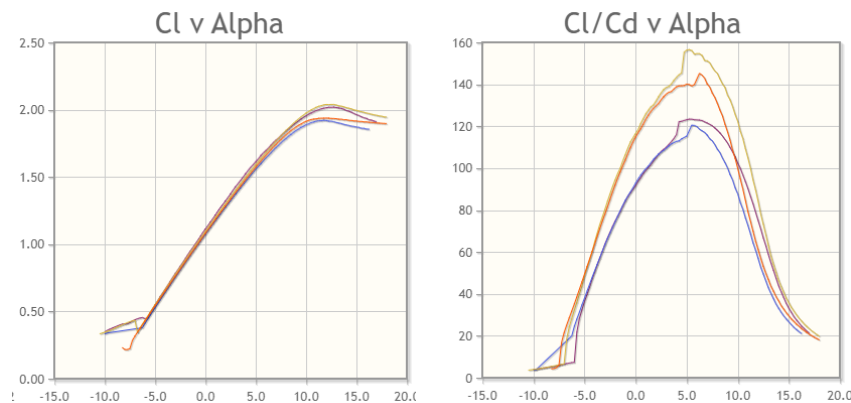


Figure 8.1.1– Graphs taken from airfoil tools website, displaying the coefficients of lift and efficiency of the Eppler e423 profile. The different colour lines represent different Reynolds numbers from 500,000 to 1,000,000. ranging from around 20m/s to 35m/s. (Airfoil tools, 2020)

The graphs in figure 8.1.1 show that even at low angles of attack between 0 – 5, the Eppler e423 profile should already be reaching a coefficient of lift of 1.5, with a peak of around 2.0 at 11-12 degrees angle of attack, the stall characteristics beyond this point are also shown to be relatively shallow. The profile also has a very high-efficiency ratio at low angles of attack making it a good candidate for the project as drag can be kept to a minimum. The graphs were taken from the airfoil tools website and will be used to validate CFD studies.

8.2 Wortmann FX74 CL5

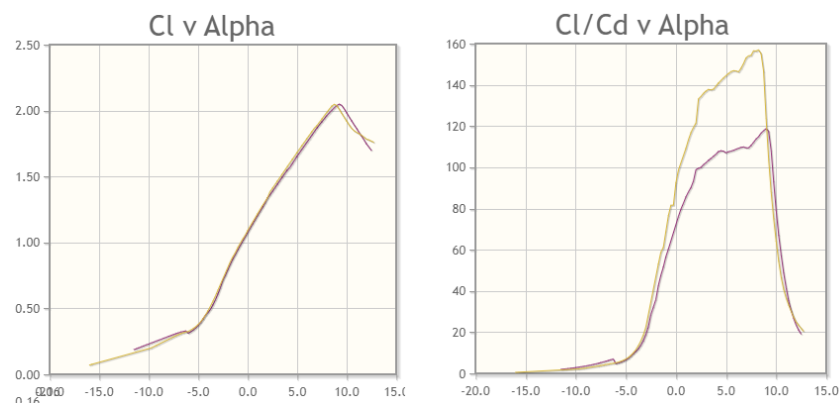


Figure 8.2.1 -Graphs taken from airfoil tools website, displaying the coefficients of lift and efficiency of the Wortmann FX74 CL5 profile. The different colour lines represent different Reynolds numbers from 500,000 to 1,000,000. (Airfoil tools, 2020)

The FX74 profile has a similar coefficient of lift pattern, reaching slightly higher peak values than the E423. The FX74 does, however, stall abruptly at around 9 degrees and lose a significant amount of lift. The efficiency graph also shows similar levels of efficiency to the E423, but a much sharper decrease than the E423 at around 8 - 9 degrees angle of attack. This profile looks to be strong but unstable at high angles of above 9.

8.3 Angle of attack study

3D CAD models of the Eppler e423 and Wortman FX74 wing were created in SolidWorks with a span of 1554mm and chord length of 350mm which was then imported into ANSYS for an angle of attack study. The settings used in ANSYS are shown in table 8.1.

Table 8.1 - Table showing the settings used during angle of attack study on the Eppler and Wortmann wings

Geometry		
domain sizes		
front to origin		5
front to back dom		7
bottom to top		3
axis to bottom		3
FD1 depth		4
Mesh		
scale factor		1
max element size		0.5
hell mesh parameter		
mesh type		all tri
mesh method		patch independant
volume mesh parameter		
mesh type		tetra mixed
mesh method		octree
edge criterion		0.09
prism meshing		
growth law		expo
initial height		0
height ratio		1.2
num of layers		7
total height		0
parts mesh setup	max size	prism layers
wing	0.003	7
density box		
size		0.01
fluent		
model		
		viscous laminar
		K-Omega
		SST
velocity		
		35m/s
reference value		
compute from		inlet
planar area		projection in y direction
solution control		
flow courant number		50
momentum		0.3
pressure		0.3
turbulent kinetic		0.5
turbulent dissipation		0.5
turb viscosity		0.9

The results of the angle of attack study are shown in table 8.2 and figure 8.3.1 below

Eppler E423

Table 8.2 - table showing the results from the angle of attack study of the Eppler e423 wing

aoa	cl	cd	lift	drag	l/d
0	1.1866	0.0967	483.47	39.38	12.277
5	1.860657	0.216419	758.0851	88.1755	8.597458
7	2.090832	0.216419	851.865	88.1755	9.661017
8	2.182535	0.240262	889.2275	97.88975	9.083969
9	2.314588	0.271441	943.0295	110.593	8.527027
10	2.453977	0.287948	999.8205	117.3183	8.522293
11	2.404457	0.322795	979.6448	131.516	7.448864
12	2.420964	0.344804	986.37	140.483	7.021277

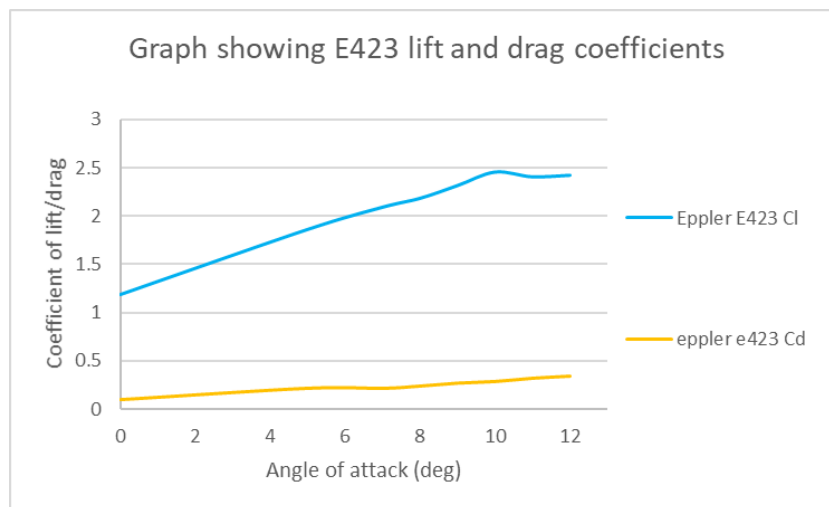


Figure 8.3.1 - Graph showing the coefficient of lift and drag for the Eppler e423 wing over a range of angles of attack

Comparing the graph in figure 8.3.1 to that in figure 8.1.1, it can be seen that both follow the same trend with a steady linear increase in lift before tailing off and beginning to stall at around 10 degrees at a coefficient of lift value of 2.45 which is higher than the data from the airfoil tools website states, but this is a 3D model, whereas figure 8.1.1 is a 2D profile. The stall is relatively controlled, and the lift value does not drop by much between 10 and 12 degrees. As shown in table 8.2, the maximum lift force produced is 999.8N at 10 degrees which is a lot more than is necessary for the project. The drag shows a relatively low increase as the angle of attack changes, ranging from 39N at 0 angle of attack, to 140N recorded at 12 degrees.

However, these studies are conducted with the wing in isolation and results from a study with the wing in situation will most likely be observed to produce lower lift due to air flow disturbance from the body of the vehicle.

Wortmann FX74

Table 8.3 -Table showing the results from the angle of attack study of the Wortmann FX74 CL5 wing

aoa	cl	cd	lift	drag	l/d
0	1.195809	0.124716	487.207	50.813	9.588235
5	2.041312	0.205415	831.6893	83.692	9.9375
7	2.292579	0.249433	934.0625	101.626	9.191176
8	2.4283	0.271441	989.359	110.593	8.945946
9	2.510833	0.298952	1022.985	121.8018	8.398773
10	2.453977	0.287948	999.8205	117.3183	8.522293
11	2.503496	0.359476	1019.996	146.461	6.964286

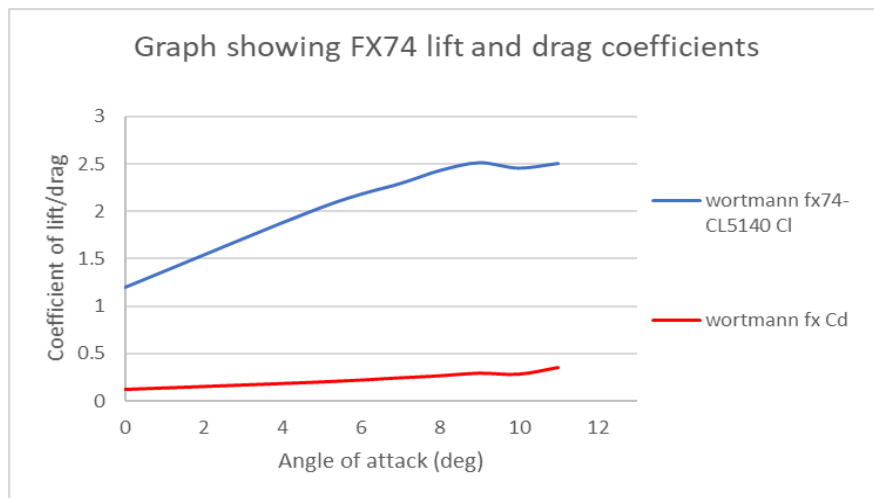


Figure 8.3.4 - Graph showing the coefficient of lift and drag for the Wortmann FX75 wing over a range of angles of attack

The graph in figure 8.2.1 shows the profile was expected to stall dramatically at around 9 degrees, with a coefficient of just over 2.0. Looking at the results from the angle of attack study on the Wortmann FX74 wing in figure 8.3.4, the coefficient of lift follows an almost identical trend to the Eppler e423, but peaks higher than expected at a maximum of 2.51. The FX74 does begin to separate earlier than the e423 at an angle of attack of 9 degrees, but not as dramatically as predicted. Maximum lift force values are seen at 9 degrees at 1023N, which is also too powerful for what the project requires.

The drag coefficient doesn't see much of an increase as the angle of attack increases, but when comparing the drag force figures to those of the e423, it can be seen the FX74 produces slightly higher drag values, with maximum drag force of 146N at 11 degrees.

Both the Eppler E423 and the Wortmann FX74 are overly powerful for what the project requires, and due to an error when calculating the SRCC wings coefficients and force values, which was discovered after the CFD runs of all three wings were completed and the data was compiled into graphs, It was initially thought that a total of around 600N of downforce was required to balance the vehicle aerodynamically when only 300-400N was needed. With hindsight, a chord length of 350mm was to large, and reducing the chord lengths of the two wings would have had the effect of reducing the amount of downforce seen.

Looking at the results of the angle of attack study, and using the result calculated in equation 7, a coefficient value of around 1.4 at 4 degrees angle of attack would have produced around 600N of downforce for the Eppler e423. And the same downforce would have been seen at an angle of attack of around 1.5 degrees for the Wortmann FX74. However, as mentioned previously, this is subject to change once the wing is studied in situation on the car due to the effect the car's body would have on the onset airflow. It was also decided at this point that the Eppler e423 was the more suitable wing to carry forward as the airflow showed better overall attachment, and stall less suddenly.

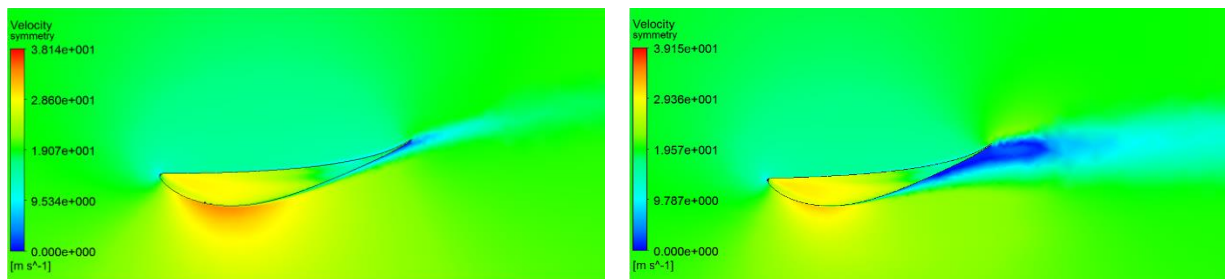


Figure 8.3.5 – Post processing images of the Wortmann FX74 starting to show separation between 8 degrees on the left, and 9 degrees on the right

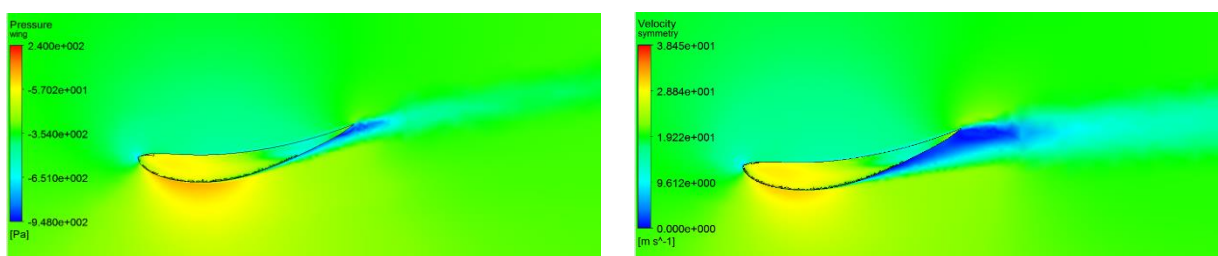


Figure 8.3.6 – Post processing images of the Eppler E423 starting to show separation between 9 degrees on the left, and 10 degrees on the right

9) Wing placement study

Moving forward, the next step of the project was to identify several positions to test the overall effectiveness of the e423 wing within the confines of the regulation zone. A matrix of positions, which can be seen in figure 9.1, was measured up by splitting the height and width of the regulation box in half vertically and horizontally, then the upper two sections horizontally in half again. Positions were taken from a zero point at the corner of the central axis of the rear wheel and the bottom of the tyre, the coordinates can be seen in table 9.1.

Table 9.1 - Table showing the location coordinates for the position of the rear wing

Position	X (mm)	Y (mm)
1	0	900
2	375	900
3	750	900
4	375	675
5	750	675
6	750	450

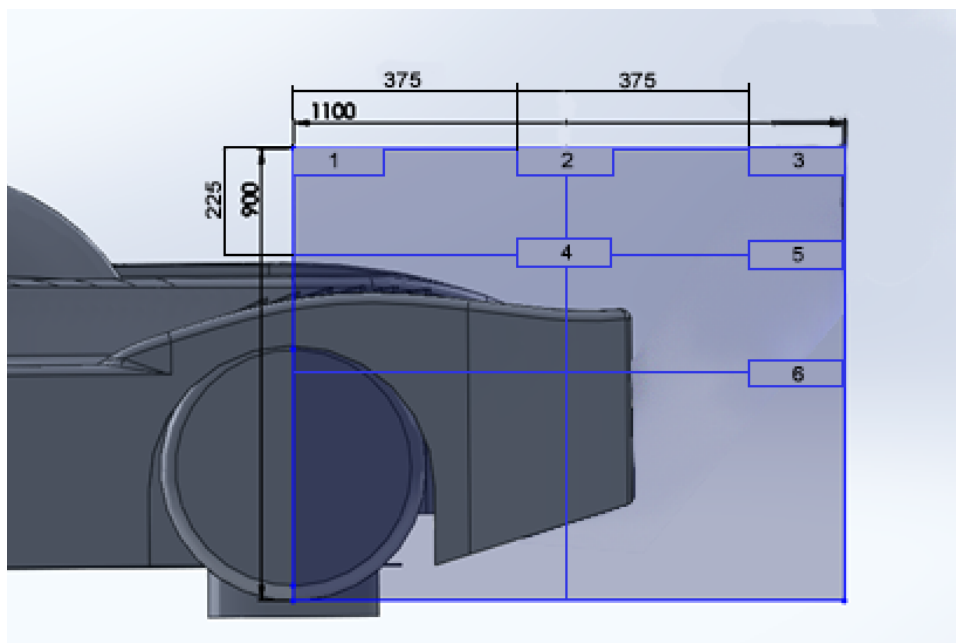


Figure 9.1 - Picture showing the proposed positions for wing placement within the rule box coordinates are taken from the bottom left corner

Once the necessary CAD files with the wings in position on the car were created, they were each loaded into ANSYS to analyse how the shape of the body affected wing performance. Several attempts to run position 1 were made, each resulting in either floating-point errors when attempting to calculate, or “partition cell count = 0” when loading the mesh into Fluent. Initially thinking this was due to the bad orthogonal quality seen around the louvres on the car, the CAD file of the car was modified, and the louvres removed and smoothed over. Doing so improved the orthogonal quality and skewness, but floating-point errors were still occurring.

9.1 Floating-point errors

To try and understand what was causing the issue, research was conducted into floating-point errors, as the usual approach of re-meshing and trying again was showing no progress. Floating-point errors are not, as initially thought, a tetrahedral element within the mesh that has become unattached from those around it, but is a problem within the maths used by the software.

As a 32bit computer can only store 23 significant figures, numbers after this point are then rounded or truncated. As computers are binary machines, they use base 2 to calculate, whereas human minds tend to work in base 10. So where a human mind sees hundreds, tens, and units, and beyond the decimal tenths, hundredths, thousandths, a computer that uses base 2 will see fours, twos, and ones, then halves, quarters, and eighths and so on. Some calculations will produce numbers that cannot be represented properly in base 2, for example 0.1 would be represented in base 2 as 0.0001100110011 recurring. Any recurring numbers would be cut off after 23 decimal places, meaning the true number is represented approximately, giving a rounding error. (Goldberg, 1991) These rounding errors and approximations can cut off extremely small numbers resulting in the approximation returning as zero, and when used in calculations within ANSYS can lead to divergence as the calculations may attempt to divide by zero.

10) COVID-19 pandemic impact

Unfortunately, due to the outbreak of the COVID-19 pandemic within the UK, the decision was taken by UWTSD to suspend all further classes and disallow access to the buildings on campus on the 16th March. This decision had an impact on the project from this point onwards, as the author could no longer use the university computers to set up and run any further CFD studies or access the wind tunnel for validation on new wing models. From this point onward most of what is detailed is work the author had expected to carry out and results that were hoped to have been observed compared with literature had the outbreak not disrupted society.

11) Future planned work

11.1 Wing placements continued

The effect of the wing's location can have a large impact on the aerodynamic characteristics of the vehicle. Simon McBeath conducted a similar study on a Stohr WF1 which has a similar shape to that seen on the MCR sports 2000. The first study was a height study with the leading edge of the wing overlapping the rear deck of the body by 50mm, McBeath refers to this point as $X = 0$, starting at $Y = 900\text{mm}$ from the ground, as per Hillclimb regulations. The results showed that as the height of the wing was reduced, the downforce generated by the wing was also reduced, dropping by around 1250N as the wing was reduced in height by 500mm. (fig 11.1)

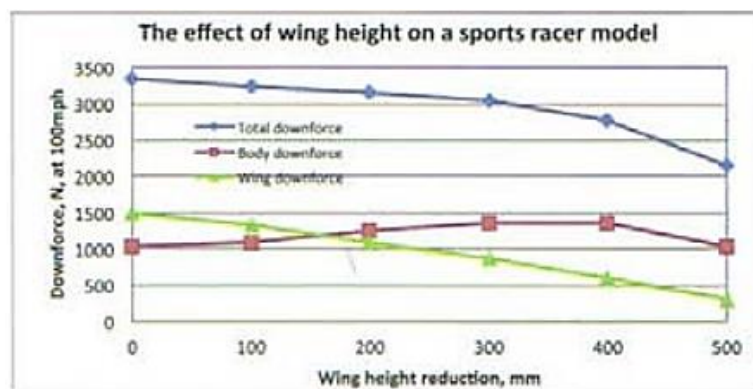


Figure 11.1 - Graph showing the results of Simon McBeath's wing height study at $X = 0\text{mm}$. wing downforce can be seen in green. (McBeath, 2014)

McBeath then went on to repeat the same height study with the wing moved 150mm and 300mm further back in the X direction and found that with the wing at $X=150\text{mm}$ overall car downforce stayed relatively unaffected as the wing was lowered by up to 300mm, before decreasing as the wing was lowered further. Results are shown by the purple line in figure 11.2. the position of the wing also had some effect on drag, with the further back the wing was positioned, less drag was observed. Although, this was only by a relatively small amount. Centre of pressure was also seen to move toward the rear as the wing was positioned further back. (McBeath, 2014)

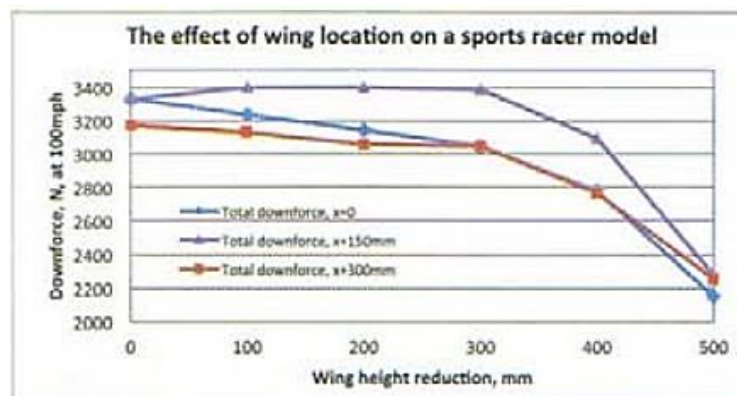


Figure 11.2 - Graph showing the effect of moving the wing rearward in 150mm steps and repeating the height study. $X = 150\text{mm}$ showed the best downforce even as height was lowered. (McBeath, 2014)

had the placement study been undertaken, similar trends to those seen in McBeath's study would also have been expected to be seen on the MCR car. Position 1 on the MCR is, however, a lot further forward than any of McBeath's investigations. It would be assumed that although the wing is in a high location, it would produce less downforce as air would be disrupted coming over the roll hoop, airbox, and the drivers head, leading to turbulent conditions. Also, according to McBeath's study, position 1 would not have the desired effect of bringing the centre of pressure far enough back. (McBeath, 2014). According to the McBeath investigation, an ideal location would be seen somewhere between positions 3 and 5, as per figure 11.3.

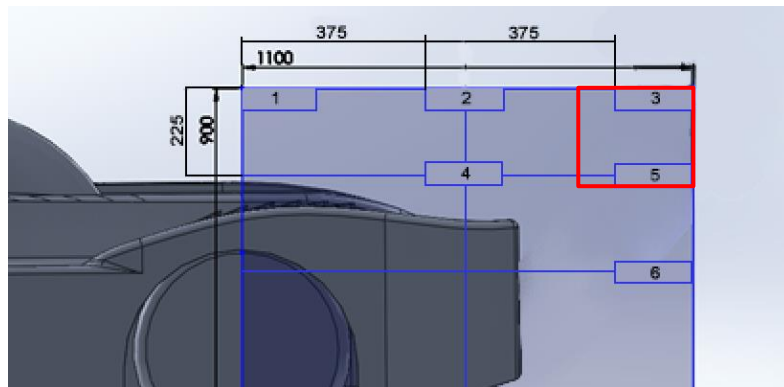


Figure 11.3 - Ideal location for wing placement according to McBeath study.

11.2 Designing for onset flow

Once an ideal position was chosen, the next step of the project would be to design the wing in accordance with the onset flow by adding a spanwise angle of attack twist. This is seen in GT and touring cars regularly to account for airflow over the roof approaching the wing at different angles to the air approaching from around the side of the vehicle.

Earlier in the year, a brief run was conducted of the MCR car with no wing or mounts attached. Using post-processing, data was gathered detailing the X and Y velocities of the airflow at several points within the regulation zone. Using basic trigonometry, the angle of the onset flow at these locations was calculated. Table 11.1 shows the data at that was calculated for wing position 3.

Table 11.1 - Table of X and Y air velocities at position 3. Trigonometric calculations were then used to calculate onset flow. Values in red are to be ignored as these extend past the regulation zone.

location Z [m]	Velocity X	Velocity Y	tan (x/y)	onset angle
-1.00E+00	3.55E+01	-1.04E+00	-2.93E-02	-1.6780443
-7.78E-01	3.53E+01	-2.05E+00	-0.05807	-3.3236423
-5.56E-01	3.49E+01	-3.21E+00	-0.09198	-5.2551126
-3.33E-01	3.37E+01	-3.65E+00	-0.10831	-6.1815298
-1.11E-01	2.99E+01	-2.60E+00	-0.08696	-4.9697407
1.11E-01	2.91E+01	-2.01E+00	-0.06907	-3.9512677
3.33E-01	3.23E+01	-2.88E+00	-0.08916	-5.0952515
5.56E-01	3.50E+01	-2.92E+00	-0.08343	-4.7690608
7.78E-01	3.53E+01	-1.95E+00	-0.05524	-3.1618507
1.00E+00	3.55E+01	-9.49E-01	-0.02673	-1.5312887
		avg	-0.08028	-4.588432

The data in table 11.1 split the wing into 8 spanwise sections 222mm apart in the Z-axis. the wing can either be twisted in 8 locations to account for the different airflow, or the wing can be adjusted to a suitable angle of attack in accordance with the average airflow angle. The E423 wing was shown to produce around 400N at AoA 4, making 0.58 degrees AoA the most appropriate for the average onset flow. However, if the wing were designed to twist for each section so that every section was at 4 degrees AoA to the onset flow, this would maximise the potential of the wing, giving it the optimised shape that can be seen in figure 11.2.1.

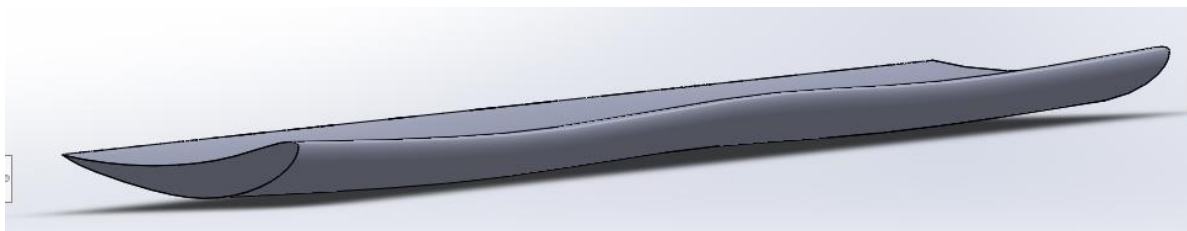


Figure 11.2.1 - Eppler E423 wing designed to suit onset flow. the wing has a spanwise twist that maximises efficiency.

The twist angle of each section was calculated simply by adding 4 degrees to the onset flow angle. The wing can then be run at an overall AoA of 0, and each section of the wing will be producing the correct amount of downforce, showing an even low-pressure distribution across the lower surface of the wing. (Marques, 2013)

Table 11.2 - Table showing the resultant AoA across the span, this was used to design the twist so each part of the wing sits at 4 degrees AoA to the onset flow.

Z axis	onset angle	resulting AOA
-7.78E-01	-3.32	0.7
-5.56E-01	-5.26	-1.3
-3.33E-01	-6.18	-2.2
-1.11E-01	-4.97	-1.0
1.11E-01	-3.95	0.0
3.33E-01	-5.10	-1.1
5.56E-01	-4.77	-0.8
7.78E-01	-3.16	0.8

11.3 Endplates

the final piece of work that was planned for the project, was to investigate the effect of the size of wing endplates. Standingford states that endplates are used to contain high and low-pressure regions on the upper and lower surfaces of the wing, stopping the high pressure leaking over the edges of the wing and creating strong vortices that result in more drag, effectively reducing the aspect ratio of the wing. (Standingford, 1997)

For this study, the author wished to investigate 3 endplate sizes, and the effect they each had on lift force produced. It was expected that as endplate size increased, higher lift and less drag would be observed, as more pressure gradients were contained, and vortices minimised.

Another study by McBeath looked into the effects of wing endplate depth. A study was set up with a wing profile with a chord length of 300mm and span of 1380mm, at 44m/s. Several endplate sizes ranging from 0-600mm were investigated, measured in depth from the lowest part of the wing profile. An increase in downforce was noticed, but a strange detail happened where the downforce spiked at an endplate depth of 125mm. Figure 11.3.1 displays McBeath's results.

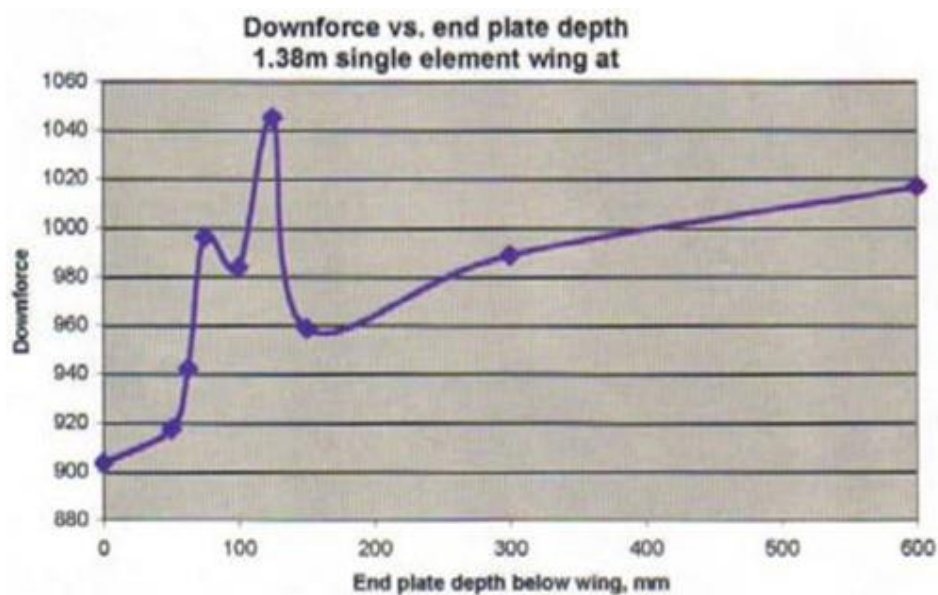


Figure11.3.1 - Graph displaying McBeaths results of endplate depth against downforce. An increase in downforce is seen as depth increases, but a large spike in downforce is seen between 75mm and 125mm. (McBeath, 2008)

Further analysis in post-processing shows that the spike in downforce is due to high pressure leaking underneath the endplate and creating a vortex with an upwash at such an angle that it helps to re-energise the local flow on the low-pressure surface of the wing, and increase velocity. As endplate size was increased, drag was observed to increase very slightly as the frontal area was also increased. (McBeath, 2008)

Whilst this may be true for simple square endplates, the shape of an endplate, along with its positioning can massively influence the flow around it. Endplate design can also be adapted to account for motorsports such as drifting or rally where high yaw angles are seen when cornering, allowing air to bleed into either the high or low-pressure sides with cut-outs or slots in the endplates. Although this warrants enough work for another full study in itself. (Gogel & Sakurai, 2006)

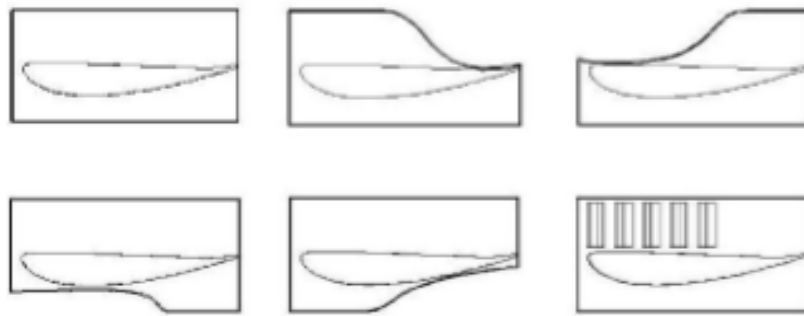


Figure 11.3.2 - Examples of different shapes or slots cut out of endplates to improve performance under high yaw manoeuvres.

12) Conclusion

During the course of this aerodynamics project the author was tasked with designing a new wing to relocate the centre of pressure on an MCR Sports2000 for use in UK Hillclimb championship. The performance of three wings was investigated using ANSYS computational fluid dynamics software, which was validated with experimental data on the Ahmed body, and a scale model wind tunnel test.

Straight line test data gathered from the car showed the splitter was creating 900N of front-end downforce. The current SRCC wing was found to be inefficient to provide enough rear end downforce to balance the car aerodynamically and an extra 500N was needed. The Wortmann FX74 CL5 and the Eppler E423 were the two candidate profiles chosen to replace the SRCC wing and after an angle of attack study in isolation between 0-12 degrees it was discovered that both wings with a chord length of 350mm were too powerful for what the project required. Both producing more than the required 400N of downforce at 0 degrees angle of attack. Although, due to a miscalculation when calculating coefficients which was not noticed until late on, it was initially thought the new wing to be chosen would need to produce 600N of downforce, and at low angles of attack, both of the two candidate wings were able to produce 600N of downforce, and more. Had the author had more time to investigate another profile, one with less camber, and a smaller chord length would have been a better choice for this application as less downforce would have been produced.

Although the project aim was not met, a lot was discovered about wing theory, wing geometry, and the basics of computational fluid dynamics software. Due to the COVID-19 outbreak in March, the project was cut short and access to CFD software was restricted. However, further investigations into wing performance such as the effect of position of the wing in relation to the car, endplate depth, and designing to suit onset airflow were investigated via similar experiments and literature, and it was found that these factors play an important part in wing performance.

References

Ahmed, S. R., 1984. *Some Salient Features Of The Time Averaged Ground Vehicle Wake*, Detroit: SAE.

Airfoil tools, 2020. *E423 (e423-il)*. [Online]

Available at: <http://airfoiltools.com/airfoil/details?airfoil=e423-il>

[Accessed 30 3 2020].

Airfoil Tools, 2020. *FX74_CL5_140 (fx74cl5140-il)*. [Online]

Available at: <http://airfoiltools.com/airfoil/details?airfoil=fx74cl5140-il>

[Accessed 20 3 2020].

Airfoil Tools, 2020. *FX74_CL5_140 (fx74cl5140-il)*. [Online]

Available at: <http://airfoiltools.com/airfoil/details?airfoil=fx74cl5140-il>

[Accessed 20 3 2020].

Anderson, J. D., 1985. Fundamentals of inviscid, incompressible flow. In: *Fundamentals of Aerodynamics*. Singapore: B & Jo enterprise PTE LTD, pp. 164-166.

ANSYS, 2006. Introductory fluent training.. In: *Modeling turbulent flows*. s.l.:Ansys, pp. 12-13.

ANSYS, 2006. *southampton.ac.uk*. [Online]

Available at:

http://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Turbulence_Notes_Fluent-v6.3.06.pdf

[Accessed 16 1 2020].

Babinsky, H., 2003. How do wings work. *Physics Education*, 38(6), pp. 497-498.

Baehmann, P. W. S. S. M. G. K. a. Y. M., 1987. Robust, geometrically based, automatic two-dimensional mesh generation. *International Journal for Numerical Methods in Engineering*, 24(6), pp. 1043-1077..

Chaparral cars, 2005. *Chaparral cars*. [Online]

Available at: <http://www.chaparralcars.com/>

[Accessed April 2020].

Engineering.com, 2015. *3 Criteria for Assessing CFD Convergence*. [Online]

Available at:

<https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/9296/3-Criteria->

[for-Assessing-CFD-Convergence.aspx](#)

[Accessed 14 1 2020].

Fefferman, C. L., 2017. *Existence and smoothness of the Navier-Stokes equations*. [Online]

Available at: <http://www.claymath.org/sites/default/files/navierstokes.pdf>

[Accessed 11 1 2020].

Fozan Ur Rab, M., 2018. *Comparative analysis of high lift airfoils for motorsport applications*, Pakistan: WIT press.

Gogel, D. & Sakurai, H., 2006. *The Effects of End Plates on Downforce in Yaw*, Dearborn: SAE.

Goldberg, D., 1991. What Every Computer Scientist Should Know About Floating-Point Arithmetic. *Computing Surveys*.

Hepperle, M., 2018. *aerotoools*. [Online]

Available at: https://www.mh-aerotoools.de/airfoils/jf_multi.htm

[Accessed 19 02 2020].

Katz, J., 1995. aerodynamics and vehicle performance. In: *Race car aerodynamics*. s.l.:bentley publishers, p. 156.

Kersale, D. E., Unknown. *Fluid dynamics lecture notes - lift forces*. [Online]

Available at: http://www1.maths.leeds.ac.uk/~kersale/2620/Notes/chapter_8.pdf

[Accessed 13 11 2019].

L.N.V.M. Society Group of Institutes, 2007. Airfoils and Wings (subsonic). In: *Theory of Flight*. New Dheli: L.N.V.M. Society Group of Institutes, p. 27.

Lanfrit, M., 2005. *Best practice guidelines for handling automotive external aerodynamics with FLUENT*, s.l.: Fluent.

Löhner, R. & Parikh, P., 1988. Generation of three-dimensional unstructured grids by the advancing-front method. *International journal for numerical methods in fluids*, 8(10), p. 1135.

Marques, P., 2013. *Wing design with a twist: Optimised geometric twist of the Grand Touring sports car wing*, s.l.: International Journal of Unmanned Systems Engineering.

Matthews, R., 2012. *The secret to airplane flight? No one really knows*. [Online]

Available at: <https://www.thenational.ae/uae/science/the-secret-to-airplane-flight-no-one-really-knows-1.358230>

McBeath, S., 2008. The end is nigh. *Racecar Engineering*, august, pp. 67-72.

McBeath, S., 2014. How to wing it. *Racecar Engineering*, december, pp. 57-62.

Meile, W. et al., 2011. Experiments and numerical simulations on the aerodynamics of the Ahmed body. *CFDLletters*, 3(1), pp. 32-39.

Motorsport UK, 2019. *Motorsport UK yearbook 2019*. 63 ed. Staines: the royal automobile club motorsports association ltd.

NASA, 2015. *Euler equations*. [Online]

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

Reynolds, O., 1895. On the Dynamical Theory of Incompressible Viscous Fluids and the Determination of the Criterion. *Philosophical Transactions of the Royal Society of London. A*, Volume 186.

Schöberl, J., 1997. NETGEN An advancing front 2D/3D-mesh generator based on abstract rules.. *Computing and visualization in science*, 1(1), pp. 41-52.

Shewchuk, J., 1996. Triangle: Engineering a 2D quality mesh generator and Delaunay triangulator. In: *Applied computational geometry towards geometric engineering*. s.l.:s.n., pp. 203-222.

SimWiki, 2016. *What are the Navier-Stokes Equations*. [Online]

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

[Accessed 15 11 2019].

Singh, N., Sasikumar, K. & Janardhan, P., 2018. *Clearing certain misconceptions in the common explanations of the aerodynamic lift.*, Ahmedabad: Physical Research Laboratory.

Sports Racing Car Club, 2019. In: *SRCC SPORTS 2000 DURATEC CHAMPIONSHIP SPORTING & TECHNICAL REGULATIONS 2019*. s.l.:s.n., p. 6.

Standingford, D. W. F., 1997. *Optimal lifting surfaces including endplates, ground effect, and thickness*, Adelaide : University of Adelaide.

Swanson, W. M., 1961. *The Magnus Effect: A Summary of Investigations to Date*, St. Louis: Washington University.

Thomas, N., 2020. *diffuser dissertation project*. s.l.:s.n.

Tudor, T., 2018. *CFD External Aerodynamics 2 Domain and Geometry*. Swansea: UWTSD.

Tudor, T., 2019. *Llandow test data*, s.l.: s.n.

Woodford, C., 2012. *Aerodynamics: an introduction*. [Online]

Available at: <https://www.explainthatstuff.com/aerodynamics.html>

[Accessed April 2020].

Appendix A

NACA 0012 scale wind tunnel pressure distribution data

A small experiment was undertaken to see how the pressure across the top and bottom surfaces of a symmetrical NACA 0012 profile wing differs. The wing had 11 pressure taps along its surface, and was adjusted in increments of 2 degrees, between 0- and 10-degrees angle of attack. Air speed was set to 6m/s and ambient pressure was taken, along with pressure at each tap. The results were collected into a spread sheet and the following data produced.

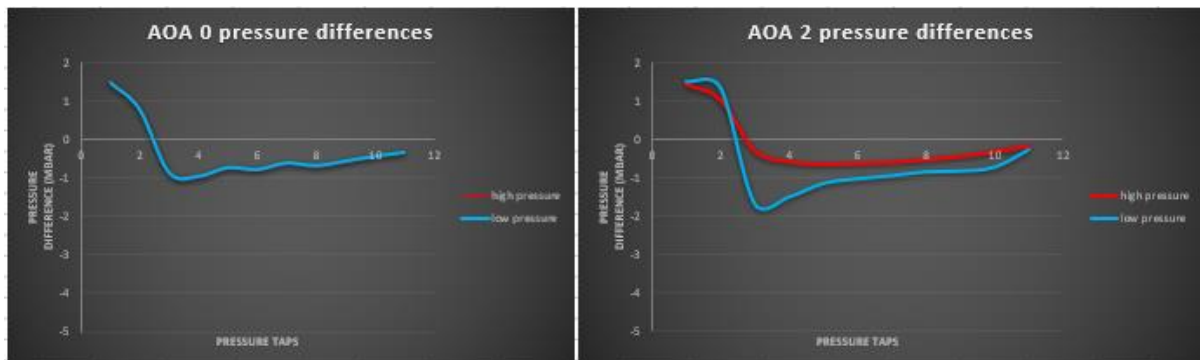


Figure 13.1 - pressure distribution on the NACA 0012 at AoA 0 and 2

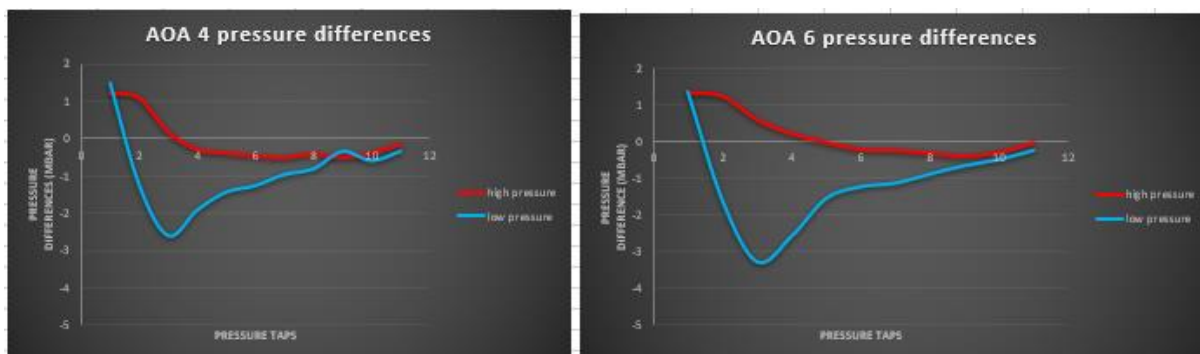


Figure 13.2 - pressure distribution on the NACA 0012 at AoA 4 and 6

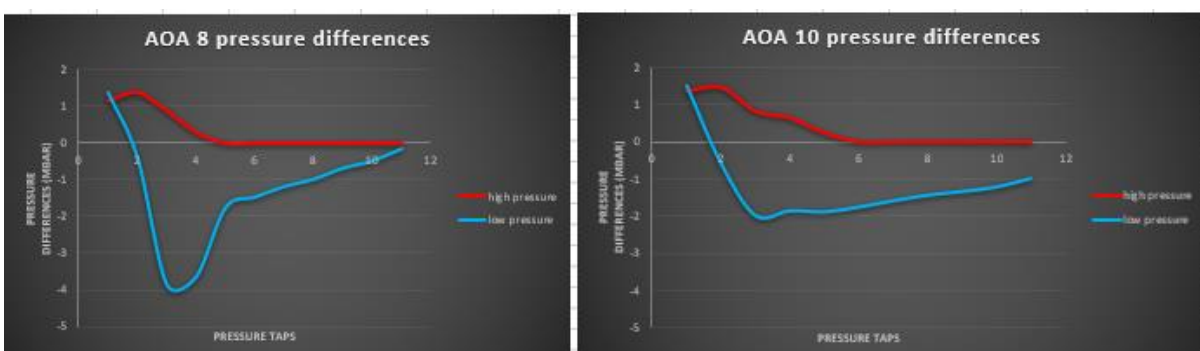


Figure 13.3 - pressure distribution on the NACA 0012 at AoA 8 and 10

The graphs show the pressure distribution on each side of the wing at 0 angle of attack is identical, which is to be expected, as the profile is symmetrical. As the angle of attack increases, the high-

pressure side of the wing holds a higher pressure across its chord for longer, before dropping to close to ambient at around pressure tap 6 for the higher angles of attack.

The low pressure is seen to drop sharply before making a gradual return to ambient pressure by the time it reaches the trailing edge. As the angle of attack increases, the initial pressure drop increases, except at angle of attack 10, where separation begins to occur, and a pressure drop of only -2Mbar is shown.

Appendix B

Comparison between SRCC, FX74, and E423 wings

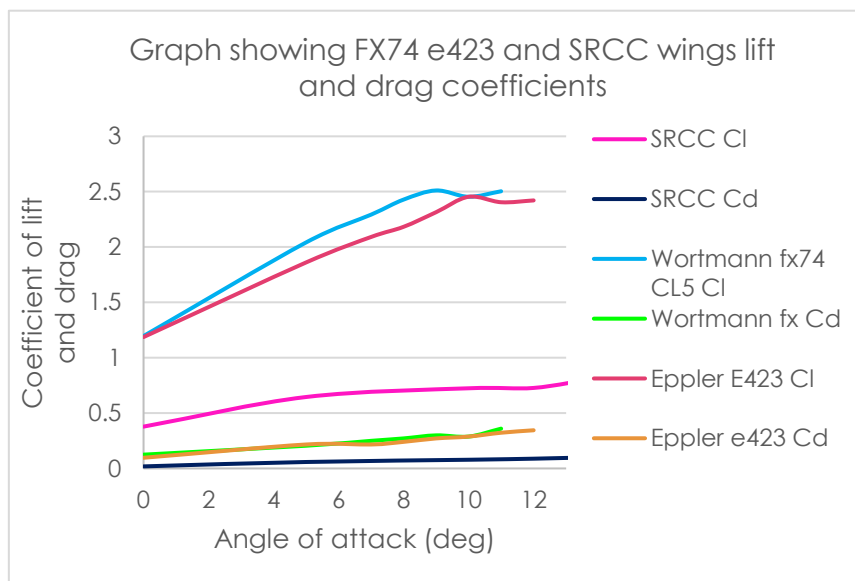


Figure 14.1 – Graph showing the coefficients of drag and lift for the SRCC wing, the Wortmann FX74 wing, and the Eppler E423 wing.

Comparing all three wings together it can be seen just how badly the SRCC wing was performing overall. Although it showed a low drag coefficient, it also showed poor lift coefficient, and changing the angle of attack showed only small improvements.

The Wortmann and Eppler wings, however, showed a lot more responsiveness to angle of attack change. And maintained relatively low drag coefficients throughout.