



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

University of Wales Trinity Saint David School of Engineering

*Aerodynamic Design of the Underbody of an
MCR Racecar for the British Hillclimb & Sprint
Championship*

Major Project

M6X00417

Nicholas David Thomas

1702567

BEng Motorsport Engineering

Supervisor: Tim Tudor

20/04/2020



Abstract

This report is centred around the design of the underbody of an open regulations Hillclimb prototype car. Using Ansys CFD, first two validations cases were made in an attempt to correlate the lift, drag and centre of pressure values to real life data and previous studies data. Then using Solidworks CAD a number of design iterations were created and analysed again using ANSYS CFD to balance the frontal downforce of a newly designed splitter by (Iley, 2019) so that the centre of pressure was similar to the original race car it is based upon. This is because the new splitter created too much negative lift so a newly designed floor can not only balance the downforce but also increase the total negative lift of the vehicle whilst keeping the total drag force below a threshold. Three design iterations were created in an iterative process.

Acknowledgements

I would like to thank Tim Tudor for his guidance and advice throughout the project and driving the MCR car to gain the data needed. To add to that I want to acknowledge fellow students John Hughes and Ainsley Martin for their time in assisting me with tips and advice with the ANSYS Software. I would also like to take this time to thank Alison and Nigel Thomas for their support throughout my studies.

Confirmation of own work

This thesis is the result of the author's original research. It has been composed by the author and has not been previously submitted for examination which has led to the award of a degree.

Signed: Nicholas David Thomas

Date:20/04/2020

Contents

1	Introduction	1
1.1	Aims.....	2
1.2	Objectives.....	2
2	Literature Review.....	3
2.1	Rules and Regulations	3
2.1.1	Motorsport UK Technical Regulations	3
2.1.2	British Sprint Championship	3
2.2	Venturi Effect.....	4
2.3	Ground effect.....	4
2.4	Boundary Layer	6
2.5	Separation	7
2.6	Diffusers	8
2.7	CFD Methodology	10
2.7.1	Domain.....	10
2.7.2	Meshing.....	11
2.7.3	Solving.....	13
2.7.4	Post processing	16
3	Work Carried Out.....	18
3.1	Ahmed Validation Study.....	18
3.2	MCR Validation Study	21
3.2.1	Approach 1.....	21
3.2.2	Approach 2.....	23
3.3	Simplification of geometry	24
4	Results and analysis.....	27
4.1	Design 1	27
4.2	Design 1.1	27
4.3	Design 2	29
5	Conclusions	32

6	Pandemic Impact.....	33
7	Further Work.....	33
8	References	36
9	Appendices	41
9.1	Appendix A	41
9.2	Appendix B	43

Figure 1- Comparison between a race car in CFD (Left) and a Wind Tunnel (Right) (Multimatic Engineering, n.d.).....	2
Figure 2- Class Structure of British Sprint and Hillclimb Association (British Sprint Championship, 2019).....	3
Figure 3-Rule Box Dimensions Around Car	4
Figure 4-Venturi Tunnels of Lotus 79 (Historic Motorsport Central, 2015).....	5
Figure 5- Diagram of the effect of height on ground effect (Katz, 2006).	5
Figure 6- Boundary layer on a flat plate (Schlichting & Gersten, 2017)	6
Figure 7-Diagram of Separation (Helicopters & Aircrafts, 2016)	8
Figure 8-Effect of Changing Diffuser Angle on an Ahmed Body (Moghimini & Rafee, 2018).....	8
Figure 9-Graph of Diffuser Length to Pressure Coefficient (Schuetz, 2016).....	9
Figure 10-Example of Curve Diffuser on a Pagani Zonda R (Rapid Racer, n.d.)	9
Figure 11-Example of a Domain used for Aircraft (Dash, 2006).....	11
Figure 12-Example of an Unstructured Mesh (Tu, et al., 2018).....	12
Figure 13-Example of Isogeometric Analysis (Cottrell, et al., 2006).....	13
Figure 14-Finite Volume Method Element (Haider, et al., 2018)	14
Figure 15- Y^+ Against U^+ in a boundary layer (M, 2018).....	16
Figure 16-Ahmed Body (Meile, et al., 2010)	18
Figure 17-Results from 25° Ahmed Study.....	18
Figure 18-Flow around 35° Ahmed Body	19
Figure 19-Flow around 25° Ahmed Body	19
Figure 20-Results from 35° Ahmed study	20
Figure 21-Results from Correlation Study 1.....	21
Figure 22-Simplification of Louvres on MCR Geometry.....	22
Figure 23-Mesh Approach 2	23
Figure 24-Results from Correlation Study 2 for Lift and Drag Values	23
Figure 25-Results from Correlation Study 2 Centre of Pressure	24
Figure 26-Simplified MCR Geometry	25
Figure 27-Results from Simplification Study of John Iley Design MCR Geometry.....	25
Figure 28-Results of Simplification Study of Design 1.1	26
Figure 29-Pressure Contour of floor of Design 1	27
Figure 30-Design 1.1	28
Figure 31-Side View of Velocity Contours on Midplane of Design 1.1	29
Figure 32-Side View of Pressure Contour around Rear End of Design 1.1	29
Figure 33-Design 2	30

1 Introduction

This report centres around the design and development of the under floor of an MCR race car for the British Hill Climb and Sprint Championship. Aerodynamics in motorsport have been used since the late 1960's to gain an advantage by creating downforce. This results in the tyres being forced into the ground to create more grip, as witnessed in the equation below. The 1970's saw a significant step forward in motor racing aerodynamic research occurred when Peter Wright and Tony Rudd discovered the promise of using ground effect to create greater amounts of downforce for very little drag penalty (McBeath, 2006). This changed racing aerodynamics technology permanently, even after it was banned in the 1980's with the introduction of diffusers.

$$F = \mu N \text{ Equation 1- Fiction Force (Engineers Edge LLC, 2020)}$$

F=friction force

μ =Coefficient of Friction

N=Applied Normal Force

The two main criteria when it comes to aerodynamics in motorsport are lift and drag. The aim of an aerodynamicist, in most motorsport, is to get the most negative lift for as little drag as possible. However, to achieve one the other must occur. The effects of lift and drag on the car can determine if a car is a winning one or not, as it directly affects the handling characteristics of the vehicle. Another consideration is where on the vehicle this negative lift occurs as this will also affect the handling of the vehicle which is known as centre of pressure.

The way aerodynamics were traditionally investigated were through wool tufts on cars and having another running next to it to see the flow structure from the movement of the tufts. This was then taken into wind tunnels so that the results are more repeatable and as a result obtain an actual value for lift and drag. In more recent times Computational Fluid Dynamics (CFD) has become much more accessible as computing power has increased enough to get results which are reasonably accurate. However, all this must still be validated against each other until you can be confident with the results being accurate.



Figure 1- Comparison between a race car in CFD (Left) and a Wind Tunnel (Right) (Multimatic Engineering, n.d.).

1.1 Aims

The main aim of this project was based on balancing the aerodynamic downforce of the car whilst creating minimal drag. This should have been achieved using underbody aerodynamics and the ground effect phenomenon. This should be done in accordance with the rules of the British Sprint Championship (Motorsport UK, 2019). Another consideration is where on the car the negative lift will impact on. This causes a huge difference on how a vehicle handles, so it must balance out longitudinally along the car (Divila, 2020). If there is too much load on the rear tyres, then the car will give more grip on acceleration and in our case will also cause understeer. If there is too much load on the front then the front tyres will grip more and the car will tend to oversteer (Quinarelli, 2018). Furthermore, the aerodynamic devices must be able to work under differing conditions so that it can be optimised for different track uses. An example of this is under braking the car will pitch forward raising the rear and dropping the front. This may cause the car to produce more downforce on the front and less on the rear. However, if the pitch is too much then this means the aerodynamic devices can stop working and then for a moment the car will have significantly reduced grip. The same is in reverse for acceleration where there can be an increased ride height on the front and reduced ride height on the rear. The devices should also work well whilst turning. As the car yaws whilst turning the air is not going normal to the car. If very sensitive devices are used, then sometimes the devices may stop working or work at a reduced amount.

1.2 Objectives

The objectives of this project were to create a car with a minimum of an extra 300N of downforce acting on the rear axle of the car. This was with a maximum increase of drag coefficient of 0.5. Whilst achieving these two criteria the car must also still conform to the rules and regulations of the Motorsport UK and the British Sprint Championship. To add to that the centre of pressure needs to be 1.29m from the front axle line.

2 Literature Review

2.1 Rules and Regulations

2.1.1 Motorsport UK Technical Regulations

Motorsport UK (formally Motorsport Association) is the main governing body of motorsport in the UK. Every year they release a rule book which gives the base sporting and technical regulations for motorsport series to use and develop on. The regulations that affected the project are sated below (Motorsport UK, 2019). The rules that relate to the series which the car is based on is sections B, C H, J, K, L & Q but the only ones concerning the project is sections J, K & Q.

- J [5.2.7] Any Aerodynamic device cannot be the maximum height or higher the ROPS (roll hoop). Also, they cannot extend more than 1100mm from the rear axle line and no more than 100mm from the rear bodywork.
- J [5.20.10] The car cannot have any skirts, bridging device or any other aerodynamic device between the track/ground and chassis. Any part of the car influencing aerodynamics must:
 - A) Comply with the rules relating to coachwork
 - B) Be rigidly secured to the entirely sprung part of the vehicle
 - C) Remain immobile in relation to the vehicle
- J [5.20.11] No part of the bodywork or suspended part can be below the horizontal plane passing 40mm above the ground.

2.1.2 British Sprint Championship

The British Sprint Championship is organised by the Hillclimb and Sprint Association (HSA) which holds around 18 events throughout the year at circuits around the UK. It is the most prestigious titles in the UK for Sprint championships (British Sprint Championship, 2019). The regulations are split into 7 classes which can be seen below

CLASS	DESCRIPTION
A	Modified Production Cars
B	Sports Libre Cars
C	Racing Cars up to 1100cc
D	Racing Cars 1100cc to 1600cc
E	Racing Cars 1600cc to 2000cc
F	Racing Cars up to 1600cc forced induction car engine
G	Racing Cars over 2000cc

Figure 2- Class Structure of British Sprint and Hillclimb Association (British Sprint Championship, 2019)

The regulations for the sprint championship are completely open within the Motorsport UK's regulations (British Sprint Championship, 2019). This does mean that the minimum ground clearance must be 40mm.

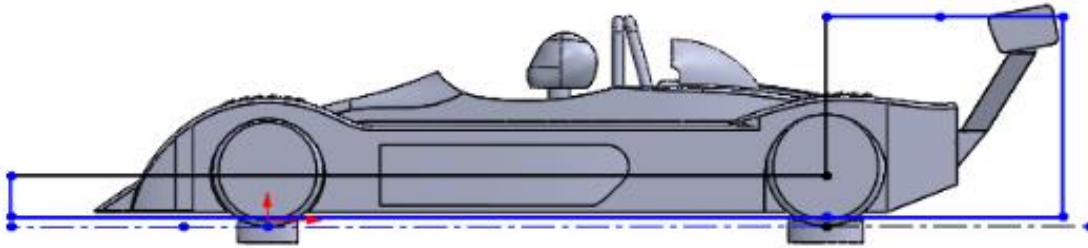


Figure 3-Rule Box Dimensions Around the Car

2.2 Venturi Effect

The effect occurs when a fluid which is subsonic (as is in most cars with exception to land speed record) and is incompressible (such as air) is forced through a pipe which has a narrow section, it causes the static pressure to decrease (Felfoldi, 2019). This pressure drop is caused because the conservation of mass (The Editors of Encyclopaedia Britannica, 2018) and the conservation of mechanical energy or momentum (Cruz, et al., 2019) causes the fluid to speed up, shown by the Bernoulli's equation:

$$P + \frac{1}{2}\rho V^2 + \rho gh = \text{constant}$$

Equation 2- Bernoulli's equation (Princeton University, n.d.)

- P=Pressure
- ρ = Density
- V=Velocity of fluid
- g=gravity
- h=height

2.3 Ground effect

This is the change of aerodynamic properties when the vehicle is in proximity of the ground. It is taken advantage of by aircraft on take-off and landing to aid in stopping and help it take off by increasing lift (R.Stewart, 1987). Span related ground effect is in reference to aircraft but in the case of the underside of race cars it is length-related ground effect. Incoming air is directed through in an inlet which is then accelerated through a nozzle which uses the ground and the car as a venturi tunnel (Schuetz, 2016). This creates low pressure under the car accompanied with high pressure above, creates a pressure difference vertically on the car. This is the negative lift on the vehicle. This was most famously used in the motorsport on the Lotus 79

(figure 4). As can be seen this is optimised by forcing the air through a smaller gap and then gradually reintroducing the fluid to the free stream (fluid flowing in normal conditions not affected by vehicle) gently. This is to prevent separation of the air from the bodywork which will be referenced later in this paper. It also uses sliding skirts that dragged on the ground as this prevents the high pressure on the outside of the car bleeding underneath and increasing the pressure in the tunnels. If the pressure under the car increases this reduces the pressure difference and therefore making the downforce smaller. This is also one of the reasons these types of car were banned, as the skirts could get stuck going around a corner and then unexpectedly, the driver has significantly less downforce than expected and can cause an accident.

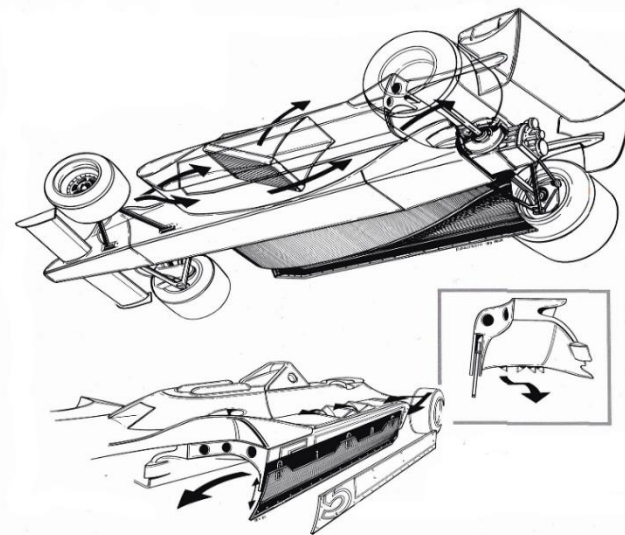


Figure 4-Venturi Tunnels of Lotus 79 (Historic Motorsport Central, 2015)

The closer the bodywork can get to the ground the stronger this effect becomes to a point as can be seen in figure 5.

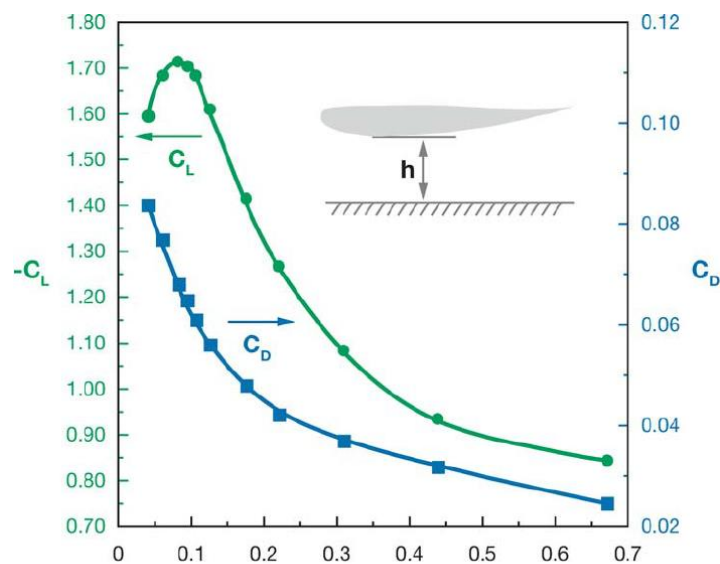


Figure 5- Diagram of the effect of height on ground effect (Katz, 2006).

As the wing gets closer to the ground the CI increases up until a critical point. This is around 20-30mm because the larger the compression ratio (area of inlet compared to area of choke point) the more downforce it produces. Until the compression ratio becomes so great the air cannot travel through the gap and then there is very little flow going through the tunnel. As the minimum ride height in the series the car was designed for is 40mm the height was not too much of an issue. Even though this is a different geometry than the one used in the project the actual theory remains the same as most ground effect cars have a wing shape underneath. This is because wing shapes are effective at accelerating the air under the car whilst avoiding separation if designed properly.

2.4 Boundary Layer

The surface of a vehicle is never completely smooth there will always be small microscopic divots. This accompanied by the viscosity of a fluid means that the velocity directly on the surface is at 0m/s, compared with the free stream causes a velocity difference. Because of shear forces it stops the velocity flow parallel to the surface which has a knock-on effect until it reaches free stream velocity (U_∞) again.

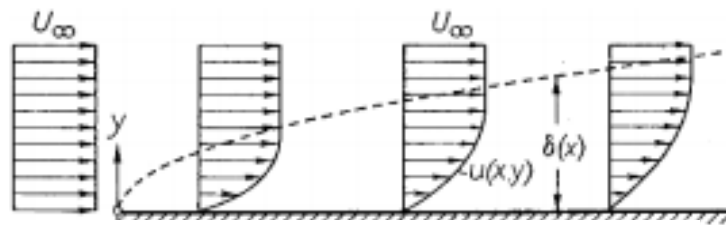


Figure 6- Boundary layer on a flat plate (Schlichting & Gersten, 2017)

Figure 6 shows that as you get further away from the surface the velocity gradient become smaller, until the shear stress has decreased sufficiently that the surface drag no longer affects the fluid. This is where the boundary layer has reached free stream velocity again. This is where the fluid is considered laminar. Also, as the fluid gets further along the plate the larger the boundary layer grows. This is because the shear stresses are no longer strong enough to hold the flow layers in a laminar fashion and they start to rotate. This is where turbulence can emanate from which will be discussed later. When the car gets too close to the ground the boundary layer starts to interact with the ground causing turbulence which causes the choking effect.

$$Re = \frac{\rho V d}{\mu} = \frac{V d}{\nu} \text{Equation 3-Reynolds Number Calculation (Benson, 2014)}$$

- ρ =density
- V =Fluid Velocity
- D =characteristic or working length

- μ =Dynamic Viscosity
- ν =kinematic viscosity

External Flow transitions from laminar to turbulent between 3×10^5 and 3×10^6 Reynolds number but this does depend on the factors acting on the body. In internal flow laminar is considered up to 2300, transition is between 2300 and 4000 and turbulent is considered above 4000 (Simscale, 2017) (Schlichting & Gersten, 2017). Laminar is when the shear forces are strong enough to keep all the layer so that they are parallel to each other. The transition layer is when the energy in the fluid begins to not be strong enough and the fluid starts to become turbulent where it is a mix between laminar and turbulent. Carry this through and it begins to become fully turbulent as the fluid has small vortices in it that causes random fluctuations (Schuetz, 2016).

However, this can be split vertically. The bottom layer (very close to the surface) is called the laminar sub-layer. This is a section which stays laminar along the plate. It is formed as the shear forces are still acting on the flow and causing the velocity of the stream to be almost zero. Then above that is the buffer layer and then the turbulent region. Although, this is a simplified model and the boundary layer can still stay laminar through if it has enough energy to overcome the shear forces. Furthermore, as the flow goes along the plate the drag resistance increases. This is because the effective frontal area increases as the boundary layer height increases, and the skin friction increases the drag because there are higher shear forces.

2.5 Separation

Separation occurs when the boundary layer loses enough energy that it can no longer stay attached to the surface. This is because the energy is lost to the shear friction forces acting on the fluid and then into heat. Because the fluid is travelling so slowly the velocity gradient becomes too great and this means that the flow detaches from the surface (Houghton, et al., 2013). This can be a key part of causing turbulence as this creates a swirl as the air tries to fill the void left behind which grows into turbulent air flow which is evidenced in figure 7.

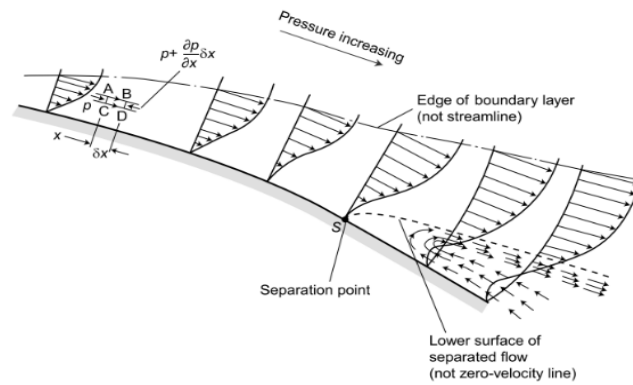


Figure 7-Diagram of Separation (Helicopters & Aircrafts, 2016)

2.6 Diffusers

A diffuser's main use is to introduce the air flow from under the car back to the mainstream to prevent separation occurring. In doing so this speeds up the air under the car even more as there is a pressure difference between underneath the car and the atmospheric pressure. This then draws the air from underneath the vehicle and is known as diffuser pumping. This in turn creates more downforce through the acceleration of the air (McBeath, 2018). This has been theorized to do this in different ways.

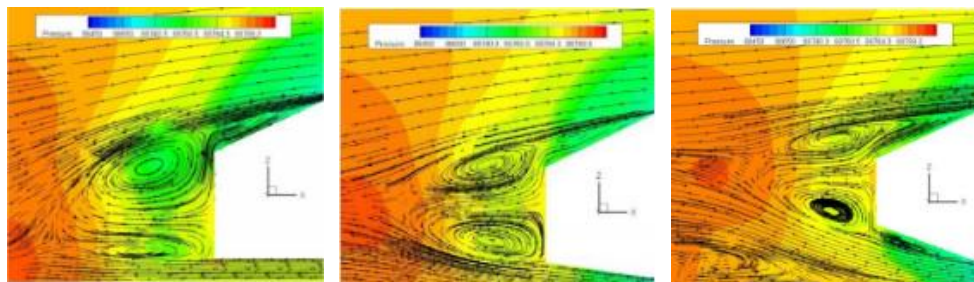


Figure 8-Effect of Changing Diffuser Angle on an Ahmed Body (Moghimi & Rafee, 2018)

As shown in the middle case above having a diffuser lets the air stay attached to a point. However, because it has stayed attached this means that the turbulence at the rear has a much smaller area meaning there is less drag. To add to that you can also see that on the underside of the model the pressure has dropped which would increase downforce. Finally, as can be seen in the last picture when the diffuser angle becomes too great the flow can begin to separate from the surface. This can in fact increase the turbulence and reduce the effect of the diffuser. This paper states that above eight degrees is when the diffuser becomes less effective (Moghimi & Rafee, 2018), this does change on a case by case basis though.

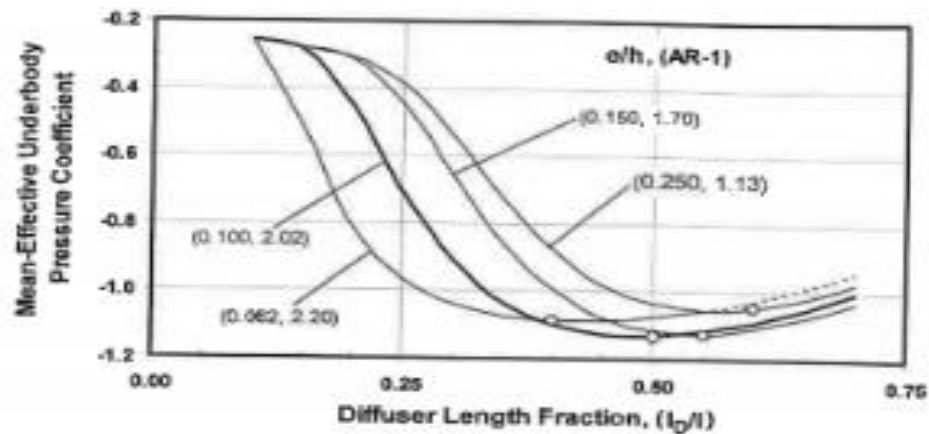


Figure 9-Graph of Diffuser Length to Pressure Coefficient (Schuetz, 2016)

Another factor which can affect the performance of the diffuser is its length. According to (Schuetz, 2016) the longer the diffuser the lower the pressure of the underbody is up to a point which in this case is around 0.5m. However, with most things in aerodynamics this is a case by case basis so may be different on this project. The longer the diffuser the longer the flow stays attached which gives it more time to get back to free stream velocity. However, this applies to straight diffusers. A curved diffuser is beneficial as it means that the angle of it can produce around 22% more downforce than a traditional straight diffuser as well as a small decrease in drag (Huminic & Huminic, 2019). This also means that the diffuser is less pitch sensitive as the pressure is spread out, so it does not hinge on one point. The curved diffuser is also less peaky, as in that the peak value of downforce is across a bigger range compared to a straight diffuser if looking at a pressure plot longitudinally, this means a greater overall downforce can be produced for a similar centre of pressure.



Figure 10-Example of Curve Diffuser on a Pagani Zonda R (Rapid Racer, n.d.)

2.7 CFD Methodology

In this project CFD took a large role by testing the performance changes in different designs. However, there are many different parts of CFD which need to be validated against other tests such as wind tunnels and real-life data to see that the answers we are getting are close to accurate. There is a phrase coined by George Fuechsel who was a programmer in IBM, 'Garbage in, Garbage Out'. This means that a computer will give an inaccurate result out if you put inaccurate data in (Techopedia, n.d.). So, the methodology study used later is to make sure that the author was putting in the correct data. The process of CFD can be split into three distinct sections, Pre-processing (domain and meshing), Solving (actually calculating the result) and Post-processing (analysis of results) (Schuetz, 2016).

2.7.1 Domain

To first start the process a domain should be created and the model be simplified. The domain is the working area that will be solved. So, this restricts an area around the car where the equations will take place. This could be a number of shapes such as the shape of a wind tunnel, by comparison in aircraft design they sometimes use curved domains (figure 10). But in automotive cases generally just a plain box is used as it is fairly similar to a wind tunnel. Lanfrit suggest that the domain should extend five car lengths behind and at least three car lengths in front. He also suggests that the displacement of the car should be no more than 1-1.5% of the total cross-sectional area (Lanfrit, 2005). However, some use 2 lengths in front and 4 behind and 2 times the length in the Y and Z direction (Sung, et al., 2014). This will change from a case to case basis as one model may have a larger wake region than another so it may be the case, there is a need for extending the rear of the domain. Generally, the bigger the domain the more flow you can capture but it gets to a point where there is no flow affecting that area so there is little point having it there. There is also the issue of having a larger domain as it is more computationally expensive so it would take much longer to calculate. The size of the domain can be halved, if the body being calculated is symmetrical, then it can be assumed that the flow is as well. So, if the body is split in half then you can get away with a bigger domain as the number of elements is halved.

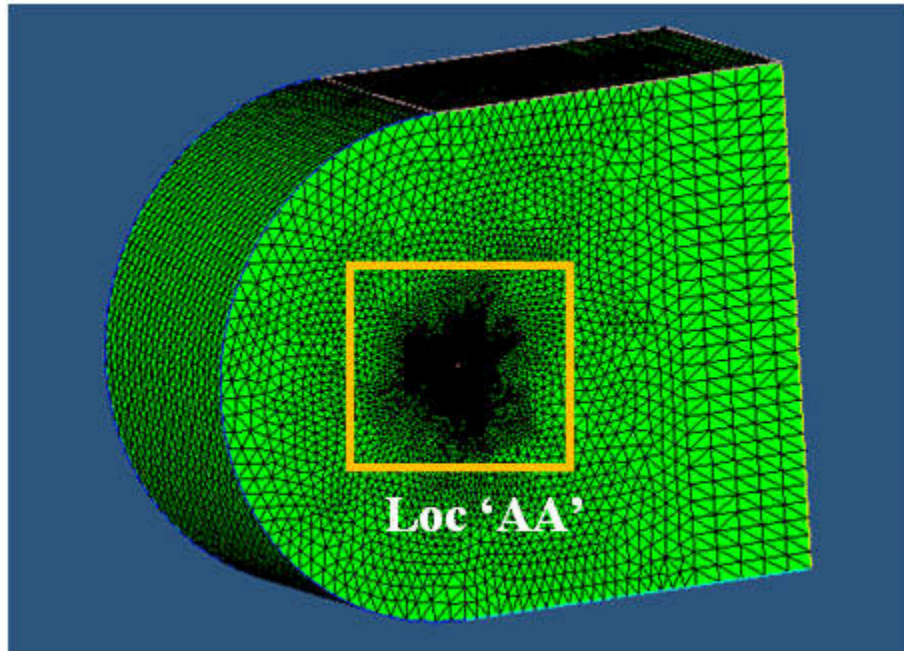


Figure 11-Example of a Domain used for Aircraft (Dash, 2006)

2.7.2 Meshing

This is the second part of pre-processing. This is where the body and domain are split into segments or 'elements' to calculate. This comes in a combination of different forms and shapes. One of the first considerations is the shape of the elements themselves. This depends if the problem is in 2D or 3D. For 2D there are triangles ("Tri") and 2D prism (Quadrilateral or "quad"). Then these have their 3D counterparts Tetrahedron ("tets") or prism with quadrilateral base (hexahedron or "hex") respectively. There are also variations on the pyramid shape and prism with triangular base (wedge). There is also another shape which is an arbitrary polyhedron, which is a 3D 6 faced shape (Bakker, 2006). These all have their advantages and disadvantages. One such example is polyhedrons have reduced accuracy, but it can reduce the number of elements in half. Generally, as in aerodynamics geometry are quite complex so the meshing process is automated, but it can be done manually. This is a lot of time and effort, but it can reduce the total number of elements used. But in the case of this project an automated mesh was used.

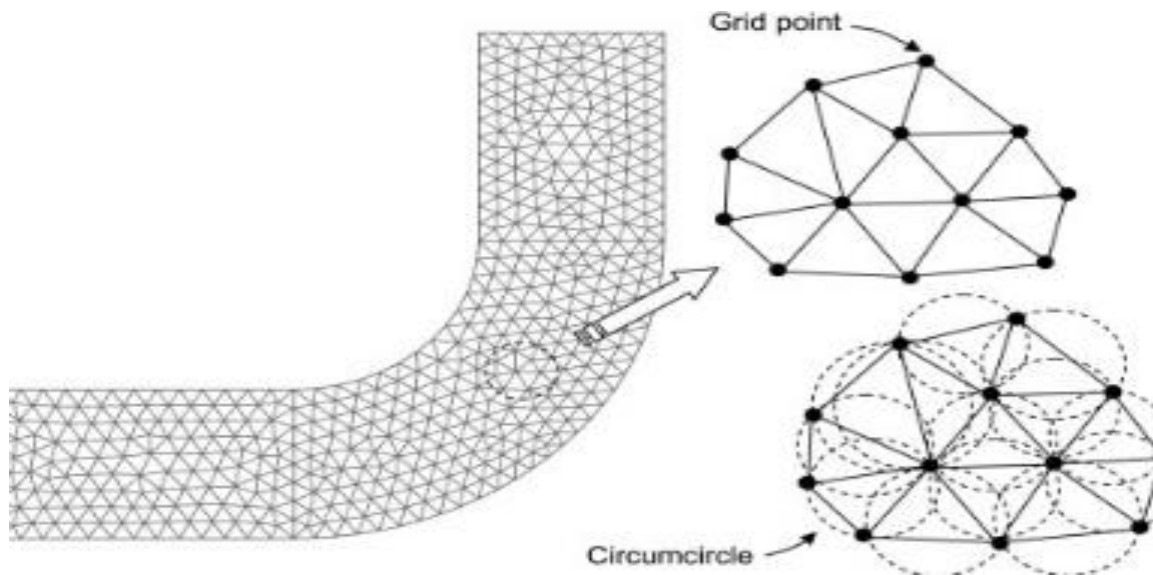


Figure 12-Example of an Unstructured Mesh (Tu, et al., 2018)

The elements can form either an unstructured or a structured mesh. A structured mesh is a uniform pattern of elements where the unstructured is generally more 'natural' and changes shape depending on the body being analysed. The benefit of an unstructured mesh is that it is more flexible so for a detailed complex geometry it will pick up more surface detail. However, it may make the results marginally less linear as there may be some areas where the mesh is tighter than others (Tu, et al., 2018). The use of a structured mesh usually uses quadrilaterals in 2D and hexahedral in 3D. The use of a structured mesh can give a higher convergence and a higher resolution over unstructured (Introduction to CFD Meshing, 2016).

There is also the area of study of isogeometric analysis which could improve the mesh quality. This is a meshing strategy which uses splines rather than straight lines to create the mesh elements, therefore creating a mesh much closer to the actual shape rather than just an estimation (Salmoiraghi, et al., 2016). This would be beneficial as it could mean a more accurate mesh is produced so giving higher accuracy results whilst possibly using a smaller number of elements. It also means it has a higher order degree of continuity achieved. However, this is a fairly new field and was not available to the author to use in this project.

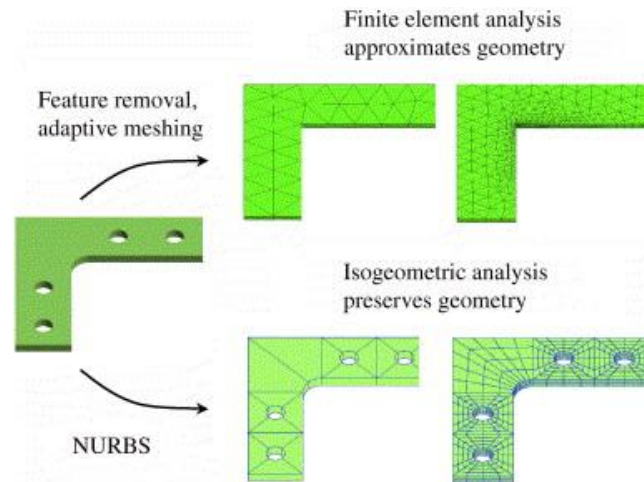


Figure 13-Example of Isogeometric Analysis (Cottrell, et al., 2006)

2.7.3 Solving

This process is the act of calculating the conditions on the edges of each node. It does this by converting the differential equations seen in equation 4 into linear equations using matrix algebra. This calculates at each node and then goes through another iteration, each time making a guess at the answer. As the iterations go through the guesses become closer and closer together until it reaches convergence and the answer is deemed accurate by the program. The convergence criterion controls the difference in the computer's guesses so theoretically the lower it is then the more accurate it will be, but it will take more time to come close to the solution. This is why some people such as (Lanfrit, 2005) suggests using a 1st order for the first 50 iterations then finish off with the 2nd order to get the answer as accurate as possible. This changes which order the equation goes to. The higher the order the more accurate it is but has a longer run time.

$$\text{Continuity: } \frac{dp}{dt} + \frac{d(\rho u)}{dx} + \frac{d(\rho v)}{dy} + \frac{d(\rho w)}{dz} = 0$$

$$X - \text{Momentum: } \frac{d(\rho u)}{dt} + \frac{d(\rho u^2)}{dx} + \frac{d(\rho uv)}{dy} + \frac{d(\rho uw)}{dz} = -\frac{dp}{dx} + \frac{1}{Re_r} \left(\frac{d\tau_{xx}}{dx} + \frac{d\tau_{xy}}{dy} + \frac{d\tau_{xz}}{dz} \right)$$

$$Y - \text{Momentum: } \frac{d(\rho v)}{dt} + \frac{d(\rho uv)}{dx} + \frac{d(\rho v^2)}{dy} + \frac{d(\rho vw)}{dz} = -\frac{dp}{dy} + \frac{1}{Re_r} \left(\frac{d\tau_{xy}}{dx} + \frac{d\tau_{yy}}{dy} + \frac{d\tau_{yz}}{dz} \right)$$

$$Z - \text{Momentum: } \frac{d(\rho w)}{dt} + \frac{d(\rho uw)}{dx} + \frac{d(\rho vw)}{dy} + \frac{d(\rho w^2)}{dz} = -\frac{dp}{dz} + \frac{1}{Re_r} \left(\frac{d\tau_{xz}}{dx} + \frac{d\tau_{yz}}{dy} + \frac{d\tau_{zz}}{dz} \right)$$

$$\text{Energy: } \frac{d(E_T)}{dt} + \frac{d(uE_T)}{dx} + \frac{d(vE_T)}{dy} + \frac{d(wE_T)}{dz} = -\frac{d(up)}{dx} - \frac{d(vp)}{dy} - \frac{d(wp)}{dz} - \frac{1}{Re_r Pr_r} \left(\frac{dq_x}{dx} + \frac{dq_y}{dy} + \frac{dq_z}{dz} \right) + \frac{1}{Re_r} \left(\frac{d}{dx}(u\tau_{xx} + v\tau_{xy} + w\tau_{xz}) + \frac{d}{dy}(u\tau_{xy} + v\tau_{yy} + w\tau_{yz}) + \frac{d}{dz}(u\tau_{xz} + v\tau_{yz} + w\tau_{zz}) \right) \quad \text{Equation 4-}$$

Navier-Stokes Equation (NASA, 2015)

This uses the cartesian coordinate system to calculate the x, y, z momentum the conservation of mass (continuity) and the conservation of kinetic energy. It does this by using couple partial differential equations using the finite volume method (Zuo, 2005).

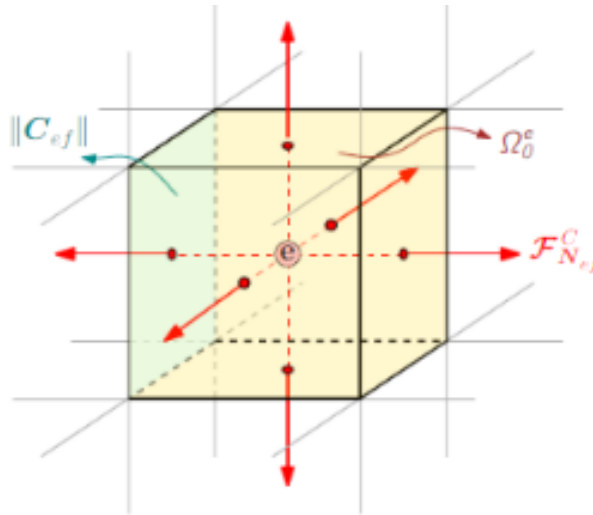


Figure 14-Finite Volume Method Element (Haider, et al., 2018)

The finite volume method calculates an entire section rather than points. It then uses the value of the edge of the element to calculate the one next to it. The benefit of using this method is that what you lose on one edge you gain on the other, so it is very good at conserving energy flow. There is also an added benefit that for complex problems the finite volume method is easier to solve than the finite difference method (Bishop, 2013). This also means that for complex problems there are less calculations evoking the idea that it has quicker run times as there are less nodes to calculate. Rather than having four points on a face there is only one to calculate. This is for 3D problems; for 2D or 1D as such as in a wing profile a finite difference method is often used. In the case of this project only 3D was used as a car is a 3D shape and could not be simplified as the flow is not 2D

On top of the basic Navier-Stokes equation there is a turbulence model. It simplifies the Navier-Stokes equation to decrease run time. This comes in a number of models which use different or variations of an equation. The ANSYS package comes with a number of pre-determined models but in the high levels of motorsport the teams create their own to suit their needs. The way CFD solves can be split into three groups. Direct Numerical Simulation (DNS) solves using the Navier-Stokes equations and does not require a turbulence model (Roelofs & Shams, 2019). As it does not use a turbulence model it is very computationally expensive

and is therefore not practical in industry. Then there is Large Eddy Simulation (LES) which solves using a Navier-Stokes equation which is averaged spatially (Gel & Celik, 2000). This is less computationally expensive than DNS but is still too expensive for average computers. Then there is Reynolds Averaged Navier-Stokes Simulation (RANS). This is a time averaged Navier-Stokes equation which uses a turbulence model so that the computation time is not too long (ANSYS, 2009). This is the option most use in industry apart from those companies making their own, as mentioned previously. The RANS approach has a number of different equations to add on as the turbulence model. These vary from one and two equation models and they vary in computational cost. The recommended for a base run is a realisable K- ϵ model with K being a transport equation and ϵ being the dissipation rate equation. The realizable part is in reference to the conditions applied whilst using this. The benefits of using this model is that it can give good modelling of flows with rotation, separation and boundary layers (ANSYS, 2009) which are often seen in automotive cases.

$$\frac{\partial \epsilon}{\partial t} + U_j \frac{\partial \epsilon}{\partial x_j} = \frac{\epsilon}{k} \left(C_{1\epsilon} \frac{\mu_t}{\rho} S^2 - C_{2\epsilon} \epsilon \right) + \frac{\partial}{\partial x_j} \left[\frac{1}{\rho} \left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] \text{Equation 5-K-Epsilon (Stanford University, n.d.)}$$

Along with the turbulence model there is also the treatment of near wall interaction. As near wall interaction is quite similar between cases a function is used to model it which were found from empirical data. The modelling of the wall starts from the mesh where the elements closer to the surface are smaller. (ANSYS, 2009) states that the first grid cell needs to be around $y^+=1$ to get an accurate viscous sublayer. This will however increase the cell count significantly but is used with a low-Reynolds number turbulence model such as K-Omega. Y^+ is the non-dimensional distance from the wall (Datawave Marine Solutions, 2019). (ANSYS, 2009) also states that there is another approach having a Y^+ of between 30 and 300. This would be used with a turbulence model such as SKE, RKE or RNG. This is suggested to be used if the middle of the domain is more interesting rather than the forces on the wall. Wall functions can be split into two with being linear and logarithmic based. Logarithmic has the advantage of being very accurate near the wall as the cells are very small near the wall and gets bigger the further out you get. This means it is very detailed next to the wall if that is needed. However, in some situations the logarithmic is not the best function, such as when there is separation as it cannot model the boundary layer very well. In this project the author used the non-equilibrium wall function as this was suggested by (ANSYS, 2009) as being good for modelling separation and adverse pressure gradients which is what often happens on aerodynamics of a car. However, this does have an issue, namely a coarse mesh near the wall so matters like skin friction drag are not very accurately modelled. (M, 2018) also states that the first grid cell should have a Y^+ of between 30 and 300 whilst using high Reynolds number model. This is because if the value is too low then the model can become invalid and if it is too high the wall is not properly

resolved. With the Y^+ value a wall function is used to calculate the flow close to the wall. This is split into three sections.

When $Y^+ < 5$: Viscous sublayer using the linear velocity law

$$U^+ = Y^+ \text{ Equation 6-} U \text{ value when } Y^+ < 5$$

When $Y^+ > 30$: Logarithmic area which uses the function below with $B=5.2$ and $k=0.41$

$$U^+ = \frac{1}{k} \ln(Y^+) + B \text{ Equation 7-} U^+ \text{ value when } Y^+ > 30$$

When Y^+ is between 5 and 30: Buffer layer is the transition section between the turbulent dominated flow and the viscosity dominated flow (M, 2018).

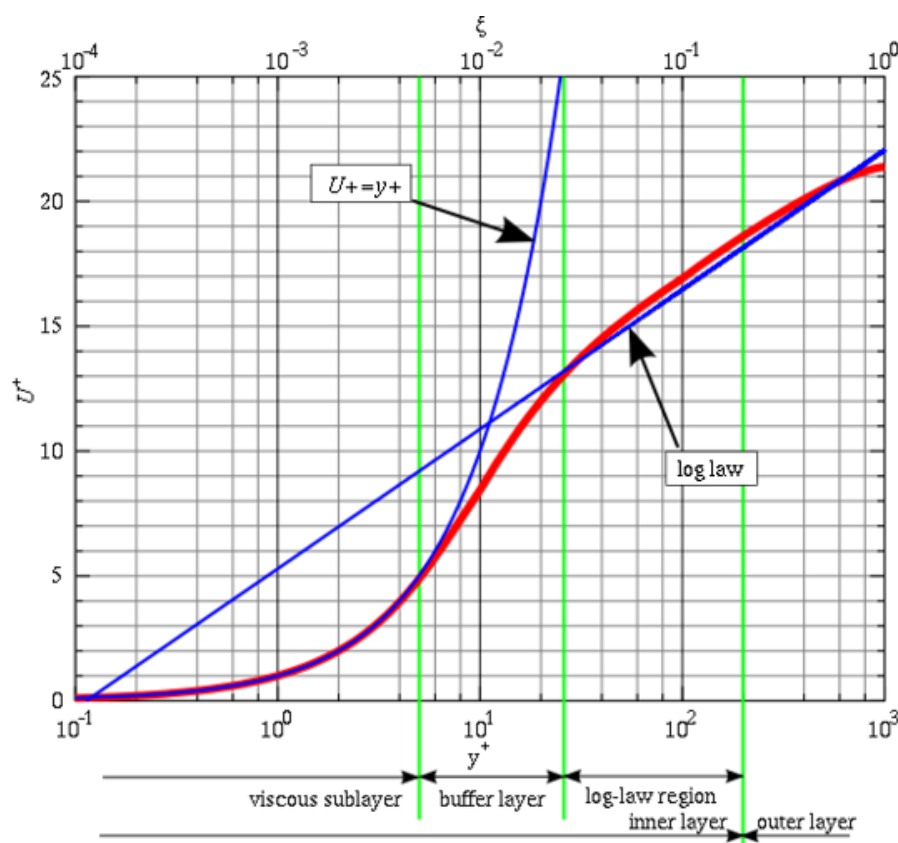


Figure 15- Y^+ Against U^+ in a boundary layer (M, 2018)

2.7.4 Post processing

This section is where the results from the solver can be seen and analysed. It can be analysed through a number of different ways. It can display the flow structures given from the body, the vortices and streamlines. This is highly important as it can give you the answers on how to change your geometry to suit your need. Another way to analyse is the pressure distribution. This is a useful tool, especially in motorsport, as it can make a serious impact on the balance of the car as referenced earlier. Not only does it display pressure distribution but also velocity, turbulent kinetic energy and many more, each displaying different attributes which can be used

to analyse the simulation results. Post processors can also give tabulated results so that further analysis and comparison can be performed outside the software (Ozen Engineering Inc, 2020).

3 Work Carried Out

3.1 Ahmed Validation Study

The author first made a bluff body validation using the Ahmed body (Ghaedamini, 2010) to check that the CFD methodology was correct. This body has been used to do aerodynamic research in many studies such as (Huminic & Huminic, 2019) and to check the accuracy of CFD methodology. He did this with using two variations of the body, at two different slant angles as this would further make sure that the methodology is correct. In this study the C_d and C_l were compared to a previous studies results (Meile, et al., 2010). It may not seem that this is a very representative shape however, the rear slant angle is very similar to a diffuser but upside down so this can be useful as a basis to the methodology that was used within the project.

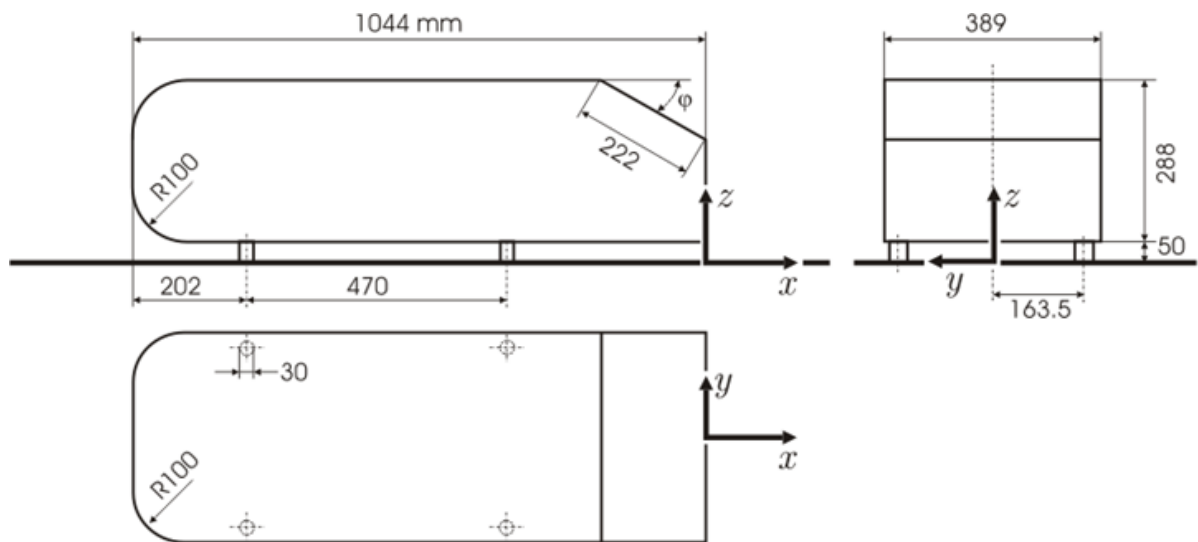


Figure 16-Ahmed Body (Meile, et al., 2010)

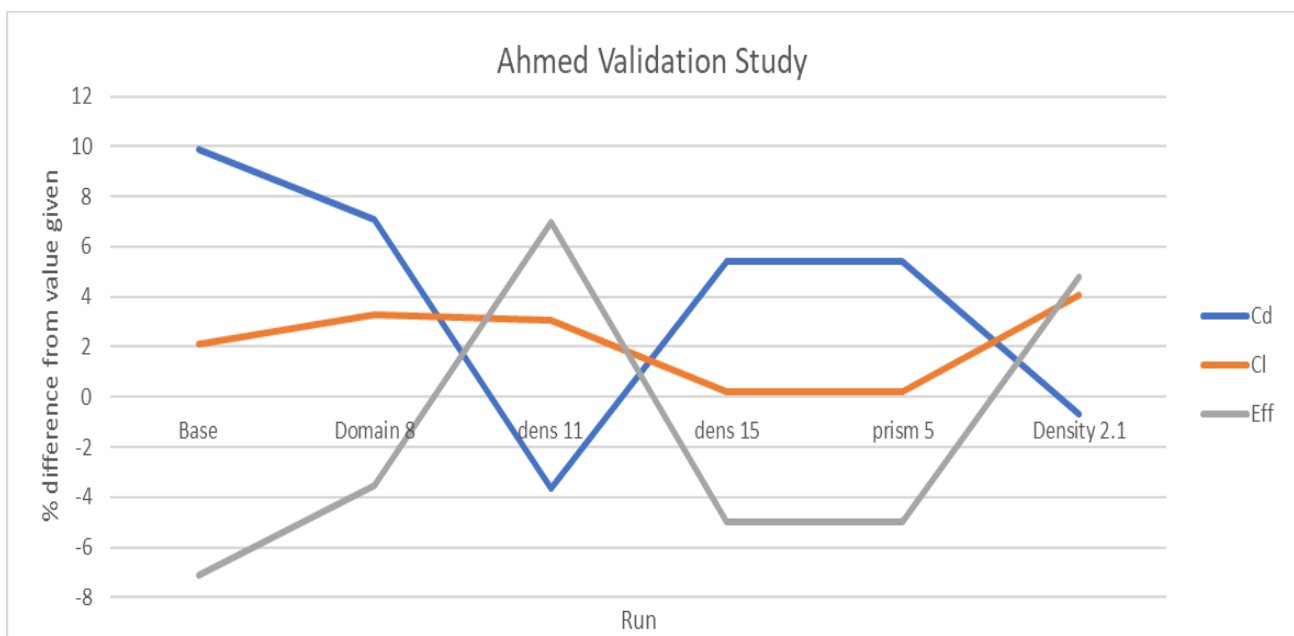


Figure 17-Results from 25° Ahmed Study

As shown, as the study progressed the author got closer and closer to the results. The author was unable to get the exact results, but he deemed that within 5% accuracy tolerance was enough as the previous studies results may also have a margin for inaccuracy. This margin is also what is deemed accurate enough within industry standards. The author started by changing the domain so as to pick up all the flow and turbulence from the rear but not too big to increase computing time. Then he went on to changing the mesh first by the global density and then the mesh refinement. The biggest affecter of the results was the density of the mesh. This is because there is a trade-off between accuracy with a finer mesh and an increase in computation time. The author then made a sweep of the number of prism layers. To add to that as there is very little separation occurring on 25° slant angle so the suggested turbulence model for general cases (K- ϵ realizable) was the best after going through a number of models. After this the writer thought there could be more accuracy gained from further mesh refinement, so he again changed the density boxes location to get better results. As can be seen above in figure 17 the CI was slightly less accurate but with the Cd being much closer it was judged to be a better mesh. Although the CI was still within the target of 5% accuracy which is judged acceptable by industry.

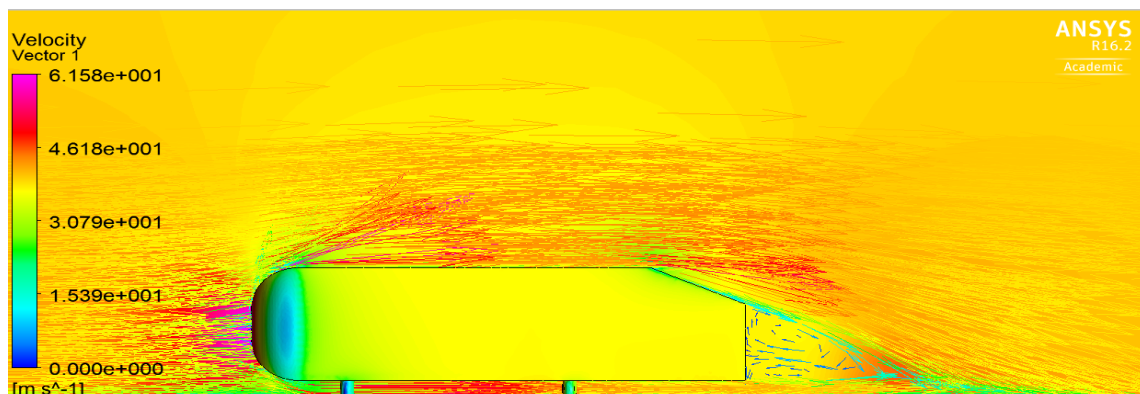


Figure 19-Flow around 25° Ahmed Body

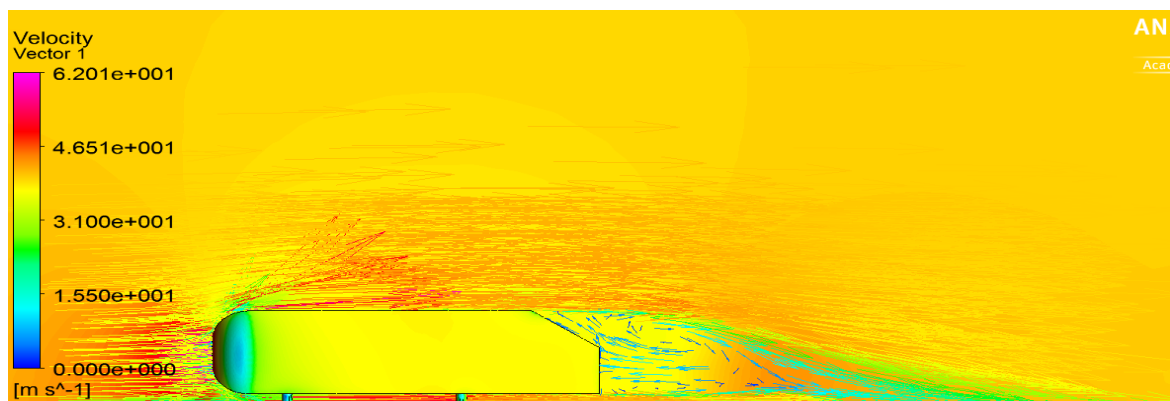


Figure 18-Flow around 35° Ahmed Body

The author then did an investigation into the 35° slant. After doing another domain sweep, he found that the mesh density did not change the results, but the number of prism layers had a large effect. This is because the larger the boundary layer the more prism layers are needed to correctly model it. As there is much more separation the boundary layer is bigger as mentioned earlier. This is because there is not enough energy within the fluid to keep it attached to the surface. There is a limitation with having a large number of prism layers as the more layers the greater skew is introduced affecting the overall mesh (Chalasani & Thompson, 2004). However, in this case it was more beneficial to have more layers. The author also found a significant gain with adjusting the turbulence model. The author found that changing to the K- ϵ RNG had a significantly closer to the given result. This is because this turbulence model significantly aids the modelling of swirling motion in turbulence (ANSYS, 2009) which suggests that for high separation cases this is the best suited turbulence model. This just shows that to get an accurate simulation a base case needs to be run and depending on the flow regime the turbulence model needs to be changed. There was an issue to get the CI close to the value of the previous example. This was because the value from the study was 0.004 which the author then assumed to be negligible.

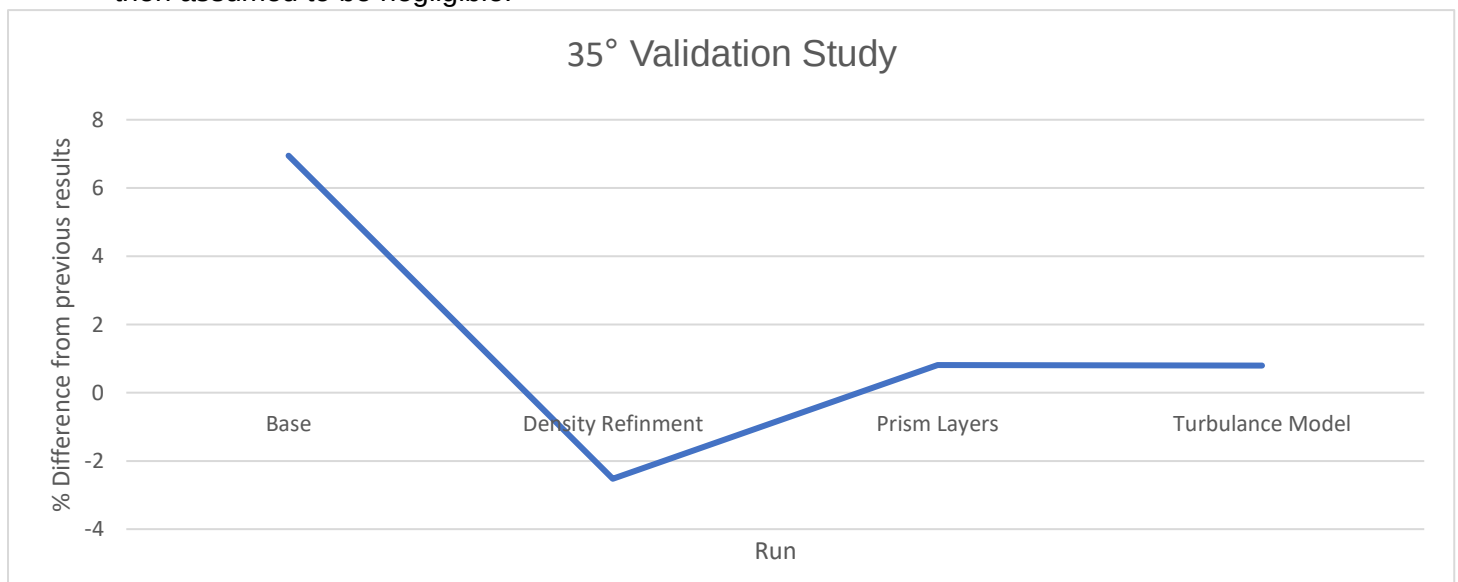


Figure 20-Results from 35° Ahmed study

The author believes if he were to do this study once again, he would make modest adjustments. As the run time on such a simple geometry was relatively short then a different approach to the mesh sweep could be taken. The sweep could be done by making the mesh smaller and smaller until the results did not change significantly. This would mean that the analyst could be sure that the mesh is as accurate as possible. The issue with the method that the author uses was that each step affects the other so for a general case this was not a very good approach but for the Ahmed body it still attained reasonable results. For a more general case the new meshing method would be more appropriate. Because of the complexity of the

MCR case a mesh sweep could not be easily done within the confines of this project so the author accepted a modest error within the results.

3.2 MCR Validation Study

3.2.1 Approach 1

The author attended a straight-line test where the standard splitter and the new Iley Design (Iley, 2019) splitter was tested to obtain values for the lift of the vehicle. The drag of the car could not be obtained as the straight at the test track was not sufficiently long to do a coast down test. The time it took to decelerate would have been used to calculate the deceleration and along with the frontal area an average drag coefficient could be calculated. From these values the author was then able to make comparisons to make sure that his methodology was correct. The test was carried out on the different splitters at three ride heights 78mm, 66mm and 54mm. so to make sure the trend was the same the author ran the geometries at 45mm, 60mm and 80mm to take into account dynamic ride height. The author calculated the load on the front then put that weight on the car to see the change in ride height. When the car was at 54mm static it was more like 46mm dynamic ride height. The actual values are not critical as the trends are the most important consideration to make sure that changes in geometry affects the lift values as predicted.

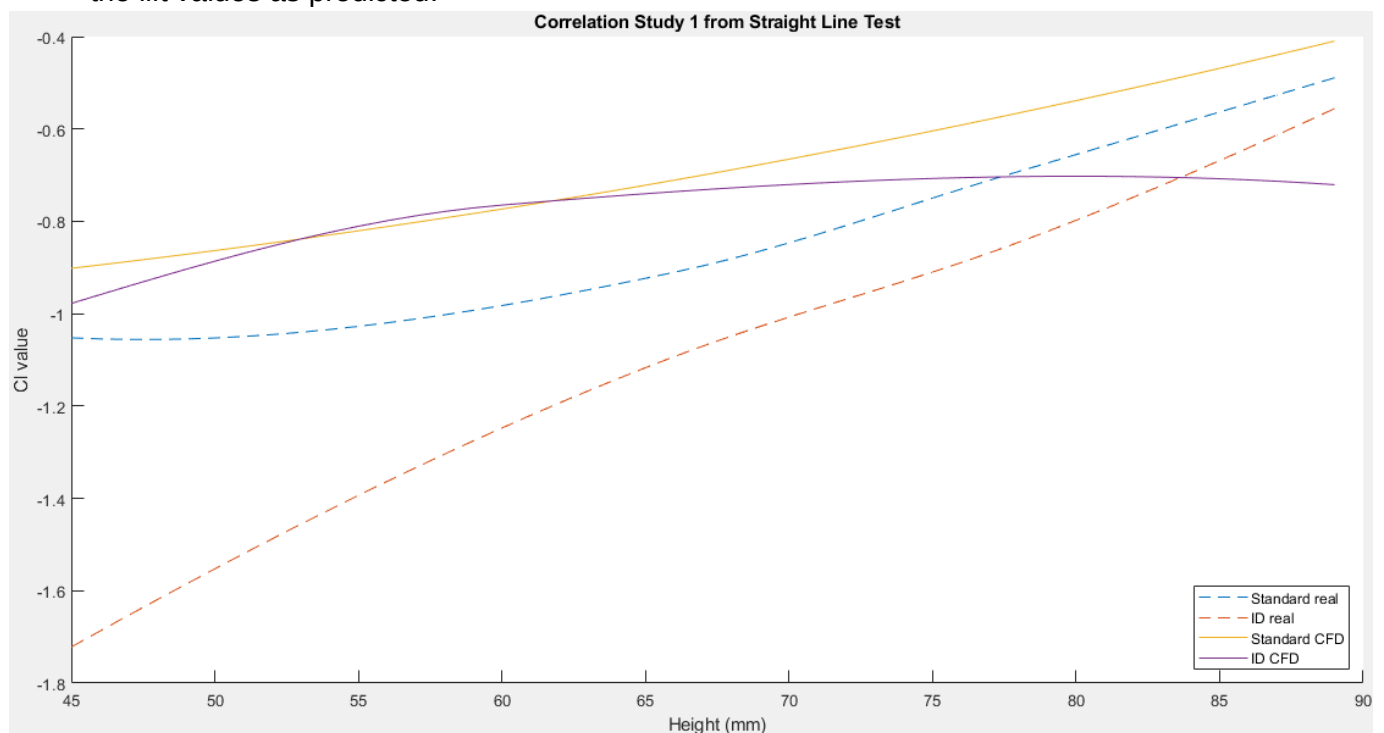


Figure 21-Results from Correlation Study 1

Figure 21 demonstrates that the author's initial methodology was not very accurate, with the trend being very different. However, there were still some positives to this correlation as the CFD also shows that the Iley splitter is better than the standard but not to the extent the straight-line test demonstrated. To add to that both trends show that as the height of the car

increases so does the lift value which was a good starting point. To try and get the results closer the author then changed his meshing strategy to try and get closer to the results. The author used a meshing strategy previously used by (Martin, 2019). The author believes that the mesh used in the first sweep was too detailed in the domain and not fine enough on the car itself. So, the second method was much more detailed on the body (0.015m previously 0.02m) and on the wheels (0.01m previously 0.02m). The domain then had slightly larger elements within the density boxes so that the element count did not get too high, as there was a reduction in size of elements in other places. The global mesh was kept the same at 1m but two density boxes of 0.5m and 0.1m respectively was included with the 0.5m being the bigger density box. There may also be very small error due to the fact that the author simplified the geometry by deleting the fillets around the louvres. This was because around this area there was consistently skewed elements reducing the quality of the mesh. So, following an example from (Altair university, 2019) the author deleted the fillets, and this significantly helped with the meshing approach. This is because the mesh uses straight lines so removing the fillets made the louvres have straight edges which are easier to mesh against. However, this would have made a small difference the author deemed that the difference was so small it had a negligible effect. After this change the mesh was much easier to create with meshing time slightly reduced.

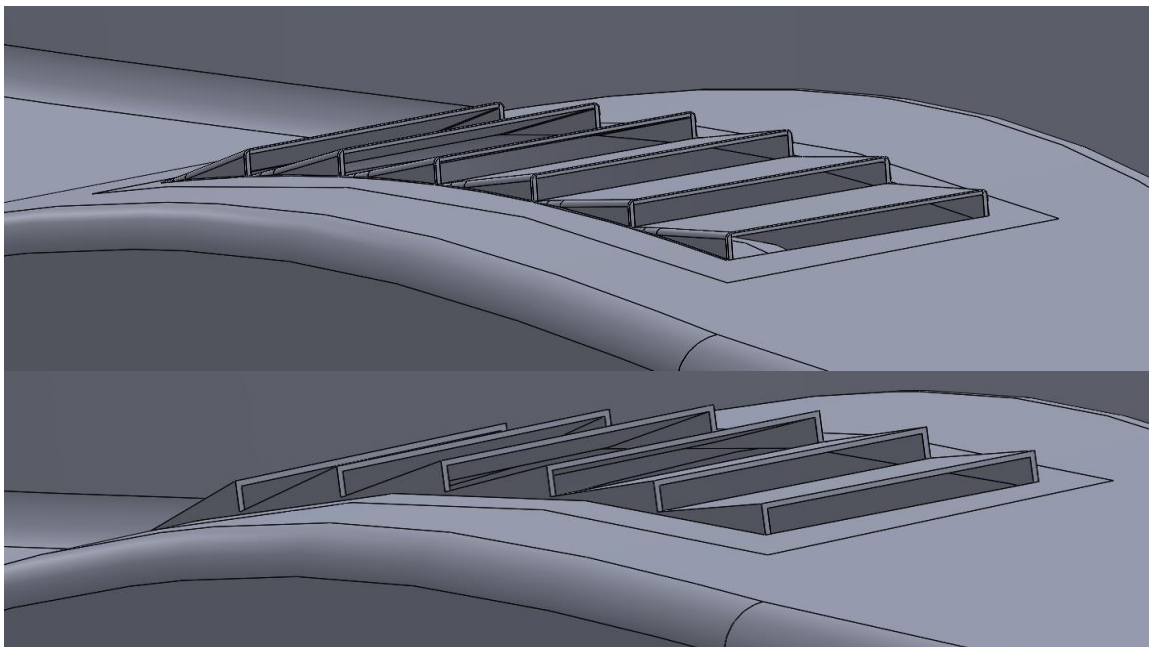


Figure 22-Simplification of Louvres on MCR Geometry

3.2.2 Approach 2

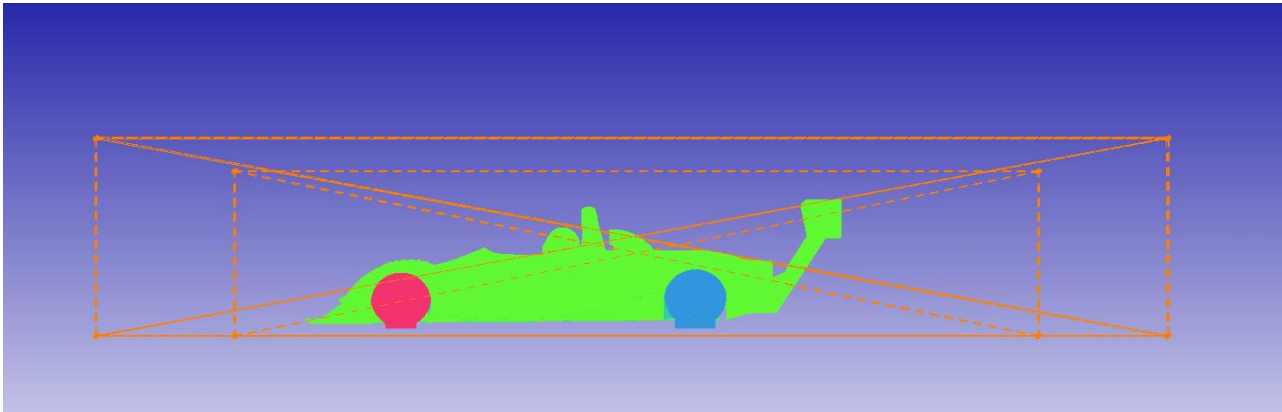


Figure 23-Mesh Approach 2

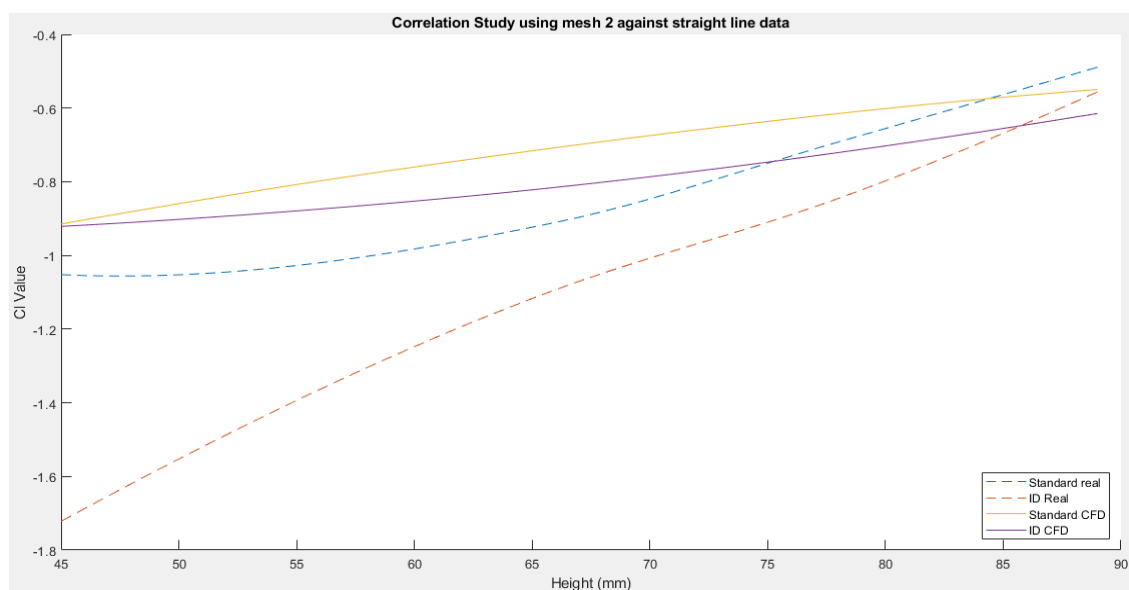


Figure 24-Results from Correlation Study 2 for Lift and Drag Values

This gave better results from the previous mesh. There was also an added benefit that the mesh was much easier to create so the process of meshing was sped up considerably. This was useful for the purposes of this project as meshing was the main time usage of the author. As a consequence, he was able to leave a simulation run over night whereas for meshing the author needs to be at the computer for many hours. Using this the mesh time was cut down from 4-5 hours to a maximum of 2. The correlation is not perfect but is much closer than the first approach. If more time were available this mesh could have been improved. Perhaps a higher density mesh much closer around the vehicle together with a smaller density inner box, but dependent upon higher computing capacity being available to use.

Another point of validation could be the centre of pressure.

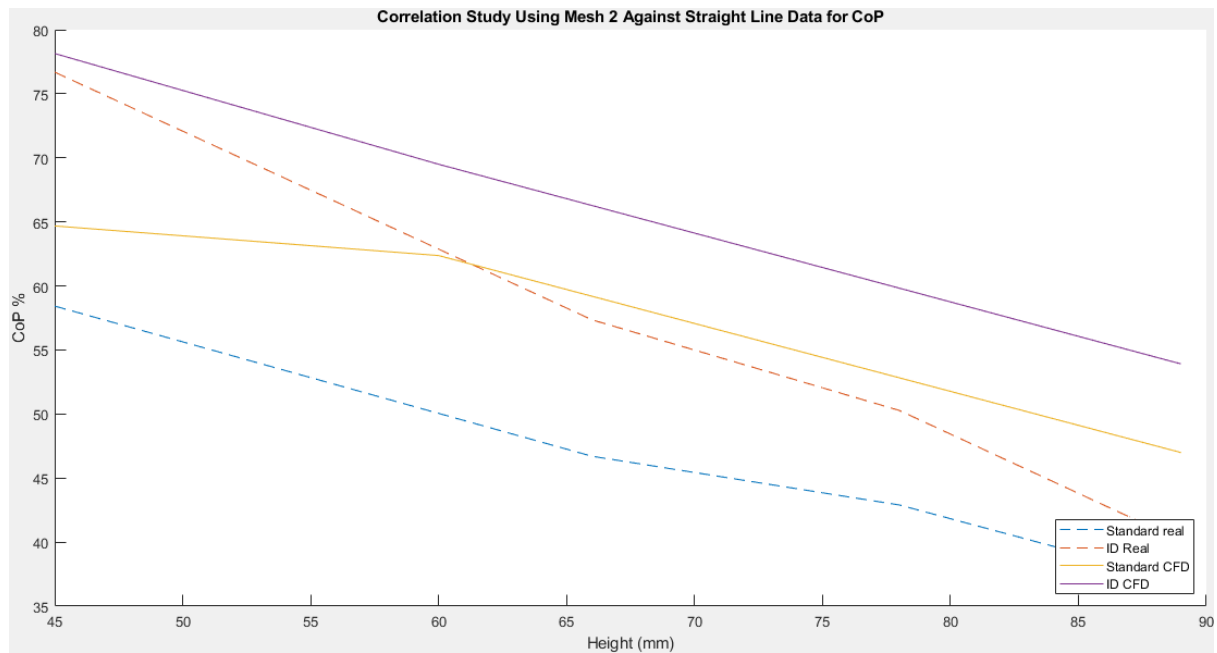


Figure 25-Results from Correlation Study 2 Centre of Pressure

As can be seen above there is some correlation between the CFD and the straight-line data as both negative lifts increase as the ride height increases. This is expected as the lower the splitter gets the greater the downforce it can produce from ground effect. Because of this it brings the centre of pressure forward and therefore increase the percentage of total aerodynamic load acting on the front. This is what the centre of pressure (CoP) is, it is the percentage of the total aerodynamic load acting on the front compared with the rear. This can have a major effect on the handling in corners and in a straight line. In a corner it needs to balance the centre of gravity so that the total load can be correct on the tyres. The CoP should be behind the CoG as it allows for over steer in slower corners with quicker turning and more stable in high speed corners (Rehnberg, et al., 2012). In the same way in a straight line it also gives more stability having the CoP behind the CoG (Evans, 2019). In extreme cases if this is set up incorrectly it can create a 'fish tail' effect which causes the rear of the vehicle to swing side to side possibly even making the vehicle crash.

3.3 Simplification of geometry

Each computer run took a considerable amount of time to calculate (50 seconds per iteration with around 2000 iterations needed for convergence ~27hours) as a result the author decided to simplify the geometry further. This reduces the number of elements needed therefore less calculations to be done in each iteration. This can change the results, but if the results of the simplified geometry change along with the new designed geometry then all the runs can be done with the simplified geometry. The author first removed the driver's head and seat headrest to see the effect that that had the lift and drag values on a car at 45mm ride height

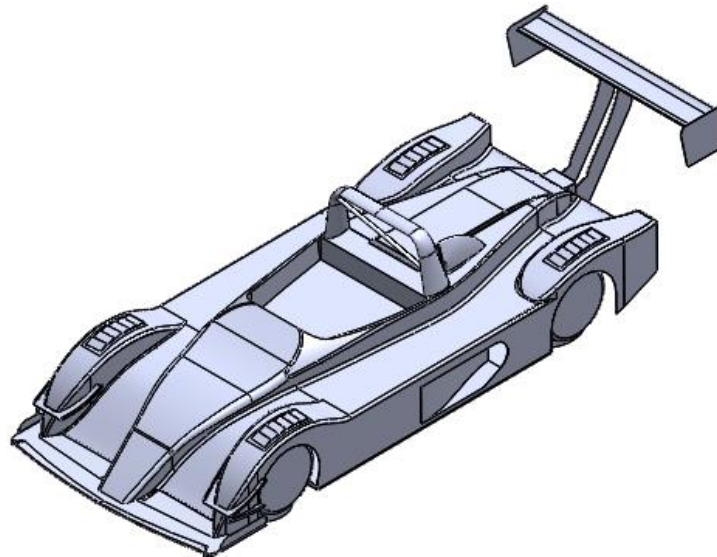


Figure 26-Simplified MCR Geometry

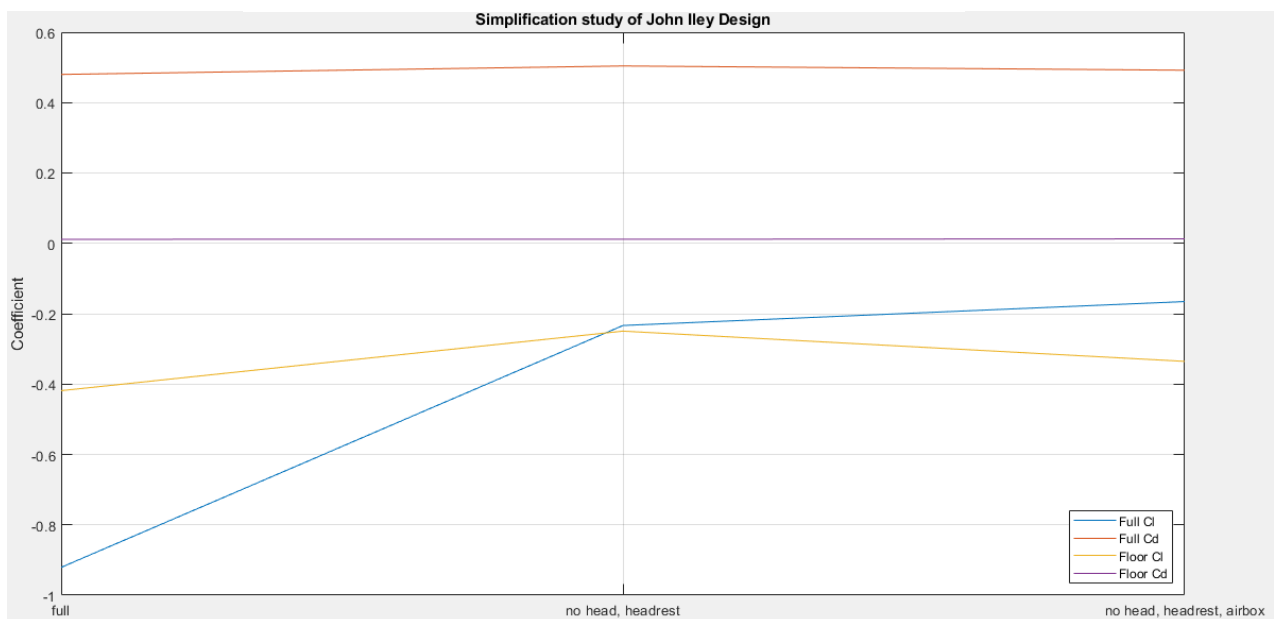


Figure 27-Results from Simplification Study of John Iley Design MCR Geometry

The simplification was done initially on the John Iley Design car at 45mm ride height. Interestingly removing the head rest had a large effect on full car lift value. The author believed that this was an erroneous result though as he would expect the lift value to decrease as the air flow would be smoother going toward the wing enabling the wing to work more efficiently. This would be because the air would be turbulent before it reaches the wing so the air would separate as soon as it hit the wing. Interestingly though there was not much of a difference in removing the airbox. This was unexpected as the author thought again the Cl would decrease. As these were unexpected results the author made another study on design 1.1. This design

will be explained later in the paper, but this was chosen as it has a diffuser so the effects on the floor and diffuser could be seen as the project concentrates in this area.

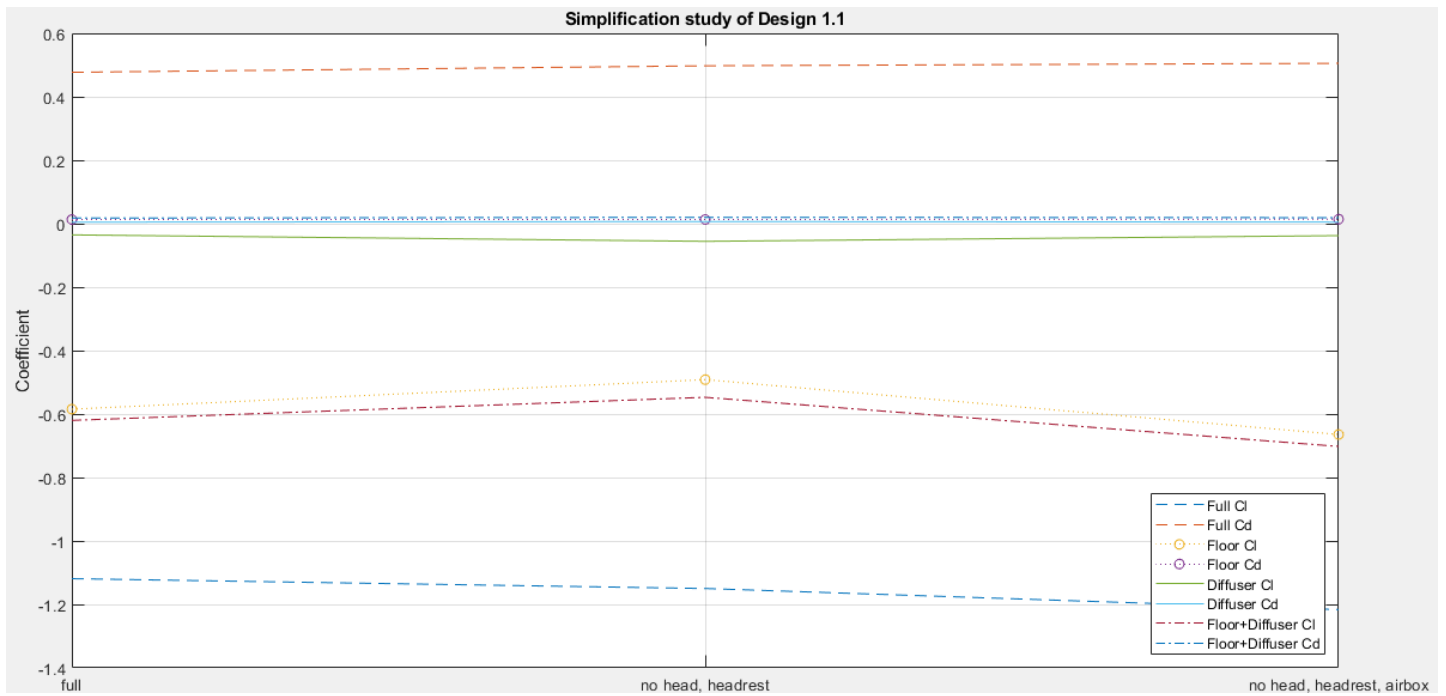


Figure 28-Results of Simplification Study of Design 1.1

As can be seen that there is less of a change when a diffuser is added. There is a slight change in total Cl. This is probably because, as theorised earlier there is cleaner flow getting to the wing which means that it can work more efficiently. However, it may be the case the wing is high enough to be out of the turbulent air from the airbox and headrest to as not be affected by the air coming off them. The author expected the total Cd to decrease as the frontal area had decreased, but it actually increased slightly from 0.477 to 0.506. This could be due to the cleaner flow going to the rear of the car is interacting poorly with the air coming from the floor and is creating turbulence. Nonetheless simplifying the geometry did not reduce the overall computing time by very much. So the author took the decision that the small time saving was not worth the reduction in accuracy.

4 Results and analysis

4.1 Design 1

The first design was a fairly basic straight diffuser which expanded laterally with an area ratio of 3.2. The author used this as it was in a similar range used by (Toet, 2017). It also included two strakes on the edges as the author believed that this would prevent the flow from bleeding out of the edges and get caught in the turbulent air from the wheel wake. There is also an intake for the radiator at this point, so they were also placed to prevent the turbulence from these elements. The author decided to use an angle of 7° for the diffuser as from the research this is around the angle the diffuser can be before separation occurs (Huminic & Huminic, 2019). It is also the angle that the MCR race cars OSS car has their diffuser angle set at.

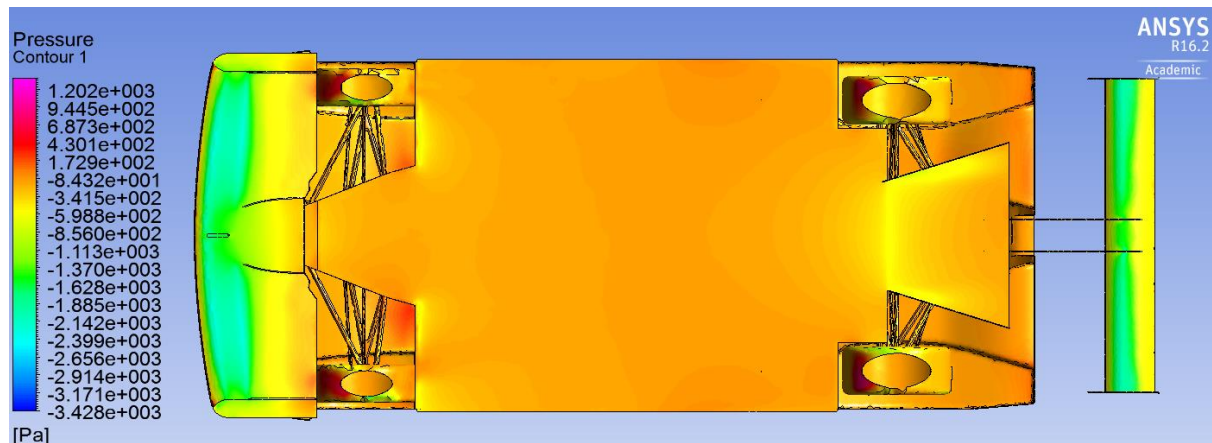


Figure 29-Pressure Contour of floor of Design 1

As shown in figure 29 the diffuser does have a small effect on the pressure under the car. There is a decreased pressure region in front and on the sides of the diffuser. As a result, the total downforce increased from -0.9211 to -1.068. It also had a centre of pressure of 75%. This is an improvement, but the author believed there could be further development. As there does not seem to be high pressure on the edges of the diffuser this suggests the strakes work.

4.2 Design 1.1

Design 1.1 was a development from design 1 by reducing the angle down to 5 degrees and extended the start of the diffuser laterally so that it can speed up a wider section of the whole floor. The author initially designed the diffuser to expand laterally as it was used in other designs previously. However, the author believes this was done because the floor of the car was narrow, so it was needed. In this case the floor is relatively wide so a full width diffuser could be used.

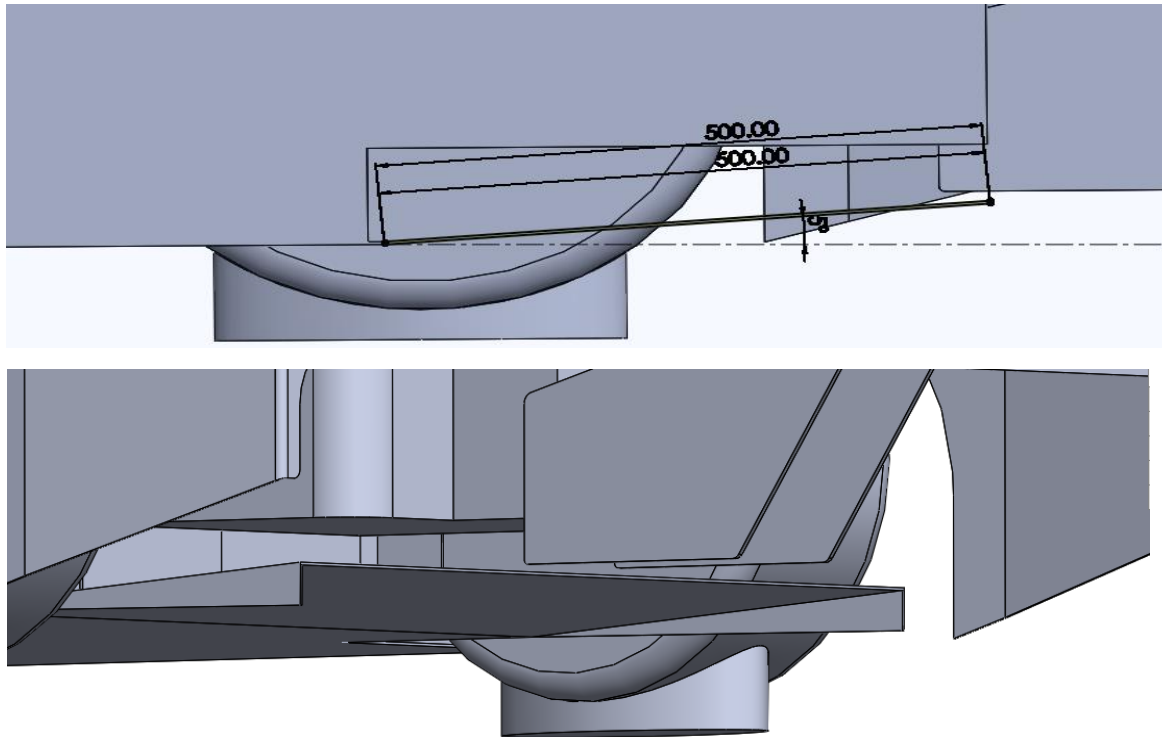


Figure 30-Design 1.1

There are still strakes in place so that the pressure does not bleed out of the edges but also to stop the turbulence from the wheels bleeding into the flow of the diffuser. This produced a much greater improvement on that of the first design. The C_L was -1.12 with a C_d of 0.477. So not only did it increase the downforce, but it also reduced the drag slightly. The C_L increased a large amount on the floor as with it producing -0.419 of lift with but with the new diffuser that went down to -0.45067. The diffuser itself did not produce a huge amount of lift (-0.0458) but as can be seen below it did drop the pressure significantly in the area in front of the diffuser and around the kick point. This brought the centre of pressure to 67.67%. This is significantly better as the aim is to get it to around the 64% mark so as to make it similar to what the CFD calculated as the standard splitter distribution. The author believes that because the width of the diffuser is larger it can affect a larger portion of the floor. This was an aspect that the author chose to carry through. One other thing to note is that the author believed that there was some wing interaction. This could be because the air is coming off the diffuser and is carrying on the same path as the diffuser. This then interacts with the air coming off the wing. This could perhaps be used to increase the effectiveness of both the wing and diffuser. So, to see if this was the case the author increased the angle of the diffuser back to 7° as this could perhaps give performance gains (Christoffersen, et al., 2011).

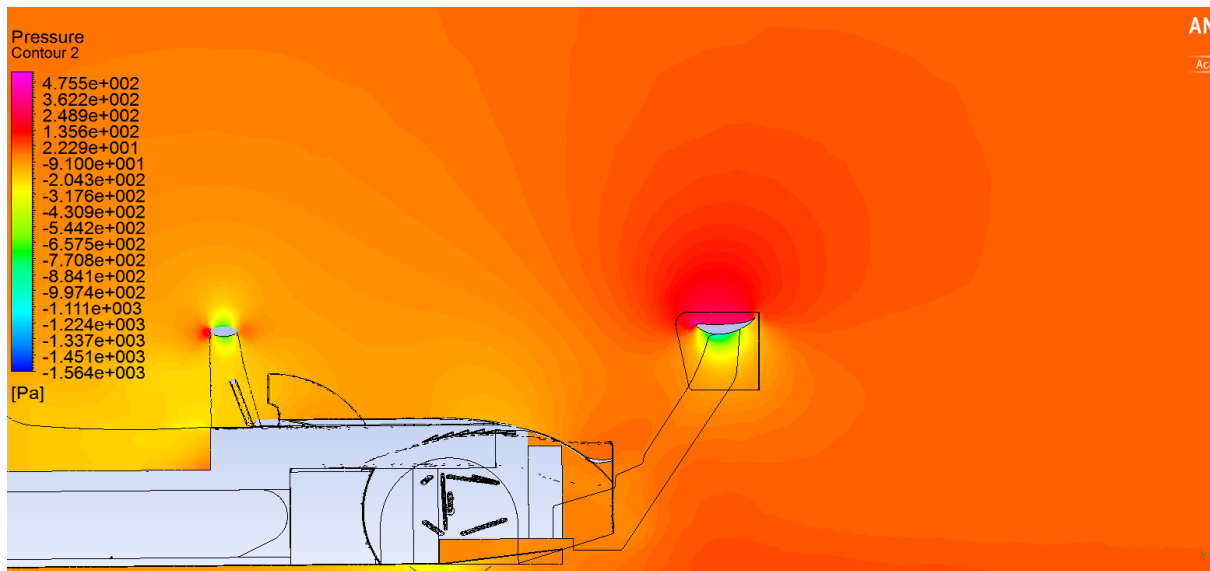


Figure 32-Side View of Pressure Contour around Rear End of Design 1.1

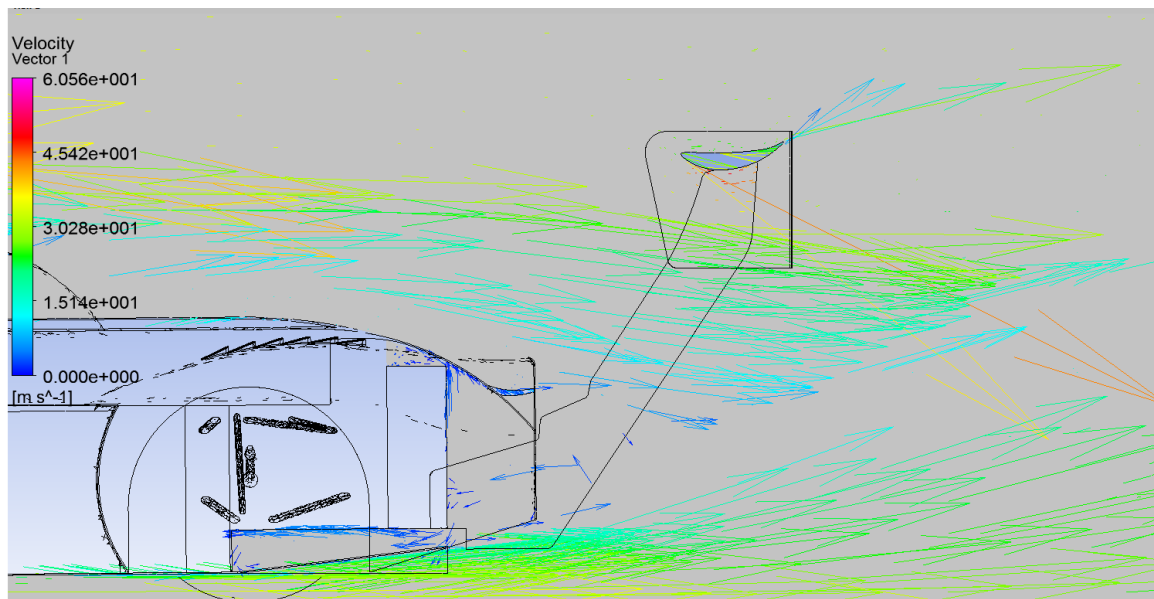


Figure 31-Side View of Velocity Contours on Midplane of Design 1.1

Changing the diffuser to 7° actually reduced the lift marginally to -1.10. This suggests that it is starting to separate at this angle. The author believes that if the angle went greater than this then the C_L would increase even more. However, further investigation showed that there is very little interaction witnessed between the diffuser and wing. This is most likely because the air flowing off the body is interacting with the air coming off the diffuser rather than the wing.

4.3 Design 2

The author's second design is a curved diffuser. He chose this as curved diffusers can have much higher angles than straight diffusers, as separation does not occur until a lot later. Also giving around 22% extra downforce on a bluff Ahmed body compared to a straight diffuser (Huminic & Huminic, 2019). Another benefit of using a curved diffuser is that they are less

peaky. This means that the pressure is more spread along the diffuser so there is no single high point but rather a long slightly lesser high point. This means that the car is less sensitive giving more confidence to the driver, but it can also mean that a higher downforce can be produced with a similar centre of pressure as it is spread out more. There is also a side benefit of reducing skin friction because of local separation and reattachment (Ehirim, et al., 2019).

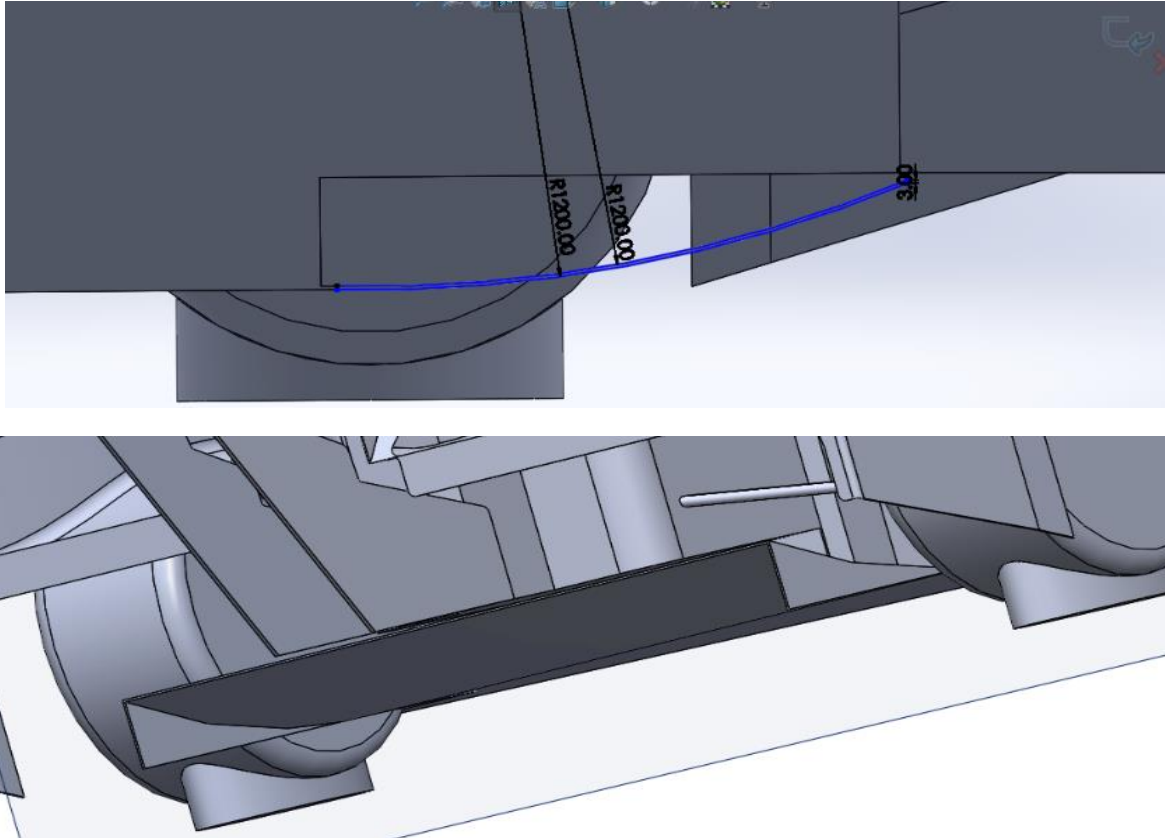


Figure 33-Design 2

This gave good results as it had a total CI of -1.17. This brought the centre of pressure to 62.25% which is a little over what is needed. However, this also gives more opportunity to develop the design to balance the centre of pressure out. This could be done with design or setup by changing the wing angle. Moreover, the C_d is also within the objectives with a value of 0.492. This is a little more than the other designs but it does produce greater downforce and a lift/drag efficiency of 2.39. In comparison to the other designs, design 1.1 had an efficiency of 2.34 and design 1 had an efficiency of 2.22, so design 2 is the most efficient. The author also began a sweep of changing the angle of the diffuser. The reason the curved diffuser is better than the straight is that it is a smoother transition from the underbody of the car. This creates less separation, so the diffuser works more effectively.

The author initially increased the radius to 2000mm. This decreased the total lift to -1.22 and increase the total drag to 0.48 bringing the efficiency to 2.54 which is an improvement again.

It also had a centre of pressure of 62.77%. This demonstrates that there can be more downforce whilst keeping a similar centre of pressure. This could be due to diffusers working by smoothly introducing the air back to freestream so the larger the radius the smoother the reintroduction is.

5 Conclusions & Evaluation

After running simulations and analysing them the following conclusions can be made:

A design was created that produced an extra 32.45% of total downforce. This was done by using an iterative process of diffuser design. The most promising final design was a curved diffuser with a radius of 2000mm and two vanes on the edges off the diffuser with a kick point at the shear plate of the car. The other two iterations were an expanding straight diffuser and full width straight diffuser. The full width diffuser performed better than the expanding as it dropped the pressure across a much larger area of the floor. The author also achieved his aim as the centre of pressure was taken to its original value of the standard car (~64%). The author also achieved the aim of keeping the drag coefficient below 0.5 having a value of 0.48. This in turn increased the efficiency of the car from ~1.85 to 2.54 which is an increase of efficiency of 37.3%. The increase in downforce is from the diffuser pumping effect where it speeds the air in front of the diffuser because the relatively low pressure under the car goes to the relatively high pressure in the atmosphere hence speeding it up. This creates a chain reaction where it drops the pressure further down the floor which then increases the pressure gradient across the car. The difference in pressure above and below the car creates a pressure difference which creates downforce. The reason it does not change the drag too much as it has none to very little effect on the frontal area and does not cause much turbulence which is the main source of drag.

The author also concluded that simplifying the geometry had a minor effect on the overall change in the lift and drag values, but it did not benefit the run time of the model. One piece of simplification that was a benefit was the removal of fillets on the louvres which made the model run much better as it reduces the skewness associated with the part. This is because the mesh is made from straight lines, therefore removing the fillet gives a straighter edge, so it is easier to mesh. This also uses less elements as the mesh does not need to have very small elements to try and get around the fillet.

One part which could have been better within the project was the process of getting the CFD methodology. The author did not take a very structured process to this which meant that the model was not fully accurate. Perhaps if the MCR model was made in a similar method to the Ahmed body then perhaps a greater accuracy could be achieved. An iterative process was not taken as it is such a complex geometry it is difficult to mesh and run so there was not enough time factored into the project timeline which can be seen in appendix A. Along with this the author believes that the timetable originally set was a bit too ambitious as he had not expected such great difficulty in getting the initial model to run.

6 Pandemic Impact

Due to unexpected time restraints from the Covid-19 virus not all studies could be finished. This is because the writer did not have access to the Ansys CFD package to run simulations and look at detailed results of some other studies previously made. One study that would have been undertaken is a diffuser radius study. As partially shown within the report, there can be a difference made from changing the radius of the diffuser. This could be used to see at which radius the optimum results could be found. Having the data for the design is also useful, so if at any point there is a need to change the aerodynamic setup, then that information is there to use. To add to this the writer would also have investigated multi-radius diffusers. As the gearbox is in the way this limits the radius of the diffuser. So perhaps there could be gains in having a steeper radiused diffuser on the side and a shallower angled one in the middle. The author would have also made a simulation of the design seen in appendix B. This design was inspired by the ground effect cars of the 1980's using two venturi tunnels inserted in the side pods of the car. The idea would be to accelerate the air in these pod's by reducing the area as stated in the literature review. Another study the author would have done is the number of channels on the diffuser. Published studies have shown that the number of channels can increase the downforce significantly with little drag penalty. This is because it helps prevent separation and therefore increases the effect of diffuser pumping (Jowsey & Passmore, 2010).

7 Further Work

The author would also suggest including within this study the start points or 'kick' point of the diffuser. This can not only change the lift and drag of the diffuser but can also change the centre of pressure. So, changing both the radius and kick point an ideal setup of highest downforce for the needed centre of pressure. Along with this, a study can be made on the length of the diffuser. As mentioned in the literature review the length of the diffuser can have a big effect on the lift created (Schuetz, 2016). In general, the longer the diffuser the smoother the transition from the low pressure under the car to the relative high pressure in the atmosphere. There is a limit, if the diffuser is too long then it can reduce the energy of the flow too much and this causes separation. There are also limitations from regulations with the length as the maximum size is 1100mm from the rear axle or 100mm from the rear body work.

Furthermore, the writer would also suggest including a study on the expansion ratio of the diffuser. There may be a possible gain in expanding the diffuser behind the rear wheels as this could increase the pumping action. Along with this the front of the underside could be optimised. Currently the start of the floor is quite sharpened edge. An interesting investigation could be to look into the benefits of optimising the inlet as it can have an effect on the rest of the floor making the air leading onto it more linear rather than turbulent.

The writer would also advise doing a ride height study. As discussed in the literature review the ride height can have a big effect on the aerodynamic qualities. So, to examine what the optimum ride height is, a sweep could be beneficial. Not only will this help with the setup of the vehicle, but it will also give an indication of how the car will react when driving around a track where the ride height is constantly changing. A rake (the angle the car is at relative to the ground) study could also help as this changes around the track as it pitches and dives under braking and accelerating. Some studies show that a high rake angle (rear higher than the front) can give good aerodynamic performance (Dzijan, et al., 2019). To add to this the angle of the incoming air could be changed. This is to simulate the car going around a corner and the incoming air is not perpendicular to the front of the car. This could be used to see if the diffuser would work going around a corner as this could become dangerous if the aerodynamic load cut out unexpectedly. The other area to examine is the roll of the vehicle. This could see how sensitive the devices are when the car is going through a corner. This could be very important as if the devices do not work going in a corner then it defeats the object of having the device in the first place. Even if the roll is very small, it picks up one side of the car and lowers the other. This again may cause the aerodynamics to drop off. The author predicts that this will however only have a negligible affect as the roll on the vehicle is so small that it would have very little effect on the aerodynamics. The author would also propose to vary the speed of the incoming air. As downforce and drag is relative to speed squared it would be beneficial to see if the design could work at low speed and high speeds (Divila, 2020). To add to that the faster the vehicle is going the more energy the fluid has. This may affect the flow structures which are formed and where it separates on the diffuser.

$$C_l = \frac{F_L}{\frac{\rho_L V^2}{2} \infty A_x} \text{Equation 8-Coefficient of Lift (Schuetz, 2016)}$$

Another piece of further work the author could recommend is improve the methodology. The method used in this report was enough to have reasonable correlation but there was still room for improvement. So, the author suggests using a mesh with a higher resolution in the floor area so that the details around the floor and diffuser can be seen more accurately. There could also be slightly more detail in the wake region so that again that can be modelled more accurately. As can be seen in some of the pressure contours previously they are a bit boxy so if smaller elements were used it could be smoother and more accurate as it picks up finer details. There could also be more research into isogeometric analysis. As mentioned previously there can be gains in the accuracy of the mesh by doing isogeometric analysis. Along with this perhaps another methodology study on the MIRA bluff body. This is a closer representation to a car so perhaps this could be used to further develop the model to make it more accurate.

If the project were taken even further, then perhaps exhaust blowing could be investigated. This is where the exit of the exhaust is in the diffuser to reenergise the air on the diffuser. This means that much greater angles can be achieved on the diffuser without separation. It also improves the pumping affect as it gets the air out from under the car quicker in turn reducing the pressure under the car (Filisetti, 2010). However, without reconfiguring the engine this would not work as it relies on the engine to be blowing exhaust gasses out and if the driver is off the throttle through the corner then it would not work. As the use is for an open series it is possible to change the engine to blow air through the exhaust when the driver is off throttle. (Motorsport.com, 2011) says there are two types, cold and hot blowing. Cold is where the engine is pumping normal air through and then hot is where there is a spark and fuel is put in the engine to give energy to the gas. This increases fuel consumption by about 15%. However, in F1 it gave a time advantage of around 1 second. As this car would be for a sprint series fuel usage is not such a great concern.

8 References

Airfoil Tools, 2019. *Airfoiltools.com*. [Online]

Available at: <http://airfoiltools.com/airfoil/details?airfoil=n0012-il>

[Accessed 22 November 2019].

Altair university, 2019. Introduction to Meshing. In: *Practical Finite Element Analysis*. s.l.:Altair University.

Ansys, 2019. *Ansys*. [Online]

Available at: <https://www.ansys.com/>

[Accessed 20 November 2019].

ANSYS, I., 2009. *RNG K-e Model*. [Online]

Available at: <https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node59.htm>

[Accessed 24 January 2020].

Bakker, A., 2006. *Lecture 7- meshing*. [Online]

Available at: <http://www.bakker.org/dartmouth06/engs150/07-mesh.pdf>

[Accessed 14 November 2019].

Benson, T., 2014. *NASA Reynolds Number*. [Online]

Available at: <https://www.grc.nasa.gov/WWW/BGH/reynolds.html>

Bishop, J., 2013. *8.2.2-PDEs: Finite Volume Method (Control Volume Approach)*, s.l.: s.n.

British Sprint Championship, 2019. *British Sprint Championship*. [Online]

Available at: <http://www.britishsprint.org/default.asp>

[Accessed 05 November 2019].

British Sprint Championship, 2019. *Britsh Sprint Championship*. [Online]

Available at: <http://www.britishsprint.org/docs/bscregs2019.pdf>

[Accessed 05 November 2019].

BRSCC, 2018. *BRSCC OSS Championship*. [Online]

Available at: <https://brscc.co.uk/formulae/oss-championship/>

[Accessed 10 September 2019].

BRSCC, 2019. *BRSCC Sporting and Technical Regulations*. [Online]

Available at: <https://brscc.co.uk/wp-content/uploads/2019/03/2019-OSS-Championship-Regs-Published-Copy.pdf>

[Accessed 05 November 2019].

Chalasani, S. & Thompson, D., 2004. Quality improvements in extruded meshes using topologically. *INTERNATIONAL JOURNAL FOR NUMERICAL METHODS IN ENGINEERING*, Volume 60, pp. 1139-1159.

Christoffersen, L., Soderblom, D. & Lofdahl, L., 2011. *Wing-Diffuser Interaction on a Sports Car*, s.l.: SAE International.

Cottrell, J., Reali, A., Bazilevs, Y. & Hughes, T., 2006. Isogeometric analysis of structural vibrations. *Computer Methods in Applied Mechanics and Engineering*, 15 August, 195(41-43), pp. 5257-5296.

Cruz, S., Godinez, F. A. & Navarrete, M., 2019. Study of a Cavitating Venturi Tube by Lumped Parameters. *Journal of Fluids Engineering*, 01 July.141(7).

Dash, P., 2006. TRAINER AIRCRAFT WING AERODYNAMIC ANALYSIS CONSIDERING TURBULENCE. *International Journal of Current Research*, 8(07), pp. 34959-34967.

Datawave Marine Solutions, 2019. *Guts of CFD: Near Wall Effects*, s.l.: s.n.

Divila, R., 2020. Forward Thinking. *Racecar Engineering*, April, 30(04), pp. 62-72.

Dzijan, I., Pasic, A., Buljac, A. & Kozmar, H., 2019. aerodynamic Forces Acting on a Race Car for Various Ground Clearances and Rake Angles. *Jorunal of Applied Fluid Mechanics*, 12(2), pp. 361-368.

Ehirim, O. H., Knowles, K. & Saddington, A. J., 2019. A Review of Ground-Effect Diffuser Aerodynamics. *Journal of Fluids Engineering*, 141(2).

Engineers Edge LLC, 2020. *coeffients of friction*. [Online]
Available at: https://www.engineersedge.com/coeffients_of_friction.htm
[Accessed 24 March 2020].

Evans, D. B., 2019. *Designing and testing the world's fastest car*,. Swansea: Swansea University.

Felfoldi, A., 2019. *What is the Venturi Effect*. [Online]
Available at: <https://www.simscale.com/blog/2018/04/what-is-venturi-effect/>
[Accessed 05 November 2019].

Filisetti, P., 2010. Blown diffusers explained. 30 June .

Gel, A. & Celik, I., 2000. *Parallel Implementation of a Commonly Used Internal Combustion Engine Code*. Amsterdam: Elsevier Science B.V..

Ghaedamini, H., 2010. *CFD analysis of Ahmed reference body by several turbulent models*, s.l.: s.n.

Haider, J. et al., 2018. *Explicit solid dynamics in OpenFoam*. Hamburg, ResearchGate.

Helicopters & Aircrafts, 2016. *Boundary Layer Separation*. [Online]
Available at: <http://heli-air.net/2016/02/21/boundary-layer-separation/>

Historic Motorsport Central, 2015. *#Focus- Ground Effect for Dummies*. s.l.:s.n.

Houghton, E., Carpenter, P., Collicott, S. H. & Valentine, D. T., 2013. *Aerodynamics for Engineering Students (Sixth Edition)*. Waltham(Massachusetts): Elsevier.

Huminic, A. & Huminic, G., 2019. Computational Study of Curved Underbody Diffusers. *XII International Conference on Computational Heat, Mass and Momentum Transfer (ICCHMT 2019)*, 08 November, 128(E3S Web Conferences).

Illey, J., 2019. *John Illey Design splitter*. s.l.:s.n.

Introduction to CFD Meshing. 2016. [Film] India: Center for Professional Development Learning and Research.

Jowsey, L. & Passmore, M., 2010. *Experimental study of multiple-channel*, Loughborough: Sage.

Katz, J., 2006. Aerodynamics of race cars. *Annu. Rev. Fluid Mech*, 01 01, 38(1), pp. 27-63.

Lanfrit, M., 2005. *Best practice guidelines for handling Automotive External Aerodynamics with FLUENT*. 1.2 ed. Darmstadt: s.n.

Martin, A., 2019. *Development of the bodywork of an MCR for S200 series*, Swansea: s.n.

McBeath, S., 2006. *Competition Car Aerodynamics, A Practical Handbook*. Sparkford(Yeovil): Haynes Publishing.

McBeath, S., 2018. Diffusing situations. *Racecar Engineering*, February, 28(2), pp. 60-68.

Meile, W. et al., 2010. *Experiments and numerical simulations on the aerodynamics of the Ahmed body*, Graz: s.n.

M, J., 2018. *What is y+ (yplus)?*. [Online]
Available at: <https://www.simscale.com/forum/t/what-is-y-yplus/82394>

Moghimy, P. & Rafee, R., 2018. Numerical and Experimental Investigations on. *Journal of Applied Fluid Mechanics*, 3 February, 11(04), pp. 1101-1113.

Motorsport UK, 2019. *British Sprint Championship*. [Online]

Available at: <http://www.britishsprint.org/>

[Accessed 04 November 2019].

Motorsport UK, 2019. *Motorsport UK*. [Online]

Available at: <https://www.motorsportuk.org/assets/bluebook2019revisedversion.pdf>

[Accessed 05 November 2019].

Motorsport.com, 2011. *Blown Diffuser- So what exactly is being banned*. [Online]

Available at: <https://www.motorsport.com/f1/news/blown-diffusers-so-what-exactly-is-being-banned/3218465/>

Multimatic Engineering, n.d. *Multimatic Engineering*. [Online]

Available at: <https://www.multimatic.com/engineering/multimatic-engineering/predictive-methods/computational-fluid-dynamics/>

[Accessed 04 November 2019].

NASA, 2015. *NASA Navier-Stoke Equation*. [Online]

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

[Accessed 14 November 2019].

Ozen Engineering Inc, 2020. *Ansys CFD-Post*. [Online]

Available at: <https://www.ozeninc.com/products/fluid-dynamics/ansys-cfd-post/>

[Accessed 03 March 2020].

Petrany, M., 2018. *Road and Track*. [Online]

Available at: <https://www.roadandtrack.com/car-shows/geneva-auto-show/a19125403/mclaren-senna-gtr-photos-specs-info/>

[Accessed 2019 June 2019].

Princeton University, n.d. [Online]

Available at: https://www.princeton.edu/~asmits/Bicycle_web/Bernoulli.html

[Accessed 18th October 2019].

Quinarelli, A., 2018. *Aerodynamic efficiency, balance and center of pressure*. [Online].

R.Stewart, V., 1987. *Effect of Ground Proximity on the Aerodynamic Characteristics of the STOL Aircraft*. s.l., SAE International.

Rapid Racer, n.d. *Rapid Racer*. [Online]

Available at: <http://www.rapid-racer.com/aerodynamic-upgrades.php>

[Accessed 19 March 2020].

Rehnberg, S., Borjesseon, L., Svensson, R. & Rice, J., 2012. *Race Car Aerodynamics - The Design Process of an Aerodynamic Package for the 2012 Chalmers Formula SAE Car*, Chalmers: SAE International.

Roelofs, F. & Shams, A., 2019. 6-CFD--Introduction. In: F. Roelofs, ed. *Thermal Hydraulics Aspects of Liquid Metal Cooled Nuclear Reactors*. s.l.:Woodhead Publisher, pp. 213-218.

Salmoiraghi, F., Ballarin, F., Heltai, L. & Rozza, G., 2016. Isogeometric analysis-based reduced order modelling for incompressible linear viscous flows in parametrized shapes. *Advanced Modeling and Simulation in Engineering Sciences*.

Schlichting, H. & Gersten, K., 2017. *Boundary-Layer Theory*. 9th ed. Berlin: Springer.

Schuetz, T., 2016. *Aerodynamics of Road Vehicles*. 5th ed. Warrendale(Pennsylvania): SAE International.

Simscale, 2017. *What is the Reynolds Number*. [Online]
Available at: <https://www.simscale.com/docs/content/simwiki/numerics/what-is-the-reynolds-number.html>
[Accessed 06 November 2019].

Stanford University, n.d. *Simulation of Turbulent Flows*. s.l.:s.n.

Sung, L. G. et al., 2014. Aerodynamic Analysis of the Preliminary Design of SURIKAR 4. *Applied mechanics and materials*, Volume 629, pp. 507-512.

Techopedia, n.d. *Techopedia*. [Online]
Available at: <https://www.techopedia.com/definition/3801/garbage-in-garbage-out-gigo>
[Accessed 13 November 2019].

The Editors of Encyclopaedia Britannica, 2018. *Encyclopaedia Britannica*. [Online]
Available at: <https://www.britannica.com/science/conservation-of-mass>
[Accessed 05 November 2019].

Toet, W., 2017. *Race Tech Motorsport Engineering*. [Online]
Available at: <https://www.racetechmag.com/2017/08/willem-toet-explains-motorsport-diffusers/>
[Accessed 03 December 2019].

Tu, J., Yeoh, G.-H. & Liu, C., 2018. Chapter 4 - CFD Mesh Generation: A Practical Guideline. In: *Computational Fluid Dynamics (Third Edition)*. s.l.:s.n., pp. 125-154.

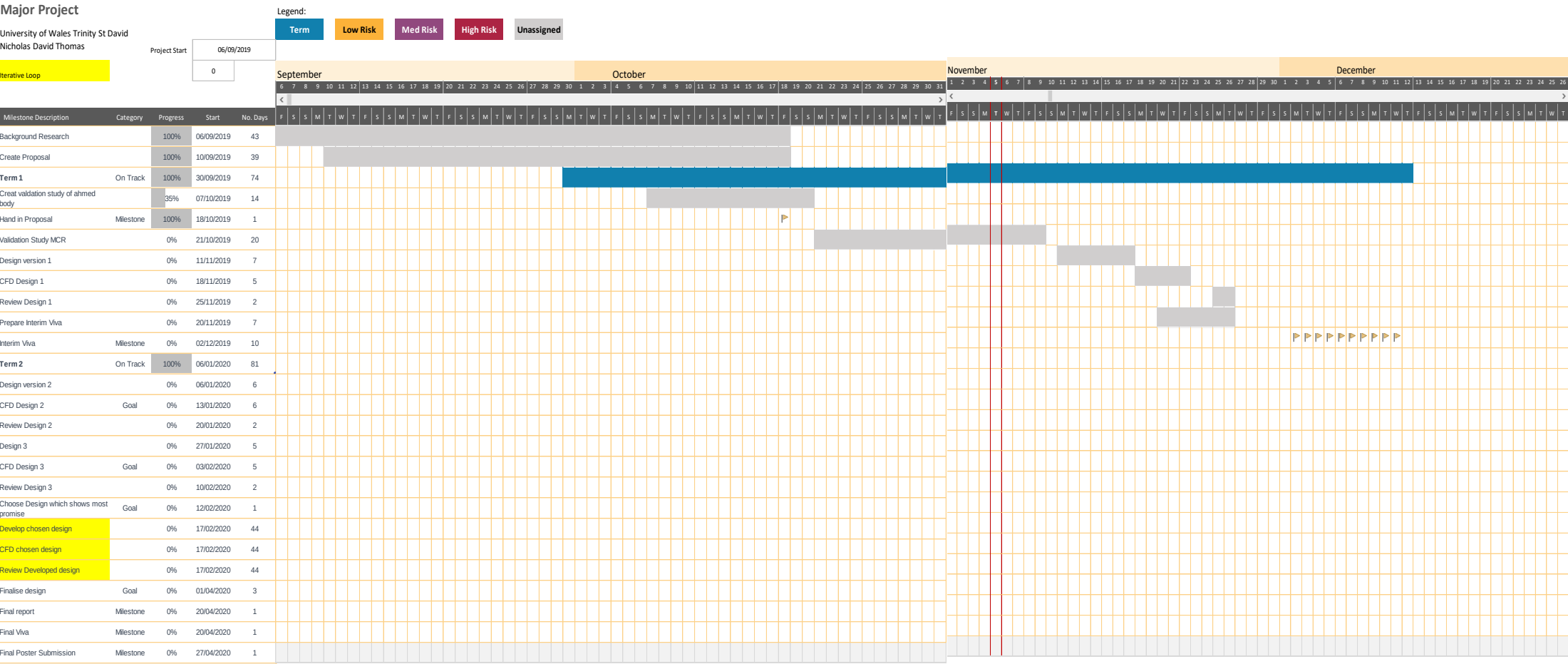
Zuo, W., 2005. *Introduction of Computational Fluid Dynamics*, St Petersburg: s.n.

9 Appendices

9.1 Appendix A

Major Project

University of Wales Trinity St David
Nicholas David Thomas



1702567

Major Project

Nicholas Thomas

Major Project

University of Wales Trinity St David
Nicholas David Thomas

Project Start

06/09/2019	
110	

Legend:

Term	Low Risk	Med Risk	High Risk	Unassigned
------	----------	----------	-----------	------------

December																															January														February														February														March														April																																											
25	26	27	28	29	30	31	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24	25	26	27	28	29	30	31	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24	25	26	27	28	29	30	31	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24	25	26	27	28	29	30	31	1	2	3	4	5	6	7	8	9	10	11	12	13	14	15	16	17	18	19	20	21	22	23	24	25	26	27	28	29	30	31
<																																																											>														<																																										>															

Milestone Description	Category	Progress	Start	No. Days
Background Research		100%	06/09/2019	43
Create Proposal		100%	10/09/2019	39
Term 1	On Track	100%	30/09/2019	74
Creat validation study of ahmed body		35%	07/10/2019	14
Hand in Proposal	Milestone	100%	18/10/2019	1
Validation Study MCR		0%	21/10/2019	20
Design version 1		0%	11/11/2019	7
CFD Design 1		0%	18/11/2019	5
Review Design 1		0%	25/11/2019	2
Prepare Interim Viva		0%	20/11/2019	7
Interim Viva	Milestone	0%	02/12/2019	10
Term 2	On Track	100%	06/01/2020	81
Design version 2		0%	06/01/2020	6
CFD Design 2	Goal	0%	13/01/2020	6
Review Design 2		0%	20/01/2020	2
Design 3		0%	27/01/2020	5
CFD Design 3	Goal	0%	03/02/2020	5
Review Design 3		0%	10/02/2020	2
Choose Design which shows most promise	Goal	0%	12/02/2020	1
Develop chosen design		0%	17/02/2020	44
CFD chosen design		0%	17/02/2020	44
Review Developed design		0%	17/02/2020	44
Finalise design	Goal	0%	01/04/2020	3
Final report	Milestone	0%	20/04/2020	1
Final Viva	Milestone	0%	20/04/2020	1
Final Poster Submission	Milestone	0%	27/04/2020	1

9.2 Appendix B

