

University of Wales Trinity Saint David

Wales Institute of Science & Art



Research Proposal

Development of a diffuser for the UWTSD Motorsports MCR Race Vehicle

Owain Williams

1922500

Contents

1	Table of Figures.....	3
2	Table of Tables.....	4
3	Introduction.....	5
3.1	Project Solution.....	6
4	Project Relevance.....	8
5	Literature Review.....	12
5.1	Key Aerodynamic Considerations.....	12
5.2	Design Strategies	14
5.3	Bluff Bodies and CFD	16
5.3.1	Design of Experiment.....	16
6	Research Methodologies	19
6.1	Modelling and Meshing.....	19
6.2	Alternative Considerations.....	20
6.2.1	Wind Tunnel Testing	20
6.2.2	Questionnaires	21
7	Results Collection and Analysis Methods.....	22
7.1	Contingency Planning.....	22
7.2	Conclusion.....	24
7.3	Evaluation.....	24
8	Bibliography.....	25
9	References	28
10	Appendices	31
10.1	Appendix 1 – Project Ideas.....	31
10.2	Appendix 2 – Project Plan to Date.....	32

1 Table of Figures

Figure 1 - Indianapolis Speedway One Lap Speed Records (Katz, 2006)	5
Figure 2 - Example of Rake Angle in a 1969 Lola T70 (Classic Drive Ltd, 2020)	6
Figure 3 - An Example Diffuser from the Williams FW19 (Williams Grand Prix Engineering Limited, 2020)	8
Figure 4 - Example of an Extended Diffuser on the Williams FW19 by Dropping the Wing Height and Removing the Main Element (Viveen, 1997)	9
Figure 5 - Example of Diffuser Side Flow Which Generates Vortices (Katz, 2006).....	12
Figure 6 - Rear Diffuser Area for Remodelled MCR Aerodynamic Package.....	14
Figure 7 - Rear Diffuser Area for Remodelled MCR Aerodynamic Package.....	15
Figure 8 - Example of MCR Initial Ride Height DoE Test Variations using Develve	17
Figure 9 - CFD Work Cycle	19
Figure 10 – CFD Process Flow Chart	19
Figure 11 - Recommended Domain Size (Simanungkalit, 2016)	20
Figure 12 - Project Ideas Mind Map	31

2 Table of Tables

Table 1 - Maximum Lap Times Achievable to Balance Vehicle Performance 9

Table 2 - Test Variables (bold areas are definite test variables, non-bold are after thoughts
if enough time is available) 9

3 Introduction

The utilisation of aerodynamics has been present in race vehicles for some 100 years (Katz, 2006), by better understanding the way air interacts with the vehicle body top speeds can be broken, corners can be taken faster, and in some cases a reduction in fuel consumption can be seen. The prevalent increase in vehicle performance that led to the science we see today in modern motorsports was seen first in the 70s as front and rear wing design pioneered from the experimentation done in the 60s, heralding cars such as the Lotus T98, Peugeot WM-P88, and Brabham BT46; into breaking what was expected and possible with race vehicles.

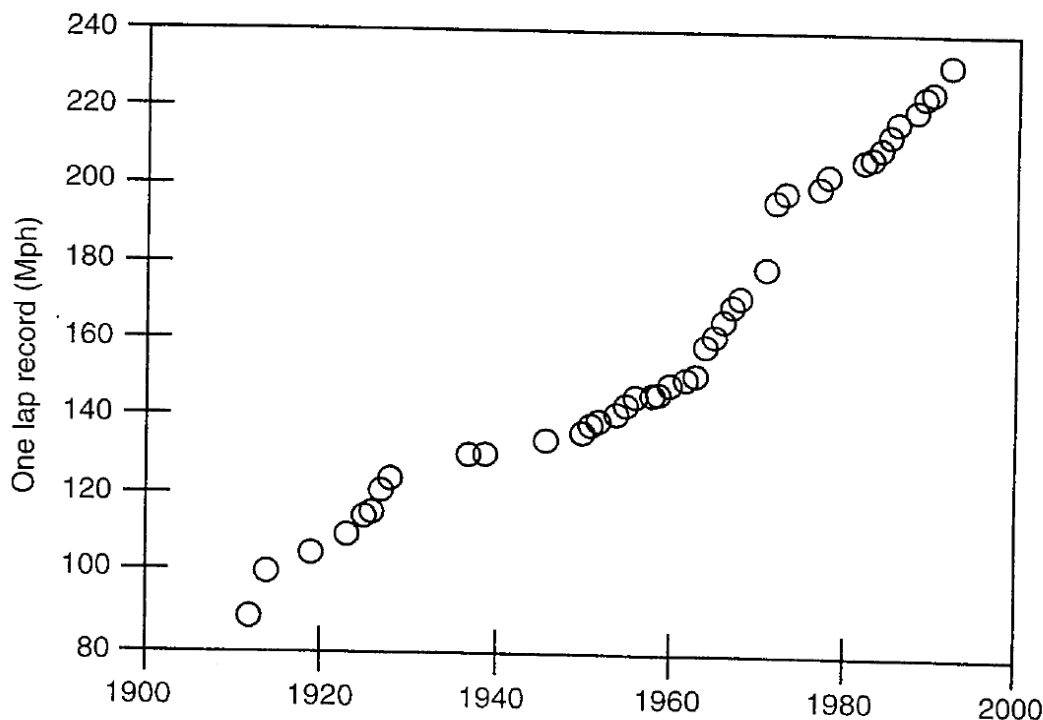


Figure 1 - Indianapolis Speedway One Lap Speed Records (Katz, 2006)

Although in the public eye the visible aspects of aerodynamics are sought to be the most important such as big wings, aggressive diveplanes, and wide front splitters, the underside of a race vehicle has equally beneficial yields if optimised properly, this project aims to quantify that in terms of downforce generated and drag reduced on the UWTSD Motorsports MCR.

With the new wind tunnel and CFD corroborated bodywork the MCR produces some 442% more downforce than before, from front to rear the bodywork has been redesigned to properly optimise;

- Front bodywork and splitters
- Pontoons
- Floor and rear deck
- Rear wing
- Cockpit

With the increased aerodynamic load these factors have generated a large balance issue within the vehicle, with a lack of underside aerodynamic aids the force balance is more front biased, which can generate a plethora of issues to be overcome by the driver and technicians such as oversteer, or difficulty setting up suspension due to such a largely differing static and driving load distribution (Smith, 1978).

3.1 Project Solution

To properly balance the vehicle the addition of a diffuser is the approach of this study. Initially understanding the theory is the first hurdle, a diffuser works by increasing the cross-sectional area of a port to aid in the change of fluid velocity, to reduce drag, the basic theory can be explained in Bernoulli's theorem.

$$P_1 + \frac{1}{2}\rho v_1^2 = P_2 + \frac{1}{2}\rho v_2^2$$

With the secondary velocity (V_2) and pressure (P_2) being a product of the change in cross sectional area, and as the velocity reduces due to increased cross-sectional area, the pressure increases allowing it to be vented to atmosphere and generate less drag (Jones, et al., 2017). Joseph Katz describes their similarity to a venturi tube, with the latter end of the tube being described as a diffuser as it diffuses the pressure between ambient and dynamic air, much like a rocket diffuser does to reduce back pressure generated by the ambient air (Jones, et al., 2017).



Figure 2 - Example of Rake Angle in a 1969 Lola T70 (Classic Drive Ltd, 2020)

Being first utilised as early as the 1930s in applications such as engine cooling and rocket motors, evidence of testing and development was seen fully as early as 1941 to better optimise vehicle trailer efficiency (Grisworld, 1941), but was dropped due to having no wartime applications. From this it was first properly utilised in the form of rake, by lifting the rear of the vehicle up this would generate a diffused area under the vehicle, with a bottleneck at the front and larger area at the rear (Katz, 2006), evident in a variety of early race vehicles such as Porsche, Lola, Dallara etc.

4 Project Relevance

The diffusers design must be considered alongside a variety of factors, namely how it interacts with the rear wing, and the existing underfloor aerodynamics. As the MCR has had a refresh of the aerodynamic package to optimally compliment the entire vehicle, optimising its performance regarding this interaction is of the utmost importance.



Figure 3 - An Example Diffuser from the Williams FW19 (Williams Grand Prix Engineering Limited, 2020)

A wide variety of angles are to be considered, some of which are too expensive, difficult to manufacture, or limited due to fitment constraints, all of these will be considered for the key reason of propelling the project to be as beneficial as possible and explore as many design routes as possible, looking towards F1 yielded a variety of interesting facts, such as that which Katz maintains that allowing the rear wing to act as an extension to the diffuser increased velocity to interact with the underside of the wing, essentially increasing the size or length of the diffuser while manoeuvring outside of regulatory or fitment constraints, he goes on to state that this can be incredibly beneficial to vehicles that utilise ground effect due to the extension of the diffuser by a wing.



Figure 4 - Example of an Extended Diffuser on the Williams FW19 by Dropping the Wing Height and Removing the Main Element (Viveen, 1997)

By modelling and designing an appropriate diffuser, the performance of the front splitter, upper body, and wing can be increased dramatically, and with the gains visible from the remodel in terms of downforce, this will hopefully be reciprocated in the development of the diffuser.

The limitations of this project stem from a few factors, the largest being time and fitment. The issue of regulations is somewhat of a null point since the regulations are incredibly open, with only the standard regulations of no aerodynamic device exceeding the ROPS, or no aerodynamic device exceeding 1100mm behind the vehicle (Motorsports UK, 2021), as well as no moveable aerodynamic device. The physical limitation that can be considered is the set times the vehicle cannot exceed, but these are up to the driver to either morally oblige to, or not worry if the vehicle isn't capable:

Table 1 - Maximum Lap Times Achievable to Balance Vehicle Performance

Circuit	Maximum Lap time
Donnington GP	1:30.1
Silverstone GP	n/a
Silverstone International	1:01
Portimao	1:45
Paul Ricard	2:05

Within the original setup 44.08% of the aerodynamic force balance was based towards the front, the newer configuration boosted this to 49.4% giving the car an almost 50/50 setup, in terms of setup a front biased centre of pressure induced oversteer, with a rearward bias yielding more straight-line stability (Smith, 1978).

The considerations for testing are detailed below, due to limitations in time not all factors can be tested, with the most pertinent highlighted with accompanying sources of research and background data.

Table 2 - Test Variables (bold areas are definite test variables, non-bold are after thoughts if enough time is available)

Testing parameter	What's adjusted	Focus	Referenced Test
--------------------------	------------------------	--------------	------------------------

Diffuser length	• The starting point (ties in with curvature radius)	• The effect on vortices development • Centre of pressure • Diffuser Volume	• Aerodynamic Effects of Indy Car Components – Katze & Garcia - 2002
Lateral Expansion	• The angle of the sides of the diffuser	• Pressure and Velocity • Surface area • Flow Separation	• Optimal Diffuser Design for Formula SAE Race Car Using an Innovative Geometry Build-up and CFD Simulation Setup with On-Track Testing Correlation – Ehirim - 2012
Straight vs Curved	• Diffuser Radius • Constant vs Non-linear Curve	• Flow Separation • Surface Area • Choke Points	• Performance Comparison of Straight and Curved Diffusers – Sullerby et al. - 1983
Overhang	• The Distance it Hangs Out the Rear of the Vehicle	• Wing Interaction • Upper body Turbulence Interaction • Diffuser Length	• Aerodynamic Effects of Indy Car Components – Katze & Garcia – 2002 • Optimal Diffuser Design for Formula SAE Race Car Using an Innovative Geometry Build-up and CFD Simulation Setup with On-Track Testing Correlation – Ehirim - 2012
Angle	• Tied in with the Radius of a Curved Diffuser • Angle where Separation Occurs	• Flow Separation • Flow Velocity •	• A Review of Ground-Effect Diffuser Aerodynamics – Ehirim - 2018
Wing Interaction	• Reducing wing height to better interact with the Diffuser • Wing Angle • Wing Distance from Rear of Car	• Active Diffuser Length • Wing underside Pressure and Velocity	• Aerodynamics of race cars – Katz - 2006

Vanes	• Spacing of Vanes • Length of Vanes	• Fluid Velocity • Drag	• Aerodynamic Effects of Indy Car Components – Katze & Garcia – 2002
Ride Height	• Minimal adjustments in ride height for low/high speed downforce • Low and high velocity airflow	• Choking and Porpoising • Overall Diffuser Angle • Effective Ground Effect Height	• Simulation and Analysis of Suspension and Aerodynamic Interactions of Race Cars – Floyd & Law – 1994
Vortices Generators	• Addition of vortices generators upstream and at the mouth of the diffuser	• Vortices turbulence • Fluid Velocity • Number of Vortices	• Aerodynamic Effects of Indy Car Components – Katze & Garcia – 2002

5 Literature Review

5.1 Key Aerodynamic Considerations

The initial consideration of the diffuser is its application as a vent, venting the high velocity underside air to atmospheric pressure to reduce drag, this is the most simplistic method of viewing it and through research it was found that there are a multitude of other methods in which they enable the car to generate more downforce and less drag.

As stated by Katz, and Smith, vortex generation and flow are intrinsic factors of diffuser operation, George and Donis develop on this describing in-depth the interaction of the upstream components of the vehicles underside, they proved that the pressure under the vehicle is controlled by the skirts, front air-dam or splitter, and rear outlet, with these factors all contributing to the vortices generation as the high pressure on the side of the vehicle travels towards the low pressure under the vehicle (fig.5), generating the vortices prevalent in diffused vehicles. These vortices aid in the elimination of flow separation behind the vehicle, and thus reduce drag. They also massively accelerate the flow downstream reducing the pressure, being further enhanced by the addition of splitter plates, generating vortices off the end of each plate opposed to just off the diffuser walls (Ehirim, et al., 2018).

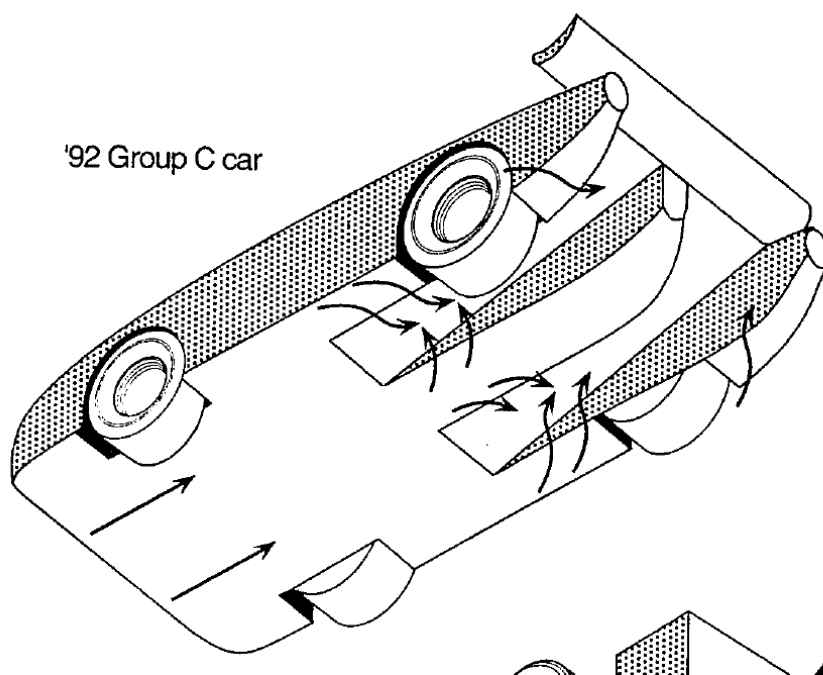


Figure 5 - Example of Diffuser Side Flow Which Generates Vortices (Katz, 2006)

In a later paper discussing the aerodynamics of a 2000 Indy car, Katz further on his previous points of component interaction, stating that the optimisation of the rear wing to work with the vortices and increased flow velocity generated by the diffuser can massively improve downforce and drag reduction. As the rear wing utilises the low pressure ejected from the diffuser, both extending the throat of the diffuser and improving wing performance, the drawback being that this also generated a high value of drag due to the increased flow under the wing, with the pertinent point additionally being that this did generate a largely

unbalanced car, due to the high concentration of downforce around the rear of the vehicle (Katz & Garcia, 2002), displaying that any number of interactions and concoctions can be thought up to improve vehicle aerodynamic performance, especially so with the relaxed regulations.

The main aspect of diffusers is the way in which they affect airflow under and out of the vehicle in terms of velocity (not regarding Reynolds Number), a sub-aspect that is utilised is how vortices can be utilised to increase flow velocity as mentioned before, but other techniques can be adopted such as those presented by Oxyzoglou, which involves the use of a second choke point just before the start of the gradient, which also helped to further accelerate air flow. This also being pertinent in the reduction boundary layers, and thus separation allowing for a greater angle as the turbulent, high velocity air suffers less from separation, but if sections were to suffer from flow separation and then reattachment, this reduction in skin friction reduces drag considerably (Ehirim, et al., 2018), this component also visible in fig.3.

5.2 Design Strategies

A multitude of existing designs are apparent within motorsports, having evolved through regulations, environments and technology, this has mean that there is no 'off the shelf' solution. Utilising different angles, positioning and constrictions will hopefully lead to a balanced and more manageable car. The current design displays a completely flat floor (fig.7), with the floor having two venturis that exit either side of the gearbox, the design is simplistic in term of it being a mixture of curves and corners and massive open areas (fig.6) so the initial study will aim to understand air velocity through this portion.

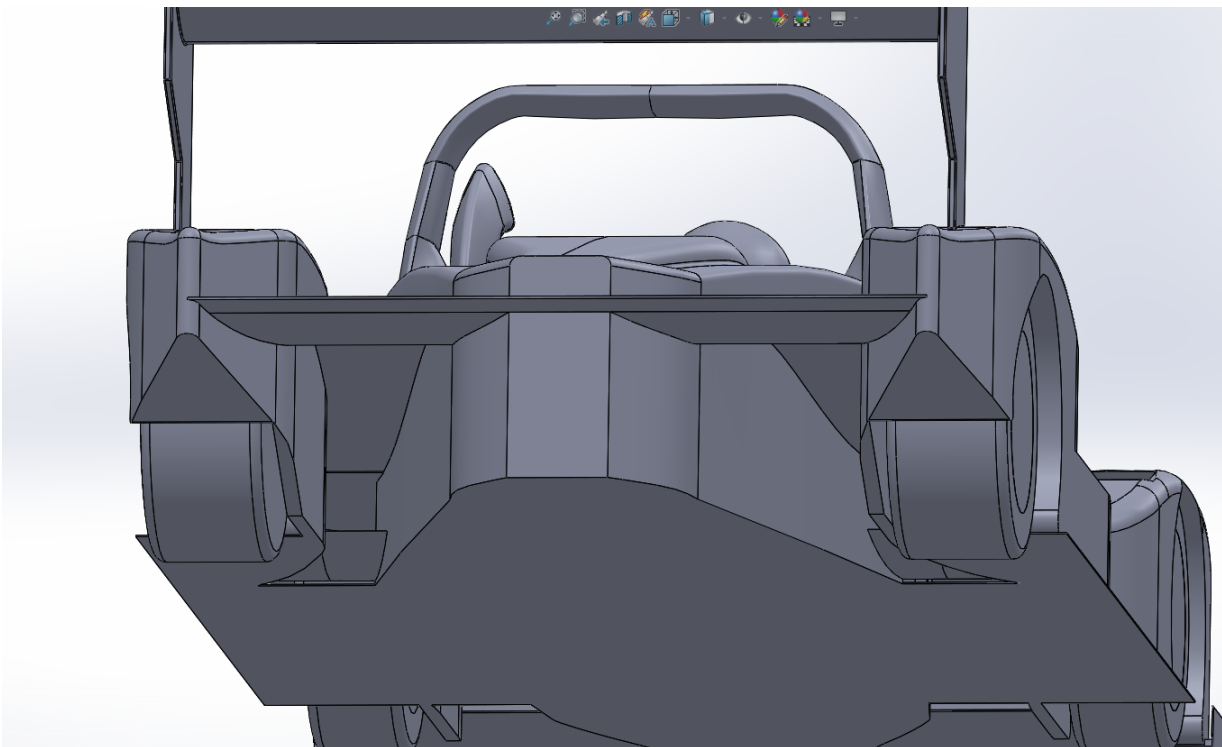


Figure 6 - Rear Diffuser Area for Remodelled MCR Aerodynamic Package

The diffuser shaping is the main consideration, looking at the basic argument of 'straight vs curved' the initial evidence that a curved edge is better is that it more closely resembles a wing profile, which is what a ground effect vehicle aims to utilise in having a wing like component very close to the ground to generate maximum downforce. In a study undertaken regarding the comparison of straight and curved diffusers it was found that straight diffusers received a higher pressure recovery as freestream turbulence increased, caused by the 'better mixing of higher stream turbulence' (Sullerby, et al., 1983), with distinct lack of flow separation due to the constant angle. Alternatively, the curved diffuser had a lower pressure recovery than the straight one due to boundary layer build up towards the rear of the diffuser, caused by the flow separation on the higher surface, finding that at more pronounced turbulence levels the curved diffuser generated less drag, this being corroborated by the inlet momentum thickness being higher in curved diffusers.

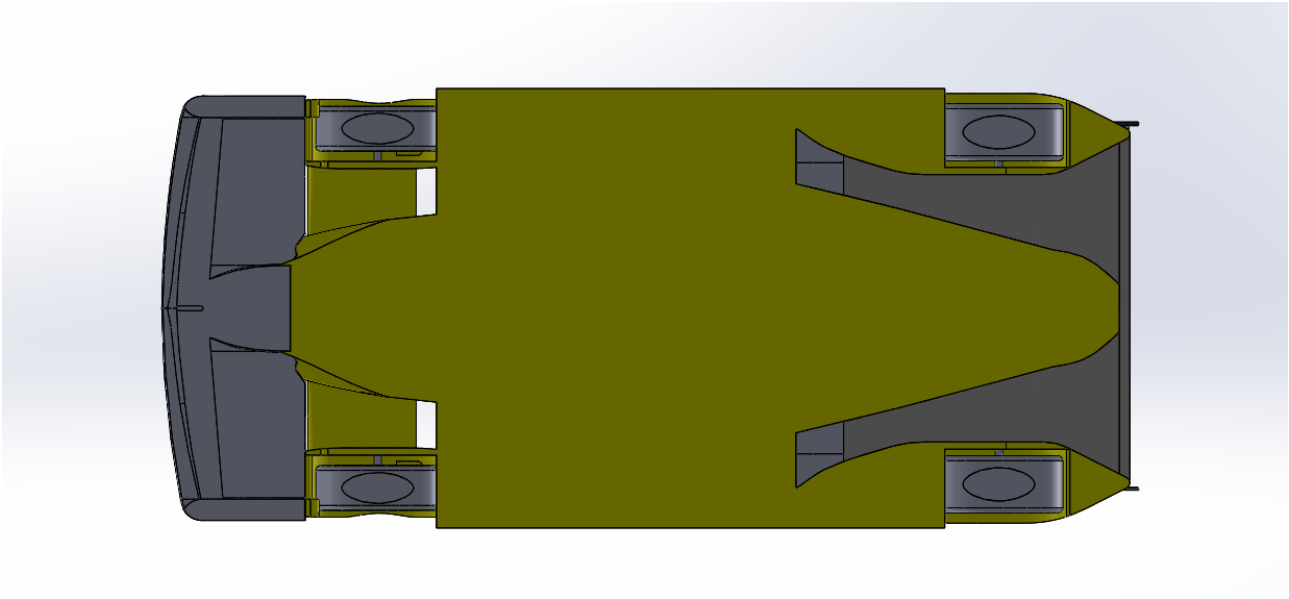


Figure 7 - Rear Diffuser Area for Remodelled MCR Aerodynamic Package

The MCR was tested with a rake setup of 0.58° , ignoring other effects this is to increase the size of the diffuser and thus the effect it has (as previously mentioned on early race vehicles). Regarding the angle of the diffuser labours the point that this angle is intrinsic to effecting the downforce generated, a development from this can be the consideration of diffuser shape, utilising a straight edged diffuser develops packaging issues, as well as an angle too large generating flow separation at the diffuser inlet (Ehirim, et al., 2018), also noting that from $5-35^\circ$ angles vortices pairs developed, while as the angle developed from 5° vortices moved closer to the inlet and strengthened, this also exacerbated with higher vehicle rake increasing the diffuser angle, which is point corroborated by Katz discussing the diffuser and vortices generators of Indy cars, without stating any specific values, but still defining that the culmination of angle, vortices generators and shape can lead to any number of characteristics of flow.

The final consideration for the study is one brought up by Dominy et al., the diffuser obviously generates specific downforce and vortices at a specific Reynolds number, utilised in testing to get the best possible level of analysis of the vehicle, the wake of the other vehicles is a potential consideration, to aid in quantifying its effect on the vehicles behaviour as it will undoubtedly effect the flow under the vehicle, as well as factors such as roll angle and pitch which will ultimately choke and modify flow under the vehicle.

5.3 Bluff Bodies and CFD

The MIRA reference vehicle functions as an aerodynamic tool for experimental and computational research (Le Good & Garry, 2004), the body is a simplistic vehicle shape, designed to be a generalised period (1970s) shape that can be modified into a hatchback or estate from the notchback to better model vehicle wake and drag (Le Good & Garry, 2004). Due to the range of research available in test results and setup it was utilised as a benchmark for the CFD to ensure reliability and estimate computing times.

Most tests researched utilised a dual-equation RANS (Reynolds Averaged Navier Stokes) model, the point being that it offers a compromise between calculation time, and computational accuracy (Palaskar, et al., 2016). Although the RANS model does fall short in terms of modelling adverse pressure gradients which could present issues should extreme flow separation come about, large eddy turbulence models (LES) can be more appropriately utilised to flow 'off-design' configurations which present larger gradients, but at the cost of more time (Jasper, et al., 2006).

Considering further the effect of flow separation presents the issue of choosing a turbulence model, its commonplace to utilise the K- ϵ turbulence model, with K representing the fluids kinetic energy and ϵ being the turbulent dissipation. Across various equations this second variable (ϵ , ω , etc) is classed as the scale of the turbulence (in terms of time or distance), the addition of transport equations helping to offset the reduction in RANS ability to model adverse pressure gradients (Bardina, et al., 1997). The appropriate modelling of this turbulence is essential as too loose of a mesh will mean that accuracy is lost in modelling the vehicles interaction and wake. This leads onto the discussion of wall functions within the model, alongside the K- ϵ turbulence model a non-equivalent wall function is often utilised, this is because like the RANS equation, it is a two part system, compensating with both kinetic energy and the turbulent dissipation, allowing a further degree of accuracy closer to the wall and boundary layer region (ANSYS. INC, 2015)

Optimisation of wall functions substantially saves computing by reducing the accuracy of turbulence and boundary layers close to the wall (ANSYS, INC., 2015), obviously this has adverse effects on the reliability. Little thought was given to which function to use as guides, papers and experts consulted all recommended the use of a non-equilibrium wall function in this application.

5.3.1 Design of Experiment

Due to the iterative approach necessary to accurately represent optimum setups of the MCRs aerodynamics, a design of experiment (DoE) will need to be undertaken to overcome the issue of testing every possible permeation and wasting time. A DoE was developed using Develve, but regarding the model design a few considerations would be needed to be made; the Box-Behnken model allows for a fractional-factorial test to be ran each with three levels (1, 0, -1), this low iterative approach allows some correlation to be

seen with a larger comparison to be made to the change of each component in a test, rather than the single adjustment of one component per test (Jasper, et al., 2006).

A	B
Front Ride Height	Rear Ride Height
0	0
-1	-1
1	-1
-1	1
1	1
0	-1
0	0
0	-1
0	1
0	1
-1	0
-1	0
1	0
1	0
0	0

Figure 8 - Example of MCR Initial Ride Height DoE Test Variations using Develve

Regarding the quantity of tests Cornell and Peipel recommend the application of a screening exercise to reduce the number of tests needed as any exceeding five means the test becomes complex, especially considering the amount of levels, the drawback to this method is that the assumption is made that the trend may be represented linearly on a surface response graph, and without using more levels, the graph may not accurately represent whether the trend is constant or curved.

In a physical wind tunnel test Roy and Landman utilised a two level DOE model, the benefit quoted was that DoE outputs a randomly generated test profile which will characterise the relationship between factors and levels, with the drawback being that using a two level methods theirs less probability of finding an interaction (Roy & Landman, 2006).

The other positive benefit to this method is the post processing, which comes in the form of regression models (as utilised by Roy and Landman) as the models are unlikely to yield constant gradients due to the complex interaction of various factors, the fallback of this is that as stated by Ando et al, due to this complex interaction any sort of response surface model (RSM) is often an untrue representation as it doesn't properly portray the correlation between component adjustment, but this factor is countered by Simpson et all in which

they state that DOE, RSM, and regression analysis are reliable methods of analysis given enough inputs due to in more advanced cases the ability to build polynomial relationships between inputs, which may often be better suited to utilising the Krigings model, but due to the complexity and time scale this would be unfeasible (Simpson, et al., 1998).

To ensure the reliability of these plots the surface plot method will be used, in which a random setup is taken, modelled, and then tested for fit, assuming the results are within the same tolerance and test procedure it should, this mediates the aforementioned issue of not knowing whether the curve and gradient is properly represented (Piepel & Cornell, 1994). The limitation compared to a chi squared test for example is the chi squares test ability to numerically define the fit of results, stating deviation from an expected value, meaning that on initial baseline tests to ensure reliability of CFD when comparing the MCR or MIRA test vehicle results to previously undertaken studies. The further limitation of the chi square tests comparably is that it can't define the cause, while a surface plot allows the trend of results to be visualised so a point-at-fault can be pinpointed, to evaluate the results of a chi square test, breaking down possible causalities in the base results is the main choice (National Institute of Standards and Technology, 2012).

6 Research Methodologies

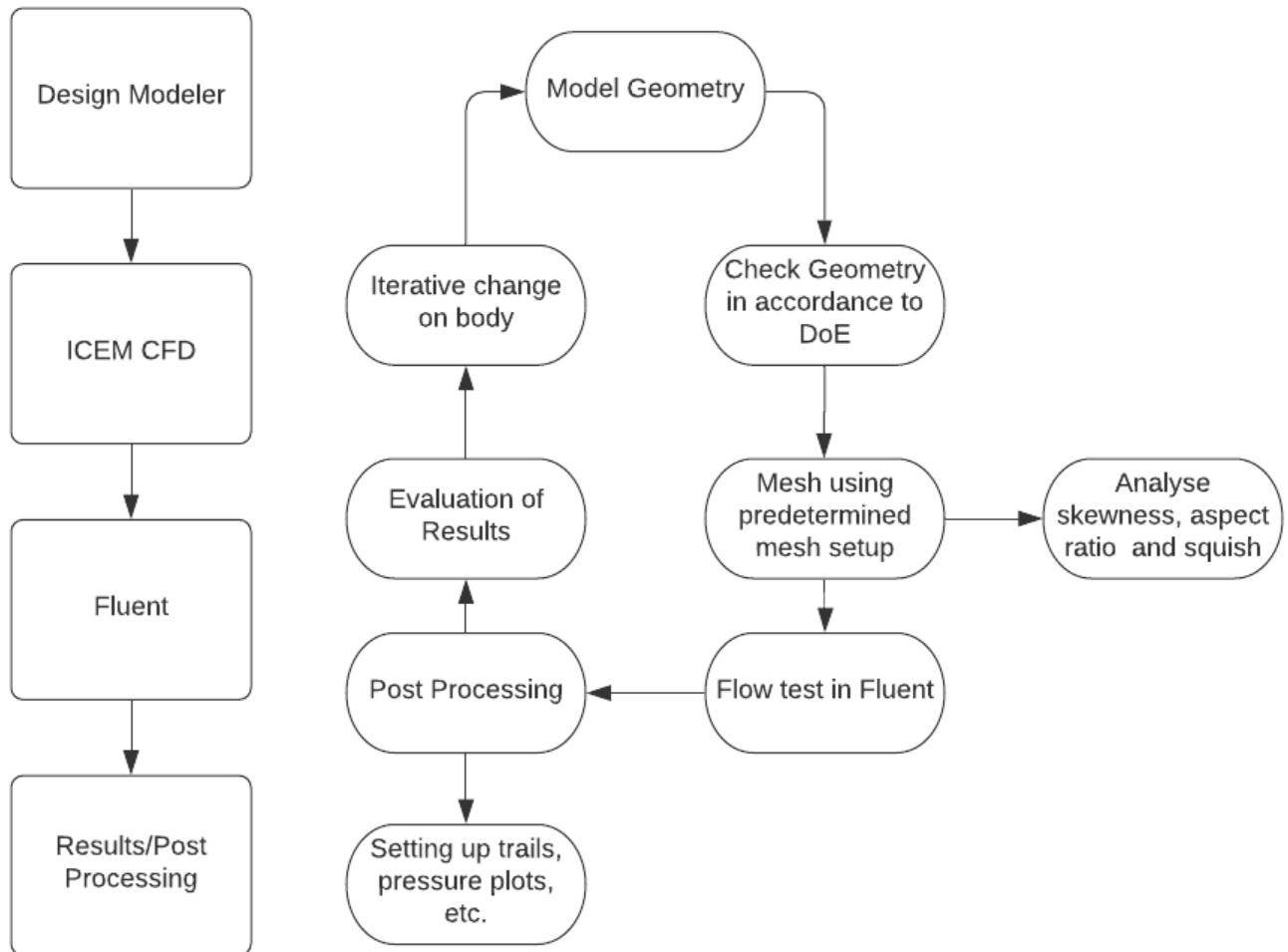


Figure 10 – CFD Process Flow Chart Figure 9 - CFD Work Cycle Chart

6.1 Modelling and Meshing

Solidworks will be the main design software due to it being a CAD package and allowing finite adjustments and appropriate tooling to generate the complex shapes necessary. Ansys will be the focus in terms of meshing though, with dedicated programmes such as ICEM.

The mesh shape and size are massively important in CFD, a hexahedral mesh may reduce computing time and have little effect on accuracy, but obviously on intricately designed components such as bodywork this method loses critical shapes and curvatures (Carrigan & Landon, 2016). Utilising a tetrahedral mesh however allows for a variety of surfaces to be mapped out in varying levels of accuracy, the level of accuracy is completely dependent on the necessity for accuracy and computing time, in the study the upper side of the vehicle will

be largely untouched, as the focus is on the rear and underside. In terms of an underlying mesh over the entirety of the model most sources recommend an initial base mesh, with as low as possible skew, aspect ratio, and squish. ANSYS recommend an aspect ratio as low as 5:1 in the bulk flow, but in terms of anisotropic flow, often higher aspect ratios can be utilised to yield accurate results with less cells (ANSYS, INC., 2015), for this project as the mesh needs to be closely matched as possible to the previous tests for continuity and ease of comparing results, not much focus will be had on these outputs, unless they signal errors and anomalous values.

Regarding the density box its main use is to “to set a local maximum size within a portion of the volume” (ANSYS, Inc, 2016), obviously the main benefit of this is to allow analysis of pressure and flow around the vehicle underside and wake region, by ensuring accurate results are outputted, this can be in the form of triangulated mesh within these sections to increase density and better map out the vehicle shape, and obviously with attention necessary to output the effect of so many iterative changes, an appropriate mesh is of the utmost importance. This density box sits within the overall domain, with the wake maintain the length of the domain but being most accurately depicted around 1-1.5 car lengths behind the vehicle.

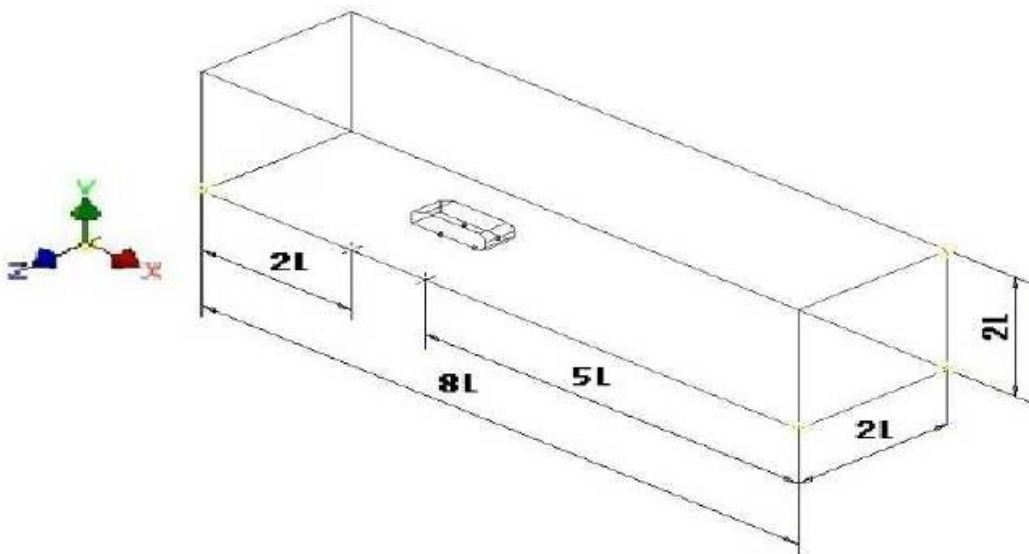


Figure 11 - Recommended Domain Size (Simanungkalit, 2016)

6.2 Alternative Considerations

6.2.1 Wind Tunnel Testing

The University wind tunnel capable of holding a scale model in wind speeds of up to 35m/s, the obvious limitation of this is that when a full vehicle is scaled down this may only prove to be an effective 10m/s wind velocity, the simple answer to this is to corroborate the vehicles low speed aerodynamics within it, for example testing the vehicle roll at low speed cornering could prove a necessary study should time allow it, to further corroborate CFD data.

6.2.2 Questionnaires

Although this study may be focusing on objective, quantative values a questionnaire could prove useful in terms of increasing reach of finding technical papers, corroborating test data and potential test resources

7 Results Collection and Analysis Methods

Within the DoE aspect of the study the main tool used will be SRM, utilising it allows trends to be seen in how adjustments are interrelated, it will be used as a simple 'idiot' check, visualising any potentially anomalous results. Developing on the use of Develve can be the comparison of results, an incredibly simplistic quantile-quantile plot can be used to ensure that the results obtained when flow testing the MIRA, and MCR are within a specified range, ideally having an R^2 around the true line of at least 0.995.

Considering the post processing further presents the issue of finite comparison, apart from the downforce and drag values differentiating the effect of diffuser aspects the main source of information will come from the flow interaction round the vehicle body, ANSYS has a host of post processing capabilities and can allow flow visualization to be completely bespoke for specific needs. To understand the effect of the change in diffuser design, the focus must be taken off statistical analysis as it was for DoE. The initial consideration is the effect on centre of pressure, with the current load being based more towards the middle returning it more rearwards generates more straight-line stability (Katz, 2006), but in turn will ultimately depend on the driver's preference. From this the pressure distribution can be mapped out to present low/high pressure areas as well as flow separation and stall points, this presented in a variety of papers as the 'go-to' methods for presenting pressure data, especially so when the results can be compared with traces emanating from the wake, underside, or from the rear wing, all sections which be necessary to analyse the flow trajectory as to better understand turbulence, vortices and flow paths.

The use of streamlines has been utilised as long as aerodynamic studies has been around (Shahnawaz, et al., 1999), but to save on computing power and reducing confusion, the use of streamline seeds will be utilised in this study, mostly because up until the front of the vehicle, the flow trajectory will be very unlikely to change, only changing amongst tests under the vehicle as the pressure will not be a constant across tests, this paired with surface plots and cut plots will be the best option for analysing the results, Shahnawaz et al. state that in smaller area to better define pressure differences modifying the colour range to stand out more is usually a viable option, with this being used opposed to a vector field as there's expected to be high turbulence and vortices and following non laminar flow with this method may become difficult and not yield as accurate presentations as needed (Ehirim, et al., 2018).

7.1 Contingency Planning

Evaluating the viability of the project proposal mainly orbits around the factor of physical testing, obtaining wool tuft, avgas and chalk, or any form of physical corroboration to determine flow characteristics is of the utmost importance, and although lap times are available, comparing some form of physical data whether its damper response or the wool tufts would allow for a better quantification of aerodynamic performance, when compared alongside CFD.

The timescale of the project has been mentioned very little due to the high number of stumbling blocks and potential deviation points. With issues in remote access connectivity,

licenses being used, and an uncertainty whether physical testing is achievable, time scaling the entire project would mean that there could be a deviation in task time of up to 2 weeks, making it difficult to do any planning apart from listing the tasks necessary to complete the project. Considerations have been made to this such as utilising Project Libra, using critical path analysis, or even planning a 'just in time' method to try and reduce lead and wait times (Burbrudge, 1971), but all of these became unfeasible with the amount of uncertainty, bar the utilisation of Project Libra to plan the project thus far, but with large amounts of deviation occurring due to the aforementioned issues above.

7.2 Conclusion

The overall scope of the project has not tried to exceed the limitations that have been set by current situations, although physical testing may be unavailable this has been circumnavigated to still deliver reliable and sensible project deliverables. The main scope of the project is to not only generate a diffuser, but to;

- Enlarge and develop understanding of aerodynamic principles not just limited to diffusers
- Develop skills and a high level of ability and critical thinking while utilising CFD software, considering the affect and necessity for using specific methods and setups
- Finish the project with a modelled diffuser and the appropriate data to back up the reasoning and methodologies utilised, allowing for a more competitive MCR race vehicle

7.3 Evaluation

Even with these considerable stumbling blocks the project still aims to supply a very much needed source of information for not only the sports2000 racing circles, but the wider sports prototype crowd. With details on the analysis, development, and evaluation of aerodynamic components it hopes to serve as a guide as although the topic may be incredibly specific, the skills, methods and software used are all cross transferable.

Furthermore, the use of DoE aims to reduce any confusion down considerably by setting out test iterations, limitations, and by giving the project a predefined scope, reducing the time of testing, as well as siphoning through hundreds of results to obtain the optimum setup and design.

8 Bibliography

ANSYS, INC., 2015. Mesh Quality. In: *ANSYS 16.2 Users Guide*. s.l.:s.n.

ANSYS, Inc, 2016. Tetra/Prism Mesh in a Fin Configuration. In: *ANSYS ICEM CFD Tutorial Manual*. Canonsburg: s.n., p. 223.

ANSYS. INC, 2015. Near-Wall Treatments for Wall-Bounded Turbulent Flows. In: *ANSYS Fluent 16.2 Users Manual*. s.l.:s.n., pp. 211-215.

Bardina, J., Huang, P. G., Coakley & T, J., 1997. *Turbulence Modeling Validation, Testing, and Development*, s.l.: NASA Technical Memorandum.

Burbrudge, J. L., 1971. *Production Planning*. 2 ed. London: Heinemann.

Carrigan, T. & Landon, M., 2016. *Meshing Considerations for Automotive Shape Design Optimization*, s.l.: Society of Automotive Engineers.

Classic Drive Ltd, 2020. 1969 Lola T70. [Online] Available at: <https://www.classicdriver.com/en/car/lola/t70/1969/786367> [Accessed 10 2 2020].

Ehirim, O., 2012. *Optimal Diffuser Design for Formula SAE Race Car Using an Innovative Geometry Buildup and CFD Simulation Setup with On-Track Testing Correlation*, s.l.: SAE 2012 World Congress & Exhibition.

Ehirim, O. H., Knowles, K. & Saddington, A. J., 2018. A Review of Ground-Effect Diffuser Aerodynamics. *Journal of Fluids Engineering*, 141(2), pp. 10-29.

Grisworld, R. W., 1941. *Visualized AERODYNAMIC RESEARCH*, s.l.: Society of Automotive Engineers.

Jasper, M., Rudell, K. & Akurai, H. S., 2006. *The Application of Design of Experiments to CFD Studies of Racecar Wing Configurations*, Fort Collins: Society of Automotive Engineers.

Jones, D. R., Allgood, D. C. & Sanders, G. C., 2017. *Passive Rocket Diffuser Testing: Reacting Flow Performance of Four Second-Throat Geometries*, s.l.: NASA.

Katz, J., 2006. Aerodynamics of race cars. *Annual Review of Fluid Mechanics*, 38(1), pp. 27-63.

Katz, J. & Garcia, D., 2002. *Aerodynamic Effects of Indy Car Components*, s.l.: Society of Automotive Engineers.

Kitoh, K., Hokkaido, O., Yamamoto, N. & Sebben, S., 2009. *A CFD Approach via Large Eddy Simulation to the Flow Field with Complex Geometrical Configurations: A Study Case of Vehicle Underbody Flows*, s.l.: The Society of Automotive Engineers.

Law, E. H. & Floyd, R. S., 1994. *Simulation and Analysis of Suspension and Aerodynamic Interactions of Race Cars*, Clemson: Society of Automotive Engineers.

Le Good, G. M. & Garry, K. P., 2004. *On the Use of Reference Models in Automotive Aerodynamics*, s.l.: Society of Automotive Engineers.

Motorsports UK, 2021. *Motorsport UK Yearbook 2021*. Bicester: s.n.

National Institute of Standards and Technology, 2012. *Chi-Square Goodness-of-Fit Test*. [Online]

Available at: <https://www.itl.nist.gov/div898/handbook/eda/section3/eda35f.htm> [Accessed 3 3 2020].

Oxyzoglou, I., 2017. *Design & Development of an Aerodynamic Package for an FSAE Race Car*, s.l.: s.n.

Palaskar, P. M., Vaidya, R. & Kumar, V., 2016. *Methodology Development to Accurately Predict Aerodynamic Drag and Lift for Passenger Vehicles Using CFD.*, s.l.: Society of Automotive Engineers.

Piepel, G. F. & Cornell, J. A., 1994. *MIXTURE EXPERIMENT APPROACHES-OPTIONS, EXAMPLES, AND RECOMMENDATIONS (shortened version)*, Richland: s.n.

Rafee, R. & Moghimi, P., 2018. Numerical and Experimental Investigations on Aerodynamic Behavior of the Ahmed Body Model with Different Diffuser Angles. *Journal of Applied Fluid Mechanics*, 11(4), pp. 1101-1113.

Roy, J. & Landman, D., 2006. *Aerodynamic Characterization of a Sportscar Prototype Racecar Using Design of Experiments in a Wind Tunnel Test*, s.l.: Society of Automotive Engineers.

Shahnawaz, V. J., Vance, J. M. & Kutti, S. V., 1999. *Visualization of Post-Processed CFD Data in a Virtual Environment*, s.l.: Iowa State University Digital Repository.

Simanungkalit, S., 2016. Effect of Active Control by Blowing to Aerodynamic Drag of Bluff Body Van Model. *International Journal of Fluid Mechanics Research*, 40(4), pp. 312-323.

Simpson, T. W., Korte, J. J., Mauery, T. M. & Mistress, F., 1998. *COMPARISON OF RESPONSE SURFACE AND KRIGING MODELS FOR MULTIDISCIPLINARY*. s.l.:Symposium on Multidisciplinary Analysis and Optimization.

Smith, C., 1978. *Tune To Win*. Fallbrook: Aero Publishers.

Sullerby, R. K., Chandra, B. & Muralidhar, V., 1983. *Performance Comparison of Straight and Curved Diffusers*, Kanpur: Department of Aeronautical Engineering, Indian Institute of Technology.

Toet, W., 2017. *Willem Toet explains....motorsport diffusers*. [Online] Available at: <https://www.racetechmag.com/2017/08/willem-toet-explains-motorsport->

diffusers/

[Accessed 7 3 2021].

Viveen, M., 1997. *Atlas F1 News (Autosports archive)*. [Online]
Available at: <http://atlasf1.autosport.com/news/1997/jul1997.htm>

[Accessed 12 2 2020].

Williams Grand Prix Engineering Limited, 2020. *Williams Conference Centre*. [Online]
Available at: <https://www.williamsconferencecentre.com/>

[Accessed 10 2 2020].

9 References

- ANSYS, INC., 2015. Mesh Quality. In: *ANSYS 16.2 Users Guide*. s.l.:s.n.
- ANSYS, Inc, 2016. Tetra/Prism Mesh in a Fin Configuration. In: *ANSYS ICEM CFD Tutorial Manual*. Canonsburg: s.n., p. 223.
- ANSYS. INC, 2015. Near-Wall Treatments for Wall-Bounded Turbulent Flows. In: *ANSYS Fluent 16.2 Users Manual*. s.l.:s.n., pp. 211-215.
- Bardina, J., Huang, P. G., Coakley & T, J., 1997. *Turbulence Modeling Validation, Testing, and Development*, s.l.: NASA Technical Memorandum.
- Burbrudge, J. L., 1971. *Production Planning*. 2 ed. London: Heinemann.
- Carrigan, T. & Landon, M., 2016. *Meshing Considerations for Automotive Shape Design Optimization*, s.l.: Society of Automotive Engineers.
- Classic Drive Ltd, 2020. 1969 Lola T70. [Online] Available at: <https://www.classicdriver.com/en/car/lola/t70/1969/786367> [Accessed 10 2 2020].
- Ehirim, O., 2012. *Optimal Diffuser Design for Formula SAE Race Car Using an Innovative Geometry Buildup and CFD Simulation Setup with On-Track Testing Correlation*, s.l.: SAE 2012 World Congress & Exhibition.
- Ehirim, O. H., Knowles, K. & Saddington, A. J., 2018. A Review of Ground-Effect Diffuser Aerodynamics. *Journal of Fluids Engineering*, 141(2), pp. 10-29.
- Grisworld, R. W., 1941. *Visualized AERODYNAMIC RESEARCH*, s.l.: Society of Automotive Engineers.
- Jasper, M., Rudell, K. & Akurai, H. S., 2006. *The Application of Design of Experiments to CFD Studies of Racecar Wing Configurations*, Fort Collins: Society of Automotive Engineers.
- Jones, D. R., Allgood, D. C. & Sanders, G. C., 2017. *Passive Rocket Diffuser Testing: Reacting Flow Performance of Four Second-Throat Geometries*, s.l.: NASA.
- Katz, J., 2006. Aerodynamics of race cars. *Annual Review of Fluid Mechanics*, 38(1), pp. 27-63.
- Katz, J. & Garcia, D., 2002. *Aerodynamic Effects of Indy Car Components*, s.l.: Society of Automotive Engineers.
- Kitoh, K., Hokkaido, O., Yamamoto, N. & Sebben, S., 2009. *A CFD Approach via Large Eddy Simulation to the Flow Field with Complex Geometrical Configurations: A Study Case of Vehicle Underbody Flows*, s.l.: The Society of Automotive Engineers.

Law, E. H. & Floyd, R. S., 1994. *Simulation and Analysis of Suspension and Aerodynamic Interactions of Race Cars*, Clemson: Society of Automotive Engineers.

Le Good, G. M. & Garry, K. P., 2004. *On the Use of Reference Models in Automotive Aerodynamics*, s.l.: Society of Automotive Engineers.

Motorsports UK, 2021. *Motorsport UK Yearbook 2021*. Bicester: s.n.

National Institute of Standards and Technology, 2012. *Chi-Square Goodness-of-Fit Test*. [Online]

Available at: <https://www.itl.nist.gov/div898/handbook/eda/section3/eda35f.htm> [Accessed 3 3 2020].

Oxyzoglou, I., 2017. *Design & Development of an Aerodynamic Package for an FSAE Race Car*, s.l.: s.n.

Palaskar, P. M., Vaidya, R. & Kumar, V., 2016. *Methodology Development to Accurately Predict Aerodynamic Drag and Lift for Passenger Vehicles Using CFD.*, s.l.: Society of Automotive Engineers.

Piepel, G. F. & Cornell, J. A., 1994. *MIXTURE EXPERIMENT APPROACHES-OPTIONS, EXAMPLES, AND RECOMMENDATIONS (shortened version)*, Richland: s.n.

Rafee, R. & Moghimi, P., 2018. Numerical and Experimental Investigations on Aerodynamic Behavior of the Ahmed Body Model with Different Diffuser Angles. *Journal of Applied Fluid Mechanics*, 11(4), pp. 1101-1113.

Roy, J. & Landman, D., 2006. *Aerodynamic Characterization of a Sportscar Prototype Racecar Using Design of Experiments in a Wind Tunnel Test*, s.l.: Society of Automotive Engineers.

Shahnawaz, V. J., Vance, J. M. & Kutti, S. V., 1999. *Visualization of Post-Processed CFD Data in a Virtual Environment*, s.l.: Iowa State University Digital Repository.

Simanungkalit, S., 2016. Effect of Active Control by Blowing to Aerodynamic Drag of Bluff Body Van Model. *International Journal of Fluid Mechanics Research*, 40(4), pp. 312-323.

Simpson, T. W., Korte, J. J., Mauery, T. M. & Mistress, F., 1998. *COMPARISON OF RESPONSE SURFACE AND KRIGING MODELS FOR MULTIDISCIPLINARY*. s.l.:Symposium on Multidisciplinary Analysis and Optimization.

Smith, C., 1978. *Tune To Win*. Fallbrook: Aero Publishers.

Sullerby, R. K., Chandra, B. & Muralidhar, V., 1983. *Performance Comparison of Straight and Curved Diffusers*, Kanpur: Department of Aeronautical Engineering, Indian Institute of Technology.

Toet, W., 2017. *Willem Toet explains....motorsport diffusers*. [Online] Available at: <https://www.racetechmag.com/2017/08/willem-toet-explains-motorsport->

diffusers/

[Accessed 7 3 2021].

Viveen, M., 1997. *Atlas F1 News (Autosports archive)*. [Online]
Available at: <http://atlasf1.autosport.com/news/1997/jul1997.htm>

[Accessed 12 2 2020].

Williams Grand Prix Engineering Limited, 2020. *Williams Conference Centre*. [Online]
Available at: <https://www.williamsconferencecentre.com/>

[Accessed 10 2 2020].

10 Appendices

10.1 Appendix 1 – Project Ideas

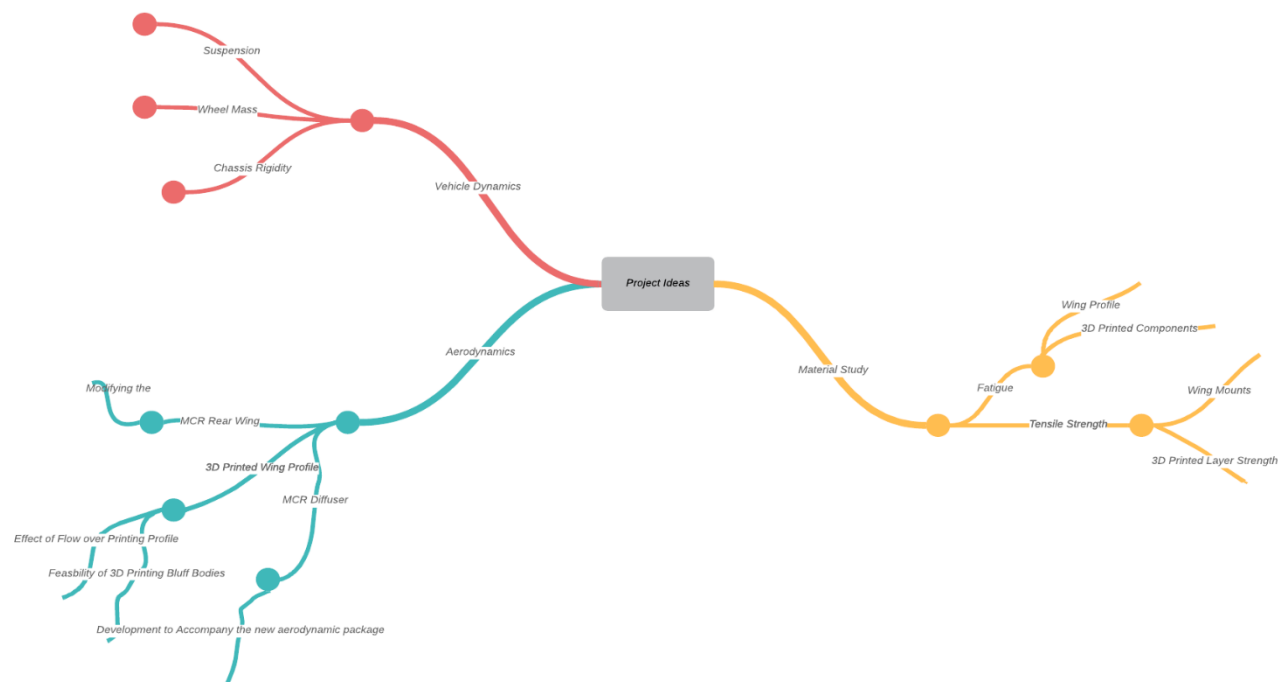


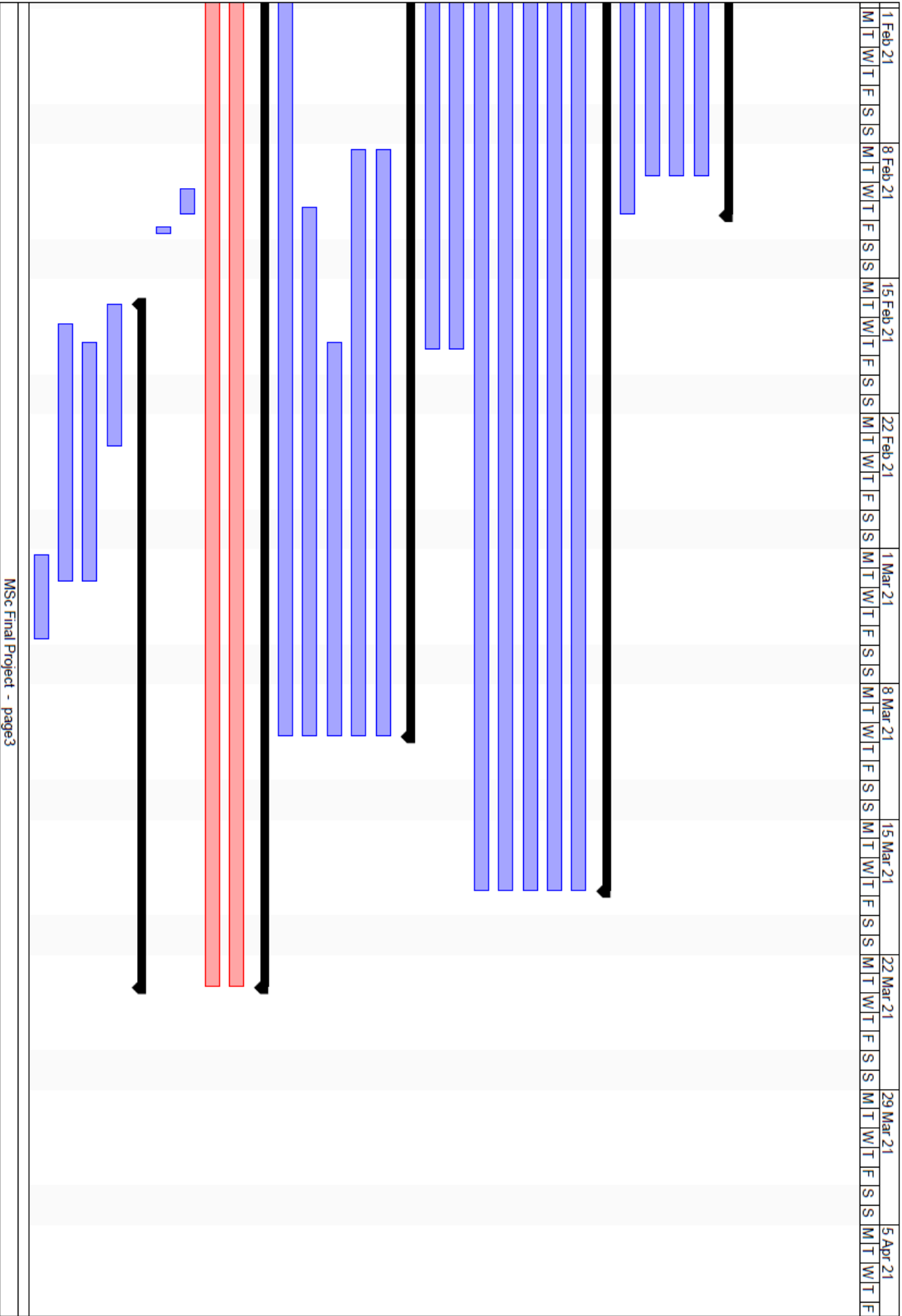
Figure 12 - Project Ideas Mind Map

10.2 Appendix 2 – Project Plan to Date

		Name	Duration	Start	Finish	Predecessors
1		Project Ideas	23 days?	08/12/20 08:00	07/01/21 17:00	
2		Brain Storm	7 days?	08/12/20 08:00	16/12/20 17:00	
3		Mind Map	8 days?	14/12/20 08:00	23/12/20 17:00	
4		Light Research of Potential Topics	23 days?	08/12/20 08:00	07/01/21 17:00	
5		Finalise Decision of Topic	5 days?	01/01/21 08:00	07/01/21 17:00	
6		Preliminary Research Topic Research	27 days?	06/01/21 08:00	11/02/21 17:00	
7		Compile and Evaluate Literature Review Sources	25 days?	06/01/21 08:00	09/02/21 17:00	
8		Background Research Regarding CFD	22 days?	09/01/21 08:00	09/02/21 17:00	
9		Background Research Regarding Aerodynamics	23 days?	08/01/21 08:00	09/02/21 17:00	
10		Potential Project Direction Research Deepens	27 days?	06/01/21 08:00	11/02/21 17:00	
11		Chosen Topic: New Diffuser Development	49 days?	09/01/21 08:00	18/03/21 17:00	
12		Research History of Motorsports Aerodynamics	49 days?	09/01/21 08:00	18/03/21 17:00	
13		Research Theory of Diffusers	48 days?	12/01/21 08:00	18/03/21 17:00	
14		Research History of Diffusers in Motorsports	48 days?	12/01/21 08:00	18/03/21 17:00	
15		Research Papers Regarding Diffusers	48 days?	12/01/21 08:00	18/03/21 17:00	
16		Research Papers Regarding CFD	49 days?	09/01/21 08:00	18/03/21 17:00	
17		Research Papers Regarding CFD in Motorsports	25 days?	15/01/21 08:00	18/02/21 17:00	
18		Research Papers of CFD in Motorsports Diffuser Development	25 days?	15/01/21 08:00	18/02/21 17:00	
19		Consideration of Design of Experiment	37 days?	19/01/21 08:00	10/03/21 17:00	
20		Research Regarding Pros and Cons of DoE	23 days?	08/02/21 08:00	10/03/21 17:00	
21		Research Regarding DoE within CFD	23 days?	08/02/21 08:00	10/03/21 17:00	
22		Practise and Development of DoE models	15 days?	18/02/21 08:00	10/03/21 17:00	
23		Research Limitations and Understanding of Chi Square Test	20 days?	11/02/21 08:00	10/03/21 17:00	
24		Understanding of Comparison Between Chi Square Test and S...	37 days?	19/01/21 08:00	10/03/21 17:00	
25		ANSYS: Introduction	69 days?	17/12/20 08:00	23/03/21 17:00	
26		Follow Tutorials to Run MIIRA Reference Car on ANSYS	69 days?	17/12/20 08:00	23/03/21 17:00	
27		Further Research Meshing, Wall Functions, and Turbulence M...	46 days?	19/01/21 08:00	23/03/21 17:00	
28		Initial Run on MCR - New Bodywork	2 days?	10/02/21 08:00	11/02/21 17:00	
29		Evaluation of Results/Post Processing	1 day?	12/02/21 08:00	12/02/21 17:00	
30		Project Proposal	26 days?	16/02/21 08:00	23/03/21 17:00	
31		Initial Writeup/Skeleton	6 days?	16/02/21 08:00	23/02/21 17:00	
32		Filtering of Appropriate Sources Into Text	9 days?	18/02/21 08:00	02/03/21 17:00	
33		Build up of Paragraphs/Text	10 days?	17/02/21 08:00	02/03/21 17:00	
34		Initial Version Reviewed/Modified	5 days?	27/02/21 08:00	05/03/21 17:00	

The Gantt chart displays a project schedule from December 7, 2020, to January 25, 2021. The header row lists dates and day abbreviations (S, M, T, W, T, F, S) for every seven days. The chart features several horizontal bars representing tasks, color-coded in blue, red, and black. A thick black arrow points from the start of the project to the end of the first major task block.

Task	Start Date	End Date	Color
Task 1	Dec 7, 2020	Dec 14, 2020	Blue
Task 2	Dec 14, 2020	Dec 21, 2020	Blue
Task 3	Dec 21, 2020	Dec 28, 2020	Blue
Task 4	Dec 28, 2020	Jan 4, 2021	Blue
Task 5	Jan 4, 2021	Jan 11, 2021	Blue
Task 6	Jan 11, 2021	Jan 18, 2021	Blue
Task 7	Jan 18, 2021	Jan 25, 2021	Blue
Task 8	Dec 7, 2020	Dec 14, 2020	Red
Task 9	Dec 14, 2020	Dec 21, 2020	Red
Task 10	Dec 21, 2020	Dec 28, 2020	Red
Task 11	Dec 28, 2020	Jan 4, 2021	Red
Task 12	Jan 4, 2021	Jan 11, 2021	Red
Task 13	Jan 11, 2021	Jan 18, 2021	Red
Task 14	Jan 18, 2021	Jan 25, 2021	Red



MSc Final Project - page8