

Parametric Diffuser Development for the MCR S2000 with the use of CFD

Christos David Zois

P146231

University of Wales Trinity Saint David

School of Engineering

Major Project

M6X00417

25/01/19



Abstract

In this project, the MIRA reference car was used to develop and validate the CFD process and create an initial CFD settings template. Following this, the flow around a provided model of the MCR S2000 car was simulated. Geometry alterations/simplifications and mesh refinement were conducted to optimally capture the flow around the model. Boundary conditions identical to those TotalSim used to simulate the MCR model were setup and tested. The TotalSim data provided for the MCR model allowed for its validation. An initial attitude study was performed. The initial geometry was altered by lowering the wing to create a baseline adhering to the OSS Championship regulations. Parametric studies were performed analysing several diffuser parameters, allowing for a step by step geometry build up. Initially the central tunnel of the diffuser was developed followed by the side tunnels. An in depth strake study was performed. Efforts were made to reduce tyre squirt by altering the geometry around the rear wheels. Finally an optimal diffuser design was proposed and compared to the initial model without it.

Table of Contents

Parametric Diffuser Development for the MCR S2000	1
Abstract	2
Abbreviations	5
Table of Figures	6
Introduction.....	9
1 Aims & Objectives	9
2 Literature Review	9
2.1 Aerodynamic Principles.....	9
2.1.1 Drag & Lift	9
2.1.2 Boundary Layer	10
2.1.3 Bernoulli's Equation	10
2.2 Diffuser	10
2.3 Tyre Squirt.....	12
2.4 Centre of Pressure	12
2.5 CFD (ANSYS)	12
2.5.1 Process	13
2.5.2 Maths.....	13
2.5.3 Mesh quality	13
2.6 OSS Championship Regulations.....	13
3 Methodology.....	14
3.1 MIRA car Studies	14
3.1.1 Validation Criteria and Initial Settings	15
3.1.2 Domain Size Study.....	15
3.1.3 Mesh Study	17
3.1.4 Turbulence Model Study	18
3.1.5 Prism Layer Study	19
3.1.6 Polyhedra Study.....	20
3.1.7 Results Comparison to Wind Tunnel Data	21
3.1.8 Initial Optimal CFD Settings Template	21
3.2 MCR S2000 car Validation Study (Baseline)	22

3.2.1	Initial Model Setup.....	22
3.2.2	Initial Run	23
3.2.3	Wheel Blocks.....	23
3.2.4	Connected ROPS.....	24
3.2.5	Vehicle Surface Mesh Refinement.....	24
3.2.6	Density Box Study	24
3.2.7	Moving Floor and Rotating Wheels	25
3.2.8	Comparison to TotalSim Data	26
3.2.9	Final Model Setup	27
3.3	MCR car Baseline Attitude Study.....	27
3.3.1	Ride Height Study	28
3.3.2	Rake Angle Study.....	29
3.3.3	Optimum Baseline Setup	31
3.4	OSS Baseline	31
3.4.1	New Splitter	31
3.4.2	Results & Analysis.....	31
4	Diffuser Development.....	32
4.1	Centre Section.....	32
4.1.1	Diffuser Profile Study 1	33
4.1.2	Length Study	36
4.1.3	Concluded Optimum Design (Centre section)	38
4.2	Side Tunnels	38
4.2.1	Diffuser Profile Study 2 (Side Tunnels).....	39
4.2.2	Kick Point Study	43
4.2.3	Design Reconsideration	45
4.2.4	Length Study 2	47
4.2.5	Concluded Optimum Design (so far).....	49
4.3	Strakes	49
4.3.1	Centre Section.....	49
4.3.2	Side Tunnels	51
4.4	Other Studies	54

4.4.1	Lateral Expansion.....	54
4.4.2	Diffuser Flow Optimisation (Tyre Squirt Study).....	55
4.5	Proposed Final Diffuser Design	60
4.6	Final Attitude Study	62
4.6.1	Results & Analysis.....	62
4.6.2	Rake Angle.....	63
4.7	New Splitter Final Study	64
4.7.1	Results & Analysis.....	64
5	Conclusion.....	65
6	Recommended Further Work.....	66
7	References	67
8	Appendices.....	69
8.1	Appendix 1	69
8.2	Appendix 2	69

Abbreviations

CFD: Computational Fluid Dynamics

Cl: Coefficient of Lift

Cd: Coefficient of Drag

CoP: Centre of Pressure

CoG: Centre of Gravity

AR: Area Ratio

AoA: Angle of Attack

ROPS: Roll Over Protection System

Table of Figures

Figure 1. Boundary layer velocity representation.....	10
Figure 2. Lift vs AoA for a wing in and out of ground effect	11
Figure 3. Basic CFD process steps	13
Figure 4. MIRA reference car	14
Figure 5. Smallest (top) & largest (bottom) domain sizes tested	16
Figure 6. Cl & Cd vs Domain Length	16
Figure 7. Cl & Cd vs Mesh Scale Factor.....	18
Figure 8. Turbulence Model comparison to wind tunnel data	19
Figure 9. Cl & Cd vs No. of Prism Layers	20
Figure 10. Cl plot comparison: MIRA wind tunnel vs CFD.....	21
Figure 11. Domain & density box sizing (MCR).....	22
Figure 12. Wheel Blocks	23
Figure 13. Connected ROPS	24
Figure 14. Cl & Cd vs Mesh Density	25
Figure 15. Progress of Cd & Cl	26
Figure 16. Progress of CoP.....	26
Figure 17. Ride height study graphs: Cl & Cd (left), CoP & Efficiency (right).....	28
Figure 18. Pressure contours (left), velocity contours of nose (middle), velocity contours underfloor (right), of 35mm (top) & 15mm (bottom) ride heights.....	29
Figure 19. Rake angle study graphs: Cl & Cd (left), CoP & Efficiency (right).....	30
Figure 20. Pressure contours: 0 rake (left), 0.2 rake (middle), 0.6 rake (right).....	30
Figure 21. MCR S2000 diffuser centre section packaging considerations	32
Figure 22. Diffuser centre section 1st study setup (left) 2nd study setup (right)	33
Figure 23. Diffuser profiles: Radiused (left), Angled (right).....	33
Figure 24. Diffuser centre section radius vs Cl & Cd (left), CoP & Efficiency (right)	34
Figure 25. Comparison of velocity (left) and pressure contours (right) of OSS baseline (top) vs radius 1200mm (bottom).....	34
Figure 26. Diffuser centre section angle vs Cl & Cd (left), CoP & Efficiency (right)	35
Figure 27. Comparison of velocity (left) and pressure contours (right) of radius 1200mm (top) vs angle 8° (bottom).....	36
Figure 28. Vortex generation comparison of radius 1200mm (left) vs angle 8° (right)	36
Figure 29. Diffuser centre section 200mm length increase	37
Figure 30. Diffuser centre section overhang vs Cl & Cd (left), CoP & Efficiency (right)	37
Figure 31. Comparison of velocity (left) and pressure contours (right) of 0mm overhang (top) vs 300mm overhang (bottom).....	38
Figure 32. LMP1 & 2 aero. regulations (left), LMP inspired side tunnel design on MCR (right).....	39
Figure 33. Diffuser side tunnels packaging restrictions	39
Figure 34. Radius profile of diffuser side tunnels.....	40
Figure 35. Diffuser side tunnel radius vs Cl & Cd (left), CoP & Efficiency (right)	40

Figure 36. Pressure contours (left) & velocity contours (right) at underfloor level of the 6000mm radius diffuser (top) vs the optimal centre section setup	41
Figure 37. Angle profile of diffuser side tunnels.....	41
Figure 38. Diffuser side tunnel angle vs Cl & Cd (left), CoP & Efficiency (right)	42
Figure 39. Pressure contours (left) & velocity contours (right) at underfloor level of the 6000mm radius diffuser (top) vs the 8° diffuser angle (bottom)	43
Figure 40. Vortex generation comparison between 6000mm radius (left) and 8° angle (right)	43
Figure 41. Initial design (left), kickpoint 300mm back (centre), kickpoint 300mm forward (right).....	44
Figure 42. Diffuser side tunnel kickpoint movement vs Cl & Cd (left), CoP & Efficiency (right)	44
Figure 43. Velocity contours at end of diffuser (left) and vortex generation (right) for the initial model (top), the 300mm forward kickpoint (middle) and the 300mm back kickpoint (bottom)	45
Figure 44. Initial design (left), reduced angle design (centre), no angle design (right)	46
Figure 45. Vortex generation of initial design (left), reduced angle design (centre), no angle design (right).....	46
Figure 46. Velocity contours level with the rear of the MCR wheels (left) and pressure contours of underfloor (right) of Initial design (top), reduced angle design (middle), no angle design (bottom)	47
Figure 47. Length study 2 design example (125mm overhang)	47
Figure 48. Diffuser length vs Cl & Cd (left), CoP & Efficiency (right).....	48
Figure 49. Diffuser pressure contour comparison of 125mm overhang (left) and 250mm overhang (right)	48
Figure 50. End of diffuser velocity contours comparison of 125mm overhang (left) and 250mm overhang (right).....	48
Figure 51. Centre section strake study example (2 strakes)	49
Figure 52. No. of strakes vs Cl & Cd (left), CoP & Efficiency (right).....	50
Figure 53. Pressure contours of 1 strake (top left), 2 strakes (bottom left), 3 strakes (top right), 4 strakes (bottom right)	50
Figure 54. Vortex generation of no strakes (left) vs 2 strakes (right).....	50
Figure 55. Single strake setup (left), double strake setup (middle), strake depth of strake design v1 (right).....	51
Figure 56. Strake Depth of strake design v2.....	51
Figure 57. Single strake setup v3 (left), double strake setup v3 (middle), v3 strake design 3D view ..	51
Figure 58. Side tunnel strake configuration vs Cl & Cd (left), CoP & Efficiency (right).....	52
Figure 59. Pressure contours of underfloor (top) and velocity contours of diffuser ending (bottom) of no strakes (left), 1st version 1 strake (middle) and 1st version 2 strakes (right)	53
Figure 60. Pressure contours of underfloor (top) and velocity contours of diffuser ending (bottom) of no strakes (left), 2nd version 1 strake (middle) and 2nd version 2 strakes (right).....	53
Figure 61. Pressure contours of underfloor (top) and velocity contours of diffuser ending (bottom) of no strakes (left), 3rd version 1 strake (middle) and 3rd version 2 strakes (right)	54
Figure 62. No lateral expansion (left), angled lateral expansion (middle), radiused lateral expansion (right).....	54

Figure 63. Velocity contours at 20mm above underfloor level of no lateral expansion (left), angled lateral expansion (middle) and radiused lateral expansion (right)	55
Figure 64. Original bodywork (left), body modification 1 tyre shelf (middle), body modification 2 (right)	55
Figure 65. Pressure contours of underfloor of initial model with RW (left), tyre shelf (v1) with RW (middle), cutaway (v2) with RW (right)	56
Figure 66. Tyre shelf pressure contours	57
Figure 67. Vortex generation of central diffuser of initial design (top) vs version 2 (bottom)	57
Figure 68. Vortex generation of entire diffuser of initial design (left) vs version 2 (bottom)	57
Figure 69. MCR wake of unaltered design (top) vs Version 2 (bottom).....	58
Figure 70. Underbody flow streamlines bottom view (colour: velocity), no body alterations (top) & rear wheel well cutaway (bottom)	59
Figure 71. Underbody flow streamlines side view (colour: velocity), no body alterations (top) & rear wheel well cutaway (bottom)	60
Figure 72. Final proposed diffuser design side view (top), underfloor view (bottom)	61
Figure 73. Final proposed diffuser design rear view	61
Figure 74. Final proposed diffuser design 3D views	62
Figure 75. Ride height vs Cl & Cd (left), CoP & Efficiency (right)	62
Figure 76. Velocity contours of nose and central tunnel kickpoint (left & middle) and pressure contours of underfloor at 35mm ride height (top) and 15mm ride height (bottom)	63
Figure 77. Rake angle vs Cl & Cd (left), CoP & Efficiency (right)	63
Figure 78. Velocity contours (left), underfloor pressure contours (right) at 0.4° of rake (top) and at 1° of rake (bottom)	64
Figure 79. Underfloor velocity contours of proposed diffuser design with new splitter (top) and without new splitter (bottom).....	65

Introduction

Aerodynamics and racing met many years ago. Initially vehicle designers would use aerodynamics to achieve higher top speeds thanks to lower drag coefficients. In the 1960s, some engineers/racers decided to try and use aerodynamics, not just to increase straight-line speed, but to also maintain that speed through the corners to try and reduce lap times. The first person to attempt this was Jim Hall in his Chaparral 2E CanAm race car in 1966, by adding an inverted aerofoil to his car. Ever since then aerodynamics has grown to be a staple in motorsport. Over the years, many methods have been developed to try design the optimum aerodynamic packages for race cars, such as track testing, wind tunnel testing and CFD.

1 Aims & Objectives

The aim of this project is to use CFD to parametrically develop a new diffuser design for the MCR S2000 car, adhering to the OSS Championship regulations.

The project's two main objectives are:

- 1) To increase downforce while improving aerodynamic efficiency.
- 2) To bring CoP further rearwards to achieve an aerodynamic balance of 50/50, because of a new splitter design producing significantly more frontal downforce producing an aerodynamic balance of 69/31.

2 Literature Review

2.1 Aerodynamic Principles

An object travelling through free stream air is generally thought to produce two types of forces; aerodynamic drag and aerodynamic lift.

Katz observed, that a vehicles forward motion will set, the otherwise still air, into motion. This causes small pressure differences around the vehicle, the joint effect of which are responsible for the aerodynamic loads. (Katz, 2006)

2.1.1 Drag & Lift

Drag is the force that resists the motion of an object travelling through a fluid(air) and is caused by friction-like forces that act parallel to the surface of it, due to its viscous characteristics. The more viscous a fluid is, the more force is required to push a body through the fluid. Furthermore, drag is affected by the size of the vehicle. A smaller vehicle will disturb less air, hence producing lesser reaction forces from the air than a larger one at identical speeds. (McBeath, 2017)

Lift is the vertical force acting on a moving object created by the pressure differential surrounding it. Race cars are required to produce negative lift, also known as 'downforce'. To accomplish this, certain devices are added to the vehicle to create the pressure differential needed. Increasing downforce means increasing the vertical force acting on the vehicle. The loads on the tires will

increase without increasing the vehicles weight, leading to enhanced cornering ability with no weight penalty, resulting in faster cornering speeds, reducing lap times. (Katz, 2006)

2.1.2 Boundary Layer

The air particles adjacent to the surface of a vehicle have zero velocity. This is a result of the no-slip condition and viscosity. The particles in close proximity to the surface have a lower velocity than those in the free-stream air due to the shear forces generated between them because of viscosity. After a certain distance from the body surface the shear forces no longer interact with the surface layer and the particles travel at free-stream velocity. The layer in which these forces take place is called the Boundary Layer. The boundary layer initially is laminar and gradually, as it moves along the body surface losing energy due to friction, becomes turbulent. Thus, its thickness increases as it moves from the front to the rear of the car. This can cause blockage at low ride heights between the underbody and the ground, significantly effecting underbody flow and performance. (Desai, et al., 2008)

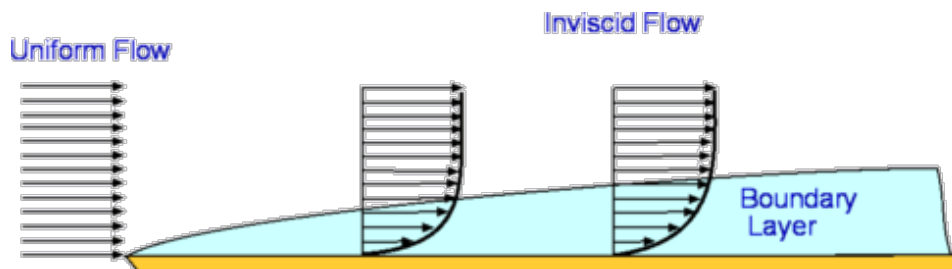


Figure 1. Boundary layer velocity representation

2.1.3 Bernoulli's Equation

Bernoulli's efforts resulted in an equation that related air speed to pressure. As described by Katz, Bernoulli's equation states that if the air speed changes around an object, then the pressure will change in an opposite trend to the square of the air speed. In simpler terms, if the speed of the flow increases around a car, the pressure will drop. (Katz, 2006)

2.2 Diffuser

Underbody diffusers have been used in racing for a number of years to generate large amounts of downforce for a minimal penalty in drag, increasing braking, cornering and acceleration performance. (Wright, 1982)

The concept of how a diffuser works has evolved over time. The 3 most common mechanisms that describe the forces produced by a diffuser that literature suggests are Diffuser Pumping, Ground Interaction and Upsweep.

1) Diffuser Pumping

A diffuser is a device whose cross sectional area increases in the direction of flow, decreasing fluid speed from inlet (diffuser kickpoint – leading edge) to outlet (trailing edge), with an equivalent increase in pressure. This upsurge of pressure can be utilised to increase the flow rate through the

system, which equates to higher underbody flow velocity reducing underbody pressures. So, the diffuser is thought to “pump-down” the underbody, increasing downforce. (Cooper, et al., 1998) This concept was presented first by Sovran in 1994 at the Pinninfarina Workshop. (Sovran, 1994)

2) Ground Interaction

In ideal flow, if a symmetrical body was brought into ground proximity, the flow around it would become asymmetrical as the flow beneath it would accelerate as a result of ground constriction. This increase in velocity under the body leads to lower underbody pressure and an increase in downforce. The smaller the distance between body and ground, the larger the increase in downforce. Downforce would increase to infinity as ride height approaches zero, as shown in Figure 2 below.

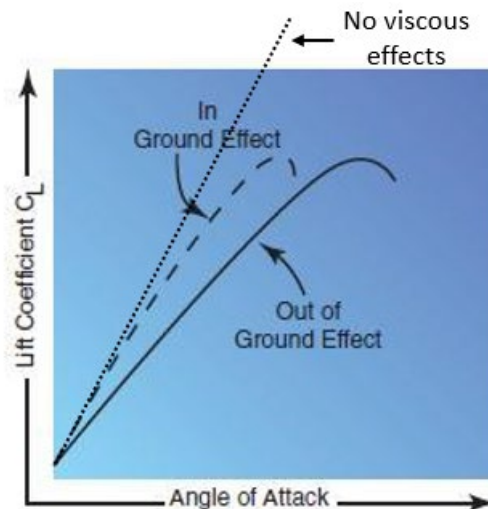


Figure 2. Lift vs AoA for a wing in and out of ground effect

Though, realistic flows are not inviscid. Viscosity at high ride heights has minimal effects, but these effects increase as ride height reduces, restricting and eventually terminating underbody flow at really low ride heights. (Cooper, et al., 1998)

3) Upsweep

Underbody upsweep has the opposite effect of a fastback vehicle. Adding an upsweep to an otherwise symmetrical vehicle shape will arc the body, ensuing in a downwards force acting on the car. (Katz, 2006)

One of the most important parameters of a diffuser is its angle. At low angles the flow of the underbody remains attached throughout the diffuser; increasing this angle though, the flow/boundary layer requires more energy to remain attached to the diffuser. When the flow separates, it turns turbulent, being detriment to the diffuser's performance.

Recently, as part of an article for Race Tech Magazine, Willem Toet expanded on how the functionality of diffusers in motorsport varies to those in the general automotive industry. Motorsport diffusers can reach significantly higher angles compare to regular diffusers (up to 20°), before stalling. This is due to the purposeful generation and use of vortices in the diffuser, hence why diffuser

analysis shouldn't be carried out just as a 2D but also as a 3D phenomenon. Unni emphasizes that a diffuser's effect cannot be explained simply by the Venturi effect, for the same reason (3D flow). (Toet, 2017) (Unni, 2017)

Vortices act in a fully three dimensional fashion as stated by George & Donis, hence the amplified interest in them by aerodynamics engineers designing race car underbodies. Trailing edge vortices were considered to have two main effects, first was the increase of negative lift (-Cl) compare to an underbody without, and second was the prevention of diffuser stalling at low ride heights. (George & Donis, 1983)

Finally, another vital diffuser parameter is the Area Ratio (AR) (how much air expansion occurs in the diffuser). AR is equal to the ratio between the areas at the diffuser kickpoint and the diffuser's trailing edge, using the ground as the barrier. Toet suggests that this ratio should not exceed 5 when designing a motorsport diffuser. (Toet, 2017)

2.3 Tyre Squirt

The flow around the tyres of a vehicle has a significant effect on its aerodynamic performance, especially the rear tyres, due to (in most cases of a race car) a diffuser running between them. Generally, when airflow enters the underbody diffuser after its kickpoint/throat, it reduces the diffuser's effectiveness of expansion. This flow "rolls up" to form vortices that can be used to increase downforce in certain cases. When a tyre is next to a diffuser, the flow ahead of it is diverted laterally due to the adverse pressure gradient. This is known as tyre squirt. This "dirty", low energy turbulent flow in conjunction with the edge vortex can have detrimental effects to a diffuser's performance. (Christoffersen, et al., 2011) (Unni, 2017)

2.4 Centre of Pressure

Centre of Pressure (CoP) caused by aerodynamic forces depicts the aerodynamic balance of a vehicle; basically specifying the longitudinal percentage split of downforce applied to the car's axles (front & rear). The importance of CoP increases at higher speeds, typically when aerodynamic forces are perceptible. The relative position of CoP to the Centre of Gravity (CoG) plays a significant role at these speeds, hence why it's an important aspect to consider when developing an aerodynamic device for a race car. For a stable and predictable race car at high speeds, the CoP is preferred to be placed behind the CoG to reduce any lateral disturbance (e.g. side winds). In the case of mid-engine race car with the CoG being significantly further back than that of a front-engine car, the previous statement can be reconsidered, and a 50/50 aerodynamic balance is preferred. (Katz, 2006)

2.5 CFD (ANSYS)

CFD, also known as Computational Fluid Dynamics, is a powerful, very regularly used tool in engineering. In motorsport and specifically aerodynamics, it is used to predict the flow around a race car and calculate the forces acting upon it.

2.5.1 Process

The CFD process includes 5 core steps:

1. Design the model to be tested in CAD.
2. Import the model into Design Modeler to define the domain in which it will be simulated.
3. Import Design Modeler file into ICEM meshing to mesh it.
4. Import the mesh into Fluent solver, setup boundary conditions and run the simulation, until the results reach convergence.
5. Import solver data into CFD Post to conduct post processing.

This process is an iterative one. Figure 3 on the right displays a basic pattern of the process.

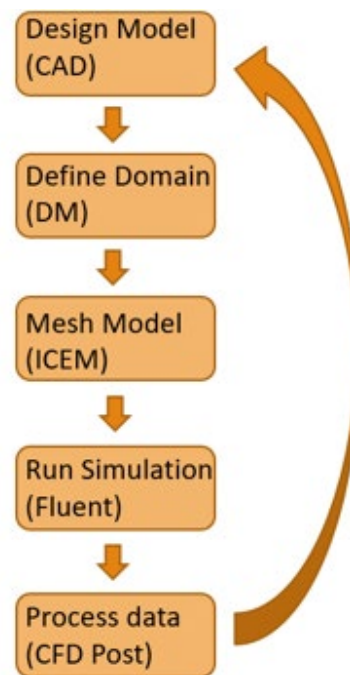


Figure 3. Basic CFD process steps

2.5.2 Maths

In order for CFD to produce this data it uses a number of equations to try and represent the fundamental principles. These are the principles of conservation of Mass, Momentum and Energy. Viscosity, boundary layer development and flow separation require an additional set of equations known as the Navier-Stokes equations, which specifically handle fluid momentum. Turbulence models try to compile these equations and use them as seen fit depending on the situation. Different turbulence models are more or less suitable for a given situation. (McBeath, 2017)

2.5.3 Mesh quality

Mesh quality plays an important role in CFD. Not capturing the important details of the geometry, not capturing the boundary layer and the difficulty to converge are all aspects of bad mesh quality. To improve mesh quality certain steps can be carried out:

- Check the mesh quality criteria such as mesh orthogonal quality and mesh skewness; perform mesh refinement/improvement if needed.
- Take into consideration the model and solver settings before generating the mesh.
- Perform a parametric mesh study.

Using these steps an acceptable mesh quality can be generated. (ANSYS, 2012)

2.6 OSS Championship Regulations

- 1) Minimum ride height is 25mm.
- 2) Devices bridging the gap between the underside of the vehicle and the road are not permitted

- 3) The tyre must not be visible from above and when viewed from the rear must be covered to at least 50mm above the axle centreline.
- 4) Aerodynamic devices must not extend beyond the maximum width of the vehicle, above the maximum height of the ROPS and longitudinally must not extend from the rear wheel axis by more than 1100mm. (BRSCC, 2018) (MotorsportUK, 2018)

3 Methodology

3.1 MIRA car Studies

CFD as mentioned earlier, simulates the flow around an object. Simulation is “the imitative representation of the functioning of one system or process by means of the functioning of another” (Merriam-Webster, 2018). So, in this case we have a computer simulation of the flow structure development around a race car. In order to generate such a simulation, human input is needed, which means the results are prone to have errors.

To try and minimize this generation of skew results and produce a baseline settings template for the CFD simulations to follow, a number of studies were conducted. A domain, a mesh, a turbulence model, a prism layer and a polyhedra study took place. The model used for these studies was the MIRA reference car, which is known to be the most popular model of a simplified car geometry used for the calibration of wind tunnels. It is based on the proportions of a 1980s “family sized” car and was designed and introduced by G. W. Carr. (Le Good & Garry, 2004)

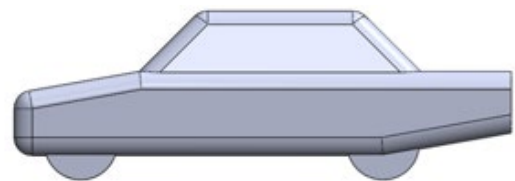


Figure 4. MIRA reference car

Experimental data for the MIRA reference car was provided by the MIRA wind tunnel. This wind tunnel is a full-scale, closed test section, open return wind tunnel. Its test section is 15m in length, 7.9m wide and 4.4m high, which estimates to a cross sectional area of $35m^2$. (MIRA, 2019)

The MIRA car was tested at 20m/s in the MIRA wind tunnel, hence the studies in this section used an inlet air speed of 20m/s.

Once the studies were complete, the results were compared and correlated to the data provided by the MIRA wind tunnel for the MIRA reference car, to try and validate the settings template suggested.

3.1.1 Validation Criteria and Initial Settings

The settings template generated will have to meet certain criteria:

- The results attained during each study must not be affected when altering a specific settings parameter i.e. increasing or reducing the length of the domain by 1m should not affect the results produced.
- Time and processing power limitations need to be considered i.e. a super fine mesh in the entire domain would increase the number of mesh cells significantly, leading to an increase in processing power and time to solve the solution required. So, to optimize the balance between computational power available and the time limit of this project, the mesh would be refined only in critical areas, such as the wake of the vehicle, and not throughout the entire domain.

The initial settings used as a starting point were suggested by Lanfrit in his “Best practice guidelines for handling automotive external aerodynamics using Fluent” (Lanfrit, 2005). These were:

- Domain size: 3 car lengths ahead of the vehicle and 5 car lengths behind for optimum wake structure development were suggested. 3 car widths on either side and 3 car heights above.
- Mesh: A maximum domain mesh size of 1000mm, a density box mesh of 100mm and a vehicle surface mesh of 10mm.
- Turbulence model: K-epsilon Realizable with non-equilibrium wall functions
- No. of Prism layers: 5

Additionally, an ideal density box size was suggested:

- Density Box size: Half a car length ahead, a full car length behind, half a car width to the side and half a car height above. (Piechna, et al., 2009)

The rest of the settings were left as default.

3.1.2 Domain Size Study

The first study performed was the domain size study. For this study and all following studies regarding the MIRA reference car, the domain was halved down the middle as the model is symmetrical. This allows for a faster solving time without effecting the results obtained.

As mentioned above, the initial domain size was set up based on Lanfrit's suggestions. The initial dimensions of the domain were: 36m long, by 6m wide, by 7m tall, with the car sitting at 12m behind the inlet.

The density box sizing was 10m long, by 1.5m wide, by 1.7m tall. For all studies the density box size remained unaltered.

The domain was scaled up and down in size, while the proportions remained the same. For this study 4 domain sizes were tested. Below in Figure 5, the largest and smallest domains tested are displayed.

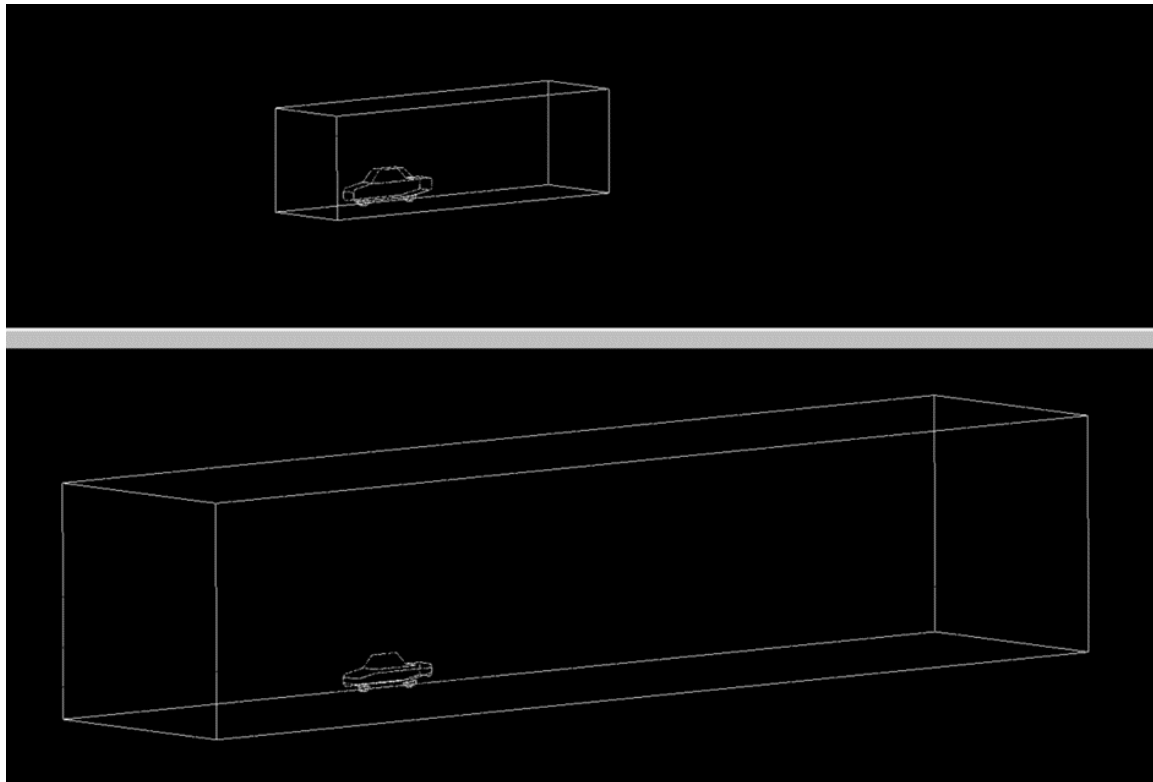


Figure 5. Smallest (top) & largest (bottom) domain sizes tested

3.1.2.1 Results & Analysis

Table 1. Domain Study

Domain Study			
Size (m)	Distance to inlet (m)	Cd	Cl
54 x 9 x 10.5	18	0.260	-0.080
36 x 6 x 7	12	0.264	-0.083
24 x 4 x 4.6	8	0.275	-0.128
16 x 2.6 x 3.1	5.3	0.301	-0.189

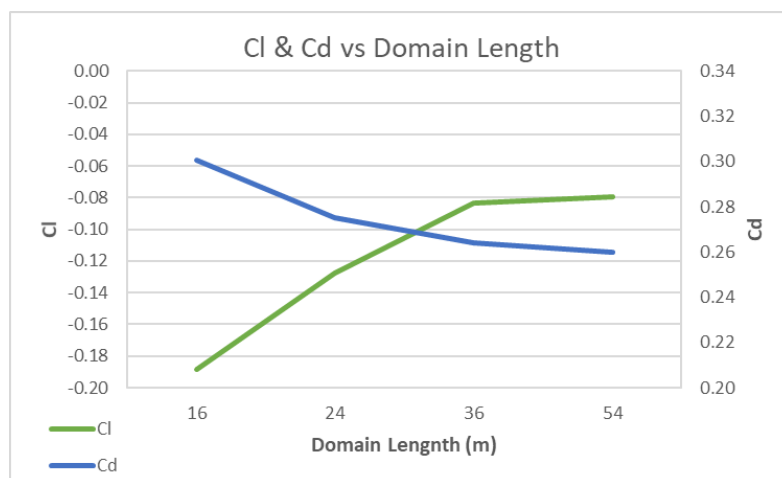


Figure 6. Cl & Cd vs Domain Length

Observing Table 1 & Figure 6 above, it is clear that the results for C_l and C_d from a 36m long domain to a 54m long domain vary at an insignificant rate, due to the blockage effects being negligible. (McBeath, 2017) For that reason, to keep computational effort low while still achieving acceptable results, the initial domain size was chosen as the optimum one for this study.

So, the domain size selected for the settings template was:

36m Length x 6m Width x 7m height, with an inlet to model distance of 12m.

3.1.3 Mesh Study

The next study conducted was the mesh study. As the optimal domain size was the original one, the settings (apart from mesh) remained identical for this study. The mesh elements used were tetrahedrals (unstructured mesh) and not hexahedrals (structured mesh), due to the complexity of the body. The meshing method used was the Robust Octree method.

The initial mesh settings, as mentioned earlier, were 1000mm maximum domain element size, 100mm density box element size, and 10mm vehicle surface mesh element size. ICEM meshing, the program used to create the mesh, has an integrated mesh scaling feature, which allows the user to scale the mesh. Using this feature the mesh was scaled from 0.85(fine) to 3(coarse) in increments. A similar mesh independence study was performed by Unni when investigating diffuser strakes. (Unni, 2017)

3.1.3.1 Results & Analysis

Table 2. Mesh Study

Mesh Study			
Scale Factor	No. of Elements	C_d	C_l
0.85	10486055	0.263	-0.086
0.9	8913963	0.263	-0.086
0.95	7795683	0.264	-0.085
1	6641163	0.264	-0.083
1.1	5112861	0.264	-0.083
1.2	3903076	0.267	-0.082
1.3	3162319	0.279	-0.074
1.5	2132616	0.286	-0.069
2	966803	0.302	-0.057
3	348005	0.327	-0.040

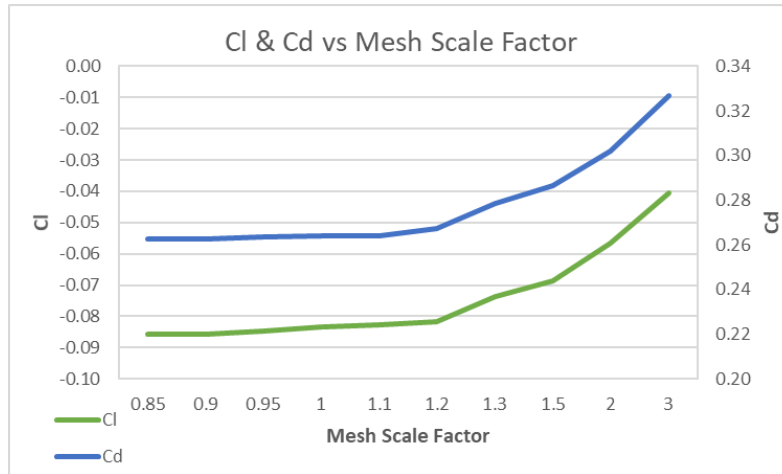


Figure 7. CI & Cd vs Mesh Scale Factor

A scale factor of 1 won't alter the mesh size, so that is the initial mesh size. Studying Table 2 & Figure 7, the CI and Cd values are almost steady from a scale factor of 0.85 to 1.2. From 1.3 to 3 the mesh element size starts to affect the results. For that reason and considering the processing power and time limitations, a scale factor of 1.2 was chosen as the setting to be used in future studies.

So, this leads to the following mesh settings:

Maximum Domain mesh element size: 1200mm

Density Box mesh element size 120mm

Vehicle surface mesh element size: 12mm

3.1.4 Turbulence Model Study

When performing CFD, each situation has an ideal Turbulence model to compliment it. As stated earlier, for an automotive vehicle, Lanfrit suggested the K-epsilon Realizable with non-equilibrium wall functions. The following study was performed using the new mesh sizing proposed in the paragraph above. The rest of the settings remained the same, and the turbulence model was changed for each run. Finally, the difference of CI and Cd results to the wind tunnel data provided for each turbulence model was plotted.

3.1.4.1 Results & Analysis

Table 3. Turbulence Model Study

Turbulence Model	Cd	CI	Δ Cd	Δ CI
Ke-Realizable	0.264	-0.083	0.032	-0.001
K-Kl-w	0.441	0.194	-0.145	-0.278
Ke-RNG	0.273	-0.012	0.024	-0.073
Kw-SST	0.290	-0.023	0.006	-0.061
Spalart-Almaras	0.308	-0.034	-0.011	-0.051
Transition-SST	0.293	-0.025	0.003	-0.059

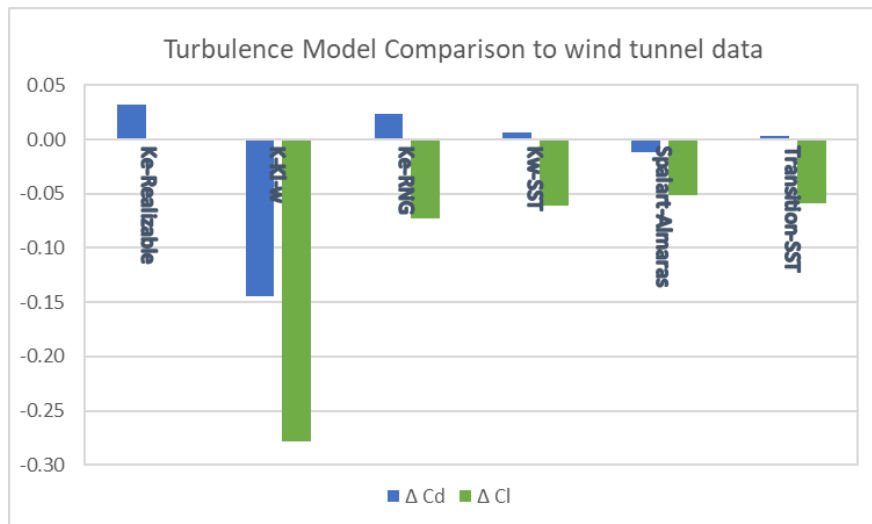


Figure 8. Turbulence Model comparison to wind tunnel data

Reviewing Table 3 & Figure 8 above, it is evident that most turbulence models tested were close to predicting the wind tunnel results, apart from the K-KI-w which deviated significantly. However, the model that produced a C_l almost identical to that of the wind tunnel data was the K-epsilon Realizable that was used initially. Other models produced more accurate results for C_d than the K-epsilon, but overall it performed better than the rest. So, the Turbulence model chosen to be the optimal one in this case was:

K-epsilon Realizable, with non-equilibrium wall functions

3.1.5 Prism Layer Study

For this study all the settings remained the same since the initial K-epsilon turbulence model was found to be the most accurate. Lanfrit suggested 5 prism layers with a 20% growth rate between them to be the optimal setup for capturing the boundary layer. 1-8 prism layers were tested. Prism layers were added to the car surface mesh and to the static floor.

3.1.5.1 Results & Analysis

Table 4. Prism Layer Study

No. of Prisms	Cd	Cl
1	0.277	-0.128
2	0.281	-0.131
3	0.273	-0.112
4	0.273	-0.120
5	0.264	-0.083
6	0.264	-0.081
7	0.264	-0.083
8	0.264	-0.082

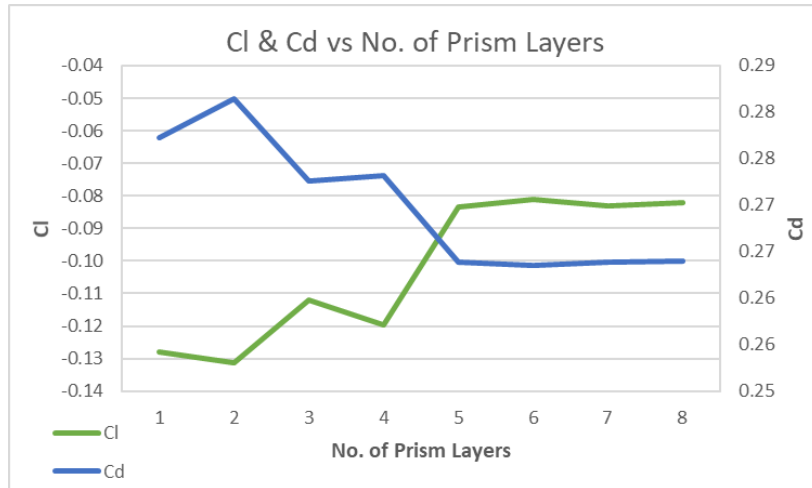


Figure 9. CI & Cd vs No. of Prism Layers

Observing Table 4 & Figure 9 above, CI and Cd remained steady when using 5-8 prism layers, while when using less the results fluctuated. Considering the time limit of this project and results accuracy, the number of prism layers chosen to be used was 5.

3.1.6 Polyhedra Study

The polyhedra mesh is used to reduce the solving time of the problem. It combines 5 tetrahedrals to create one polyhedral element. Depending on the model tested it can lead to inaccuracies. Using the optimum settings based on the 4 previous studies, a polyhedra mesh was created and a simulation was run. The polyhedra mesh results were compared to those of the optimum setup so far.

3.1.6.1 Results & Analysis

Table 5. Polyhedra Study

	Cd	CI	Δ Cd (%)	Δ CI (%)
Polyhedra	0.265	-0.081	-0.6	3.0
Optimal setup	0.264	-0.083	-	-

Based on Table 5, the polyhedra mesh predicted the results of Cd and CI with a difference of 0.6% and 3% respectively, to those of the optimum setup. Furthermore, it solved in approximately a third of the time. This suggests that perhaps the polyhedra mesh could be used for this project. However, the simple geometry of the MIRA reference car, as opposed to the complex geometry of a race car, in this case of the MCR S2000, may react differently to the polyhedra mesh. Henceforth, the polyhedra mesh will be revisited later on in this project during the MCR car model validation.

3.1.7 Results Comparison to Wind Tunnel Data

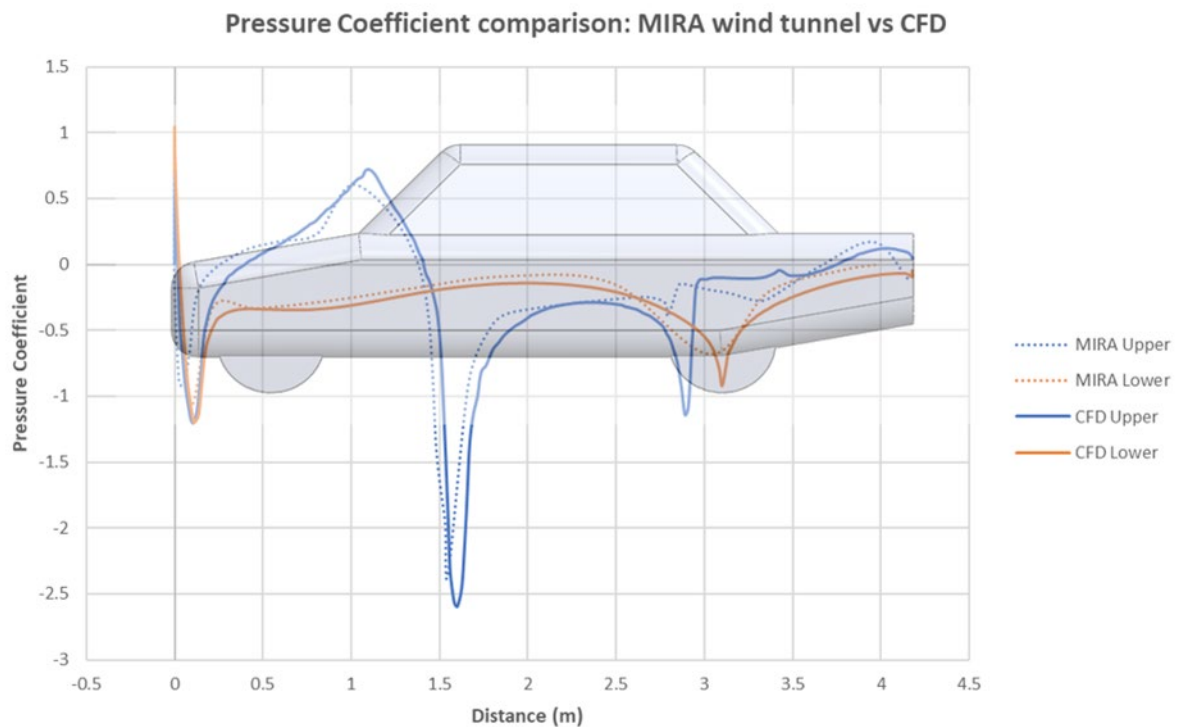


Figure 10. *Cl plot comparison: MIRA wind tunnel vs CFD*

The lift coefficient (C_l) for the length of the MIRA car was plotted and compared to that of the provided wind tunnel data. The blue colour represents the pressure plot on the upper side of the car, while the orange the pressure on the lower side. The dotted lines signify the wind tunnel data and the solid lines the CFD data. Comparing the two, it is clear that the pressure plots are very similar following each other's trends. The minimal differences observed can be justified by the lower number of pressure taps of the wind tunnel test model compare to that of the CFD model.

Table 6. *Wind tunnel data vs optimum CFD data*

	C_d	C_l	ΔC_d	ΔC_l
MIRA Wind Tunnel	0.296	-0.085	-	-
Optimal CFD setup	0.264	-0.083	-0.032	0.001

As, suggested by the table above, the optimal results setup after the studies conducted, predicted the drag and lift coefficients with a difference of 0.032 and 0.001 respectively, to the wind tunnel data.

3.1.8 Initial Optimal CFD Settings Template

Domain

36m Length x 6m Width x 7m Height, with an inlet to model distance of 12m

Mesh

Maximum Domain mesh element size: 1200mm

Density Box mesh element size 120mm

Vehicle surface mesh element size: 12mm

Turbulence Model

K-epsilon Realizable, with non-equilibrium wall functions

Prism Layers

5 Prism layers with a 20% growth rate

3.2 MCR S2000 car Validation Study (Baseline)

The studies conducted so far validated the CFD process and settings used throughout this project.

The following section focuses on validating the model/geometry provided for the MCR S2000 car.

Data for the specific model was provided by TotalSim, a world-renowned company when it comes to simulation, and in this case CFD. This provided data was used to correlate the data collected for this project.

3.2.1 Initial Model Setup

The model was set up based on the settings decided in paragraph 3.1.8 in conjunction with the TotalSim boundary conditions used to simulate the MCR car model. The domain and density box sizes were altered to match and maintain the proportions of the wider and longer MCR car compare to the MIRA car.

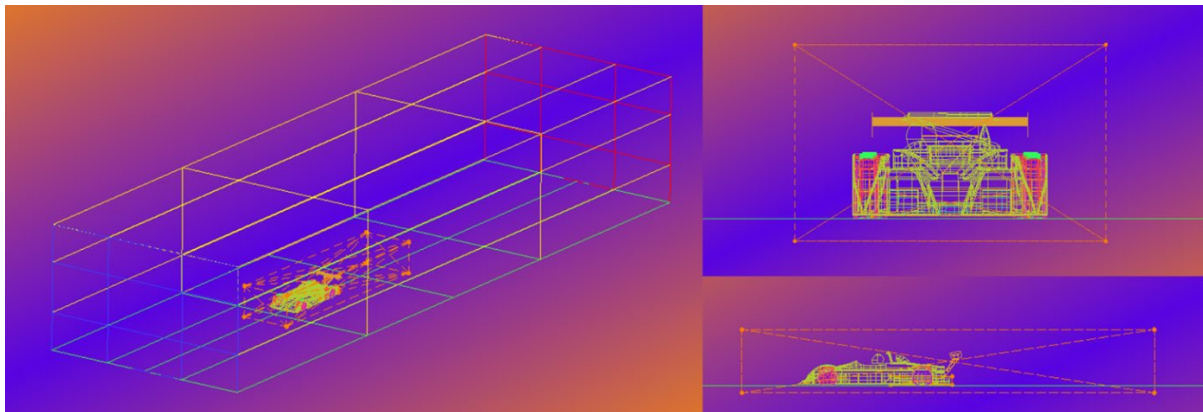


Figure 11. Domain & density box sizing (MCR)

Domain

40m Length x 13m Width x 7m height, with an inlet to model distance of 13m

Inlet Speed

35m/s

Ride Height

60mm

The rest of the settings adhered to those in paragraph 3.1.8

Note: TotalSim simulated the model with a moving floor and rotating wheels, but as this was the first “test” run of this model for this project, it was simulated with a static floor and wheels.

3.2.2 Initial Run

Table 7. Initial MCR model setup vs TotalSim data

	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance (%)
1st Static Floor	0.358	-0.119	-0.214	0.151	0.597	38.53	-5.56
TotalSim	0.477	-	-0.365	-	0.765	44.09	-

As seen in the table above, the current settings are underpredicting both the lift and drag coefficients of the car. Furthermore, the aerodynamic balance of the model is approximately 5% further rearwards than that of the TotalSim data.

These differences when comparing the data are most likely due to bad mesh quality and skewed mesh elements. In the following sections these issues were addressed with the goal to match the TotalSim data provided.

3.2.3 Wheel Blocks

Two geometry alterations were made to reduce the number of skewed cells in the mesh. The first change was the addition of blocks under the wheels. The area ahead and behind the contact patch of the wheel is very narrow, so when ICEM tries to generate a mesh in these regions it is not very successful, hence the generation of skewed elements. By adding the blocks under the wheels, the “contact patch” is extended, eliminating the narrow regions around it, preventing skewed elements to be created.

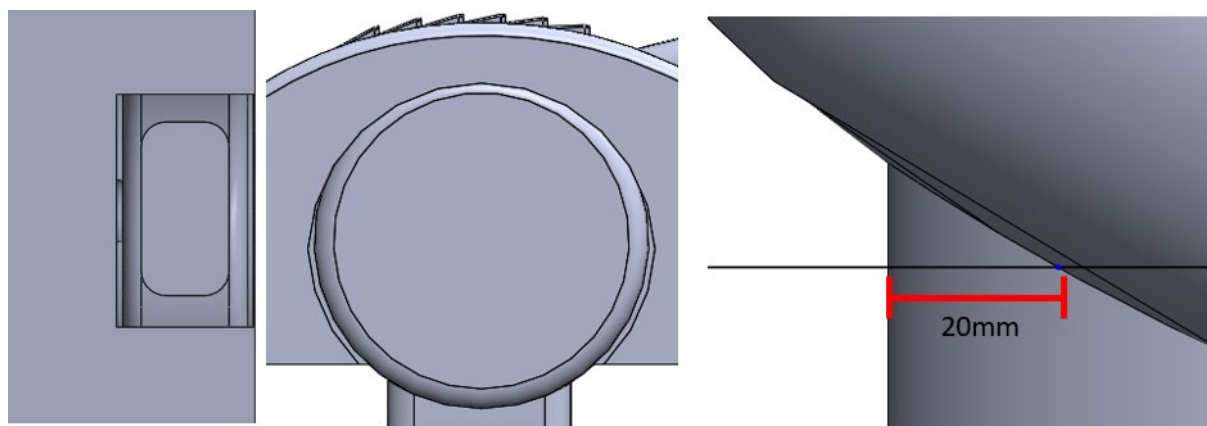


Figure 12. Wheel Blocks

As a rule of thumb, the blocks were suggested to be 1/3 of the wheel's diameter. However, the model will undergo simulation under different ride heights while the wheels remain solid as part of the car; this means the amount of wheel block interacting with the flow will vary. In order to try and eliminate this variable, the wheel blocks were resized for each ride height tested, maintaining approximately a 20mm extra radius around the contact patch.

3.2.4 Connected ROPS

The second geometry alteration to take place was connecting the ROPS. Similarly to the region around the contact patch of the wheel to the floor, the narrow region between the two roll bars was prime for skew element generation. So, to prevent this, the two roll bars were connected, as shown below, eliminating all skewed elements in that area.

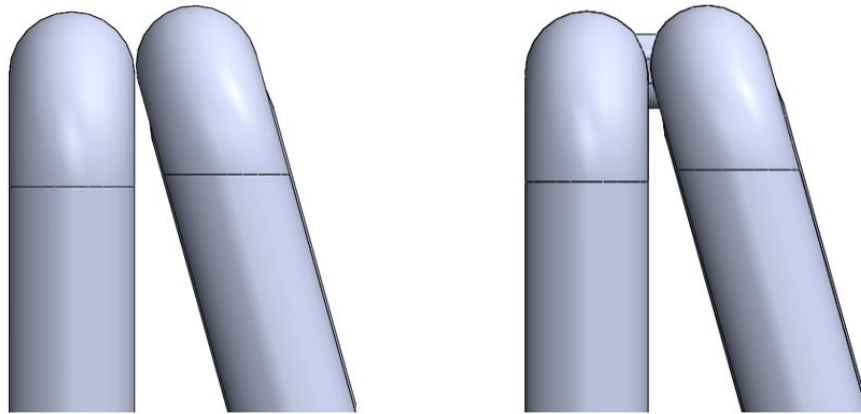


Figure 13. Connected ROPS

3.2.5 Vehicle Surface Mesh Refinement

The mesh settings of the initial run were 1200mm domain element size, 120mm density box element size and 12mm vehicle surface mesh element size. The 12mm vehicle surface mesh element size was not able to fully represent the shape of the MCR car. It struggled to mesh thin and complex shaped surfaces, such as the trailing edge of the wing, the wing endplates the wing mounts, the louvres and the newly added wheels blocks. This also led to skewed elements being generated around these edges.

Several meshes were generated in ICEM to try and find the perfect mesh element sizes needed to balance a realistic geometry representation, mesh quality and processing power/time required to mesh and solve.

The new vehicle surface mesh settings decided for the rest of the project were:

- Wing, wing endplates and wing mounts were set to a **7mm** mesh element size.
- Louvres were set to a **2mm** mesh element size.
- Wheels blocks were set to a **5mm** mesh element size.
- The rest of the model was set to the initial **12mm** mesh element size.

3.2.6 Density Box Study

The complex shape of a race car disrupts the flow around and behind it far more than what a “family” car would, such as the MIRA model. For this reason, a denser mesh is required around the vehicle to try and capture this much more aggressive and complex flow.

The initial density box element size was set to 120mm. 90mm to 60mm in 10mm increments were tested for this study.

3.2.6.1 Results & Analysis

Table 8. Mesh Density Study

Element Size	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance (%)
120mm	0.363	-	-0.261	-	0.719	40.35	-
90mm	0.377	0.013	-0.287	-0.025	0.761	40.77	0.41
80mm	0.391	0.028	-0.318	-0.056	0.812	41.14	0.78
70mm	0.400	0.037	-0.327	-0.065	0.816	41.76	1.41
60mm	0.402	0.038	-0.330	-0.069	0.821	41.78	1.43

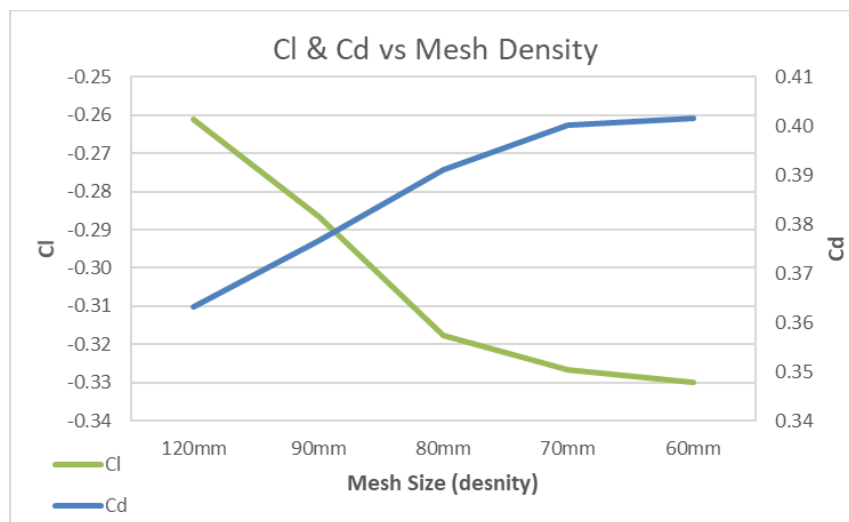


Figure 14. Cl & Cd vs Mesh Density

By observing Table 8 & Figure 14 above, it is clear that the mesh density around the MCR car affects the CFD results. In this case, Cl and Cd started to level out once the mesh reached a density of 70mm, which means its fine enough to capture the complex flow. Even though there was a slight difference in the data between 60mm and 70mm element sizes, **70mm** was chosen to be kept as the setting for the rest of the project as the difference wasn't large enough to justify the increase in solving time required.

3.2.7 Moving Floor and Rotating Wheels

So far, the MCR model was run with a static floor and wheels. As mentioned earlier, the data provided by TotalSim was simulated using a moving floor and rotating wheels.

At this point, the mesh of the car is of acceptable quality, so the next step was adding the boundary condition of a moving floor matching the speed of the flow at 35m/s. Following a successful run with a moving floor, rotating wheels were simulated for a final run before comparing the collected data to the provided data.

3.2.8 Comparison to TotalSim Data

Table 9. Collected data vs TotalSim data

	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance (%)
1st Static Floor	0.358	-0.119	-0.214	0.151	0.597	38.53	-5.56
Final Rotating Wheels	0.467	-0.010	-0.350	0.015	0.749	43.96	-0.13
TotalSim	0.477	-	-0.365	-	0.765	44.09	-

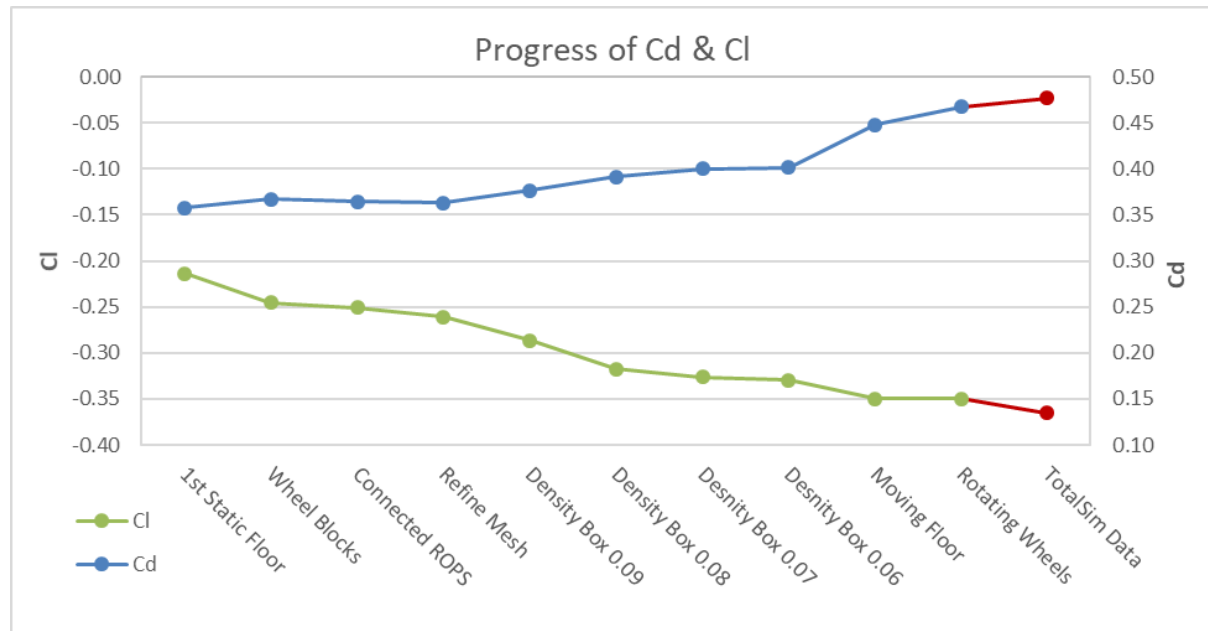


Figure 15. Progress of Cd & Cl

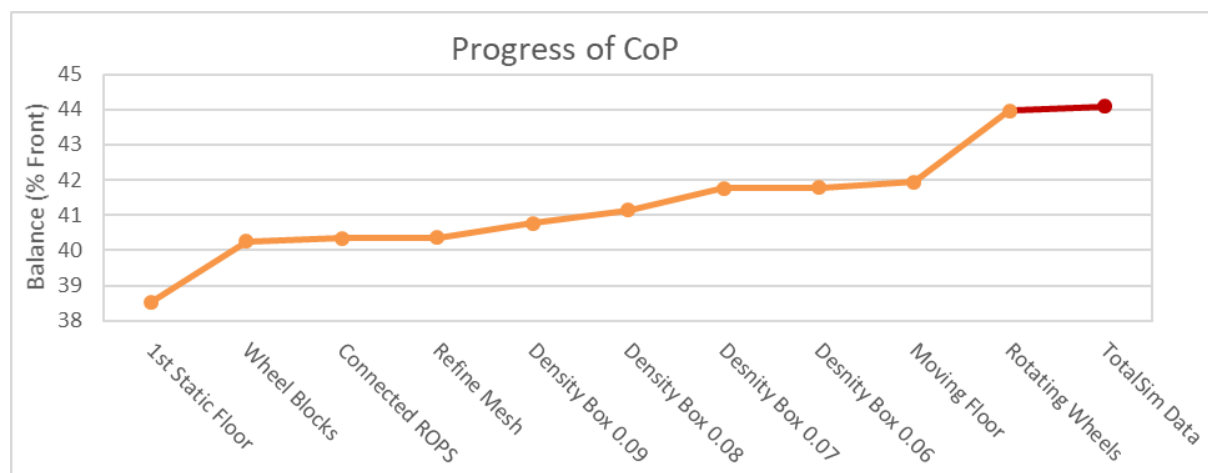


Figure 16. Progress of CoP

Displayed in Figure 15 & Figure 16 above is the progress of Cl, Cd and CoP throughout the development of the MCR CFD model in this chapter. The red line is the difference of the aforementioned values to the provided TotalSim data. It is obvious that all the steps taken in this chapter have played a role in refining the data collected, bringing them closer to the provided results and deeming the final setup acceptable for all the future simulations of this project. It was decided

however to not simulate the rotating wheels for the upcoming studies of this project, due to time limitations.

3.2.9 Final Model Setup

Domain

40m Length x 13m Width x 7m Height, with an inlet to model distance of 13m

Mesh

Maximum Domain element size: 1200mm

Density Box element size 70mm

Vehicle surface mesh element size: 12mm

Wing, wing endplates and wing mounts mesh element size: 7mm

Louvres mesh element size: 2mm

Wheel block mesh element size: 5mm

Turbulence Model

K-epsilon Realizable, with non-equilibrium wall functions

Prism Layers

5 Prism layers with a 20% growth rate

Ride Height

60mm

Boundary Conditions

Inlet Speed: 35m/s

Moving Floor: Yes

Rotating Wheels: No

The rest of the settings throughout the CFD process were left as default.

These settings were used for the rest of this project. The rotating wheels were reintroduced towards the end of the project, as they play an important part in the aerodynamic performance of a vehicle.

3.3 MCR car Baseline Attitude Study

With the latest decided settings for the CFD process, confidence can be put into the results generated. The car up to now was tested at 60mm ride height with no rake. An attitude study was performed, including a ride height study and a rake angle study.

For the ride height study, the car was set to 55mm to begin with, and then lowered in 10mm increments down to 15mm.

Once the optimum ride height was determined, the rake angle study was performed. At the optimum ride height, the back of the car was pivoted up from the front for the positive rake angle; and the front of the car was pivoted up from the back for the negative rake angle. This covered a total range of 1.6° in 0.2° increments.

Note: All the pressure and velocity contours in this study were user specified in order to achieve continuity. The legends for the pressure and velocity contours can be found in the Appendix.

3.3.1 Ride Height Study

Table 10. Ride Height Study

Ride Height (mm)	Cd	Cl	Efficiency	Balance (% Front)
55	0.429	-0.355	0.828	39.04
45	0.419	-0.361	0.862	39.31
35	0.419	-0.369	0.881	38.46
25	0.407	-0.340	0.835	38.96
15	0.406	-0.280	0.690	38.05

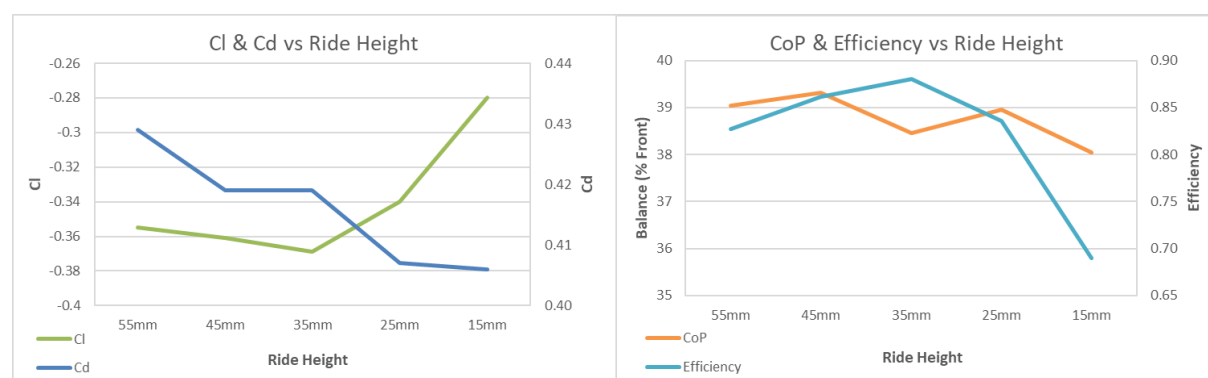


Figure 17. Ride height study graphs: Cl & Cd (left), CoP & Efficiency (right)

Studying Table 10 & Figure 17 above, it is initially clear that as the ride height is reduced, the Cd is reduced. This is primarily due to the model's frontal area being reduced. The Cl reduces (downforce increases) down to a ride height of 35mm, and then increases significantly. This is due to the reduced flow under the car because of the air's viscous characteristics (viscous blockage). This is similar to what happens to wings in ground effect. The closer the wing is to the ground, the better it performs, until it reaches a point where the air is simply too "thick" to squeeze through the gap. (McBeath, 2017)

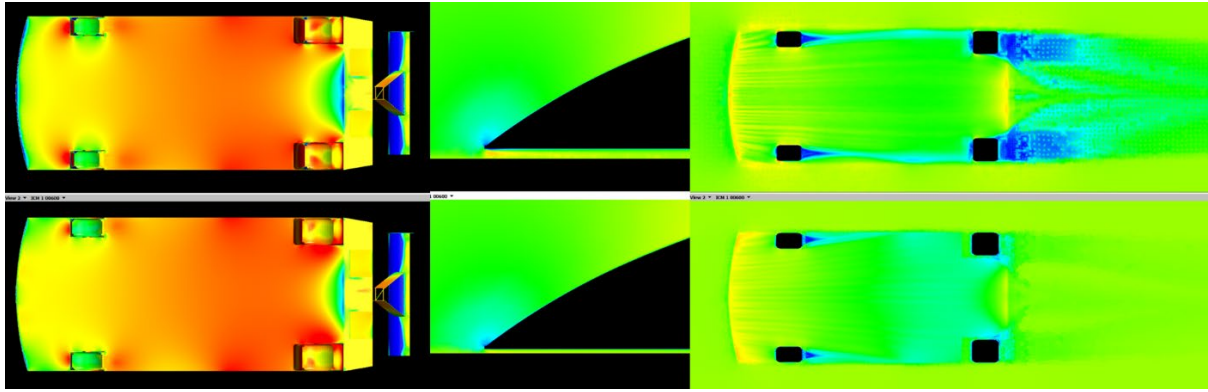


Figure 18. Pressure contours (left), velocity contours of nose (middle), velocity contours underfloor (right), of 35mm (top) & 15mm (bottom) ride heights

Observing the pressure plots on the left of Figure 18 above, it is evident that the pressure is lower at the splitter and the diffuser at 35mm ride height (top) compared to 15mm ride height (bottom). This is, as mentioned earlier, due to the air's inability to "squeeze" under the car at such a low ride height. This is also observed when comparing the velocity contours under the nose of the car and under the entire floor. Looking at the right side of Figure 18 it is obvious how much slower the airflow is under the car at 15mm compared to 35mm ride height, since there is a significantly larger amount of turquoise colour at 15mm ride height. Still, observing the right side of the figure, another reason for the decrease in drag, as the car is lowered, can be noticed; as a result of the flow reducing under the car as it is lowered, the wake created behind the car is also reduced, hence the reduction in drag.

Peak efficiency was reached at 35mm ride height. The aerodynamic balance remained reasonably stable throughout the entire study, as the pitch of the car was steady. Furthermore, the entire floor of the car is flat with no extra aerodynamic devices to lose effect and shift the CoP as the ride height reduces.

3.3.2 Rake Angle Study

Table 11. Rake Angle Study

Rake Angle (°)	Cd	Cl	Efficiency	Balance (% Front)
-0.8	0.427	-0.144	0.337	-47.26
-0.6	0.426	-0.206	0.484	-16.71
-0.4	0.424	-0.222	0.523	-6.91
-0.2	0.422	-0.317	0.753	26.30
0	0.419	-0.369	0.881	38.46
0.2	0.425	-0.399	0.938	52.55
0.4	0.428	-0.343	0.802	65.07
0.6	0.429	-0.333	0.777	70.74
0.8	0.429	-0.326	0.759	75.45

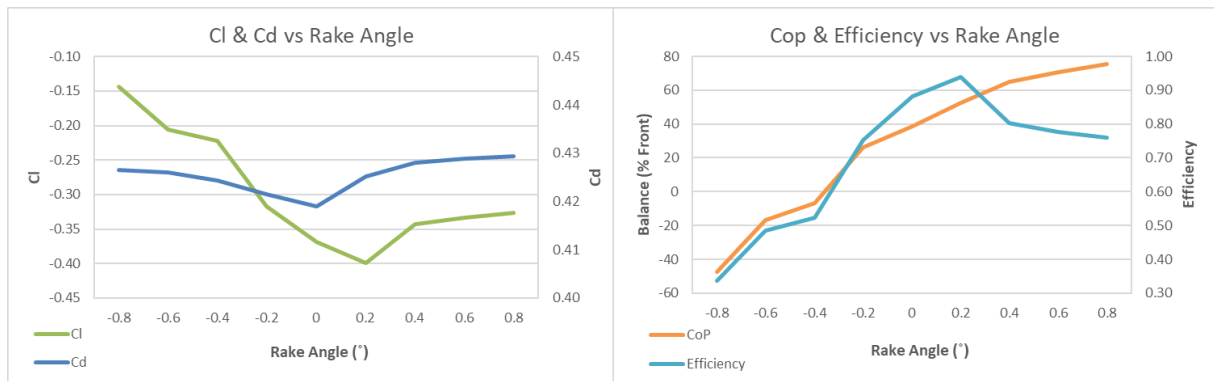


Figure 19. Rake angle study graphs: Cl & Cd (left), CoP & Efficiency (right)

Based on Table 11 & Figure 19 above, Cd (blue line) is lowest at 0° rake and is increased slightly when raking the car either way (negative rake – nose up, positive rake – rear up). This is again mainly a result of the increase in frontal area.

Cl (green line) is lowest at 0.2° positive rake, producing the highest amount of downforce, hence the peak efficiency. This happens because the entire flat floor of the car acts as a diffuser, increasing the flow and lowering the pressure at the throat/splitter as concurred by McBeath. However, the low pressure towards the rear of the floor is reduced as seen in Figure 20 below. This is a result of high-pressure air leaking into the low-pressure air under the car from the sides. Raking it further doesn't improve downforce any further as the low pressure at the splitter is not low enough to negate the increase in pressure towards the rear of the floor, due to that higher pressure air leakage. McBeath found that aerodynamic performance increased even up to rake angles of 1.3° which implies that the effect of raking is greatly dependent on the vehicle/setup. (McBeath, 2017)

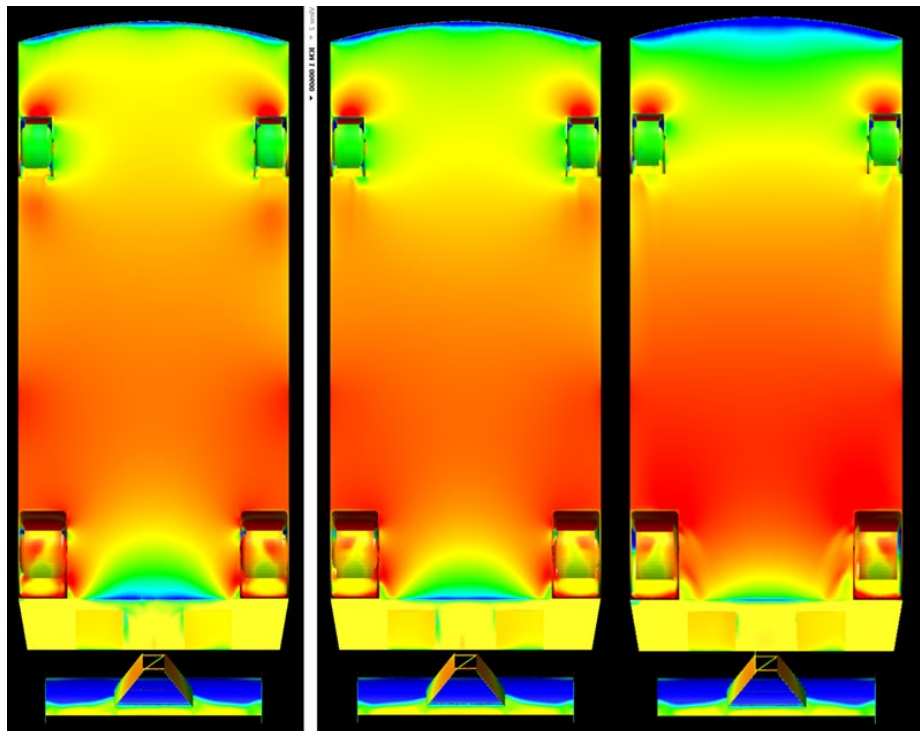


Figure 20. Pressure contours: 0 rake (left), 0.2 rake (middle), 0.6 rake (right)

When adding negative rake, lifting the nose up, downforce is lost as the air flowing under the car is trying to lift the nose up. This is very clear when observing the CoP (orange line) in Figure 19 as the nose lifts up (-ve rake) the CoP moves rearwards at a significant rate, meaning the car at its current trim is very sensitive to pitch. This was a major problem with “LM” GTP cars in 1999, which led to the cars flipping over after sudden changes in pitch. (Dominy, et al., 2000)

3.3.3 Optimum Baseline Setup

Based on the 2 studies conducted in the sections above, the optimum setup for the current model was:

Ride height: **35mm**

Rake angle: **0.2°**

3.4 OSS Baseline

The aim of this project was to develop an aerodynamic device, in this case a diffuser, for the MCR car to be competitive if it were to compete in the OSS Championship. Even though the OSS Championship is very “loose” in terms of its bodywork and aerodynamic device regulations (see paragraph 2.6), there are some limitations that needed to be considered. For this reason, the initial setup of the wing was not within regulations; it was 50mm higher than the ROPS, when the maximum height allowed was up to or below the height of the roll bar. To maintain as much aerodynamic efficiency as possible and keep the flow ahead of the wing undisturbed, it was lowered by just 50mm to match the height of the ROPS.

3.4.1 New Splitter

One of the targets of this project was to bring the CoP further rearwards, since the development of the new splitter for the car, moved it too far forward, producing an unstable rear end when driving the MCR car at speed. The effect of the new splitter can be seen in the section below.

3.4.2 Results & Analysis

Table 12. OSS Baseline Study

	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance (%)
Baseline	0.419	-	-0.415	-	0.990	42.97	-
OSS Baseline	0.416	-0.003	-0.420	-0.005	1.009	43.27	0.31
OSS Baseline New Splitter	0.436	0.017	-0.452	-0.037	1.038	69.13	26.17

The Baseline was the model which did not adhere to the regulations, the OSS Baseline was identical to the Baseline but with the wing lowered by 50mm. Finally, the new splitter was added to the OSS Baseline to see its effect.

By lowering the wing 50mm the results were affected but were considered negligible. When adding the new splitter however, the results changed significantly. Specifically, downforce increased for a drag penalty with an overall increase in efficiency. As expected, the new splitter produced a greater amount of frontal downforce compare to the old one, moving the CoP 26% further forward. The new

diffuser developed in this project aimed to bring the CoP of the car back to at least a 50/50 (50% front, 50% rear) aerodynamic balance. The original aerodynamic balance was predicted to be 43/57 and adding the splitter moved it to 69/31.

Note: The model used for the development of the diffuser was the OSS Baseline without the new splitter. The reason being the increased complexity of the new splitter leading to a significant overhead in solving time when performing CFD. The new splitter design was re-introduced at the end of the development phase of the newly developed diffuser to see if it (the diffuser) achieved its goals.

4 Diffuser Development

The development of the diffuser for this project was approached parametrically. Individual studies were performed for each stage of the development. Once the optimum setup for a stage was decided upon, it was added to the model for the subsequent development stages. This allowed for a gradual build-up of a diffuser design by studying the interaction of each development stage with the next, preventing the development of designs that would not interact and perform as expected with each other. This approach is based upon the innovative geometry build up followed by Ehirim when developing a diffuser for a Formula SAE race car. (Ehirim, 2012)

4.1 Centre Section

The first stage of development was that of the centre section of the diffuser. The reason behind having a centre section and side section (side tunnels) diffuser was the packaging. The MCR S2000 is a mid-engine race car, meaning there was an engine and a gearbox where a diffuser would ideally be. This led to the development of a shorter centre section diffuser and two longer side tunnels either side of it.

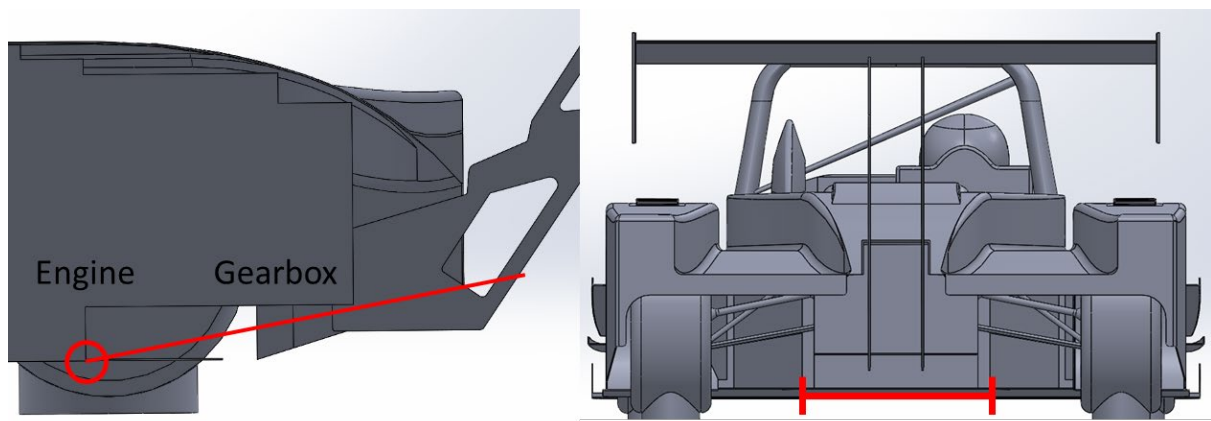


Figure 21. MCR S2000 diffuser centre section packaging considerations

In Figure 21 above, a cross section of the rear of the MCR is presented on the left, and on the right is a rear view of the car. Displayed on the left side of the image with a red circle is the base of the engine, which is as far forward as the diffuser centre section can start, so that is where the kickpoint/leading edge was decided to be. The red line on the left indicates the maximum angle the diffuser can reach before contacting the base of the gearbox. The maximum angle was 13° and the smallest

radius was 1200mm. Furthermore, the wing mount would interfere with the diffuser at high angles. Depending on the design of the diffuser the wing mount could be redesigned without affecting its role. On the right side of the figure, the red line represents the width of the engine/frame, which was also the width of the diffuser centre section. The length of the diffuser will not extend past the rear bodywork of the car (initially).

The initial idea was to develop the centre section by creating a diffuser adhering to the paragraph above but stretching the entire width between the rear wheels and adding diffuser endplates to seal the flow. The 2 first studies were performed testing a radiused diffuser profile and an angled diffuser profile. The results were not satisfying nor predictable, possibly due to tyre squirt. The 2 studies were re run with the addition of 2 strakes added separating the diffuser centre section from the rest of the diffuser (can be seen in Figure 22). The results were more promising.

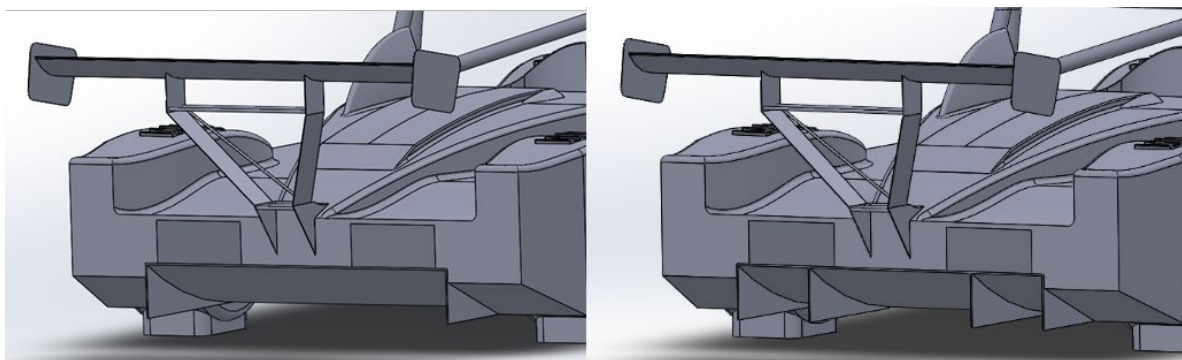


Figure 22. Diffuser centre section 1st study setup (left) 2nd study setup (right)

4.1.1 Diffuser Profile Study 1

As mentioned above, 2 diffuser profiles were tested, a radiused profile and an angled profile.

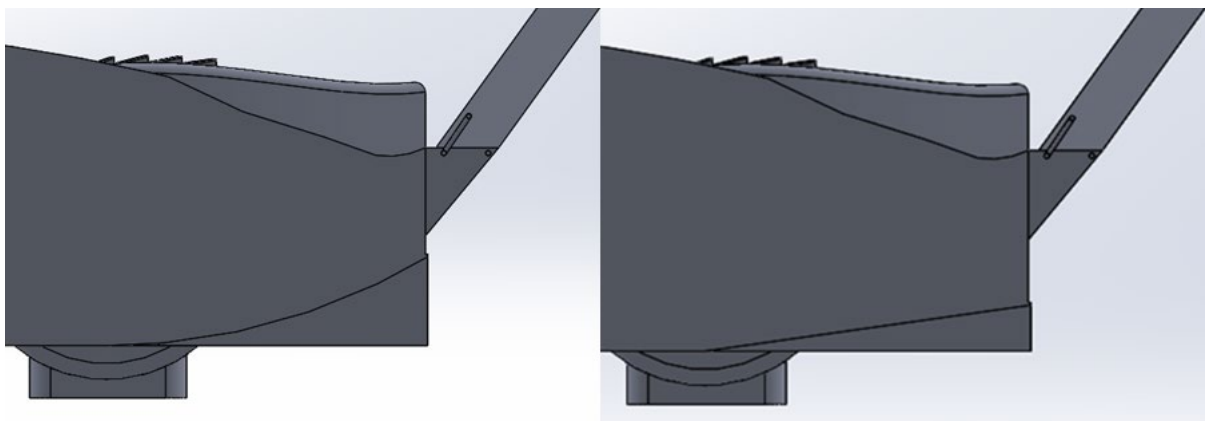


Figure 23. Diffuser profiles: Radiused (left), Angled (right)

4.1.1.1 Radius

A radiused diffuser profile is more gradual at reaching its max angle compare to an angled profile which maintains its max angle throughout the entire length of the diffuser. A radius of 2000mm to 1200mm was tested in 200mm increments.

Table 13. Diffuser centre section radius study

Radius (mm)	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance %
2000	0.401	-0.015	-0.448	-0.028	1.115	38.79	-4.49
1800	0.402	-0.014	-0.447	-0.028	1.112	37.82	-5.45
1600	0.404	-0.012	-0.446	-0.026	1.103	36.77	-6.50
1400	0.404	-0.012	-0.452	-0.032	1.119	36.55	-6.73
1200	0.404	-0.012	-0.458	-0.038	1.134	35.94	-7.33

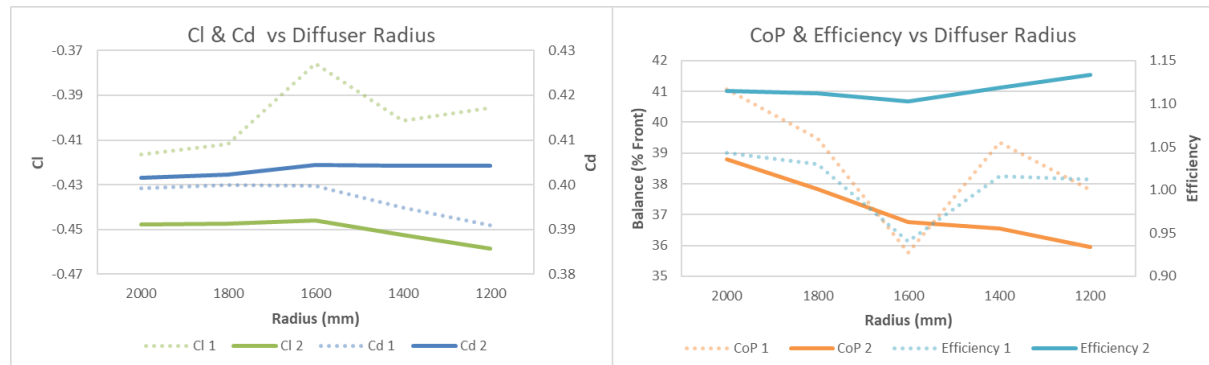


Figure 24. Diffuser centre section radius vs Cl & Cd (left), CoP & Efficiency (right)

In Figure 24 above, the dotted lines represent the initial study without the strakes separating the centre section from the rest of the diffuser. It is clear that these results were not consistent.

Overall, adding a diffuser reduced the Cd and increased downforce compared to the baseline. Decreasing the radius, so increasing the aggressiveness of the diffuser increased the aerodynamic efficiency of the MCR and moved the CoP further rearwards.

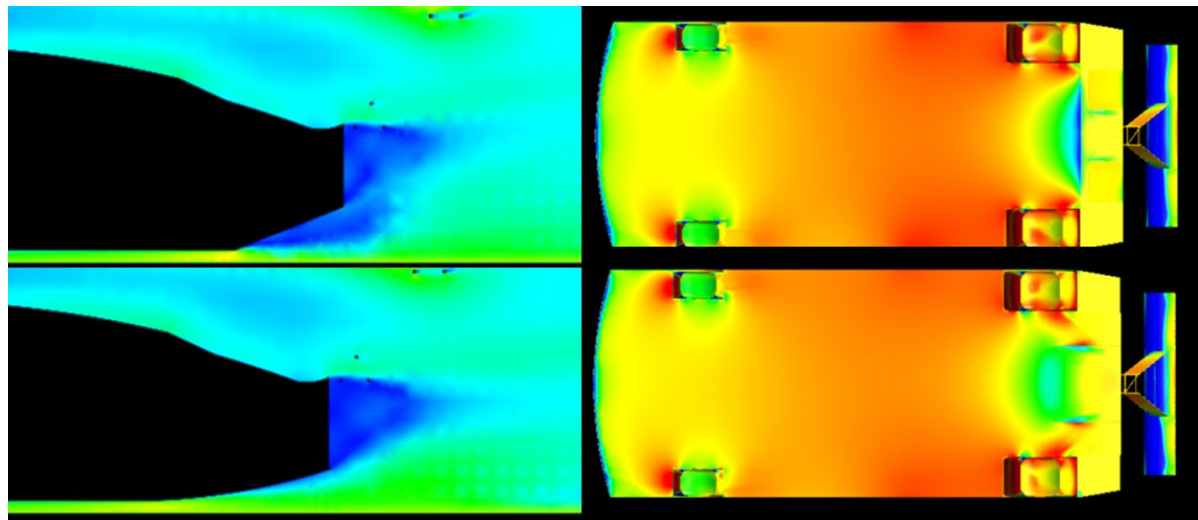


Figure 25. Comparison of velocity (left) and pressure contours (right) of OSS baseline (top) vs radius 1200mm (bottom)

Observing the velocity contours on the left side of Figure 25 above, the flow seems to remain attached for the entire length of the diffuser at a radius of 1200mm (almost stalling). Observing the pressure contours of the underbody on the right hand side, the OSS baseline seems to have a higher peak low pressure where the floor ends compared to the low pressure created by the diffuser. The

low pressure with the diffuser seems to propagate slightly further forward than the low pressure of the baseline. The differences in the overall pressure of the underbody are minimal hence the minimal increase in downforce of 0.038 of -CI when adding the radiused diffuser.

4.1.1.2 Angle

Initially an angle of 5-13 degrees was tested in 2° increments. 9° was found to perform the best out of the angles tested, identical to what Christoffersen and his team found. 8° and 10° angles were also tested for a more accurate approach, since some studies suggest 10° to perform the best.

Specifically, Cooper found that 9.64° to be the optimal diffuser angle, while a study from Cranfield University, as well as McBeath suggest 10°. (Cooper, et al., 1998) (Soares, et al., 2018) (Christoffersen, et al., 2011) (McBeath, 2017)

Table 14. Diffuser centre section angle study

Angle (°)	Cd	Δ Cd	CI	Δ CI	Efficiency	Balance (% Front)	Δ Balance %
5	0.400	-0.016	-0.415	0.004	1.038	42.83	-0.45
7	0.401	-0.015	-0.455	-0.036	1.135	42.05	-1.22
8	0.401	-0.015	-0.490	-0.071	1.222	41.69	-1.58
9	0.402	-0.014	-0.484	-0.065	1.206	40.10	-3.17
10	0.401	-0.016	-0.468	-0.048	1.169	37.63	-5.64
11	0.401	-0.015	-0.462	-0.042	1.152	37.03	-6.24
13	0.400	-0.016	-0.457	-0.037	1.142	36.62	-6.66

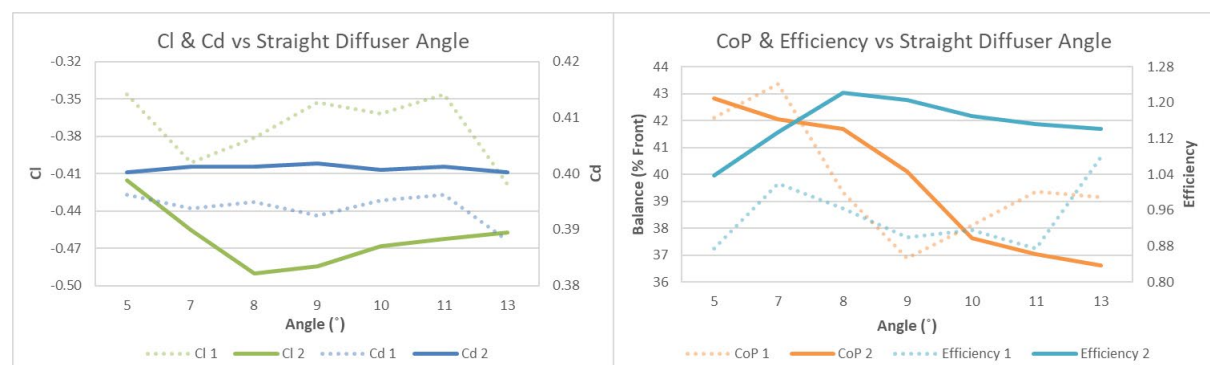


Figure 26. Diffuser centre section angle vs CI & Cd (left), CoP & Efficiency (right)

Studying Table 14, the Cd reduction when adding the angled diffuser, is similar to that noticed when adding the radiused one. At 8° the CI is the lowest with a drop of 0.071 compared to the baseline, increasing efficiency. At 8° the CoP moved rearwards by almost 1.6%. Furthermore, it was observed that even though the diffuser is starting to stall after 8° the CoP continues to move further rearwards.

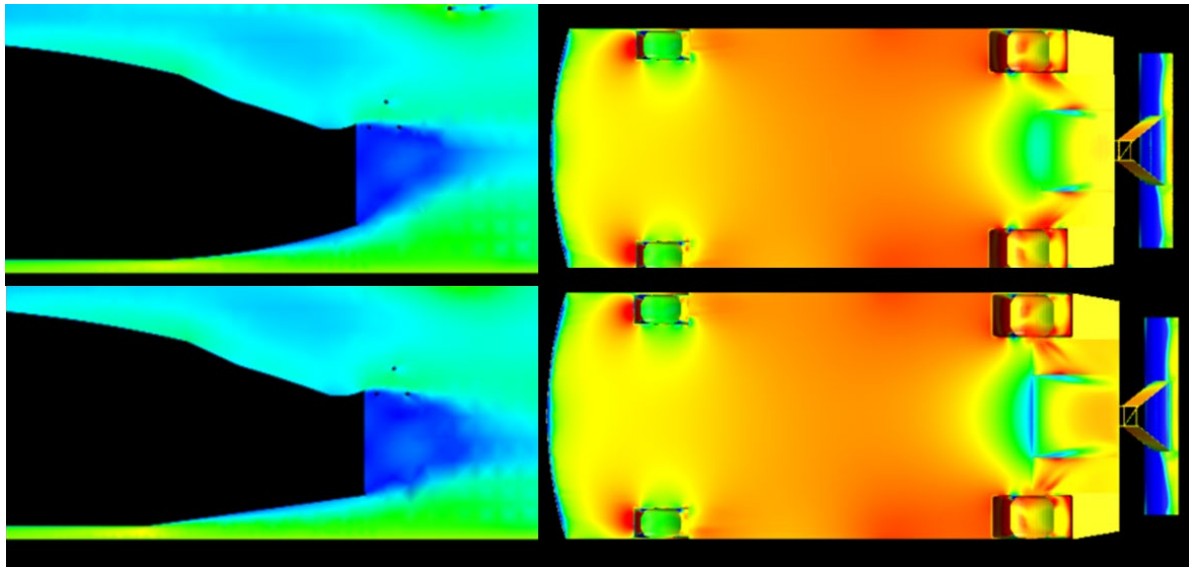


Figure 27. Comparison of velocity (left) and pressure contours (right) of radius 1200mm (top) vs angle 8° (bottom)

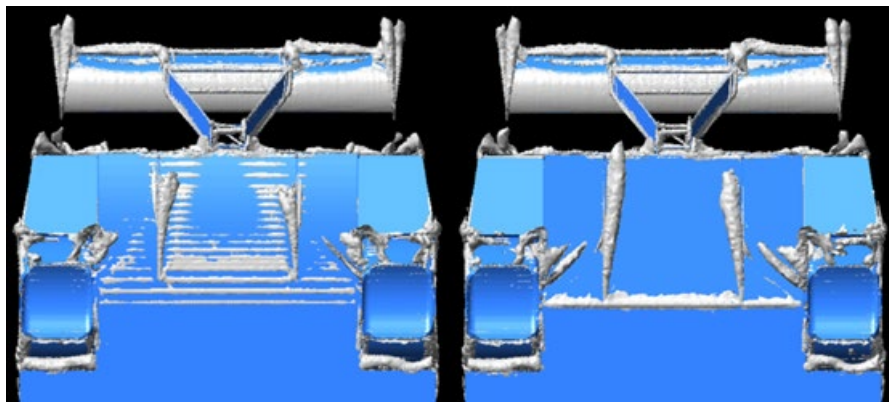


Figure 28. Vortex generation comparison of radius 1200mm (left) vs angle 8° (right)

Observing the left side of Figure 27 it is clear that the flow down the centre of the diffuser centre section is on the verge of stalling. However, the pressure contours on the right side of Figure 27 and the vortex generation plots in Figure 28 suggest that as a result of the high-pressure tyre squirt on the outside of the centre section strakes and the low pressure on the inside, two counterrotating vortices are created, one next to each strake, reenergizing the boundary layer and keeping that flow attached throughout the diffuser centre section. These vortices preventing the diffuser from stalling were previously observed by Unni, Werner and Frik. (Unni, 2017) (Werner & Frik, 1998)

From the 2 studies conducted so far, the best performing setup was that of a diffuser with an angle of 8°. The studies to follow will be based of this 8° diffuser centre section model.

4.1.2 Length Study

The increase in length of the diffuser centre section is tested in this study. The OSS regulations state that aerodynamic devices can extend up to 1100mm behind the rear axle of the car. For the MCR car this means the diffuser can extend up to 500mm behind the car body (overhang). The length of the centre section was increased by 500mm in total in 100mm increments.

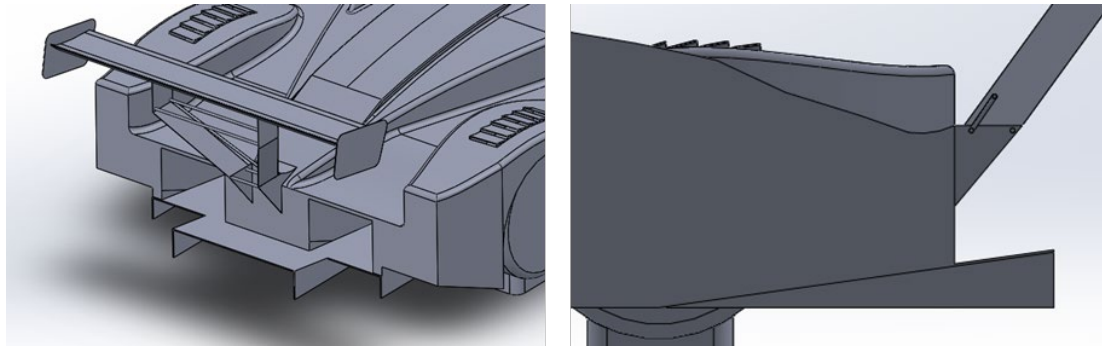


Figure 29. Diffuser centre section 200mm length increase

Table 15. Diffuser centre section overhang study

Overhang (mm)	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance %
0	0.401	-0.015	-0.490	-0.071	1.222	41.69	-1.58
100	0.397	-0.019	-0.468	-0.048	1.179	40.37	-2.91
200	0.397	-0.019	-0.425	-0.005	1.072	36.24	-7.04
300	0.395	-0.021	-0.397	0.023	1.003	42.14	-1.13
400	0.397	-0.019	-0.428	-0.008	1.077	41.20	-2.07
500	0.395	-0.021	-0.423	-0.003	1.072	41.03	-2.24

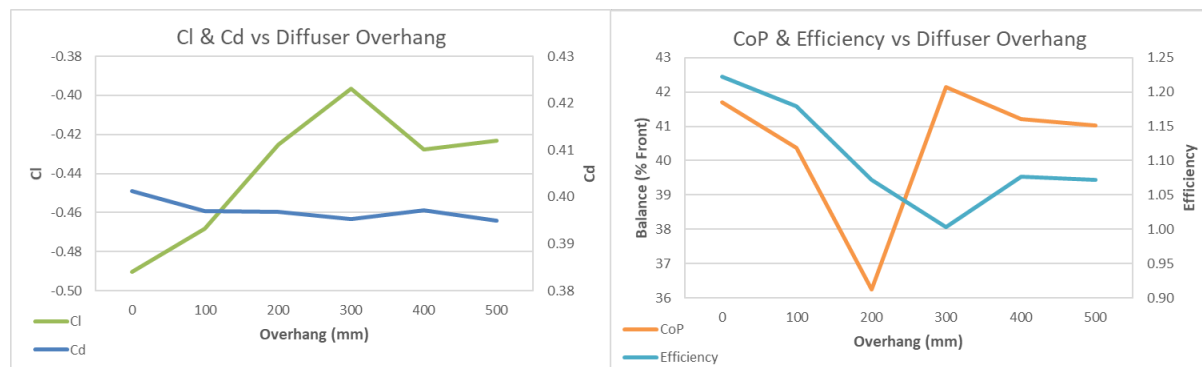


Figure 30. Diffuser centre section overhang vs Cl & Cd (left), CoP & Efficiency (right)

Analysing Table 15 & Figure 30 above, the Cd remains steady as the overhang is increased. The Cl is increased (downforce is lost) as the overhang increases. This is a result of there not being enough energy in the boundary layer of the centre section to keep the flow attached as the length and AR of the diffuser increases. On the left of Figure 31 below it is obvious that the flow stalls with an increase in diffuser length. This leads to an increase in pressure in the diffuser centre section which can be seen on the right side of the figure. More so, the diffuser stalling causes it to not work as hard compared to the 0mm overhang model, resulting in a loss of downforce. The CoP fluctuates when the overhang increases. Specifically, at 200mm overhang the aerodynamic balance moves 7% further rearwards. This anomaly in the results might be due to an error during solving.

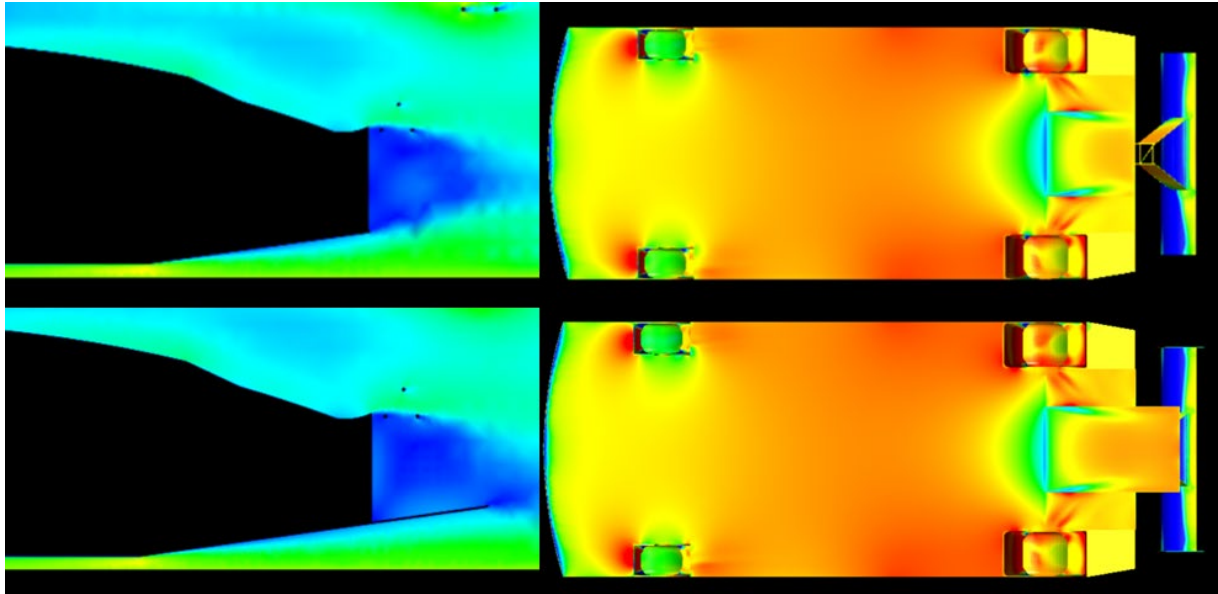


Figure 31. Comparison of velocity (left) and pressure contours (right) of 0mm overhang (top) vs 300mm overhang (bottom)

4.1.3 Concluded Optimum Design (Centre section)

The decided optimal diffuser design for the centre section, after completing the studies above was:

Diffuser angle: 8°

Diffuser overhang: 0mm

This design has an AR of 4.97, which is lower than 5 which Toet suggests should not be exceeded. (Toet, 2017)

The angle of the centre section will remain unaltered throughout the project. The diffuser centre section overhang will be revisited when a second diffuser length study is performed including the developed side tunnels.

4.2 Side Tunnels

With the decided diffuser centre section complete, the diffuser side tunnels were the next point of interest. Due to the more flexible packaging around the side of the engine assembly, there was more room to take advantage of, to develop the diffuser side tunnels.

The inspiration for this design were the diffusers developed in LPM1 and LMP2 cars after the regulation change in 2004. The new regulations allowed for the diffuser to start 1000mm ahead of the rear axle instead of at the rear axle. This was to prevent downforce being produced outside of the car's bodywork, which was leading to severe aerodynamic instability. (Fuller, 2004)

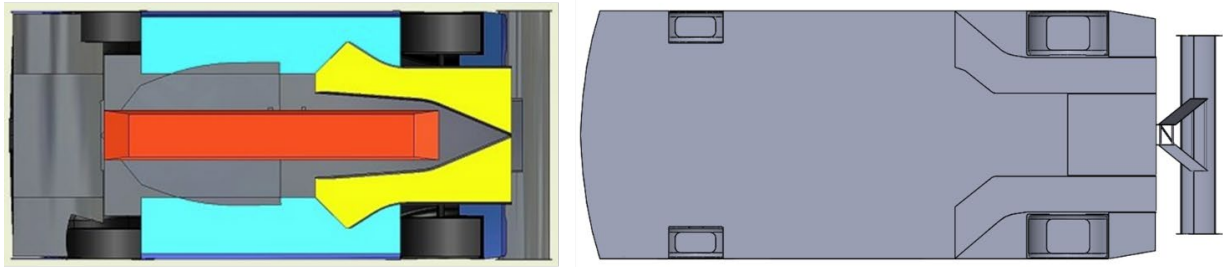


Figure 32. LMP1 & 2 aero. regulations (left), LMP inspired side tunnel design on MCR (right)

In Figure 32 above, the yellow indicates the regulation box for the diffuser of the LMP1 & 2 cars after the 2004 regulation change. On the right hand side of the image is the underfloor of the MCR car where the LMP inspired side tunnel design can be seen. This design also took into consideration the packaging restrictions of the chassis. By creating these sharp angles in the diffuser, the kickpoint was able to move further forward allowing for greater AR to be achieved.

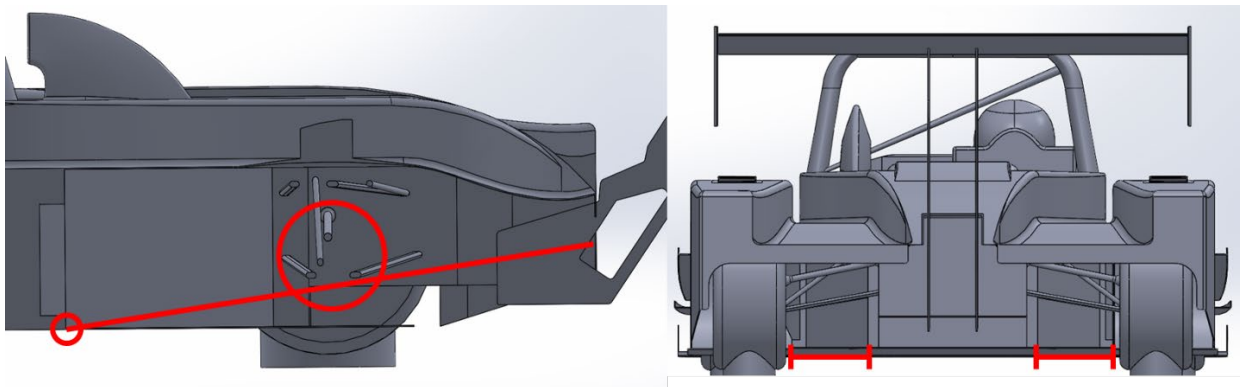


Figure 33. Diffuser side tunnels packaging restrictions

In Figure 33 above, viewing the left side of the image, the large red circle indicates the wishbones/axle assembly of the MCR. Ideally, the diffuser side tunnels will be developed under this assembly. Setting the kickpoint 1000mm ahead of the rear axle, the maximum angle the side tunnels can reach without interacting with the wishbones was 8° . A radiused profile would not have packaging restrictions when it comes to the wishbone assembly. On the right hand side of the figure, the width of the side tunnels is indicated with red.

Using this initial design, repeating the first steps of the centre section study, the two diffuser profiles (radius & angle) were tested.

4.2.1 Diffuser Profile Study 2 (Side Tunnels)

4.2.1.1 Radius

Since the kickpoint of the side tunnels was further ahead of the centre section's one, the diffuser radius had to be increased to stay within the cars bodywork and to be effective (not stall). The radiuses tested were from 6000mm-8000mm in 500mm increments.

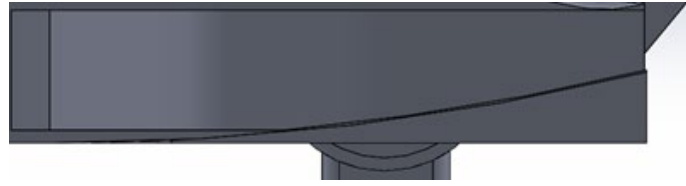


Figure 34. Radius profile of diffuser side tunnels

Table 16. Diffuser side tunnel radius study

Radius (mm)	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance %
8000	0.400	-0.016	-0.501	-0.081	1.252	41.41	-1.87
7500	0.401	-0.015	-0.525	-0.105	1.308	41.47	-1.81
7000	0.402	-0.014	-0.544	-0.124	1.352	41.53	-1.74
6500	0.403	-0.013	-0.547	-0.127	1.357	41.61	-1.66
6000	0.403	-0.013	-0.550	-0.130	1.363	41.63	-1.64

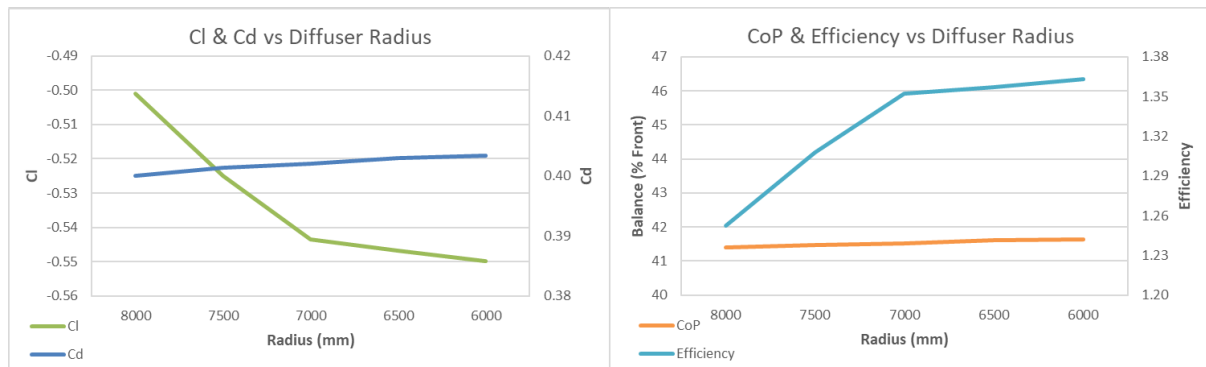


Figure 35. Diffuser side tunnel radius vs Cl & Cd (left), CoP & Efficiency (right)

Observing Table 16 & Figure 35 above, Cd remains steady as the radius decreases, while the downforce increases all the way to a 6000mm radius. The increase in downforce however is marginal compared to the optimal centre section setup from the previous studies. The CoP remains steady throughout.

Below is a comparison of the best performing radius (6000mm) vs the optimal centre section setup without side tunnels. On the left of Figure 36 the pressure contours suggest that the centre section is not working as hard when the new radiused side tunnels were added. However, the overall pressure ahead of the diffuser seems to have dropped slightly, leading to that marginal increase in downforce mentioned earlier. On the right hand side of the figure, the velocity contours at the level of the underfloor were displayed; the flow is very similar between the two cases throughout the later section of the diffuser, where tyre squirt is noticeable, as well as the high velocity vortices in the centre section. The low speed air at the kickpoint of the radius is a result of the velocity contours being measured at the underfloor level in conjunction with the slow increase in angle of a radiused diffuser, leading to the boundary layer being captured.

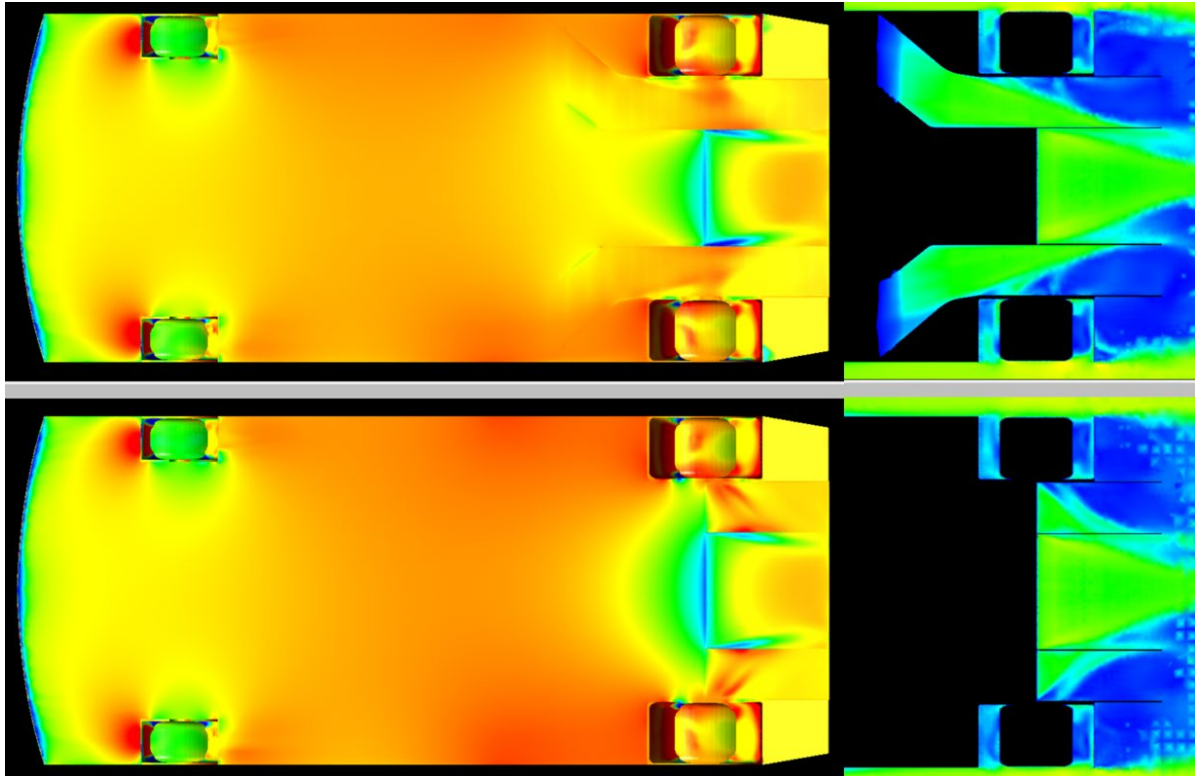


Figure 36. Pressure contours (left) & velocity contours (right) at underfloor level of the 6000mm radius diffuser (top) vs the optimal centre section setup

The optimal radius based on the data collected from this study was 6000mm.

4.2.1.2 Angle

For this study, as mentioned earlier, the maximum angle the current diffuser side tunnel design allows is 8°, in order to not interact with the wishbone assembly. However, it was decided to test angles steeper than 8° as well. If these angles performed better, the possibility of a design where the wishbones would pierce the diffuser would be considered.

The angles tested were 6-10 degrees in 1 ° increments.



Figure 37. Angle profile of diffuser side tunnels

Table 17. Diffuser side tunnel angle study

Angle (°)	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance %
6	0.414	-0.002	-1.004	-0.584	2.422	42.61	-0.67
7	0.416	0.000	-1.081	-0.661	2.599	43.13	-0.14
8	0.417	0.001	-1.129	-0.709	2.708	43.89	0.62
9	0.417	0.001	-1.113	-0.693	2.668	44.24	0.96
10	0.418	0.002	-1.117	-0.697	2.671	44.27	0.99

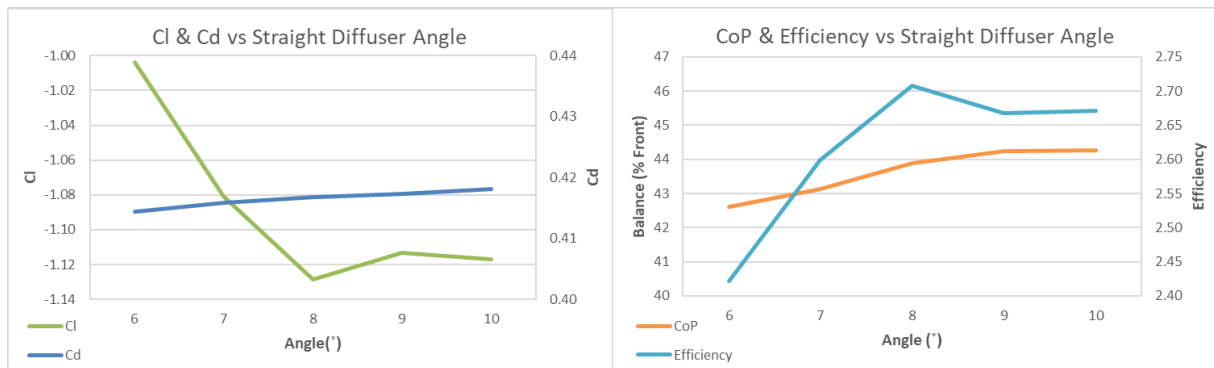


Figure 38. Diffuser side tunnel angle vs Cl & Cd (left), CoP & Efficiency (right)

Studying Table 17 & Figure 38 above, Cd increases marginally as the angle of the diffuser increases. The best performing angle was 8°, so with the current kickpoint the diffuser design does not interfere with the wishbone assembly. Furthermore, as the angle increases, the CoP moves further forward until the diffuser reaches its peak efficiency at which point it starts to stall (both the diffuser and the CoP movement). The CoP is moving forwards as a result of the side tunnel's kickpoint being ahead of the centre section kickpoint, so the harder the side tunnels work the further forward the CoP will move, up to a certain point.

The angled profile (8°) for the side tunnels worked significantly better than the radiused profile (6000mm), increasing the -Cl to 1.129 compare to 0.55 (more than double). This is a result of 2 strong vortices being generated on either side of each tunnel, “sealing” the flow, as pointed out in Figure 40 below. These vortices are a product of the sharp angles of the diffuser. A diffuser commonly does not have angles, it is straight all the way through. However, these sharp angles in unification with the angled profile, allow for adverse pressure gradients to be formed around the edges of the preliminary section of the diffuser, leading to the vortex generation mentioned above.

These vortices are being effective in two ways:

1. They help reenergise the boundary layer formed within the diffuser side tunnels allowing for the flow to remain attached throughout, preventing it from stalling.
2. The vortices “seal” the diffuser, averting any high pressure, low velocity turbulent air to seep into to the high velocity low pressure flow of the diffuser.

Observing the pressure gradient comparison in Figure 39 below, it is clear how much more effective the angled profile is compared to the radiused one. Additionally, studying the velocity contours (right side of figure), the flow velocity is increased throughout the entire length of the side tunnels and the tyre squirt is decreased significantly thanks to the vortices mentioned above. The “messy” flow in the side tunnels due to tyre squirt was also observed by McBeath during wind tunnel testing. (McBeath, 2017)

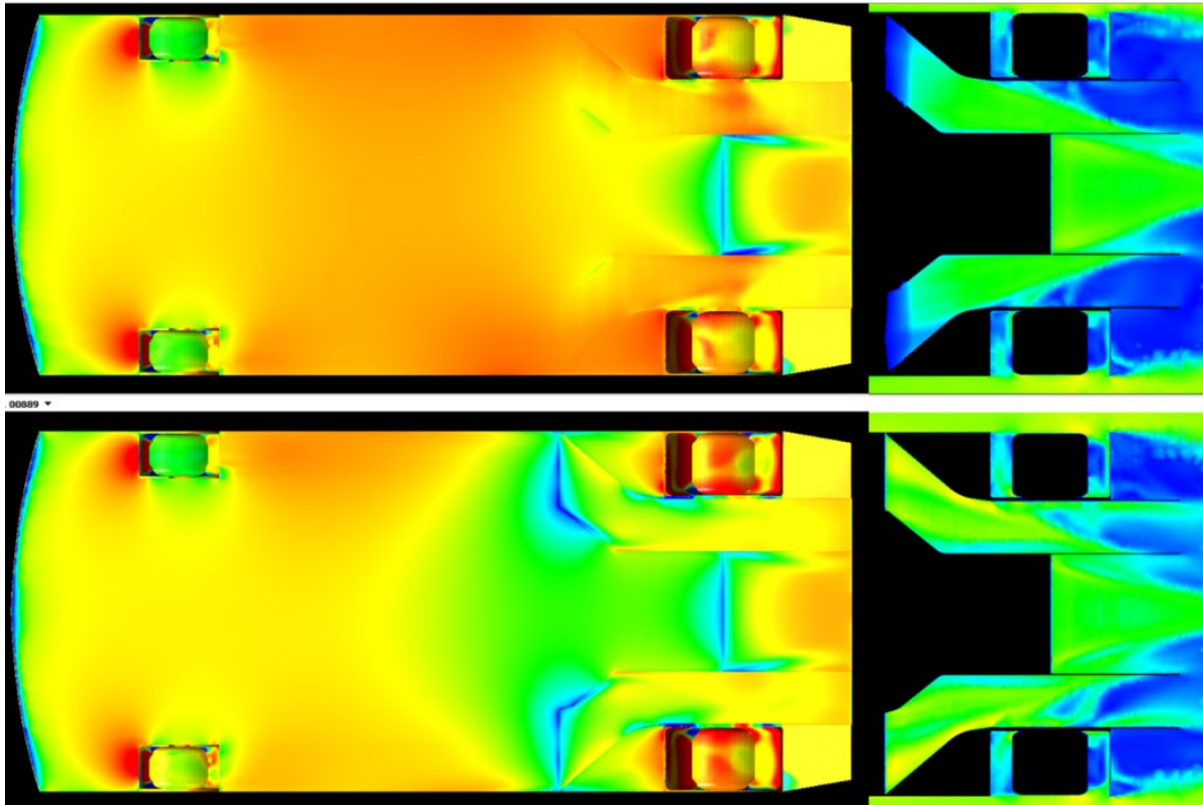


Figure 39. Pressure contours (left) & velocity contours (right) at underfloor level of the 6000mm radius diffuser (top) vs the 8° diffuser angle (bottom)

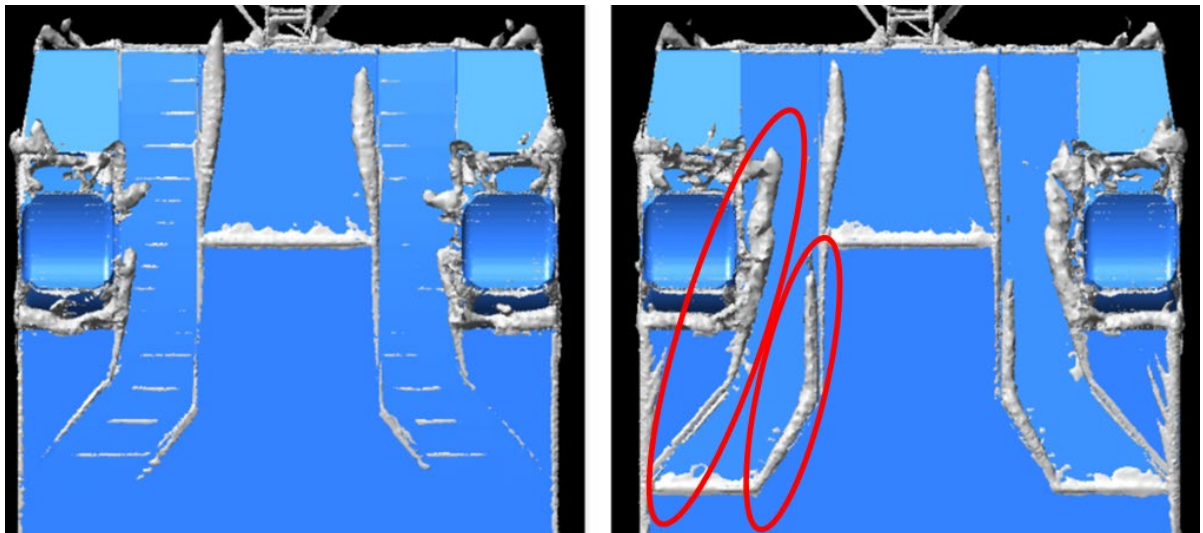


Figure 40. Vortex generation comparison between 6000mm radius (left) and 8° angle (right)

So, the optimal angle based on the data collected from this study was 8°.

4.2.2 Kick Point Study

Once the optimal diffuser side tunnel profile was decided upon (8° angle), the diffuser kickpoint was studied. The kickpoint was moved forwards and backwards 300mm each way, in 100mm increments, while maintaining the same diffuser angle. The diffuser shape was restricted by the MCR chassis/frame shape.

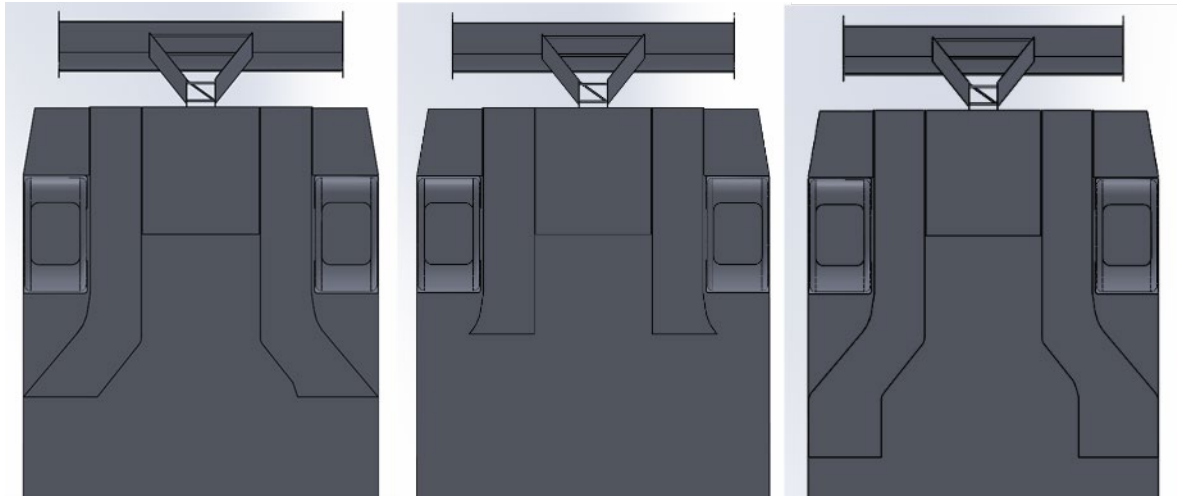


Figure 41. Initial design (left), kickpoint 300mm back (centre), kickpoint 300mm forward (right)

Table 18. Diffuser side tunnel kickpoint study

Kickpoint (mm)	Cd	Δ Cd	CI	Δ CI	Efficiency	Balance (% Front)	Δ Balance %
-300	0.404	-0.012	-0.607	-0.188	1.505	41.44	-1.83
-200	0.408	-0.008	-0.835	-0.415	2.045	42.50	-0.78
-100	0.414	-0.002	-1.089	-0.669	2.629	43.07	-0.20
0	0.417	0.001	-1.129	-0.709	2.708	43.89	0.62
100	0.414	-0.002	-1.041	-0.622	2.514	44.28	1.01
200	0.413	-0.004	-0.932	-0.512	2.258	46.23	2.96
300	0.410	-0.007	-0.884	-0.464	2.158	47.44	4.16

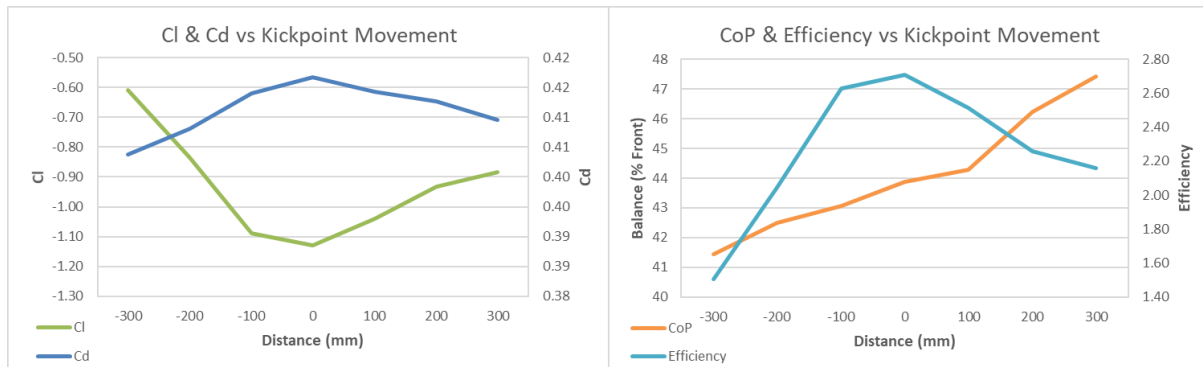


Figure 42. Diffuser side tunnel kickpoint movement vs CI & Cd (left), CoP & Efficiency (right)

In Table 18 above, the negative kickpoint values indicate the backward kickpoint movement and the positive the forward kickpoint movement. Studying Table 18 & Figure 42 above, moving the kickpoint either way has a negative effect on the diffuser's efficiency. Additionally, the CoP moves in the same direction the kickpoint moves, just like McBeath found in his studies. (McBeath, 2017)

Observing the left side of Figure 43 below, the loss in diffuser side tunnel efficiency is evident, due to the increased low velocity turbulent flow at the end of the diffuser. Studying the right side of the figure, moving the kickpoint forward, the vortex generated on the inner wall of the side tunnel seems to be stronger, however the one on the outer wall is weaker allowing for more tyre squirt to seep into to the diffuser. When the kickpoint was moved back, the sharp angles of the initial diffuser design were lost,

leading to no vortices being generated on the walls of the side tunnels, losing all the benefits they offered, hence the increased low velocity flow at the end of the diffuser compare to the initial design.

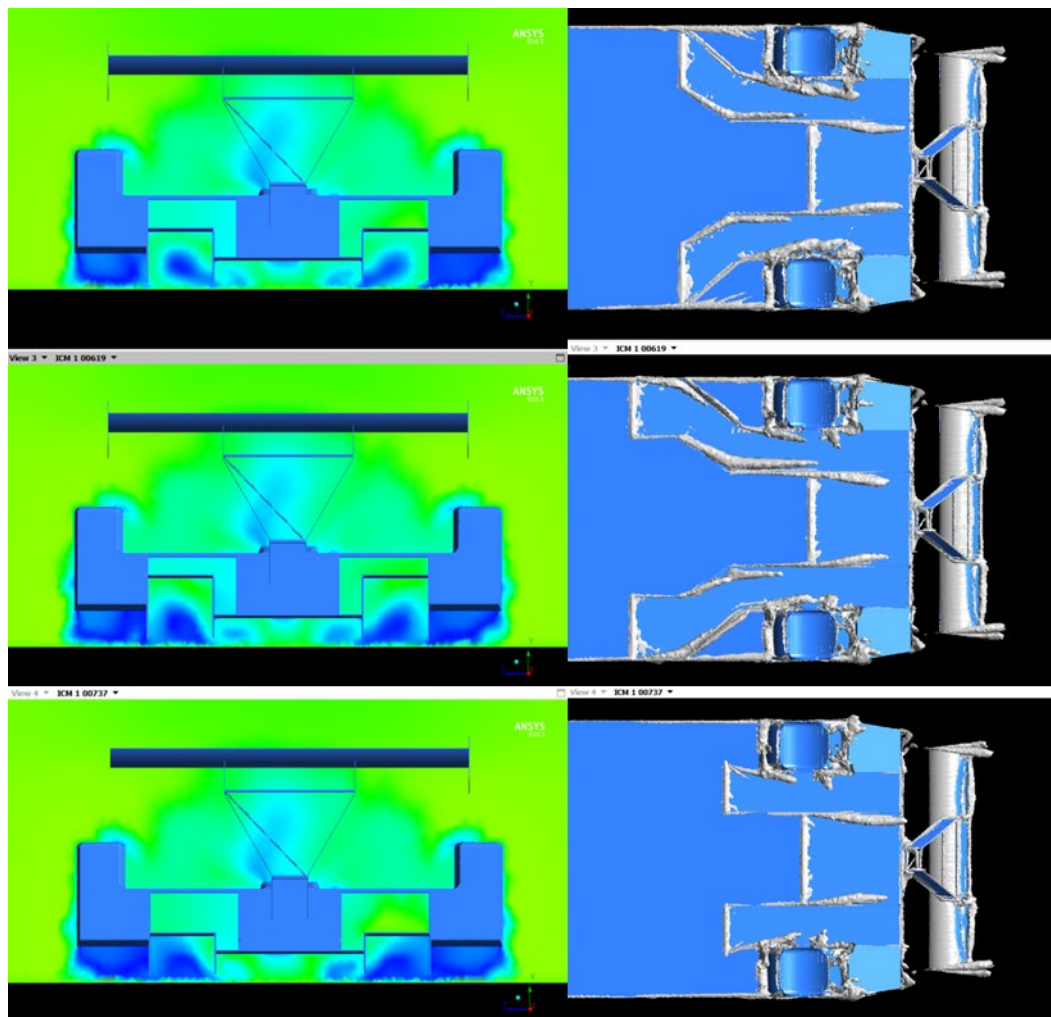


Figure 43. Velocity contours at end of diffuser (left) and vortex generation (right) for the initial model (top), the 300mm forward kickpoint (middle) and the 300mm back kickpoint (bottom)

Conclusively, the data collected in this study suggest that the side tunnel kickpoint of the initial diffuser design was placed optimally.

4.2.3 Design Reconsideration

Before conducting any further parametric studies, a design revision was performed. Two new side tunnel designs were tested:

1. Reducing the sharp angle of the side tunnels while meeting the chassis packaging limitations.
2. Maintaining the diffuser side tunnel length and angle (AR), the side tunnel kickpoint was moved back to a point where packaging allowed the diffuser to be straight throughout, without sharp angles.

The two designs can be seen below.

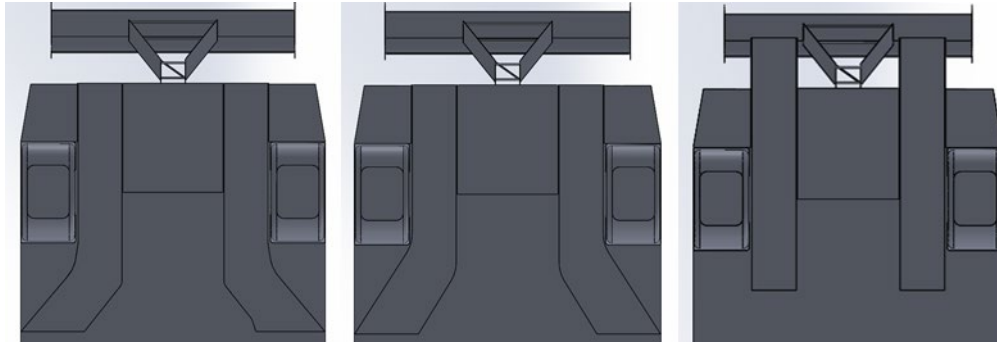


Figure 44. Initial design (left), reduced angle design (centre), no angle design (right)

Table 19. Design reconsideration study

	Cd	Δ Cd	CI	Δ CI	Efficiency	Balance (% Front)	Δ Balance %
Straight 8° Side Tunnels	0.417	0.001	-1.129	-0.709	2.708	43.89	0.62
Reduced Diffuser Turning Angle	0.415	-0.001	-1.075	-0.655	2.587	43.93	0.66
Same Length - No Turning Angle	0.400	-0.016	-0.602	-0.182	1.503	42.08	-1.20

Based on Table 19 above, reducing the sharp angle in the diffuser turn, Cd remained almost identical, while CI increased leading to a loss in efficiency compared to the initial design. This is due to the vortex generated on the outer wall of the side tunnel not being as strong as before allowing for tyre squirt to seep into the low pressure flow of the tunnel. This can be seen Figure 45 below. CoP remained steady.

The 2nd design performed poorly, losing almost half of the overall efficiency. This again is a result of no vortices being formed on the side tunnel walls allowing for tyre squirt to disrupt the flow through the diffuser.

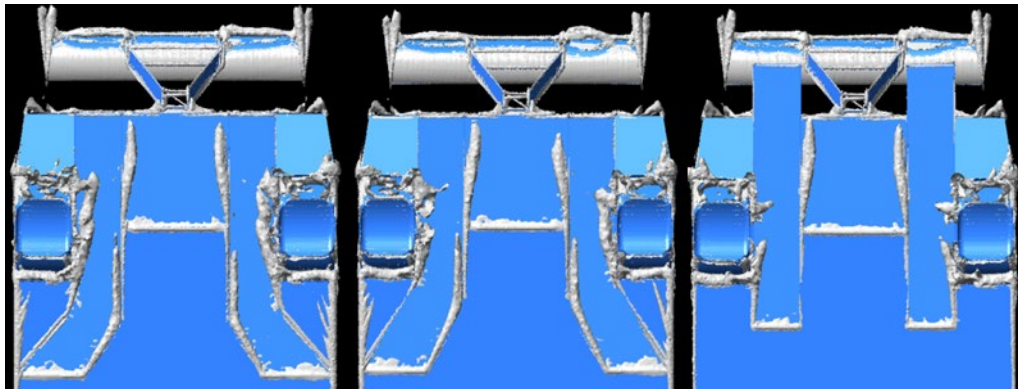


Figure 45. Vortex generation of initial design (left), reduced angle design (centre), no angle design (right)

The pressure of the underfloor is very similar when comparing the initial design to the 1st revised one, while the 2nd design clearly is ineffective. Furthermore, the increased effect of tyre squirt in both the revised designs is more prominent than the initial design's when observing the velocity contours towards the centre of the diffuser. These conclusions were drawn from Figure 46 below.

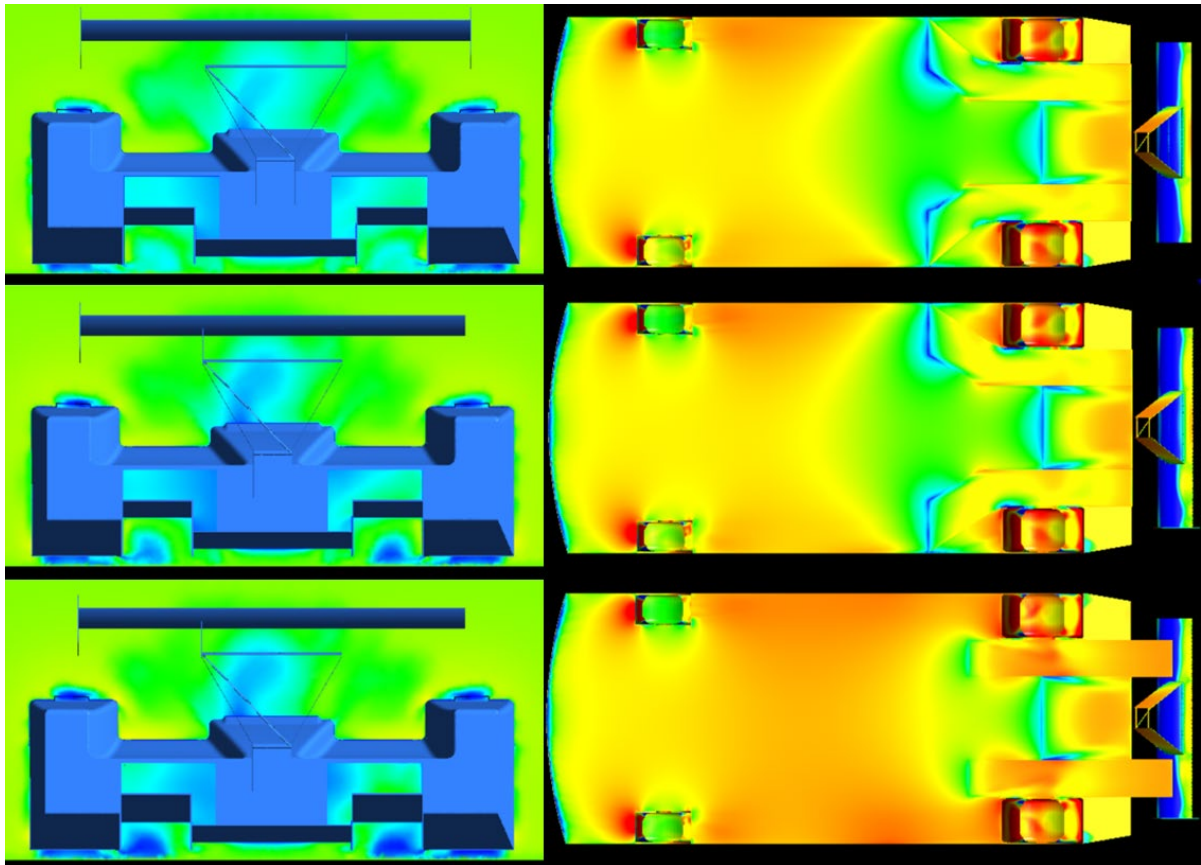


Figure 46. Velocity contours level with the rear of the MCR wheels (left) and pressure contours of underfloor (right) of Initial design (top), reduced angle design (middle), no angle design (bottom)

Due to the revised designs not outperforming the initial design, no alterations were made.

4.2.4 Length Study 2

For this study, the length of both the side tunnels and the centre section was increased incrementally by 125mm up to a 500mm overhang.

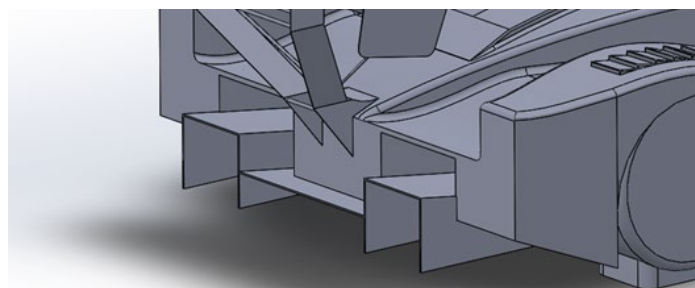


Figure 47. Length study 2 design example (125mm overhang)

Observing Table 20 & Figure 48 below, C_d remains steady and downforce increases initially at 125mm overhang but tapers off if extended any further. CoP moves back up to an overhang of 250mm, but then returns to its initial position when extending to 500mm. This is a result of flow remaining attached initially but finally separating when extending the diffuser.

Table 20. Diffuser length study 2

Overhang (mm)	Cd	Δ Cd	CI	Δ CI	Efficiency	Balance (% Front)	Δ Balance %
0	0.417	0.001	-1.129	-0.709	2.708	43.89	0.62
125	0.418	0.002	-1.133	-0.713	2.709	43.24	-0.03
250	0.416	0.000	-1.123	-0.704	2.698	43.28	0.00
375	0.416	-0.001	-1.095	-0.675	2.635	43.94	0.66
500	0.414	-0.002	-1.083	-0.663	2.614	43.98	0.70

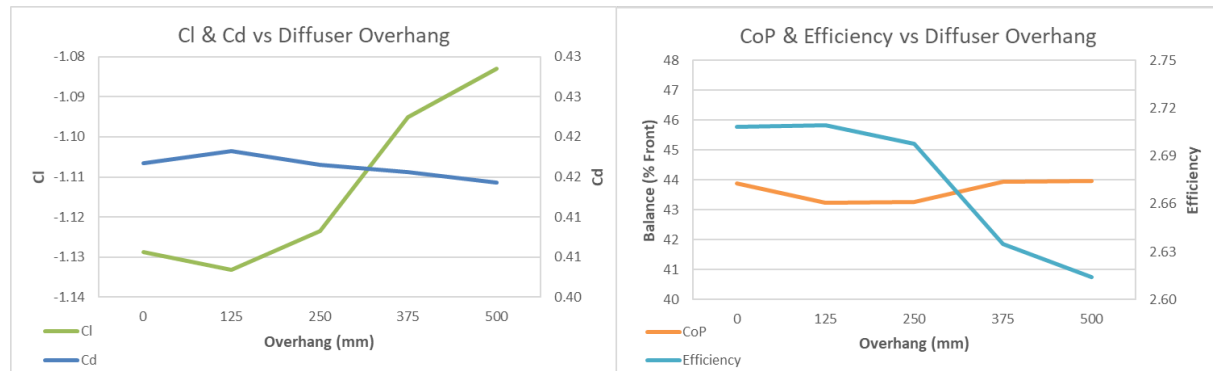


Figure 48. Diffuser length vs CI & Cd (left), CoP & Efficiency (right)

The downforce increased at 125mm overhang due the increased AR. Though, extending the diffuser further does not increase performance by virtue of flow separation due to a lower energy boundary layer turning turbulent. The obvious increase in diffuser pressure when comparing 125mm and 250mm overhang can be seen in Figure 49 below.

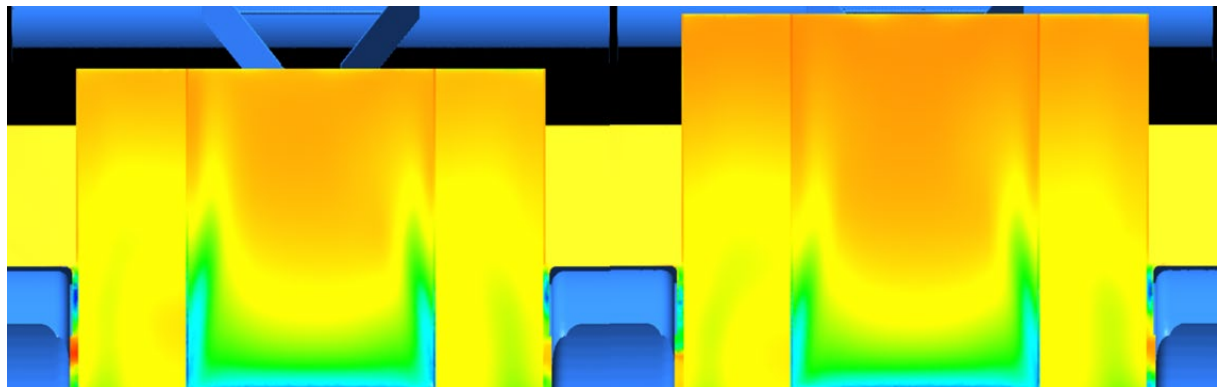


Figure 49. Diffuser pressure contour comparison of 125mm overhang (left) and 250mm overhang (right)

Studying Figure 50, the concluded diffuser flow is very similar when comparing the two cases. In the side tunnels there seems to be slightly more green colour (high speed flow) in the 125mm case. In the centre section there is more turbulent flow on the left side of the 250mm case compared to 125mm.

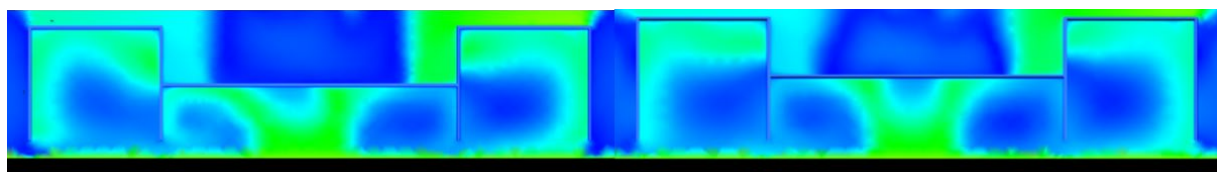


Figure 50. End of diffuser velocity contours comparison of 125mm overhang (left) and 250mm overhang (right)

The **125mm overhang** design was chosen to be implemented in all further studies, as it increased downforce and moved the CoP back compared to having no overhang.

4.2.5 Concluded Optimum Design (so far)

Based on the studies performed so far, the optimum diffuser parameters are:

Diffuser centre section angle: 8°

Diffuser side tunnel angle: 8°

Side Tunnel Kickpoint: 1000mm ahead of rear axle

Diffuser overhang: 125mm

This design has an AR of 4.

4.3 Strakes

In this study a number of strake configurations were tested, initially for the centre section followed by the side tunnels.

4.3.1 Centre Section

For the centre section, 1-4 strakes were tested spaced equally from each other.

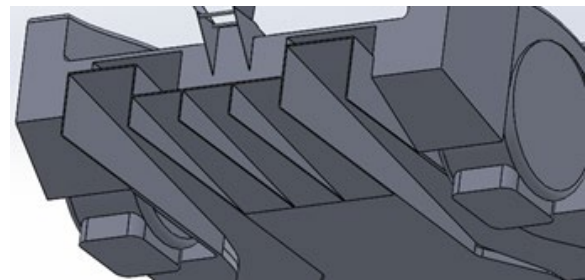


Figure 51. Centre section strake study example (2 strakes)

Observing Table 21 & Figure 52 below, the fluctuations in Cd and Cl were marginal, however one case outperformed the rest. The 2 strake setup produced the highest downforce increasing efficiency. CoP varied insignificantly.

Table 21. Centre section strake study

No. of Strakes	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance %
0	0.418	0.002	-1.133	-0.713	2.709	43.24	-0.03
1	0.416	0.000	-1.127	-0.707	2.709	43.00	-0.27
2	0.419	0.003	-1.170	-0.750	2.791	43.30	0.02
3	0.418	0.002	-1.127	-0.707	2.696	42.76	-0.52
4	0.419	0.003	-1.144	-0.724	2.732	43.44	0.17

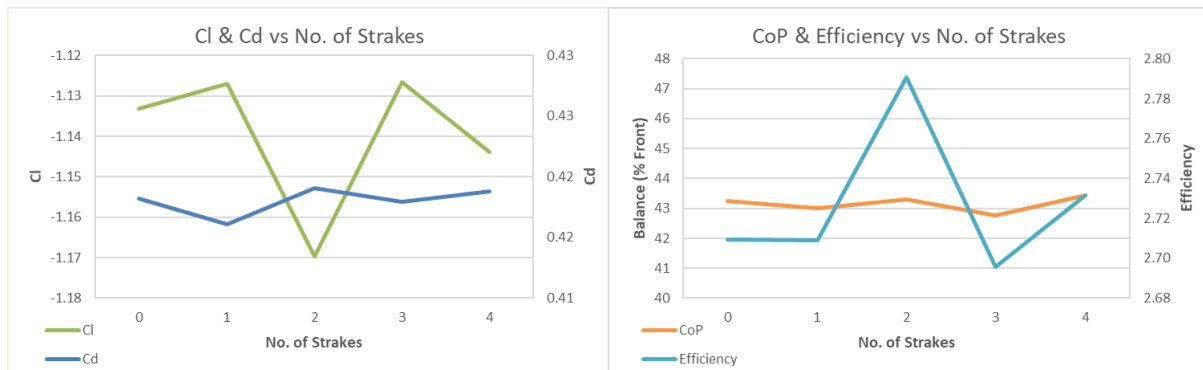


Figure 52. No. of strakes vs Cl & Cd (left), CoP & Efficiency (right)

Studying Figure 53 below, the 2 and 4 strake setup produced very similar pressures, seeming to drive the diffuser harder compared to the 1 and 3 strake setup. However, the results suggest the 2 strake setup performed the best.

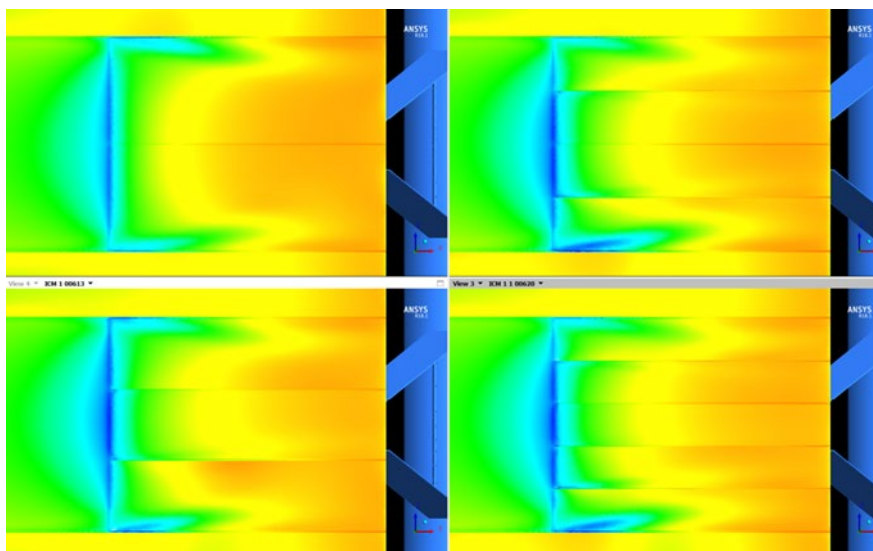


Figure 53. Pressure contours of 1 strake (top left), 2 strakes (bottom left), 3 strakes (top right), 4 strakes (bottom right)

In Figure 54 below, the vortices created by the 2 strake setup are pointed out. These vortices, act just like the ones already generated, helping in keeping the flow attached for longer in the centre of the diffuser. The 4 strake setup perhaps didn't produce better results than the 2 strake setup due to the strakes being placed too close to each other, not allowing for the vortices to grow and reach their full potential, similar to what Unni observed. (Unni, 2017)

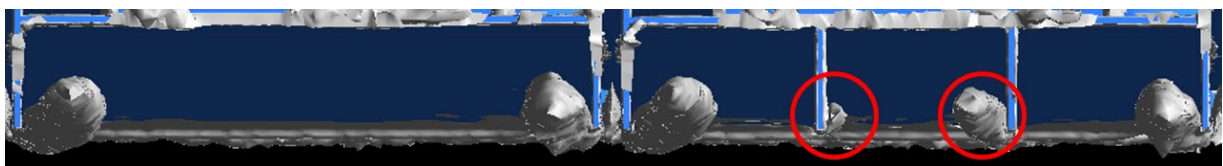


Figure 54. Vortex generation of no strakes (left) vs 2 strakes (right)

The 2 strake setup for the centre section was kept for all future studies.

4.3.2 Side Tunnels

For the side tunnel stake study, 3 stake designs were tested. One and two stake configurations were tested for each design, leading to a total of 6 stake configurations tested in this study. The stakes were equally spaced from each other.

The first version (v1) comprised of stakes starting from the point where the side tunnels straightened out. This led to the stakes not being as deep as the side tunnels, as seen in the right of Figure 55 below.

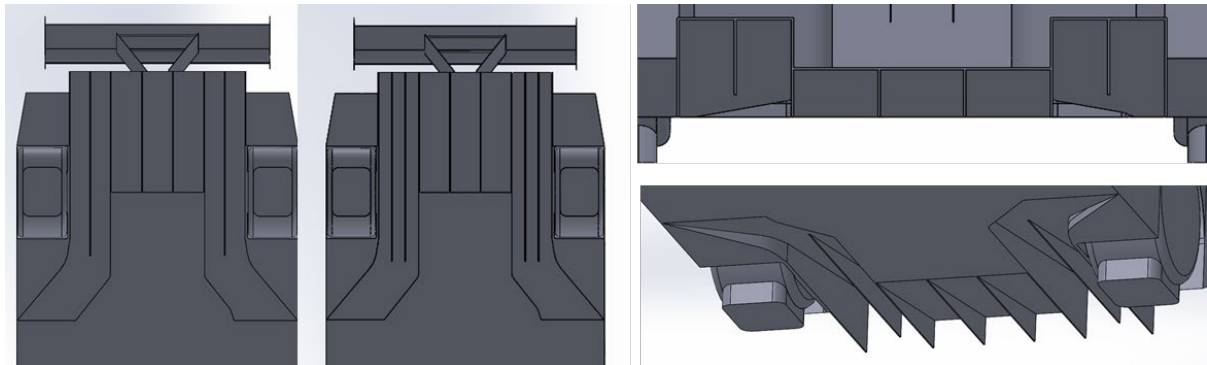


Figure 55. Single stake setup (left), double stake setup (middle), stake depth of stake design v1 (right)

The second version (v2) was identical to the first in regards of stake length and placement. The difference was that their depth was extended to meet the underfloor level, as seen in Figure 56 below.

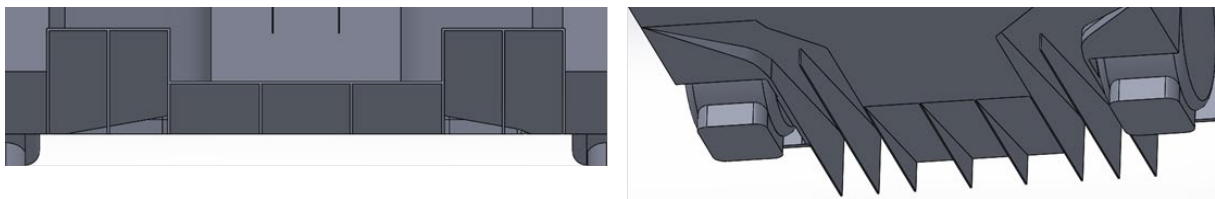


Figure 56. Stake Depth of stake design v2

For the third version the stakes were extended to the kickpoint of the side tunnels following the tunnel shape, as seen in Figure 57 below.

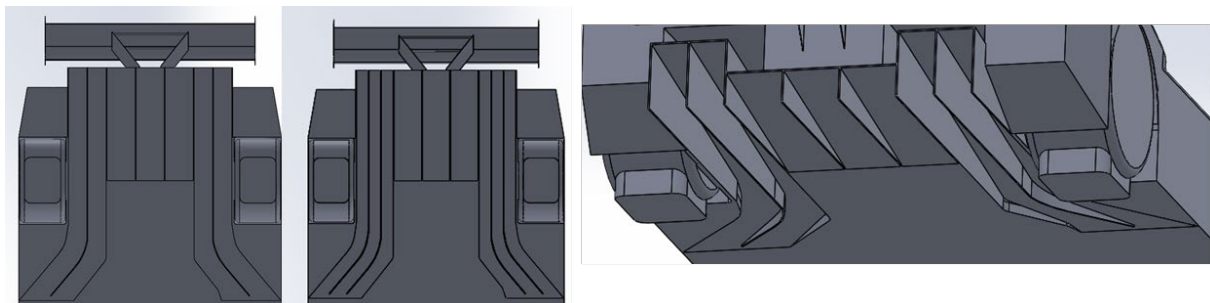


Figure 57. Single stake setup v3 (left), double stake setup v3 (middle), v3 stake design 3D view

4.3.2.1 Results & Analysis

Table 22. Side tunnel strake study

Strake Setup	Cd	Δ Cd	CI	Δ CI	Efficiency	Balance (% Front)	Δ Balance %
No Strakes	0.419	0.003	-1.170	-0.750	2.791	43.30	0.02
1 Strake v1	0.418	0.002	-1.136	-0.716	2.714	43.35	0.08
1 Strake v2	0.418	0.001	-1.153	-0.734	2.762	43.62	0.34
1 Strake v3	0.416	0.000	-1.134	-0.714	2.725	43.16	-0.11
2 Strakes v1	0.412	-0.004	-0.992	-0.572	2.408	43.59	0.31
2 Strakes v2	0.416	0.000	-1.063	-0.643	2.557	43.26	-0.02
2 Strakes v3	0.417	0.001	-1.093	-0.673	2.624	43.43	0.16

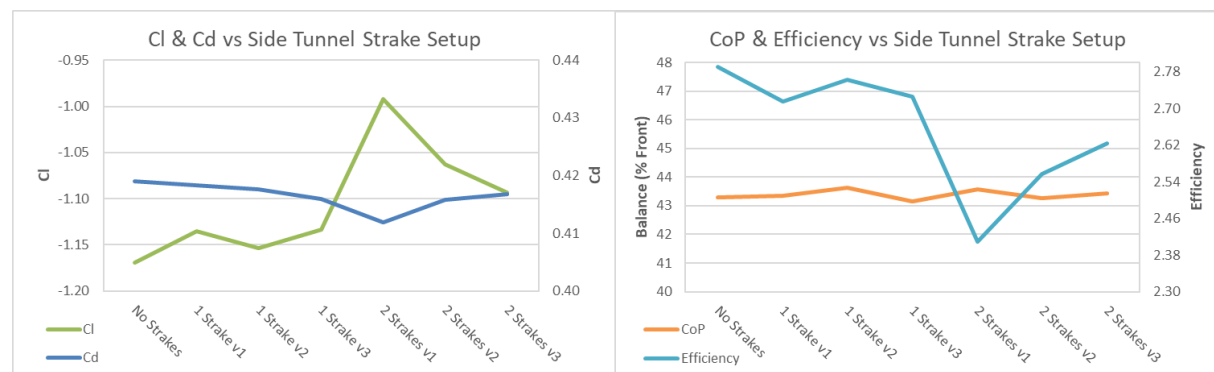


Figure 58. Side tunnel strake configuration vs CI & Cd (left), CoP & Efficiency (right)

Observing Table 22 & Figure 58 above, Cd was not affected greatly by the different strake configurations. For all 3 strake versions, the single strake setup outperformed the double strake setup. However, none of the configurations produced more downforce than the original no strake setup. The CoP movement was insignificant in this study. Version 1 with 2 strakes setup, which seems to have performed the worst, spiking the results, is likely due to skew elements in the mesh near the start of the strakes, an area prone to poor mesh quality.

Version 1 for both the single and double strake setup seemed to have increased the pressure in the centre towards back part of the side tunnels as well as in the central tunnel compared to the no strake setup, as seen in Figure 59 below. Additionally, this increase in pressure in the side tunnels has caused the centre section of the diffuser to not work as hard, leading to performance loss. Furthermore, the velocity contours below don't suggest any improvement of flow speed by adding the strakes.

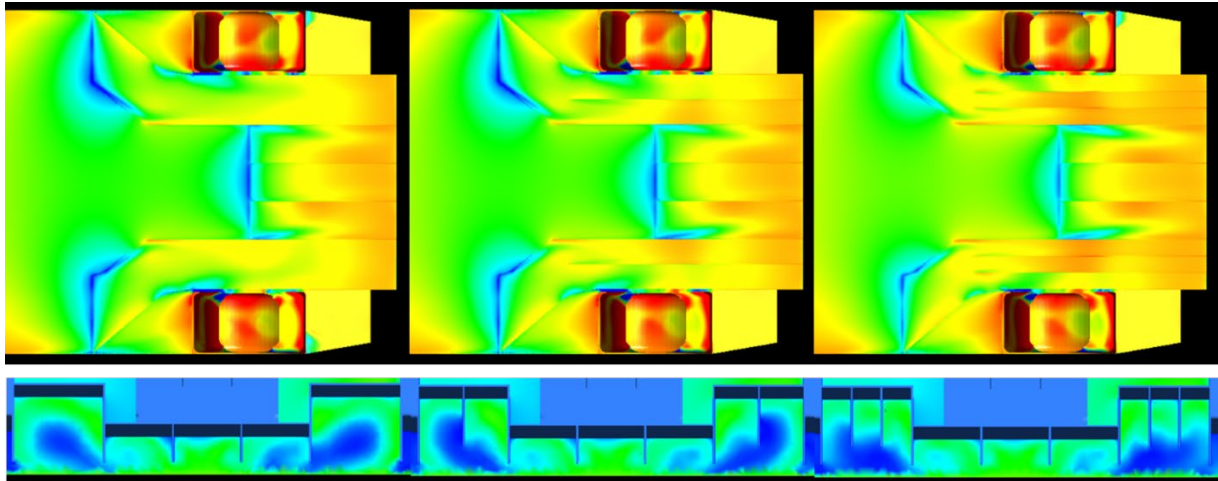


Figure 59. Pressure contours of underfloor (top) and velocity contours of diffuser ending (bottom) of no strakes (left), 1st version 1 stroke (middle) and 1st version 2 strokes (right)

The second (v2), deep version, of the stroke design, did perform better than the first version (v1) of strokes, by not increasing the pressure in the latter stages side tunnels. However, the no stroke setup centre section was working harder without strokes, as observed in Figure 60 below. The red circles point out the high velocity flow points where the flow is “ripping” around the extended stroke face in version 2 of the design. Lastly, the deeper strokes shared the low velocity flow of the side tunnels but affected and reduced the flow of the centre section, as suggested in the figure below.

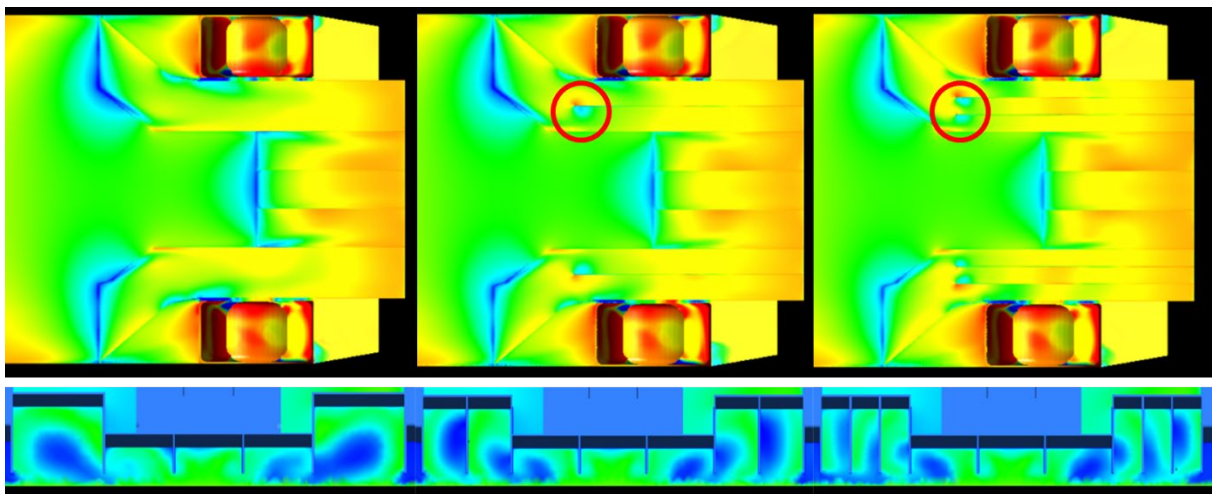


Figure 60. Pressure contours of underfloor (top) and velocity contours of diffuser ending (bottom) of no strakes (left), 2nd version 1 stroke (middle) and 2nd version 2 strokes (right)

The third design (v3) with the long strokes, performed worse than v2. The strokes starting so close to the kickpoint impacted negatively the side tunnel low pressure at the start, leading to the diffuser not working as hard overall. The red circles in Figure 61 below, indicate high pressure regions formed on the side of the strokes caused by the high pressure in front of the wheels leaking into the low pressure of the diffuser, again being detriment to the diffuser’s performance. Due to the stroke depth of version 3 being identical to version 2, the flow velocity is almost identical, affecting negatively the flow of the central section causing it to not work as hard.

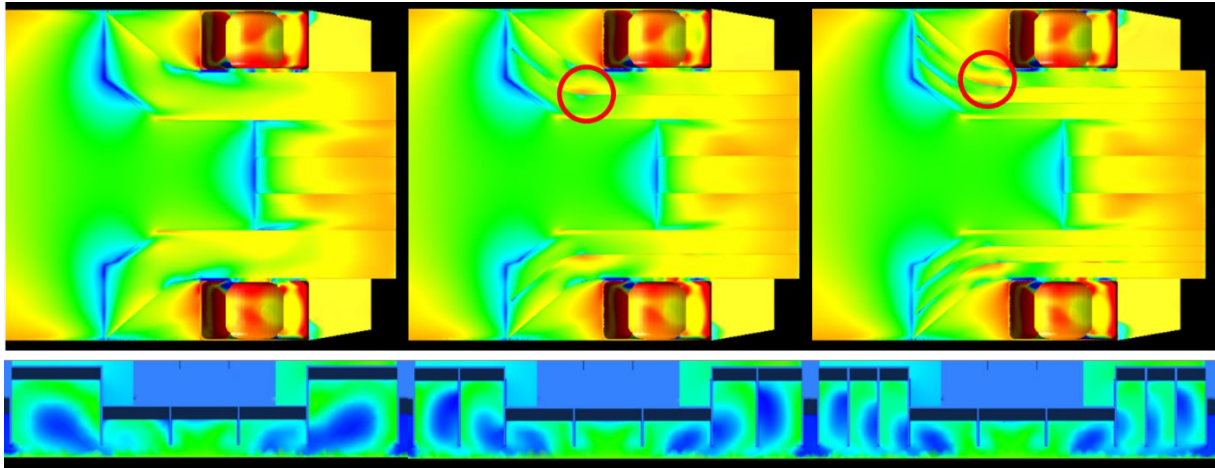


Figure 61. Pressure contours of underfloor (top) and velocity contours of diffuser ending (bottom) of no strakes (left), 3rd version 1 straw (middle) and 3rd version 2 strakes (right)

Overall, the side tunnel straw configurations did not increase performance, hence only the 2 strakes in the centre section were kept for future studies. Literature does clarify that the flow in a diffuser is complex, so by adding strakes/vanes, performance is not always enhanced as found in the study above. Furthermore, studies suggest that strakes help a diffuser perform better when it is on the verge of stalling, or has stalled, at higher diffuser angles, compared to lower diffuser angles with a fully attached flow. (Unni, 2017) (Soares, et al., 2018)

4.4 Other Studies

The studies in this section are brief and not parameterised, due to the time limitations of this project.

4.4.1 Lateral Expansion

A lateral expansion study was performed by testing an angled and a radiused lateral expansion. The expansion was kept conservative, as this study's aim was to see if in theory the design would add performance, and not to find the optimal setup.

An 8° angle of expansion laterally was added for one case, and a radius of 1200mm for the other, as can be seen in Figure 62 below.

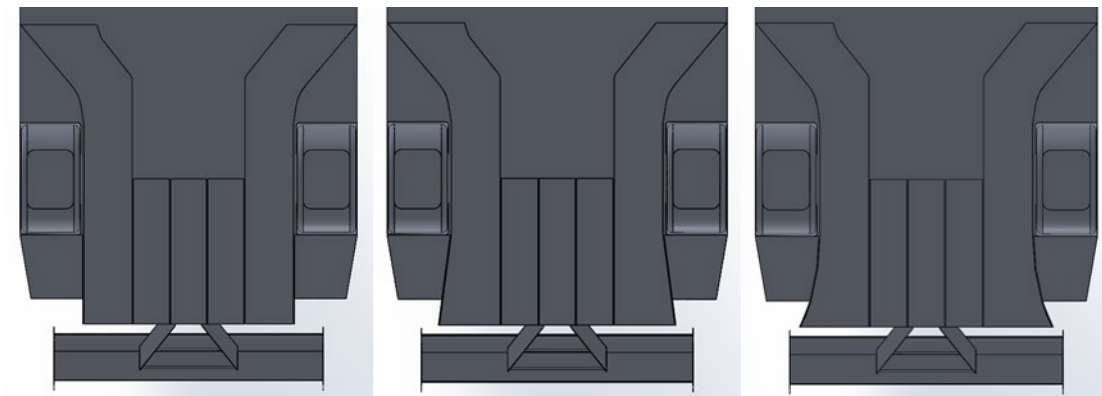


Figure 62. No lateral expansion (left), angled lateral expansion (middle), radiused lateral expansion (right)

In both cases, the Cd, CI and CoP varied marginally, however no lateral expansion outperformed the initial design, as indicated by the results in Table 23 below.

Table 23. Lateral expansion study

	Cd	Δ Cd	CI	Δ CI	Efficiency	Balance (% Front)	Δ Balance %
No Lateral Expansion	0.419	0.003	-1.170	-0.750	2.791	43.30	0.02
Lateral Expansion Angled 8°	0.419	0.003	-1.130	-0.710	2.694	43.82	0.55
Lateral Expansion Radius 1200mm	0.418	0.002	-1.156	-0.736	2.762	43.17	-0.10

Due to the differences in performance being insignificant, the velocity contours between them are very similar (Figure 63). The only noticeable difference is the flow separating laterally towards the end of the diffuser in the radiused case, possibly due the radius being too aggressive/sudden.

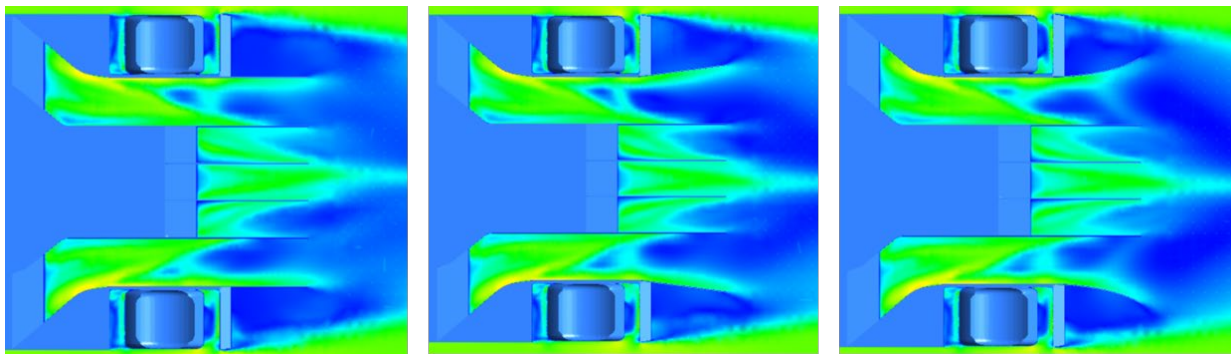


Figure 63. Velocity contours at 20mm above underfloor level of no lateral expansion (left), angled lateral expansion (middle) and radiused lateral expansion (right)

Due to the short extent of this study, a solid conclusion about lateral expansion benefiting this specific diffuser cannot be drawn, so no alterations to the design were made.

4.4.2 Diffuser Flow Optimisation (Tyre Squirt Study)

Throughout this project, tyre squirt has been detrimental to the diffuser's performance. A brief study was performed to try and reduce tyre squirt by modifying the cars bodywork within regulations. The studies so far were performed using static wheels. Rotating wheels were added as a boundary condition for this and all other studies to follow, to better understand how the flow around the wheels affects the diffusers performance.

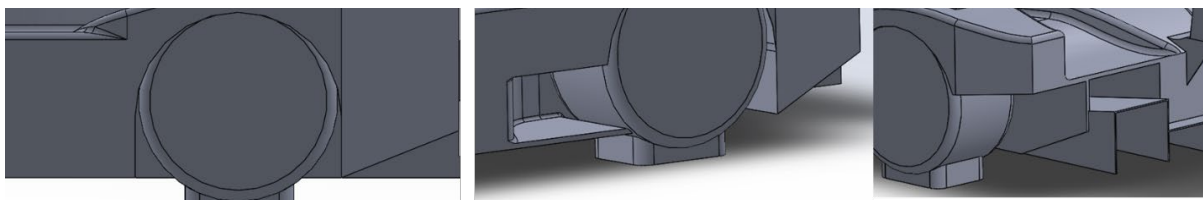


Figure 64. Original bodywork (left), body modification 1 tyre shelf (middle), body modification 2 (right)

In Figure 64 above, the two concepts to try and reduce tyre squirt can be seen. The idea behind the tyre shelf (middle) was, to remove some bodywork in front of the wheel and allow for the floor to remain so the high pressure ahead of the wheel would propagate forward and spread over (like a splitter), reducing its intensity, hence reducing tyre squirt. (McBeath, 2017)

The second idea was approached in a totally opposite manner; bodywork was removed (cutaway) from behind the wheel, to eliminate the high pressure turbulent flow in the wheel well, imitating the effect of louvres. Regulations stated that the rear wheel can be visible from the rear up to 50mm above the rear wheel axle, henceforth the bodywork removed met the regulations.

In Table 24 below, “MF” stands for moving floor, “SW” and “RW” stand for static wheels and rotating wheels respectively. Based on the table, adding the rotating wheels to the equation affected the Cd insignificantly, the Cl increased as a result of the amplified effect of rear wheel tyre squirt, while the CoP moved rearwards due lift produced by the front wheels by virtue of their rotation.

Table 24. Tyre squirt study

	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance %
Final Diffuser Setup (MF-SW)	0.419	-	-1.170	-	2.791	43.30	-
Final Diffuser Setup (MF-RW)	0.416	-0.003	-1.080	0.090	2.596	41.20	-2.10
Tyre Squirt v1 (MF-RW)	0.422	0.003	-1.013	0.157	2.402	41.50	-1.79
Tyre Squirt v2 (MF-RW)	0.418	-0.001	-1.342	-0.172	3.209	41.73	-1.57

The tyre shelf (v1) affected Cd and CoP negligibly but did however lose downforce. Reason being the high pressure on the underfloor in front of the rear wheels increased, as seen in the central image of Figure 65 below. This was caused by the overall increase in pressure on the frontal face of the tyre, which finally lead to an increase in tyre squirt, reducing diffuser side tunnel performance.

The cutaway (v2) affected the Cd and CoP negligibly as well. Though, it increased the downforce produced by a substantial amount, increasing -Cl from 1.08 to over 1.34, which corresponds to a 24% increase in downforce. This increase can clearly be seen in the right image of Figure 65 below. Both the side and central tunnel kickpoints are working harder and the overall low pressure throughout the diffuser has increased, improving its performance overall.

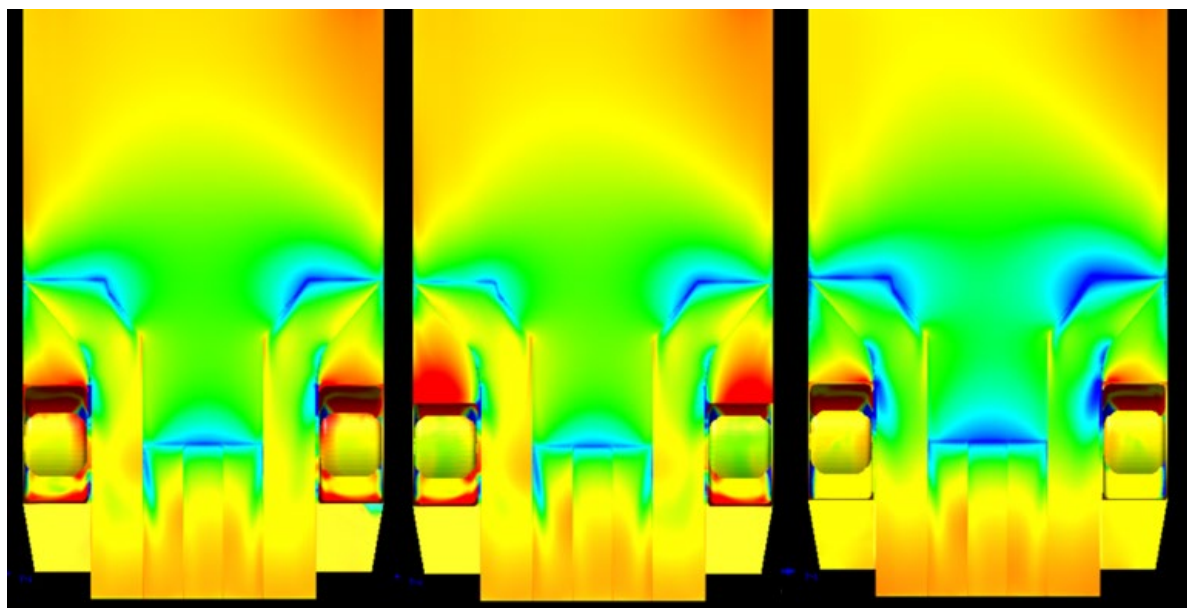


Figure 65. Pressure contours of underfloor of initial model with RW (left), tyre shelf (v1) with RW (middle), cutaway (v2) with RW (right)

The increase in pressure ahead of the wheel due to the tyre shelf can be seen in Figure 66 to the right.

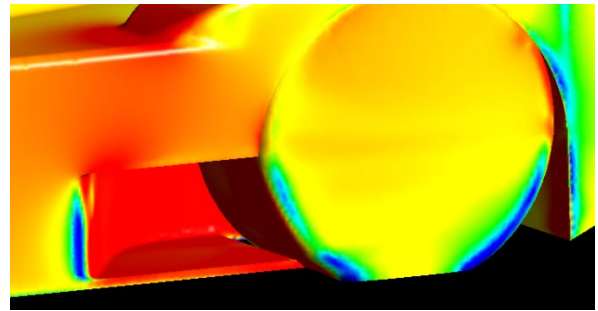


Figure 66. Tyre shelf pressure contours

In Figure 65 above, even though the cutaway produced the lowest overall pressure in the diffuser, an increase in the pressure towards the end of the diffuser is noticeable, especially in the central tunnel. This is a result of the vortices in the central tunnel not being as strong due to the reduced high pressure on the outer wall of the central tunnel due to reduced tyre squirt. The difference in vortex size can be seen in Figure 67 to the right. This applies to both sides of the central tunnel.

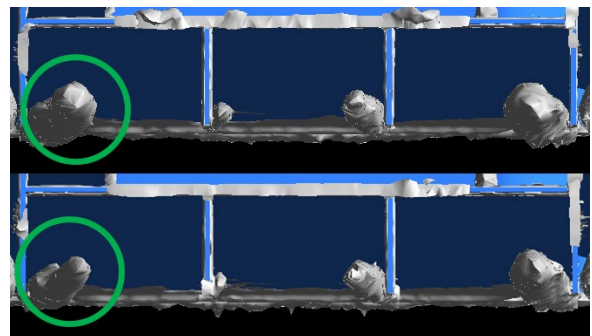


Figure 67. Vortex generation of central diffuser of initial design (top) vs version 2 (bottom)

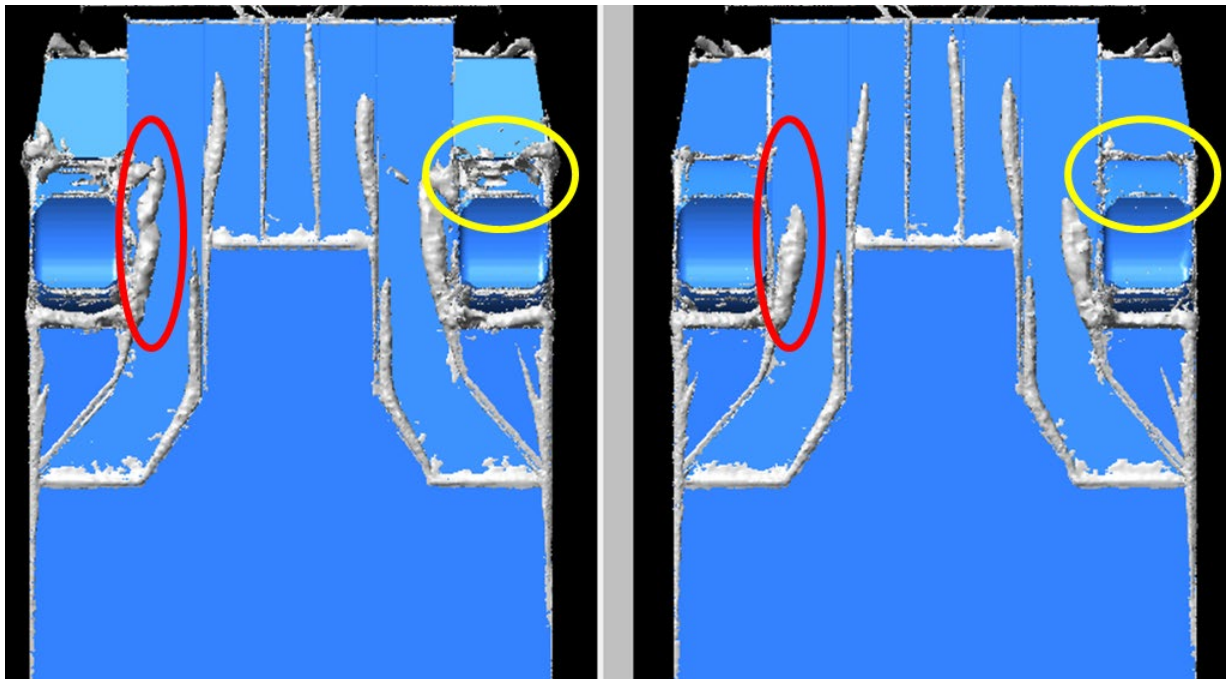


Figure 68. Vortex generation of entire diffuser of initial design (left) vs version 2 (bottom)

In Figure 68 above, the yellow circles indicate the elimination of turbulent high pressure air in the rear of the wheel well as a result of the cutaway. The red circles show the less powerful but uninterrupted outer wall vortex, as a result of eliminating the turbulent flow in the wheel wells.

In Figure 69 below, the reduction in the wake produced by the MCR car with the cutaway can be seen. This reduction in wake is due to the bodywork being shorter at the level of the diffuser which has a similar effect to extending the diffuser rearwards, allowing for a more optimal pressure recovery.

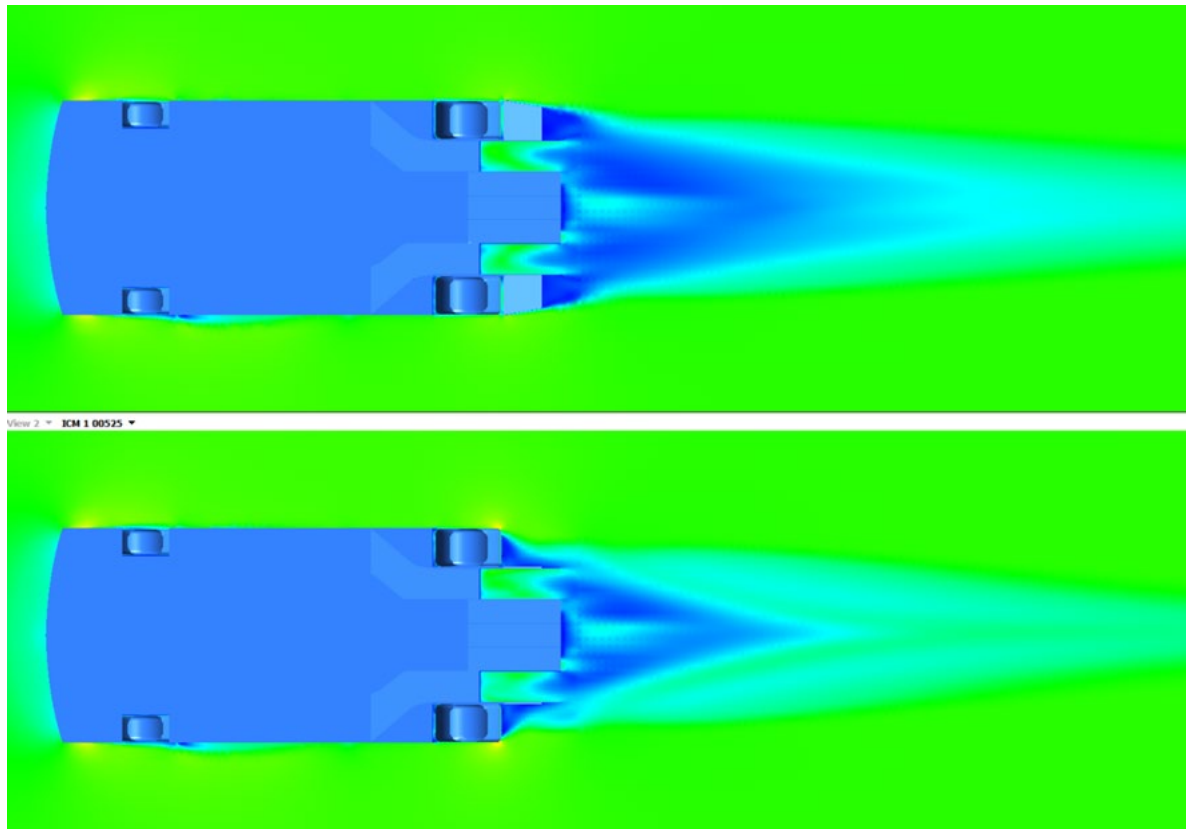


Figure 69. MCR wake of unaltered design (top) vs Version 2 (bottom)

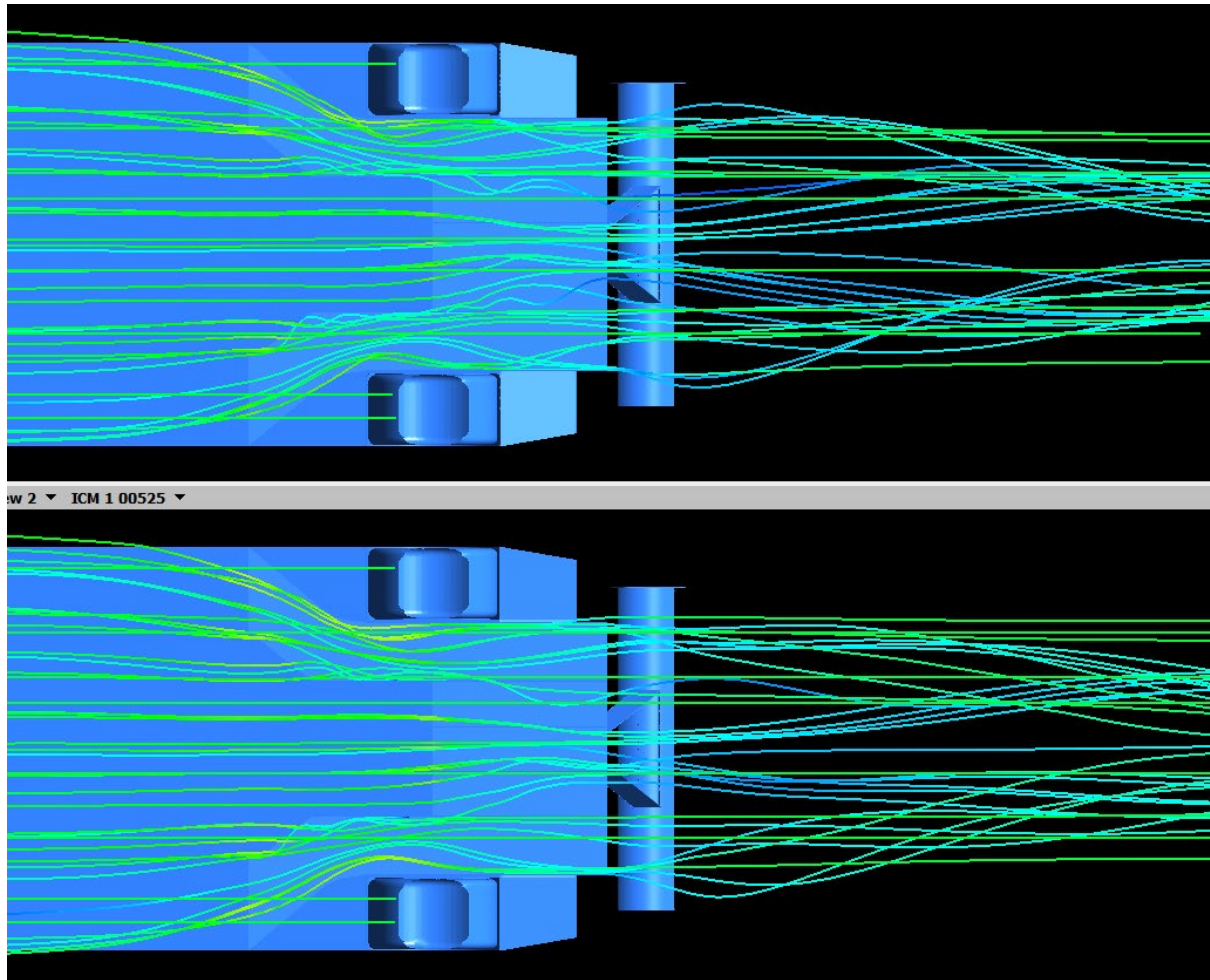


Figure 70. Underbody flow streamlines bottom view (colour: velocity), no body alterations (top) & rear wheel well cutaway (bottom)

Observing Figure 70 above, by removing part of the rear wheel well (cutaway), the increased velocity of the underbody flow is visible thanks to the yellow colour streamlines being more prominent. Additionally, the flow seems to be less “messy”, increasing flow velocity, reducing under body pressure, hence the increasing downforce.

In Figure 71 below, the streamlines from Figure 70 above were viewed from the side and from further away to observe the effect the cutaway has on the wake. It is obvious that the cutaway resulted in a stronger upwash from the diffuser, hence the increase in performance.

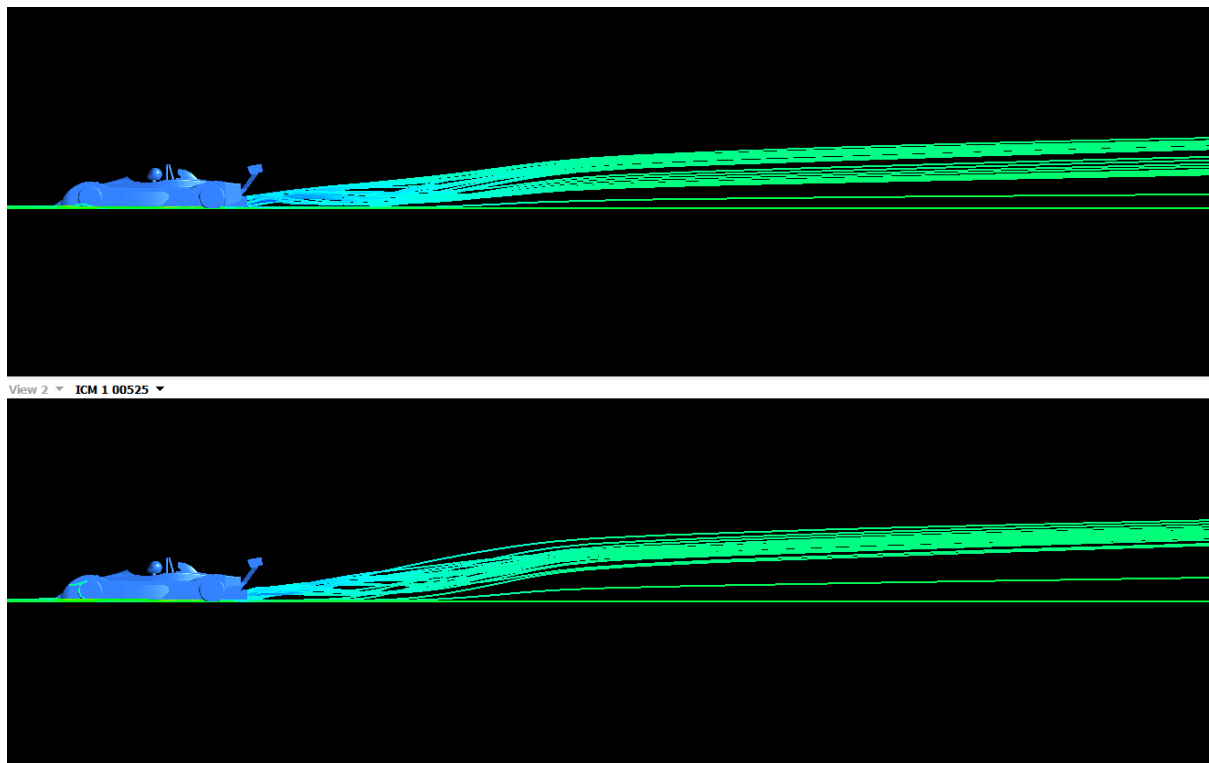


Figure 71. Underbody flow streamlines side view (colour: velocity), no body alterations (top) & rear wheel well cutaway (bottom)

To conclude, the cutaway (v2) of the tyre squirt reduction study performed significantly better than the initial design, hence it will be implemented in the final design.

4.5 Proposed Final Diffuser Design

Based on the studies and analysis performed in this project, an optimum diffuser design was created with the resources available. The details of the final design are:

Central tunnel angle: 8°

Side tunnel angle: 8°

Central tunnel Kickpoint: As forward as possible (base of engine)

Side tunnel Kickpoint: 1000mm ahead of rear wheel axle

Central tunnel width: 566mm

Side tunnel width: 247mm

Diffuser wall/strakes thickness: 5mm

Overall diffuser width: 1080mm

Central tunnel length: 740mm

Side tunnel length: 1525mm

Diffuser overhang: 125mm

Overall diffuser length: 1525mm

Central tunnel strakes: 2 equally spaced strakes

Side tunnel strakes: no strakes added

Bodywork modification to reduce tyre squirt: cutaway (v2)

This design adheres to the OSS regulations. Packaging was taken into consideration throughout the entire project, henceforth the proposed diffuser design does not interfere with any of the current car's components such as the chassis, wishbone assembly and cooling.

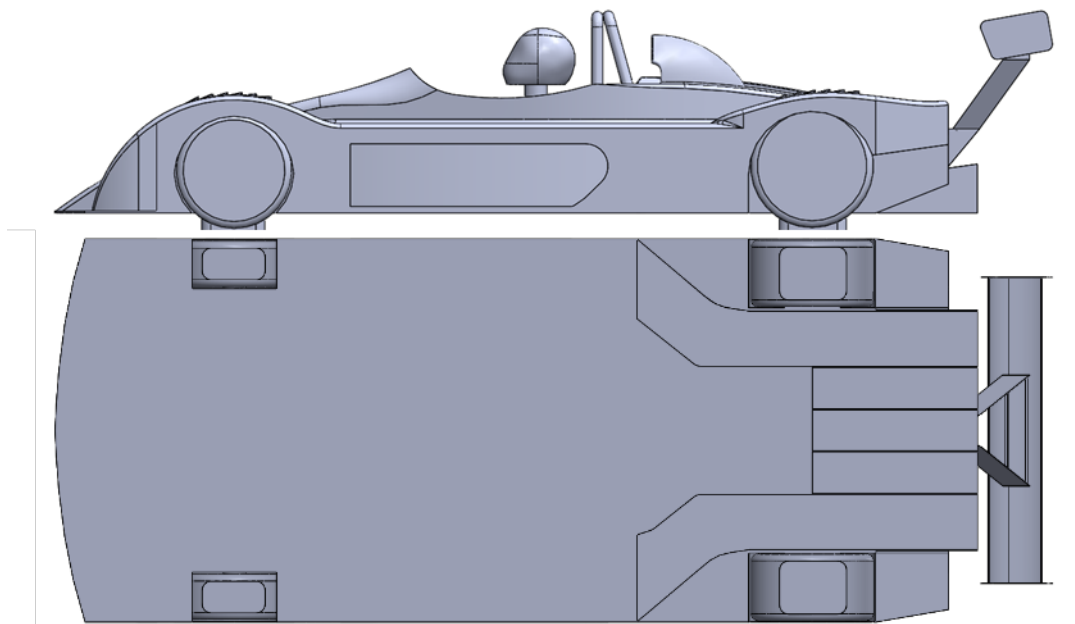


Figure 72. Final proposed diffuser design side view (top), underfloor view (bottom)

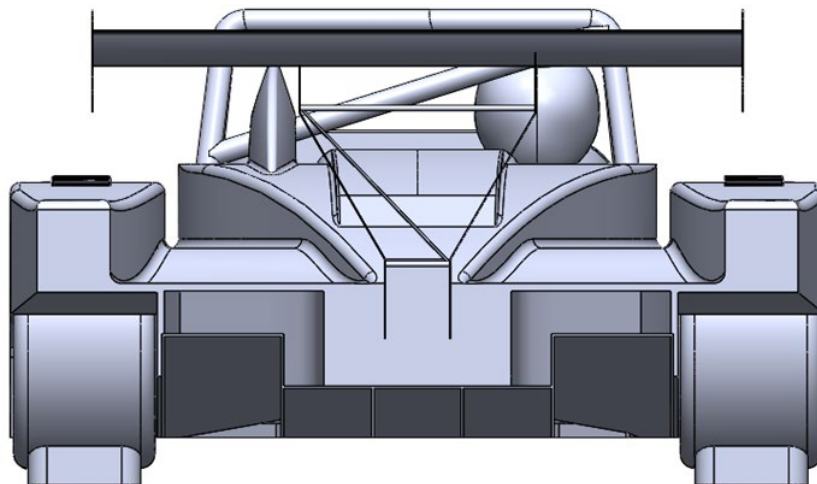


Figure 73. Final proposed diffuser design rear view

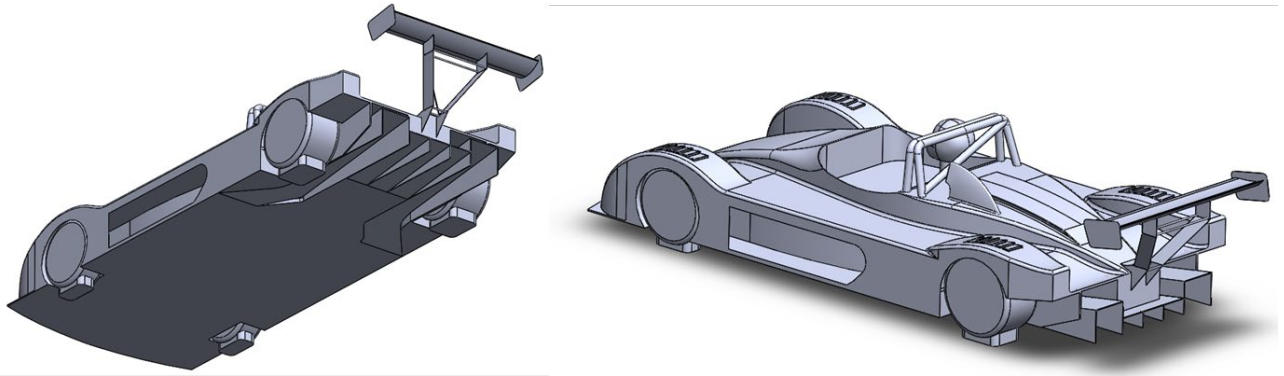


Figure 74. Final proposed diffuser design 3D views

4.6 Final Attitude Study

To solidify the performance of the proposed diffuser design, a final attitude study was performed, identical to the one performed in the earlier stages of the project.

4.6.1 Results & Analysis

4.6.1.1 Ride Height

Table 25. Ride height study 2

Ride Height (mm)	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance %
15	0.388	-0.030	-0.926	0.415	2.386	41.65	-0.08
25	0.405	-0.013	-1.280	0.062	3.160	41.68	-0.04
35	0.418	-	-1.342	-	3.209	41.73	-
45	0.419	0.001	-1.322	0.020	3.153	41.73	0.00
55	0.424	0.006	-1.312	0.029	3.096	41.74	0.01

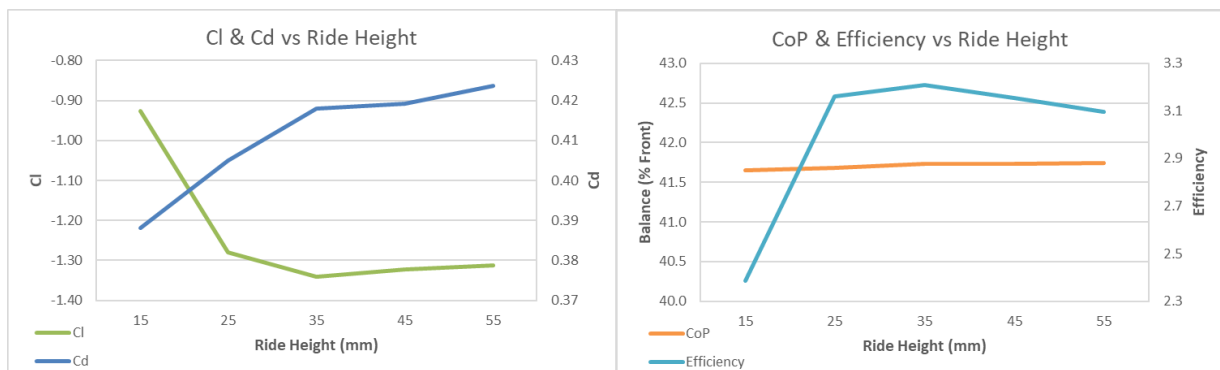


Figure 75. Ride height vs Cl & Cd (left), CoP & Efficiency (right)

Observing Table 25 & Figure 75 above, identical trends to the initial ride height study performed in chapter 3.3.1 are noticed. 35mm ride height is the optimal operating height for this diffuser design since it was developed at that ride height. From Figure 76 below, it is evident that when the car bottoms out, the flow beneath it and the effectiveness of the diffuser is reduced, especially the central tunnel. Based on the data, the optimum operating ride height range is between 30mm and 40mm,

hence the ideal static ride height for the MCR would be in the range of 45-50mm to compensate for the change in ride height due to aerodynamic loads produced at speed and dynamic weight transfer. (Unni, 2017)

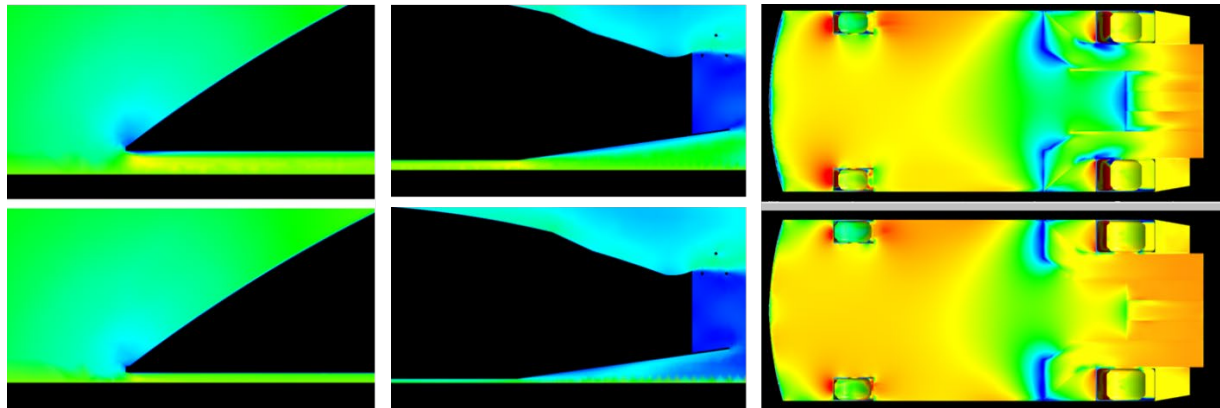


Figure 76. Velocity contours of nose and central tunnel kickpoint (left & middle) and pressure contours of underfloor at 35mm ride height (top) and 15mm ride height (bottom)

4.6.2 Rake Angle

Table 26. Rake angle study

Rake Angle (°)	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance %
-1	0.401	-0.017	-0.923	0.418	2.303	28.90	-12.82
-0.8	0.407	-0.011	-0.940	0.402	2.311	29.41	-12.32
-0.6	0.410	-0.008	-0.945	0.396	2.307	31.41	-10.32
-0.4	0.415	-0.003	-1.034	0.307	2.490	35.35	-6.38
-0.2	0.418	0.000	-1.181	0.160	2.823	36.37	-5.35
0	0.418	-	-1.342	-	3.209	41.73	-
0.2	0.431	0.013	-1.402	-0.060	3.249	44.25	2.52
0.4	0.442	0.024	-1.509	-0.167	3.417	46.16	4.43
0.6	0.444	0.026	-1.373	-0.032	3.091	48.80	7.07
0.8	0.445	0.027	-1.324	0.018	2.974	51.05	9.32
1	0.447	0.029	-1.230	0.112	2.754	52.09	10.36

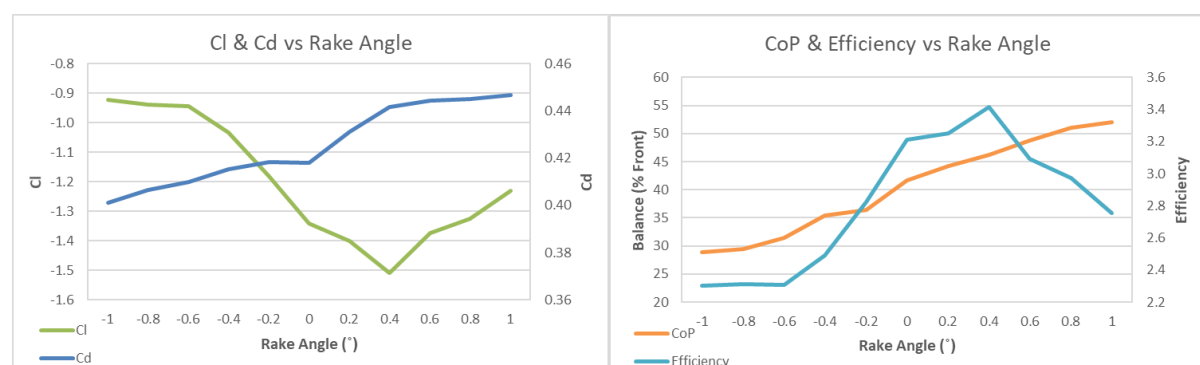


Figure 77. Rake angle vs Cl & Cd (left), CoP & Efficiency (right)

Based on Table 26 & Figure 77 above, the optimum rake angle for this diffuser is 0.4° , increasing downforce by just over 12% for a negligible penalty in drag. The CoP moves progressively forward as the rake increases throughout the entire range of angles in this study. However, the aerodynamic

balance is less sensitive in nose up situations compared to the initial model. At -1° (nose up) rake angle the aerodynamic balance is at approximately 29% front; the initial model at -0.8° was at -47% frontal aerodynamic balance generating lift at the front axle compromising the handling of the car. Based on that the diffuser has made the car significantly less sensitive to fluctuations in pitch.

Raking the car further than 0.4° does not generate more downforce. Below in Figure 78, a comparison of velocity and pressure contours of the car at 0.4° and 1° rake can be observed. Based on the velocity contours, the car at greater rake angles creates more upwash and recovers the pressure behind the car more effectively. However, when comparing the pressure contours of the underfloor the loss in performance at higher rake angles is clear. At 1° the pressure is lower around the splitter area compared to at 0.4° but the diffuser isn't working as hard at the kickpoints due to the increased distance between the kickpoint and the ground reducing ground effect and AR.

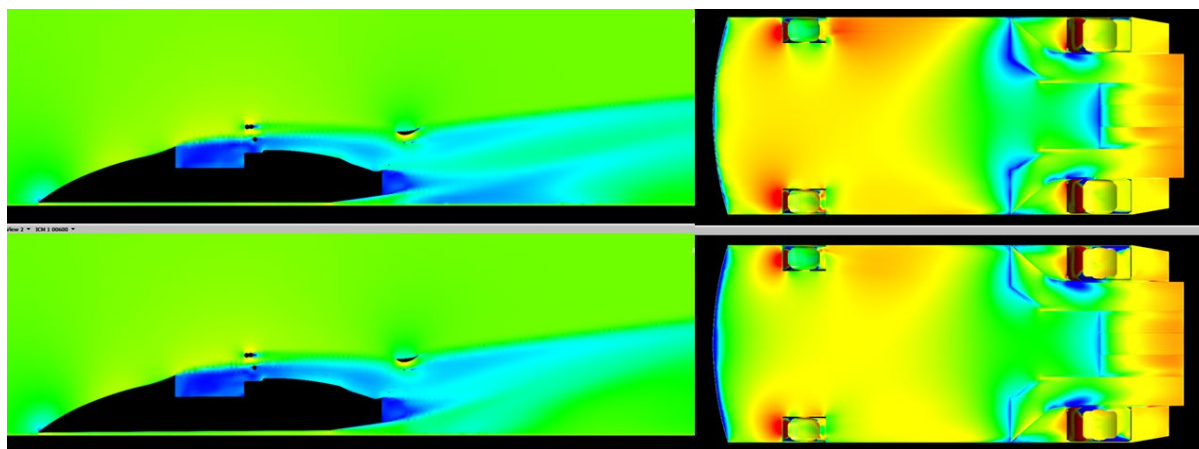


Figure 78. Velocity contours (left), underfloor pressure contours (right) at 0.4° of rake (top) and at 1° of rake (bottom)

To conclude, the optimum operating ride height for the car with the proposed diffuser design is 30-40mm with a suggested static ride height of 45-50mm. The diffuser reduced the cars pitch sensitivity and performed optimally at a rake angle of 0.4° .

4.7 New Splitter Final Study

Throughout the project due to resource and time limitations the car was run without the new splitter, hence the final study below.

4.7.1 Results & Analysis

Table 27. New Splitter Final Study

	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance %
Baseline	0.416	-	-0.420	-	1.009	43.27	-
No Diffuser New Splitter	0.436	0.020	-0.452	-0.032	1.038	69.13	25.86
New Diffuser Old Splitter	0.418	0.002	-1.342	-0.922	3.209	41.73	-1.55
New Diffuser New Splitter	0.474	0.058	-1.625	-1.205	3.430	58.66	15.39

Based on Table 27 above, adding the new diffuser to the model with the new splitter, brought the centre of pressure rearwards by the substantial amount of over 10%, leading to an aerodynamic

balance of approximately 59/41 compared to the initial 69/31. Below in Figure 79, the increase in downforce is evident as a result of the increased velocity under the car and specifically the new splitter. The $-C_l$ increased by adding the new splitter, as did the C_d , possibly due to a slightly larger frontal area and increased turbulence around the car. Additionally, the aerodynamic efficiency increased from 3.21 to 3.43.

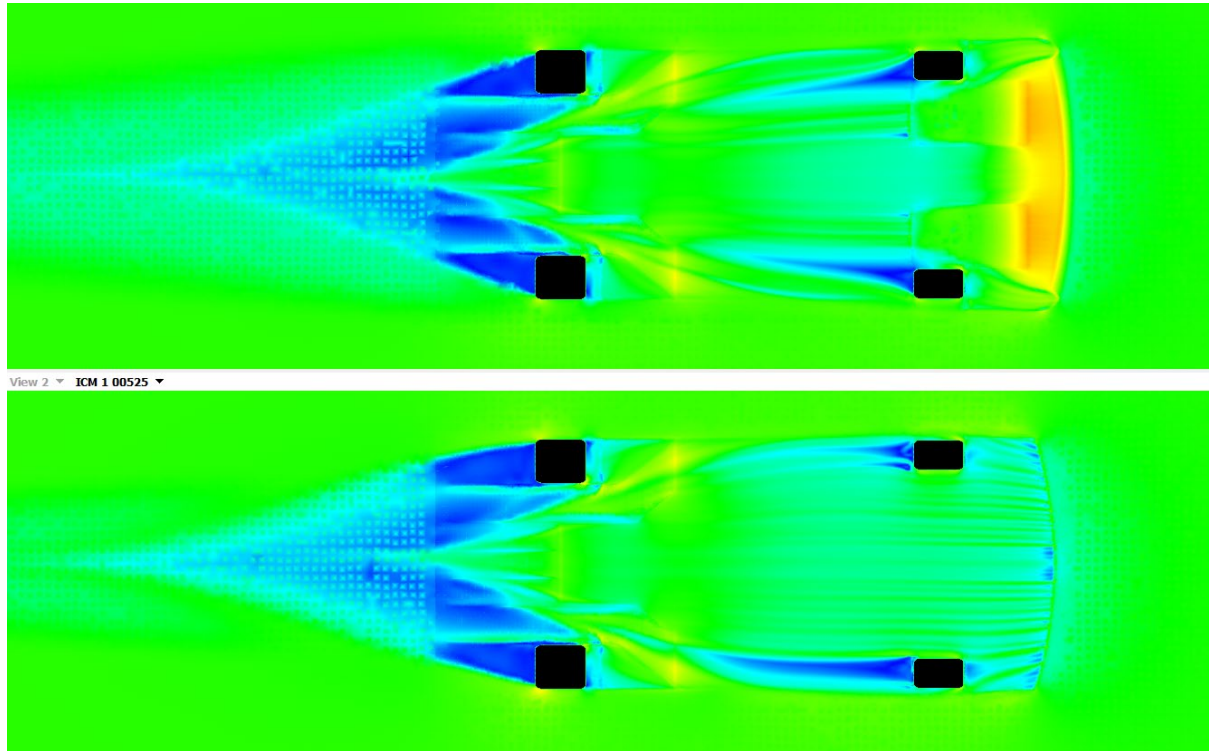


Figure 79. Underfloor velocity contours of proposed diffuser design with new splitter (top) and without new splitter (bottom)

5 Conclusion

The project aims were to increase aerodynamic performance and efficiency and to bring the CoP further rearwards due to a new splitter design that moved it too far forward.

The first aim of increasing performance was achieved by increasing downforce produced by 220% ($-C_l$ 1.342 compared to 0.42) and increasing aerodynamic efficiency from 1.01 to 3.21 at a flat ride height of 35mm.

The second aim was partially achieved, it did not reach the ideal balance of 50/50, nevertheless the new diffuser did indeed bring the CoP further rearwards accomplishing an aerodynamic balance of 59% frontal downforce vs 41% rear downforce compared to the previous 69/31.

6 Recommended Further Work

As the desirable aerodynamic balance was not achieved by adding a diffuser, a development of a new rear wing to bring the CoP further rearwards would be beneficial.

A wing location study and a wing-diffuser interaction study would yield valuable data for further development.

Adding diffuser flaps/wings to the outer walls of the diffuser in close proximity to the ground would be another viable option to achieve this. A brief investigation was performed and can be found in the appendix.

As part of the Attitude study a Yaw angle test would be of great benefit, as a vehicles aerodynamic performance can be greatly affected by yaw. (McBeath, 2017)

Overall, throughout the project 161 individual simulations were performed with an average time of 8.5 hours per simulation (ICEM & Fluent), leading to 1369 hours of CFD conducted. The data collected equated to 678GB.

Finally, if time and resources available were unlimited, the majority of the studies in this project could be revisited and analysed to a far greater extent using CFD, wind tunnel testing and track testing in conjunction. This would allow for unparalleled results to be achieved.

7 References

ANSYS, 2012. Mesh Quality. In: s.l.:s.n.

BRSCC, 2018. *British Racing & Sports Car Club*. [Online]

Available at: <http://www.brscc.co.uk>

[Accessed November 2018].

Christoffersen, L. et al., 2011. *Wing-Diffuser Interaction on a Sports Car*, s.l.: SAE International.

Cooper, K. R. et al., 1998. *The Aerodynamic Performance of Automotive Underbody Diffusers*, NRC: SAE International.

Desai, S. S., Lo, C.-M. B. & George, A. R., 2008. *A Computational Study of Idealized Bluff Bodies, Wheels and Vortex Structures in Ground Effect*, Ithaca: SAE International.

Dominy, R. G., Ryan, A. & Sims-Williams, D. B., 2000. *The Aerodynamic Stability of a Le Mans Prototype Race Car Under Off-Design Pitch Conditions*, Detroit: SAE International.

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*, Ithaca: SAE International.

Fuller, M. J., 2004. *2004 LMP1 & 2 aero. rules*. [Online]

Available at: <http://www.mulsannescorner.com/aco2004.html>

George, A. R. & Donis, J., 1983. Flow Patterns, Pressure and Forces on the Underside of Idealized Ground Effect Vehicles. II(7).

Katz, J., 2006. Aerodynamic forces and terms. In: *Race car aerodynamics*. Massachusetts: Bentley Publishers, pp. 36-38.

Katz, J., 2006. Aerodynamics and Race Cars. In: *Race Car Aerodynamics*. Massachusetts: Bentley Publishers, pp. 4-15.

Katz, J., 2006. Aerodynamics and Vehicle Performance. In: *Race Car Aerodynamics*. Massachusetts: Bentley Publishers, pp. 167-189.

Katz, J., 2006. Aerodynamics Forces - Basic Terminology. In: *Race car aerodynamics*. Massachusetts: Bentley Publishers, pp. 25-26.

Katz, J., 2006. Aerodynamics of the Complete Vehicle. In: *Race Car Aerodynamics*. Massachusetts: Bentley Publishers, pp. 197-201.

Lanfrit, M., 2005. *Best practice guidelines for handling Automotive External Aerodynamics with FLUENT*, Darmstadt: Fluent Germany.

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

- McBeath, S., 2017. Computational Fluid Dynamics (CFD). In: *Competition Car Aerodynamics*. Dorchester: Veloce Publishing Limited, pp. 46-58.
- McBeath, S., 2017. Out of thin air. In: *Competition car Aerodynamics*. Dorchester: Veloce Publishing Limited, pp. 22-23.
- McBeath, S., 2017. Removing (some of) the Guesswork. In: *Competition Car Aerodynamics*. Dorchester: Veloce Publishing, pp. 213-249.
- McBeath, S., 2017. Underbody Aerodynamics. In: *Competition Car Aerodynamics*. Dorchester: Veloce Publishing, pp. 146-183.
- Merriam-Webster, 2018. *Dictionary*. [Online]
Available at: <https://www.merriam-webster.com/dictionary/simulation>
[Accessed 22 December 2018].
- MIRA, 2019. *Full scale aerodynamic wind tunnel*. [Online]
Available at: [https://www.horiba-mira.com/our-services/full-scale-wind-tunnel-\(fswt\)](https://www.horiba-mira.com/our-services/full-scale-wind-tunnel-(fswt))
- MotorsportUK, 2018. *Motorsport UK*. [Online]
Available at: <https://www.motorsportuk.org>
[Accessed November 2018].
- Piechna, J., Rudniak, L. & Piechna, A., 2009. *CFD Analysis of the Cental Engine Generic Sports Car Aerodynamics*, Munich: EASC.
- Soares, R. F. et al., 2018. *On the Aerodynamics of an Enclosed-Wheel Racing Car: An Assessment and Proposal of Add-On Devices for a Fourth, High-Performance Configuration of the DrivAer Model*, Cranfield: SAE Technical.
- Sovran, G., 1994. *The Kinematic and Fluid-Mechanic Boundary Conditions in Underbody Flow Simulation*, Turin: s.n.
- Toet, W., 2017. Willem Toet explains... motorsport diffusers. *Race Tech Motorsport Engineering*, 8 August.
- Unni, T., 2017. *Numerical Investigation on Aerodynamic Effects of Vanes and Flaps on Automotive Underbody Diffusers*, Delhi: SAE International.
- Werner, F. & Frik, S., 1998. Aerodynamic Optimization of the Opel Calibra ITC Racing Car Using Experiments and Computational Fluid Dynamics. *Developments in Vehicle Aerodynamics*, 23 February.
- Wright, P., 1982. The Influence of Aerodynamics on the Design of Formula One Racing Cars. *International Journal of Vehicle Design*, III(4).

8 Appendices

8.1 Appendix 1

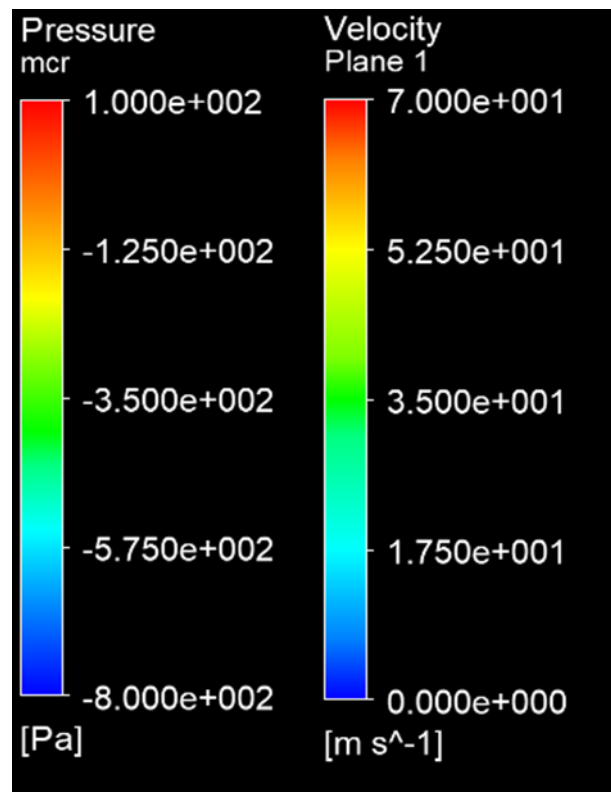


Figure 80. Pressure and velocity legends

8.2 Appendix 2

Table 28. Diffuser Flap Study

	Cd	Δ Cd	Cl	Δ Cl	Efficiency	Balance (% Front)	Δ Balance %
Final Baseline (MF-RW)	0.418	-	-1.342	-	3.209	41.73	-
Final Baseline with Flap (MF-RW)	0.432	0.014	-1.417	-0.075	3.279	40.59	-1.14

Observing Figure 81 & Figure 82, the high velocity flow in which the flap was placed can be seen.

In Figure 83 the pressure contours on the underside of the flaps is displayed which did produce more downforce based on the results in Table 28 above. Additionally, the CoP moved further rearwards.

A more in depth study of such a flap could be of benefit.

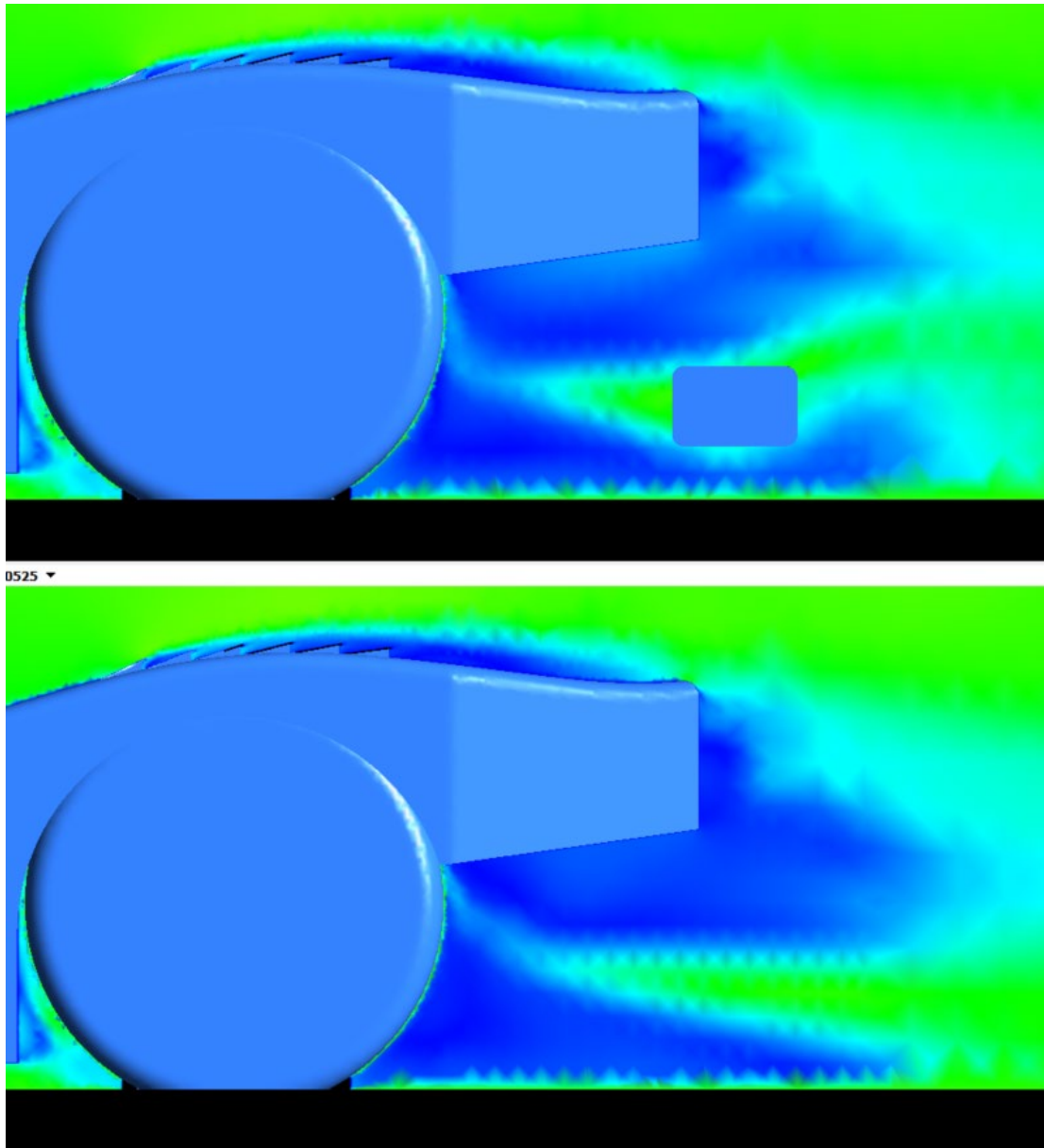


Figure 81. Velocity contours: diffuser with flap (top), diffuser without flap (bottom)

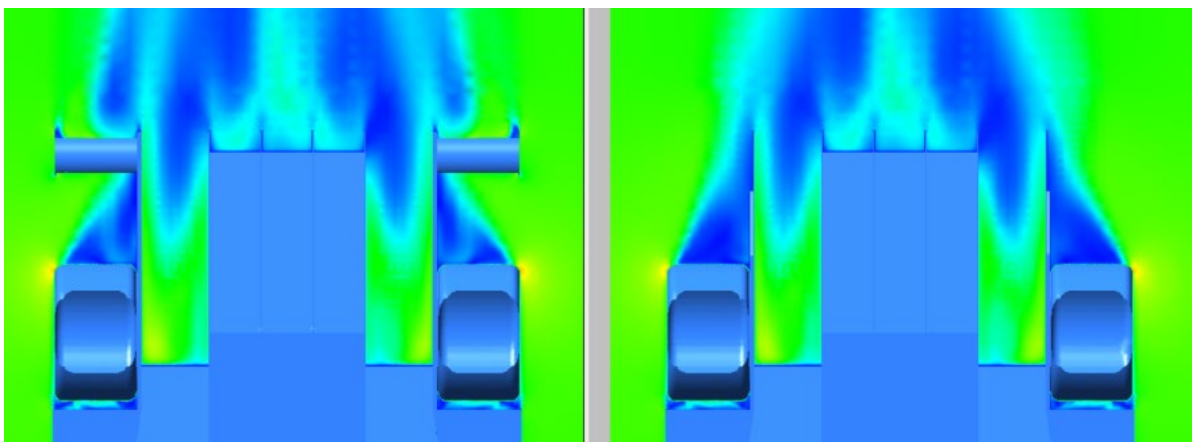


Figure 82. Velocity contours: diffuser with flap (left), diffuser without flap (right)

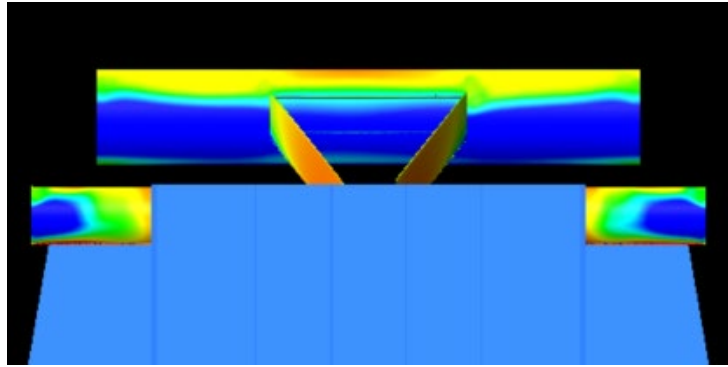


Figure 83. Pressure contours of wing and flaps

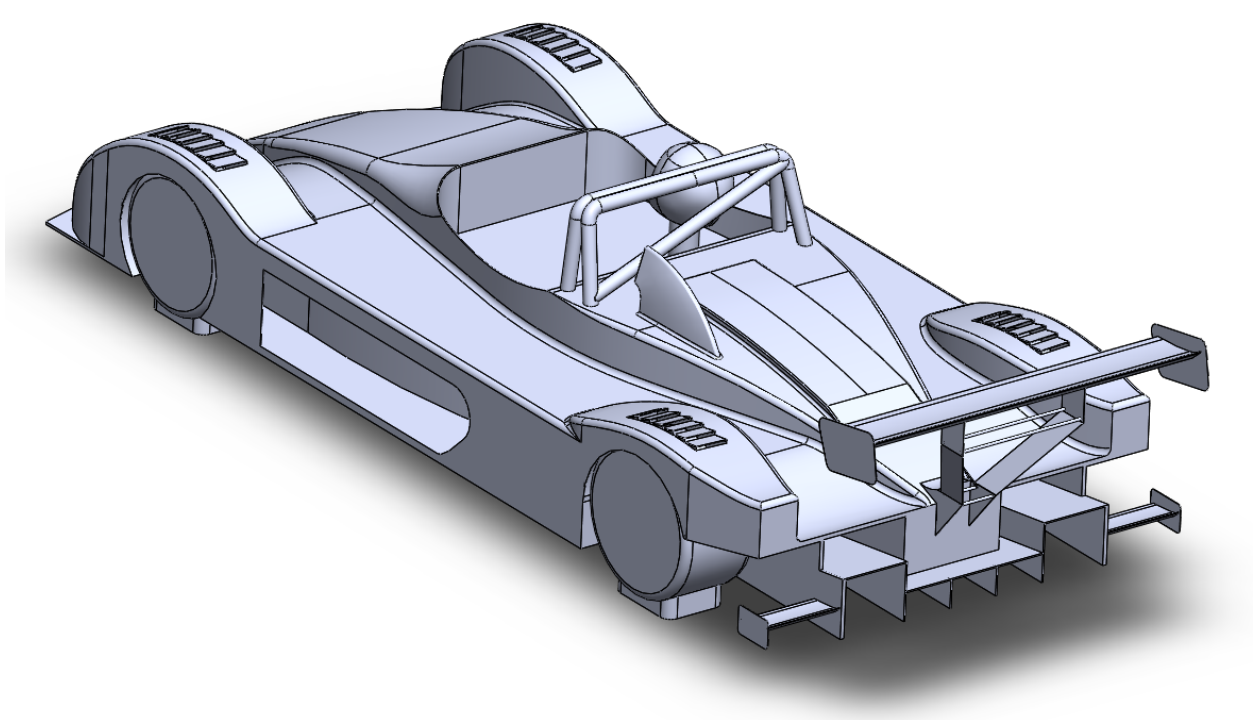


Figure 84. 3D view of diffuser flap