

Aerodynamic Design and Configuration of a Front Wing Concept to Investigate Open Wheel Near-Wake.

John Michael Hughes

MSc Motorsport Engineering

P146322

University of Wales Trinity Saint David

Major Project

Supervisor: Tim Tudor

20/8/2020



Abstract

Using numerical methods, the front wing on a modified Dallara F399/01 single seat race car was analysed. Subsequently, a design methodology was implemented in order to investigate the potential for inducing further total downforce, drag reduction and CoP relocation whilst analysing the front wheel near wake and monitoring its control.

A new front wing concept was developed, which produced a small reduction in total drag force, with total downforce suffering a reduction in value as a consequence. It was considered that the CoP could be relocated further forward, however, rear downforce suffered due to onset flow restrictions, resulting in the reduction of total downforce.

It was recognised that the front wheel near wake structure and control was heavily influenced by the front wing geometry, mainly through vortex developments generated by the front wing end plates.

It was also acknowledged that the Y250 vortex control suffered heavily without the redesign of the barge board geometry, which was left unaltered as a result of time constraints and with the purpose of simplifying the investigation.

Unforeseen Circumstances

Due to the Covid-19 virus pandemic, which commenced in February 2020, and encompassing the period of writing this report, a number of restrictions were applied nationwide.

The resulting restrictions that were introduced impacted significantly on the use of university resources. The availability of computational power became severely limited, which impeded design and simulation times. A number of investigations, therefore, could not be completed to the desired depth and detail considered necessary.

As on track testing was terminated for the duration of this time, the original validation studies could not take place. These involved the construction of a pressure pitot rake to actively analyse the generated front wheel wake of the Dallara F399/01 and validate the computational methods utilised. Validation of these methods would have provided further detail and clarification of the developed CAD and CFD models.

Another test which could not take place involved the utilisation of the Horiba MIRA wind tunnel. This would have provided numerical data which could have been used as a comparison to the predicted loadings generated by the computational methods.

Acknowledgements

I would like to thank Tim Tudor for his continued support throughout the duration of my university career and helping develop my understanding and engineering skills.

Further gratitude is given to both John Iley and Chris Bull who have both offered advice and guidance in accordance with a number of projects, including this report.

I would also like to thank Steve Broughton and Matt Hillam of SBD Motorsport Ltd for allowing me to work with them and utilise their race vehicle for the purposes of this project. Without their support and understanding, this project would not have been able to take place.

Additional appreciation is extended towards my family who have continually and consistently supported me throughout my duration at the University of Wales Trinity Saint David.

Declaration

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

Signed:.....

Date: 20/08/2020.....

Contents

Abstract.....	1
Unforeseen Circumstances.....	1
Acknowledgements.....	2
Declaration.....	2
List of Figures	5
Abbreviations	12
1.0.0 Introduction	13
1.1.0 Aims and Objectives.....	14
2.0.0 Literature Review	15
2.1.0 SBD Ltd Dallara F399/01.....	15
2.2.0 The British Sprint Championship: Technical Regulations	18
2.3.0 Wings in Ground Effect.....	20
2.3.1 Multi – Element Wings in Ground Effect	23
2.4.0 Aerodynamics of Wheels.....	25
2.4.0 Front Wing and Wheel Interaction	30
3.0.0 Methodology	33
3.1.0 Validation Study.....	33
3.1.1 Dimensionless First Cell Height (Y+)	33
3.1.2 Turbulence Model	38
3.2.0 SBD Dallara F399/01 CFD Investigation	44
3.3.0 Baseline Isolated Wing Parameter Sweeps.....	50
3.3.1 Secondary Elements AoA	50
3.3.2 Whole Assembly AoA	55
3.3.3 Proximity to Ground	59
3.3.4 Slot Gap Height.....	65
3.3.5 Secondary Elements Positive Overhang.....	69
3.3.6 Secondary Elements Negative Overhang	73
3.3.7 Gurney Flap	76
3.4.0 Parameter Sweep Conclusion	79

3.5.0 Front Wheel Study.....	80
3.6.0 Design Development	83
3.6.1 Main Plane Standardised Aerofoils	83
3.6.2 Main Plane Design Process.....	90
3.6.3 Secondary Elements Design Process.....	95
3.6.4 Design Iteration and Baseline Analysis.....	101
3.7.0 Concept Development	105
4.0.0 Concluding Remarks.....	118
5.0.0 Recommended Further Work.....	119
5.1.0 Design Concept 23.....	120
6.0.0 References.....	127
7.0.0 Appendix A.....	136
7.1.0 C2 Wind Tunnel.....	136
7.2.0 Horiba MIRA Reference Car.....	142
7.3.0 Fackrells Wheel.....	145
8.0.0 Appendix B.....	147
8.1.0 Breakdown of Study Results	147
9.0.0 Appendix C	150
9.1.0 Concept Design Geometries	150

List of Figures

Figure 1 - 2018 FIA Formula 3 Car (Coch, 2018).....	13
Figure 2 - Front Wing Design influence on C_{pt} Distribution (Nakagawa & Hojo, 2014)	13
Figure 3 - The Magnus Effect (Kundu, et al., 2016)	14
Figure 4 - SBD Ltd, Dallara F399/01.....	15
Figure 5 - CoP Relative to CoG (Quintarelli, 2018)	16
Figure 6 - SBD Ltd Dallara F399/01 Corner Weights (Kg)	16
Figure 7 – Vauxhall 2.0L XE Power Curve (SBD Ltd, 2015)	18
Figure 8 - MSUK Drawing Q-19.17 (Motorsport UK, 2020)	19
Figure 9 - MSUK Dimensions Table S-15.1.1 (Motorsport UK, 2020).....	19
Figure 10 - 2D Aerofoils in Ground Effect (Jung, et al., 2012).....	20
Figure 11 - WIG Characteristic Velocity Profile (Ranzenbach & Barlow, 1996)	21
Figure 12 - Typical WIG Results (Zerihan & Zhang, 2000)	21
Figure 13 - Unmerged Boundary Layers (Left), Merged Boundary Layers (Right) (Ranzenbach & Barlow, 1996).....	22
Figure 14 - Typical Configuration of a Multi - Element Wing (Susín, 2019).....	23
Figure 15 - Trailing Edge Vortices (Jasinski & Selig, 1998)	24
Figure 16 - Typical Tyre Construction (SPROT, 2013)	25
Figure 17 - Non-Deformable Tyre Profiles (Knowles, 2005).....	25
Figure 18 - Wheels Off Wind Tunnel Test (Edmondson, 2015).....	26
Figure 19 - Surface Pressure Distributions for Stationary and Rotating Wheels (Fackrell, 1972) (Knowles, 2005)	27
Figure 20 - Expected Flow Structures around a Rotating Wheel (Fackrell & Harvey, 1975).....	27
Figure 21 - Counter Rotating Vortical Structures (Issakhanian, et al., 2009).....	28
Figure 22 - The 6 Main Vortical Structures in the Near-Wake Flow Regime of a Rotating Wheel (Leśniewicz, et al., 2014)	28
Figure 23 - 2.5m Downstream Total Pressure (Non-Dimensional) (James, et al., 2014)	29
Figure 24 - Vortex Interaction Types (Diasinos, et al., 2014)	30
Figure 25 - Wing Geometry Impact of Vortex Formation (Nakagawa & Hojo, 2014).....	31
Figure 26 - Estimated 2019 Alfa Romeo C38 Formula 1 Car Front Wing Flow Regime (Susín, 2019).....	31
Figure 27 - Vortex Structures and Outwash used to Deflect Wheel Wake (TotalSim, 2016).....	32
Figure 28 - Change in Front Wheel Stagnation Pressures, With and Without Endplates (Petkar, et al., 2014).....	32
Figure 29 - Boundary Layer Construction (Mishra & Aharwal, 2018)	33
Figure 30 - Sub-Divisions of the Near-Wall Region (KRÁLIK, 2016).....	34
Figure 31 - Ahmed Body Velocity Profile at Distance X (Sijal, 2019)	35
Figure 32 - (Left) Logarithmic Based Wall Functions Resolution, (Right) Viscous Sublayer Based Wall Functions Resolution	37
Figure 33 - Total C_p of Wheel Wake (Top Left – SKE) (Top Right – RKE) (Bottom – SST) (Salient Edge, 2018)	42

Figure 34 - Wall Function and Near-Wall Function Approaches (Ansys Inc, 2009).....	42
Figure 35 - SBD Dallara F399 Static Pressure.....	44
Figure 36 - Main Plane Profile.....	44
Figure 37 - Front Wheel Stagnation Zone	45
Figure 38 - Oil Flow Front Wing End Plate (Static Pressure)	45
Figure 39 - 2015 Indycar Front Wing End Plate (Somerfield, 2015).....	46
Figure 40 - Front Wheel Wake	46
Figure 41 - Front Wheel Face Oil Flow in term of Static Pressure (Left - Front, Right - Rear)	46
Figure 42 - Front Wheel Face Oil Flow in terms of Static Pressure (Left - Top, Right - Bottom)	47
Figure 43 - Front Wheel Near-Wake Vortex Interaction	47
Figure 44 - SBD F399 Vortex Structures.....	48
Figure 45 - SBD F399 Outwash (Velocity)	49
Figure 46 - Front Wing Flow Field (Top – Static Pressure, Bottom - Velocity)	49
Figure 47 - Baseline Front Wing CAD Assembly	50
Figure 48 - F399 Front Wing Flap Adjustment	50
Figure 49 - Cl and Cd Values Taken from Baseline AoA	51
Figure 50 – Surface Static Pressure Contours (Left – $\alpha = 15.42^\circ$, Right – $\alpha = 20.42^\circ$)	51
Figure 51 - Minimum and Maximum AoA Pressure Distribution.....	52
Figure 52 - Pressure (Left – $\alpha = 15.42^\circ$, Right – $\alpha = 20.42^\circ$).....	52
Figure 53 - End Plate Surface Static Pressures (Left – $\alpha = 15.42^\circ$, Right – $\alpha = 20.42^\circ$)	53
Figure 54 – Static Pressure and Size Comparison of the Primary and Y250 Vortices	53
Figure 55 - Vortex Velocity Profile (Katopodes, 2019)	54
Figure 56 - Real Vortex Velocity Profile (Agrawal & Prasad, 2003)	54
Figure 57 - Cl and Cd (Whole Assembly AoA)	55
Figure 58 - Cl/Cd (Whole Assembly AoA)	56
Figure 59 - Maximum and Minimum AoA Pressure Distribution (Whole Assembly).....	56
Figure 60 - Pressure (Left - $\alpha = 0^\circ$, Right – $\alpha = 10^\circ$)	57
Figure 61 - Whole Wing Assembly Vortex Structure (Top - $\alpha = 0^\circ$, Bottom – $\alpha = 10^\circ$).....	57
Figure 62 - Static Pressures located within the Vortical Structures (Left – $\alpha = 0^\circ$, Right – $\alpha = 10^\circ$) ...	58
Figure 63 - Wing Ground Proximity.....	59
Figure 64 - Cl Percentage Increase Relative to Ground Proximity.....	59
Figure 65 - Static Pressure (Left - H/c 0.06, Right - H/c 0.64).....	60
Figure 66 - Efficiency Relative to Ground Proximity	60
Figure 67 - Cl and Cd as a Function of Ground Proximity.....	61
Figure 68 - Pressure Distribution Relative to Ground Proximity	61
Figure 69 – Peak Negative Pressures for Each Element at H/c = 0.64 and 0.06.....	62
Figure 70 - Surface Pressure Contours (Left - H/c 0.06, Right - H/c 0.64).....	62
Figure 71 – Vortex Pressure (Left – H/c 0.06, Right – H/c 0.64).....	63
Figure 72 - Velocity Vector Plane (0.7m Rear of the Axis Origin)(Top – H/c 0.06, Bottom – H/c 0.64).....	63
Figure 73 - Secondary Element Elevation.....	65

Figure 74 - Slot Gap Height C_l and C_d	65
Figure 75 - Pressure Contours (Top Left - $H/c = 0$, Top Right - $H/c = 0.08$, Bottom - $H/c = 0.15$).....	66
Figure 76 - Slot Gap Height Efficiency	66
Figure 77 - Slot Gap Height Pressure Distribution	67
Figure 78 - Slot Gap Height Vortex Formation	68
Figure 79 - Positive Overhang Positioning (Left - $S/c = 0$, Right - $S/c = 0.123$).....	69
Figure 80 - Positive Overhang C_l and C_d	69
Figure 81 - Positive Overhang Forces	70
Figure 82 - Efficiency with Respect to S/c	70
Figure 83 - $S/c = 0.123$ Pressure Distribution	71
Figure 84 - Velocity Contours (Top Left - $S/c = 0$, Top Right - $S/c = 0.07$, Bottom - $S/c = 0.123$)	71
Figure 85 - Negative Overhang C_l and C_d	73
Figure 86 - Negative Overhang Forces	73
Figure 87 - Negative Overhang Efficiency.....	74
Figure 88 - $S/c = 0$ Pressure Distribution	74
Figure 89 - $S/c = -0.099$ Pressure Distribution	75
Figure 90 - Pressure Contours (Left - $S/c = 0$, Right - $S/c = -0.099$).....	75
Figure 91 - Baseline Front Wing with Gurney Flap	76
Figure 92 - Vectors of Total Pressure (Left - Gurney Flap, Right - No Gurney Flap).....	76
Figure 93 - Suction Surface Pressure Contours (Right - No Gurney Flap, Left - Gurney Flap)	77
Figure 94 - Total Pressure Contours (Left - Gurney Flap, Right - No Gurney Flap)	77
Figure 95 - No Gurney Flap Pressure Distribution (Baseline)	77
Figure 96 - Pressure Contours (Left - No Gurney Flap, Right - Gurney Flap).....	78
Figure 97 - Vortex Swirling Strength (Top - No Gurney Flap, Bottom - Gurney Flap) (0.01Hz).....	78
Figure 98 - Front Left Wheel Pressure Distribution	80
Figure 99 - Wheel Wake Total Pressure (Top Left - 0° , Top Right - 3° , Bottom - 5°)	80
Figure 100 - Front Wheel Vortex Structures (Left - 0° , Middle - 3° , Right - 5°).....	81
Figure 101 - Velocity Vectors (Top Left - 0° , Top Right - 3° , Bottom - 5°)	82
Figure 102 - Total Pressure Wake Structure (Top Left - 0° , Top Right - 3° , Bottom - 5°).....	82
Figure 103 - Main Plane Geometries (Top Left - Dallara Baseline, Top Right - Naca 4412, Bottom Left - S1223, Bottom Right - Naca 6409)	83
Figure 104 - Pressure Contours (Top Left - Dallara, Top Right - Naca 4412, Bottom Left - S1223, Bottom Right - Naca 6409) ($H/c = 0.39$).....	84
Figure 105 - Pressure Distribution (Top Left - Dallara Baseline, Top Right - Naca 4412, Bottom Left - S1223, Bottom Right - Naca 6409) ($H/c = 0.39$).....	85
Figure 106 - Pressure Contours (Top Left - Dallara Baseline, Top Right - Naca 4412, Bottom Left - S1223, Bottom Right - Naca 6409) ($H/c = 0.14$).....	87
Figure 107 - Suction Surface Static Pressure (Left - S1223, Right - Naca 4412) ($H/c = 0.14$).....	87
Figure 108 - Pressure Distribution (Top Left - Dallara Baseline, Top Right - Naca 4412, Bottom Left - S1223, Bottom Right - Naca 6409) ($H/c = 0.14$).....	88

Figure 109 – Peak Positive Pressure.....	88
Figure 110 – Pressure Differential Increase.....	89
Figure 111 - Front Wing Main Plane Designs	90
Figure 112 - Front Wing Main Plane Front View	90
Figure 113 - 2019 Mercedes W10 Front Wing (Anderson, 2020).....	91
Figure 114 - Main Plane Top View.....	91
Figure 115 - Main Plane Design Iterations - Suction Side Static Pressure.....	92
Figure 116 - Velocity Contours of Front Wheel Wake	92
Figure 117 - Total Pressure Contours of Front Wheel Wake	93
Figure 118 - S1223 Profile Modification	93
Figure 119 - Main Plane Design Iterations (Cl and Cd).....	94
Figure 120 - Main Plane Design Iterations (CoP Location)	94
Figure 121 - Front Wing Secondary Element Designs	95
Figure 122 - Velocity Contours (0.35m from Axis Origin).....	95
Figure 123 - Static Surface Pressures (Suction Side).....	96
Figure 124 - Static Surface Pressure (Upper Surface).....	97
Figure 125 - Vortex Swirling Strength (Up to 252.5 Hz).....	97
Figure 126 - Total Pressure Contours of the Wheel Wake.....	98
Figure 127 – Wheel Wake Total Pressure Contours (0.5m Downstream of Axis Origin)	98
Figure 128 - Secondary Element Design Velocity Pathlines	99
Figure 129 - Secondary Elements Design Iterations (Cl and Cd).....	99
Figure 130 - Secondary Elements Design Iterations (CoP Location)	100
Figure 131 - Secondary Elements Design Iteration (Efficiency)	100
Figure 132 - Static Surface Pressure Contours (Top - Baseline, Bottom - Design 7).....	101
Figure 133 – Front Wing Velocity Contours on a Plane 0.45m Lateral of the Centreline (Left - Baseline, Right - Design 7)	101
Figure 134 - Whole Car Velocity Contours on a Plane 0.45m Lateral of the Centreline (Top – Baseline, Bottom – Design 7).....	102
Figure 135 - Vortex Swirling Strength (1Hz) (Top - Baseline, Bottom - Design 7).....	103
Figure 136 - Wake Velocity Contours 0.1m from the Ground Plane (Top - Baseline, Bottom - Design 7).....	103
Figure 137 - Turbulent Kinetic Energy (60J/Kg) (Top – Baseline, Bottom – Design 7)	104
Figure 138 - Wake Velocity Contours 0.2807m from the Ground Plane (Top - Baseline, Bottom - Design 7).....	104
Figure 139 - Concepts 1, 2 and 3.....	105
Figure 140 - Concept Lift and Drag Coefficients	106
Figure 141 - Concept Aerodynamic Efficiency	106
Figure 142 - Concept CoP Location.....	107
Figure 143 - Concept Suction Surface Static Pressure Contours	107
Figure 144 - Floor Static Pressure Contours.....	108

Figure 145 - Concept Floor Pressure Distributions	108
Figure 146 - Concept Venturi Tunnel Pressure Distributions	109
Figure 147 - Concept 3 Turbulent Kinetic Energy Contours (60J/Kg)	109
Figure 148 - Left - Total Pressure (Pa), Right - Dynamic Pressure (Pa)	110
Figure 149 - Concept Front Wing Velocity Contours.....	111
Figure 150 - Concept Front Wing Suction Surface Pressure Contours	111
Figure 151 - Concept Front Wing Pressure Distributions.....	112
Figure 152 - Concept Front Wing Centre Pressure Distributions	112
Figure 153 - Vortex Core Swirling Strength (1Hz)	113
Figure 154 - Velocity Contours of Vortex Formation	113
Figure 155 – Concept Front Wing Wheel Wake Velocity Contours.....	114
Figure 156 - Concept Front Wing Vortex Velocity Vectors.....	114
Figure 157 - Concept Front Wing Turbulent Kinetic Energy Isosurface (60J/Kg).....	115
Figure 158 - Concept Front Wing Front Wheel Pressure Distributions	115
Figure 159 - Concept Front Wing Whole Car Velocity Contours	116
Figure 160 - Front Wing Concept Rear Wheel Wake Turbulent Kinetic Energy (60J/Kg)	116
Figure 161 - Concept Front Wing Whole Car Isosurface (30J/Kg)	117
Figure 162 - Floor Sealing Vortex (1Hz)	117
Figure 163 - Concept 23	120
Figure 164 - Static Surface Pressures (Top - Concept 23, Bottom - Baseline)	121
Figure 165 - Front Wing Pressure Distributions	121
Figure 166 - Vortex Core Swirling Strength (1Hz) (Top - Concept 23, Bottom - Baseline).....	122
Figure 167 - Turbulent Kinetic Energy (60J/Kg) (Left – Concept 23, Right – Baseline)	122
Figure 168 - Venturi Throat Velocity Contours (Left - Concept 23, Right - Baseline)	122
Figure 169 - Velocity Contours (Top - Concept 23, Bottom - Baseline)	123
Figure 170 - Rear Wing Pressure Distributions.....	123
Figure 171 - Vehicle Wake Velocity Contours (Top - Concept 23, Bottom - Baseline).....	124
Figure 172 - Vehicle Wake Turbulent Kinetic Energy (60J/Kg) (Top - Concept 23, Bottom - Baseline)	124
Figure 173 - Wheel Stagnation Pressure Contours (Top – Concept 23, Bottom – Baseline).....	125
Figure 174 - Turbulent Kinetic Energy Isosurface (60J/Kg) (Top - Concept 23, Bottom - Baseline) .	126
Figure 175 - Front Wheel Wake Side Pod Interaction Isosurface (60J/Kg) (Top - Concept 23, Bottom - Baseline)	126
Figure 176 - C2 Open Loop Wind Tunnel (1/12 Scale)	136
Figure 177 – Naca 0012 Near-Wall Mesh Resolution	137
Figure 178 - Naca 0012 ($\alpha = 0$).....	137
Figure 179 - Naca 0012 ($\alpha = 5$).....	138
Figure 180 - Naca 0012 ($\alpha = 10$).....	138
Figure 181 - Naca 0012 ($\alpha = 15$).....	139
Figure 182 - Naca 0012 Wake Flow Circulation.....	139

Figure 183 - Naca 0012 Performance at $Re = 6E+6$	139
Figure 184 - Post Processing Wake Flow Regime	140
Figure 185 - Areas of Localised Pressure on the Wind Tunnel Sides	140
Figure 186 - MIRA Notchback Geometry	142
Figure 187 - MIRA Notchback C_p Distribution	142
Figure 188 - MIRA Reference Car C_p Distribution (Wang, et al., 2014).....	143
Figure 189 - MIRA Notchback Wake Velocity (Left - PIV) (Right - CFD) (Wang, et al., 2014)	143
Figure 190 - MIRA Notchback Wake Velocity	144
Figure 191 - MIRA Notchback Wake Velocity (Palaskar, 2016)	144
Figure 192 - MIRA Notchback Wake Structure Path Lines	144
Figure 193 - MIRA Notchback Wake Structure Streamlines (Left - Wang et al) (Right - Palaskar) ...	144
Figure 194 - Fackrell Wheel Pressure Distribution Measured on the Centreline.....	145
Figure 195 - Near-Wake Flow Recirculation	146
Figure 196 - Total Pressure Normal to Freestream.....	146
Figure 197 - Wheel Wake Total Pressure Perpendicular to the Freestream	146
Figure 198 - Concept 1	150
Figure 199 - Concept 2	150
Figure 200 - Concept 3	150
Figure 201 - Concept 4	151
Figure 202 - Concept 5	151
Figure 203 - Concept 6	151
Figure 204 - Concept 7	152
Figure 205 - Concept 8	152
Figure 206 - Concept 9	152
Figure 207 - Concept 10	153
Table 1 Comparison of Baseline Front Wing with and without a Gurney Flap	78
Table 2 - Main Plane Profile Study ($H/c = 0.39$)	86
Table 3 - Main Plane Profile Study ($H/c = 0.14$)	89
Table 4 - Main Plane Profile Study ($\Delta H/c$)	89
Table 5 - Average Velocity and Pressure at Venturi Entrance	102
Table 6 - Average Velocity and Pressure at Venturi Throat	102
Table 7 - Concept Design Downforce Balance	107
Table 8 - Concept Venturi Tunnel Entry Average Velocity and Pressure	109
Table 9 - Concept Venturi Tunnel Throat Average Velocity and Pressure	110
Table 10 - Baseline and Concept 23 Comparison.....	120
Table 11 - Rear Wheel Forces	125
Table 12 - Front Wheel Forces	125
Table 13 - C_l and C_d for Fackrells Wheel	145
Table 14 - Baseline Wing AoA	147

Table 15 - Baseline Wing Assembly AoA.....	147
Table 16 - Baseline Wing Ground Proximity	147
Table 17 - Baseline Wing Secondary Elements Height.....	147
Table 18 - Baseline Wing Secondary Elements Forward Position	148
Table 19 - Baseline Wing Secondary Elements Aft Position	148
Table 20 - Main Plane Design Study (110mm)	148
Table 21 - Main Plane Design Study (40mm)	148
Table 22 - SBD Dallara F399 Baseline Study	148
Table 23 - SBD Dallara F399 Main Plane Design Process	148
Table 24 - SBD Dallara F399 Secondary Elements Design Process	148
Table 25 - SBD Dallara F399 End Plate Design Process.....	148
Table 26 - SBD Dallara F399 Front Wing Concept Design Process	149
Equation 1 - Mass Moment (Kg.m)	16
Equation 2 – Front and Rear Mass Moments (Kg.m)	17
Equation 3 - Centre of Gravity Location (m).....	17
Equation 4 - Reynolds Number	34
Equation 5 - Boundary Layer Thickness (m).....	35
Equation 6 - Coefficient of Friction	36
Equation 7 - Wall Shear Stress (N/m ²)	36
Equation 8 - Friction Velocity (m/s)	36
Equation 9 - Dimensionless First Cell Height	36
Equation 10 - Navier Stokes Equation	38
Equation 11 - Prandtl Mixing Length	39
Equation 12 - Van Driest Hypothesis	39
Equation 13 - Mixing Length Relation to Transport Equations	40
Equation 14 - Turbulent Kinetic Energy	40
Equation 15 - Turbulence Dissipation Rate.....	40
Equation 16 - Eddy Viscosity	41
Equation 17 - Turbulent Reynolds Number	41
Equation 18 - Force (N).....	67

Abbreviations

- Cl – Coefficient of Lift
- Cd – Coefficient of Drag
- Cp – Coefficient of Pressure
- CoP – Centre of Pressure
- CoG – Centre of Gravity
- Cpt – Total Pressure Coefficient
- WIG – Wing in Ground Effect
- GF – Gurney Flap
- CAD – Computer Aided Design
- CFD – Computational Fluid Dynamics
- Re – Reynolds Number
- RANS – Reynolds Average Navier Stokes
- U – Velocity Component
- V – Velocity Component
- H/c – Height/Chord Ratio
- S/c – Distance/Chord Ratio
- c – Chord Length
- $k\varepsilon$ – K Epsilon
- $k\omega$ – K Omega
- δ – Boundary Layer Thickness
- Cf – Coefficient of Friction
- τ_w – Wall Shear Stress
- U_τ – Friction Velocity
- l_m – Prandtl and Van Driest Mixing Length
- μ_τ – Eddy Viscosity

1.0.0 Introduction

Since the introduction of the open wheel race car in the early twentieth century, the ingenuity of the engineers behind these cars has been relied upon to produce devices that will counteract the adverse aerodynamic effects of the open wheel arrangement. It has now become commonplace to include a vast array of aerodynamic devices. These devices can range from large components that generate considerable downforce, to incredibly intricate and delicate components for the use of flow management and drag reduction, with the sole purpose of gaining vehicle performance (Soares, et al., 2018).

Generally, when the terms “aerodynamics” and “motorsport” are referred to in the same sentence, the first thought is for the prestige Formula One championship. It was within this class where the inclusion of wings primarily began. By utilising an inverted aerofoil, the generated downward force produced would further increase the lateral acceleration capabilities, thus faster cornering speeds were achieved (Heyder-Bruckner, 2011). A secondary benefit would come in the form of an opposing force acting against the naturally generated lift due to such streamlined bodies for the intention of drag reduction.

With the lower formulae following the trend, it would be unusual to observe a Formula 2 or Formula 3 car which did not include front and rear wings.



Figure 1 - 2018 FIA Formula 3 Car (Coch, 2018)

Figure 1, illustrates that the front wing is a large construction often assembled from smaller components. To the untrained eye, the purpose of the front wing would often be assumed to be only for the purpose of generating downforce. However, as it is the first major component to interact with the oncoming free stream air, its influence on the resulting flow regime aft of the car could be considered a key area of interest (Nakagawa & Hojo, 2014). The reason for this is that any alteration made to the front wing will inevitably alter the aerodynamic performance at the rear of the car (*Figure 2*).

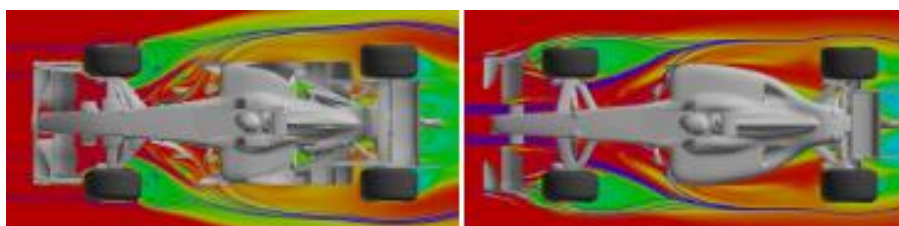


Figure 2 - Front Wing Design influence on C_{pt} Distribution (Nakagawa & Hojo, 2014)

1.1.0 Aims and Objectives

As it has been demonstrated that the front wing can have a dramatic effect on the downstream flow regime, therefore the subsequent question must be whether it can also be used as a flow management device for reducing or controlling the near-wake generated by the front wheels (Diasinos, et al., 2014). The wheels on an open wheel race car are considered to be the cause of the majority of the drag force that is produced, totalling approximately 40% of the vehicles overall drag, however it must also be stated that the front wing contributes 35% of the vehicles overall downforce (Knowles, 2005).

As the wheel rotates with an increasing angular velocity, the natural phenomenon of the Magnus effect (*Figure 3*) begins to generate lift (Wang & Fan, 2013). However, as a result of the increase in rotational velocity, an increase in skin friction is generated due to the air and wheel working in opposite directions, thus generating early onset boundary layer separation as energy is depleted. This often results in large wake formations being generated.

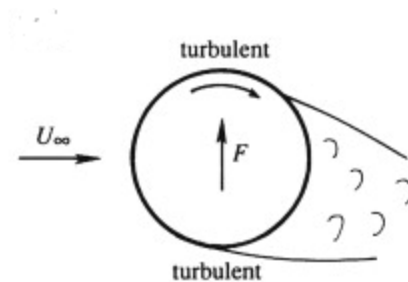


Figure 3 - The Magnus Effect (Kundu, et al., 2016)

The eddy formations within the wake structure can subsequently interfere with the downstream components, which results in a loss of component working efficiency (Leśniewicz, et al., 2014).

Therefore, the purpose of this investigation is to analyse a front wing on an open wheel race vehicle and study its interaction with the front wheels using validated computational methods.

A full parameter sweep of the front wing will be conducted in order to fully understand the aerodynamic characteristics of the current wing. A further design study will be undertaken to improve the working characteristics of the wing and its interaction with the front wheels and surrounding areas to increase the aerodynamic efficiency of the vehicle.

2.0.0 Literature Review

At the commencement of the investigation, valid research is required to be undertaken in order to understand the fundamentals of how wings function. A Dallara F399/01, Formula 3, open wheel race car will be the subject of this study, the wing being located at the extreme front of the vehicle. This will, therefore, provide information regarding the interaction between the current front wing and front wheels (plus surrounding areas).

2.1.0 SBD Ltd Dallara F399/01

Figure 4, illustrates the referenced vehicle, supplied by SBD Ltd, which is being studied for the purposes of this investigation (Davies, 2019).

It can already be seen that the Formula 3 car has been heavily modified, including the rear wing, side pods, floor, barge boards, air intake and wheels. Minor changes have been made to the front wing, such as the inclusion of end plates and flap elements. However, within the driver feedback there is a suggestion of poor front handling characteristics within fast corners with high lateral loading.



Figure 4 - SBD Ltd, Dallara F399/01

This may be as a result of the addition of an increase in ground effect due to the side pod and floor modifications which have altered the aerodynamic centre of pressure location (CoP - the point about at which resultant aerodynamic forces work), thus changing the handling characteristics of the vehicle and providing a bias of vertical force towards the rear of the car (Cai, et al., 2017) (Li, et al., 2000) (Castejon, et al., 2011) (relative to the centre of gravity location) (*Figure 5*).

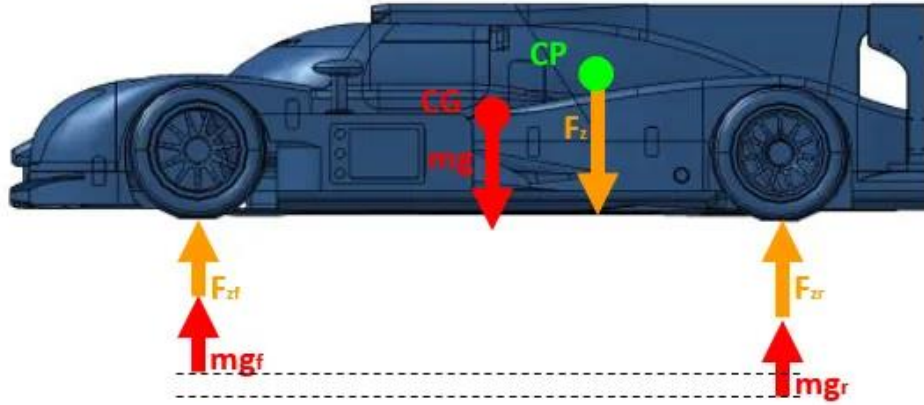


Figure 5 - CoP Relative to CoG (Quintarelli, 2018)

It must also be acknowledged, that even if the bias of downforce is forward of the centre of gravity, providing a larger torque (Y axis turning moment), this effect may not outweigh the vertical force provided by the rear downforce and mass (Håkansson & Dube, 2014). This statement assumes, however, that the centre of gravity is located towards the rear of the car.



Figure 6 - SBD Ltd Dallara F399/01 Corner Weights (Kg)

Figure 6, illustrates the corner masses of the vehicle with the total mass indicated. In order to find the location of the centre of gravity, a simple method of calculating the mass moments about a location datum must be executed.

$$Moment = M * S$$

Equation 1 - Mass Moment (Kg.m)

Using Equation 1, the mass moments of both the total front mass and total rear mass can be established, where M is the mass in Kg and S is the distance from the datum point in meters. As the masses displayed in Figure 6 are being transmitted through the tyre contact patch, the distance used is the wheelbase of the car, with the datum point being taken from the centre point of the front axle, therefore indicating a distance of 0m.

$$Moment_{front} = 171 * 0 = 0Kg.m$$

$$Moment_{rear} = 291.5 * 2.675 = 779.76Kg.m$$

Equation 2 – Front and Rear Mass Moments (Kg.m)

Using the results produced in *Equation 2*, the location of the centre of gravity along the X axis can be determined relative to the datum point when applied to *Equation 3*.

$$CoG = \frac{Mass\ Moment_{Total}}{Mass_{Total}}$$

Equation 3 - Centre of Gravity Location (m)

This suggests a CoG location 1.69m behind the front axle, providing a 63% bias of the vehicles total mass is located towards the rear of the car. To provide a neutrally loaded vehicle, the total front mass and downforce must be equal to that of the total rear mass and downforce. This would suggest a required CoP location to be 79.29% forward biased to provide a Y axis moment to counter that of the total rear loading. However, during a dynamic case, the loading distribution will change from that observed in *Figure 6*, for example, during a braking or accelerating manoeuvre, which can have a dramatic impact on the aerodynamic behaviour (equivalent mass is also not considered) (Quintarelli, 2018), therefore it would be suggested that the CoP to be located to provide a 60% forward bias (within a 5% window of movement) (Hillam, 2020).

A spokesperson at TotalSim Ltd has suggested that a downforce distribution with a 55% rear bias would be appropriate (Total Simulation, 2016), however, this is in relation to a Formula 1 car, where the mass loadings could be significantly different to those of a Formula 3 car.

It must be acknowledged that different circuits require different car handling characteristics, which are also driver dependant. As two drivers may use the vehicle in question, it is suggested that some adjustment allowance could be provided in order to move the CoP location if required, thus altering the aerodynamic handling of the vehicle per driver preference.

2.2.0 The British Sprint Championship: Technical Regulations

The British Sprint Championship is considered to be one of the most prestigious motorsport series in Britain (British Sprint, 2020). Unlike regular competitive circuit racing, only a single car is permitted onto the circuit at any time. The goal of this competition is to achieve the fastest lap time. Many types of vehicle compete within the championship, and these are segregated into the appropriate classes.

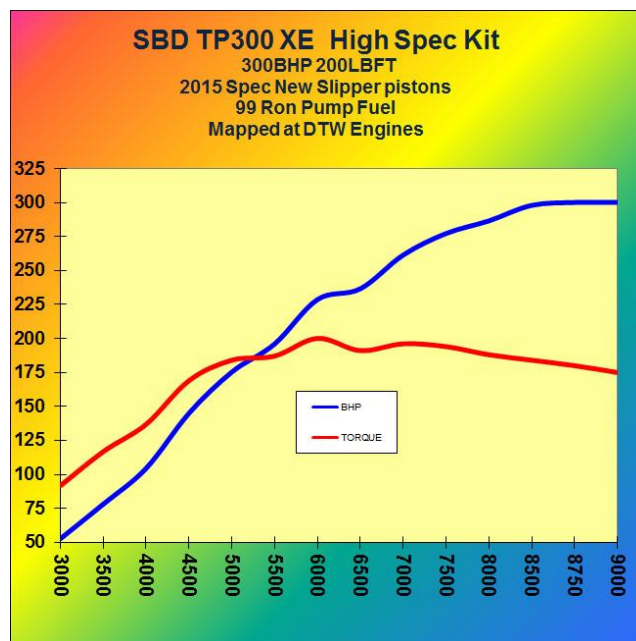


Figure 7 – Vauxhall 2.0L XE Power Curve (SBD Ltd, 2015)

The SBD Ltd Dallara F399/01 falls within the racing car class from 1600cc to 2000cc as it employs a 2.0l Vauxhall XE internal combustion engine, that produces 300BHP (Figure 7).

In British governed motorsport, a series of strict regulations must be adhered to in order to maintain the safety of the driver and to allow fair racing. The British Sprint Championship officials direct that all the relevant Motorsport UK technical regulations must be followed (SBD Motorsport Uk HSA British Sprint Championship, 2020).

As the concerned area of study with regard to the vehicle is the front wing and its immediate surroundings, only the regulations pertaining to these areas will be referenced.

Article J-5.2.7 of the MSUK Blue Book outlines that no aerodynamic device may exceed the maximum width of the vehicle or maximum height of the roll over protection system unless stated otherwise (Motorsport UK, 2020).

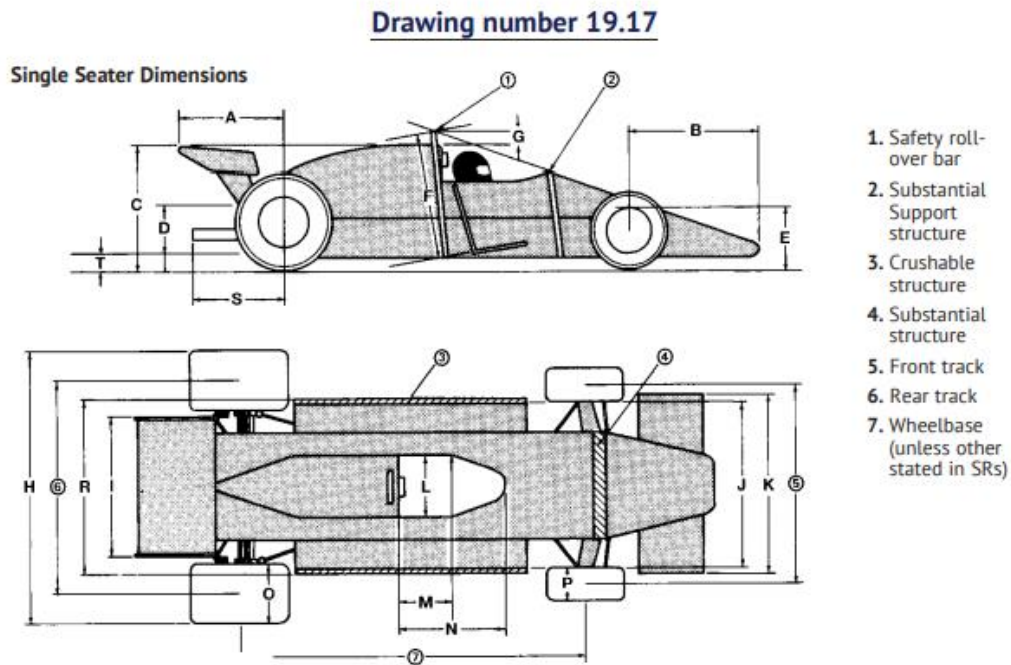


Figure 8 - MSUK Drawing Q-19.17 (Motorsport UK, 2020)

Figure 8, illustrates the maximum regulatory bodywork/aerodynamic device dimensions permitted. The maximum distance forward of the front wheel axle is unregulated, however, the width ahead of the front wheels is not allowed to exceed 1.5m and the maximum width allowed behind the front wheels is 1.4m (Figure 9).

Dimensions

15.1.1.

Maximum width ahead of front wheels	150cm
Maximum width behind front wheels	140cm
Maximum height of any part wider than 110 cm ahead of the front wheels	Not to exceed the top of the front wheel rim
Maximum height	90cm from the ground
Maximum rear overhang	150cm behind rear wheel axis.

Figure 9 - MSUK Dimensions Table S-15.1.1 (Motorsport UK, 2020)

The minimum ground clearance allowed is 40mm, therefore no part of the front wing assembly must be positioned lower than this (Motorsport UK, 2020).

2.3.0 Wings in Ground Effect

Wings have been used in motorsport since the early 1960s. The reason for their use is to increase the lateral acceleration, thus achieving faster cornering speeds and ultimately decreasing lap times as the vertical force increases.

The most common mechanical purpose for wings is their utilization within the aerospace industry, allowing planes to fly and manoeuvre in the sky. In order to accomplish this, the wing has to generate a lifting force, which is achieved as a result of the geometry and profile of the wing.

The profile geometry alters the surrounding (dynamic and static) pressures as the wing travels through the air. For a wing in a lifting attitude, the pressure on the upper surface needs to be lower than that achieved on the underside. There are many theories into what causes this differential in pressure around the wing, however, none have been fully proven (Houghton, et al., 2017). What is known is that the geometry of the wing alters the flow regime around its profile, thus changing the air pressure, causing a lifting force (Heisler, 2002).

As a consequence, the change in pressure results in a disturbance of the air. This can be in the form of pressure drag, induced drag and friction drag which is represented as an opposing force to the trajectory of the wing (Ramalingaih, et al., 2013).

A wing in ground (WIG) effect is a phenomenon that occurs when the proximity of the wing to the ground decreases. This produces a change in the flow regime compared to that of the onset freestream flow.

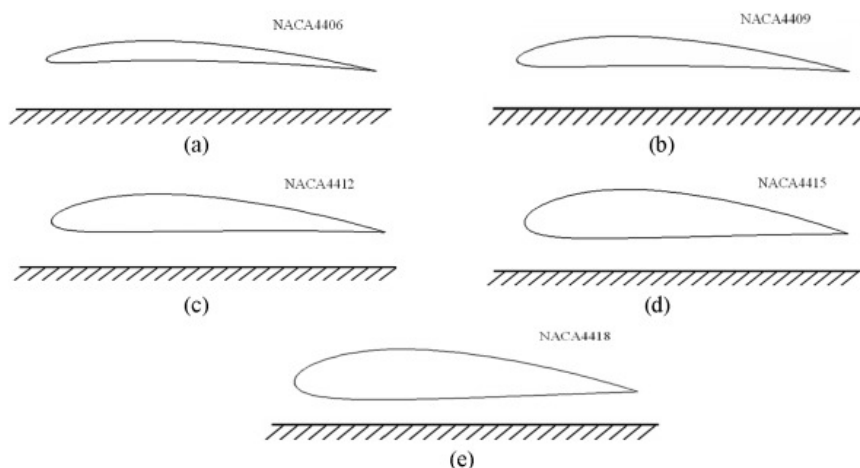


Figure 10 - 2D Aerofoils in Ground Effect (Jung, et al., 2012)

For an aviation application, when the wing is at a certain distance from the ground, a cushion of high pressure air develops on its underside, which increases its lifting force, while reducing the downwash velocity, resulting in a decrease of induced drag (Jung, et al., 2012). An example of this can be observed on the Ekranoplan, which was a ground effect vehicle developed by the Russian navy, very similar in appearance to an aircraft, but was designed to be flown very close to the surface of the sea, in order to remain undetected by radar.

WIG's are common place in motorsport and are most notably seen on single seat race cars, such as the Dallara F399/01. Such devices are often located at the front of the vehicle and can be considered to be an inverted wing, producing a downwards force. It was first proposed in 1921 by Zahm that the proximity of the ground plane had a severe effect on the suction surface of the wing (Zahm & Bear, 1921).

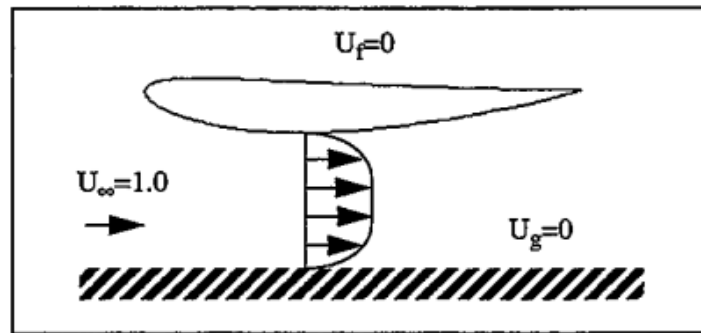


Figure 11 - WIG Characteristic Velocity Profile (Ranzenbach & Barlow, 1996)

As the wing approaches the ground, a restriction of air is apparent on its suction surface. Due to the conservation of mass, this results in an increase in air velocity due to the large expansion area (Figure 11), which causes an exponential pressure drop and dramatically increases downforce owing to the increased suction and pressure differential between top surface and bottom surface (Mokhtar, 2008).

Katz states that the front wing on a race car generally has a height over chord ratio (H/c) of 0.1 to 0.3 which generates a strong ground effect (Katz, 2006).

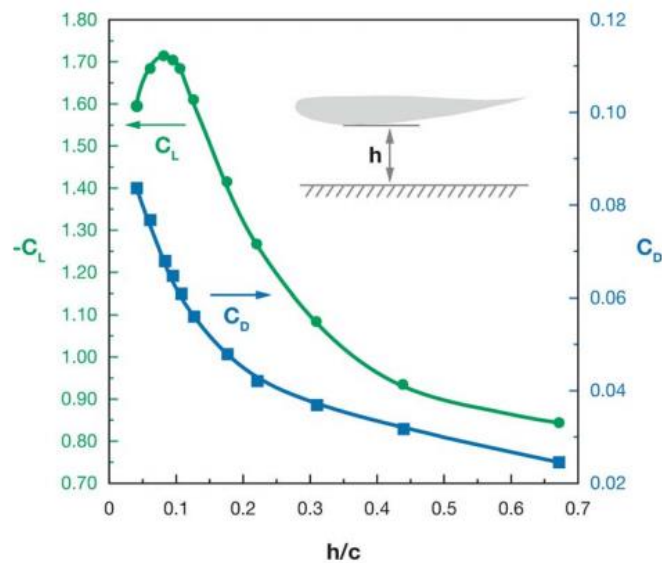


Figure 12 - Typical WIG Results (Zerihan & Zhang, 2000)

Figure 12, indicates typical results for C_L and C_D produced from experiments by Zerihan when introducing a wing into a ground effect application, an exponential increase in negative lift becomes apparent. At a certain height, a blockage effect is induced on the suction surface, which is

represented as a sudden drop in lift force, however the drag force still increases. This is as a consequence of the boundary layers of the wing and ground plane merging (*Figure 13*).

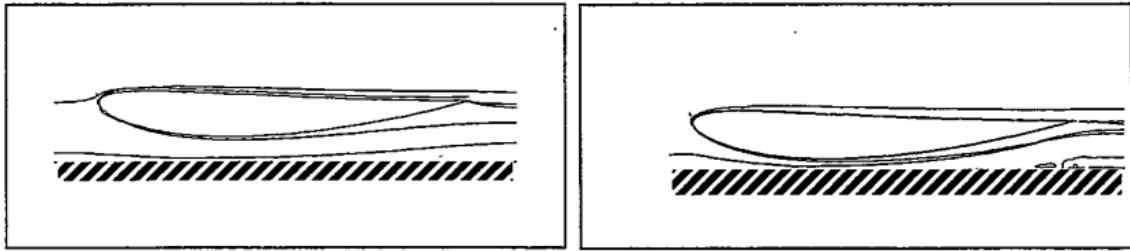


Figure 13 - Unmerged Boundary Layers (Left), Merged Boundary Layers (Right) (Ranzenbach & Barlow, 1996)

The blockage reduces the air flow to the suction surface, thereby negatively affecting the generated vertical force. Ranzenbach also discovered this characteristic when analysing a Naca 4412 profile in a ground effect scenario (Ranzenbach & Barlow, 1996).

2.3.1 Multi – Element Wings in Ground Effect

Multi-element wings are a typical feature in a motorsport application, being employed to add extra camber to the wing profile (*Figure 14*), producing a greater differential in pressure and increasing the generated vertical force. An added benefit is the re-energisation of the boundary layers. This feature delays the function of trailing edge separation which occurs as a result of the increase in camber angle or angle of attack (AoA), typically occurring at a 12° angle of incidence, and greater, which produces a larger adverse pressure gradient (Soso & Wilson, 2004). The boundary layer depletes additional energy through frictional losses when overcoming strong adverse pressure gradients, therefore increasing the occurrence of trailing edge separation.

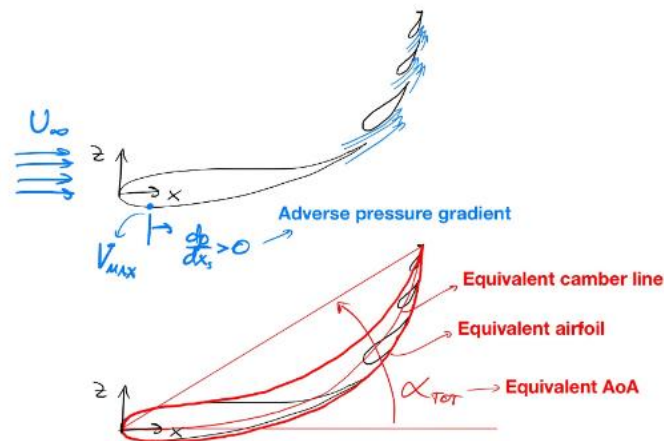


Figure 14 - Typical Configuration of a Multi - Element Wing (Susín, 2019)

Smith describes the 5 effects of the gaps between each element, which are, slat effect, circulation, dumping, off surface pressure recovery and the fresh boundary layer effect (Smith, 1975). These criteria are considered important for the correct employment of multi-element wings.

In a 2D study of a double element wing in ground effect, it was found that the main element produces the majority of the vertical force and also dominates the turbulent wake field, however, the ground proximity does not affect the wake field from the flap with any significance in terms of velocity deficit and thickness, although an increase in wake thickness and a reduction in velocity deficit are exhibited in the main element suction surface as the ground proximity reduces (Zhang & Zerihan, 2003).

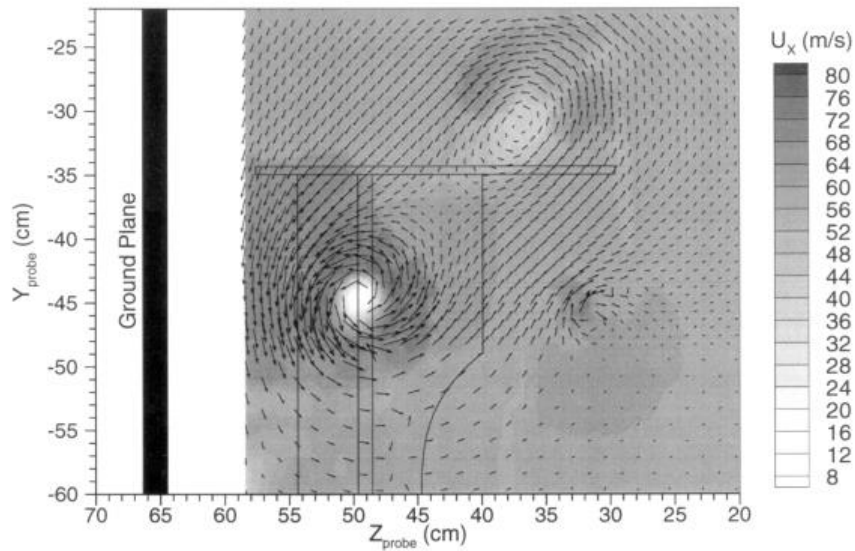


Figure 15 - Trailing Edge Vortices (Jasinski & Selig, 1998)

A 3D study conducted by Jasinski and Selig found that a multi – element wing in ground effect produced strong trailing edge vortices that were observed rolling up the end plate, increasing in size as the flap angle increased, thus increasing the induced drag force (Jasinski & Selig, 1998).

2.4.0 Aerodynamics of Wheels

Wheels are a fundamental part of a race car, as they are the primary point of contact between the ground plane and the vehicle, which when supplied with power, allow the vehicle to initiate movement.

Wheels can be considered a bluff body in terms of their robust and blunt shape; however, many complications arise when considering the aerodynamics of wheels. Firstly, for accurate measurements, tyre deformation must be considered.

It is well understood that when a wheel is loaded, its corresponding tyre will deflect, changing its contact patch and side wall profile, however, difficulties in accurately modelling this characteristic are apparent. Tyres are constructed using multiple layers (*Figure 16*) using multiple material properties, it is, therefore, difficult to obtain any analytical data for tyre deformation when introduced into a loaded scenario, even when using complex FEA techniques (Yang & Olatunbosun, 2011).

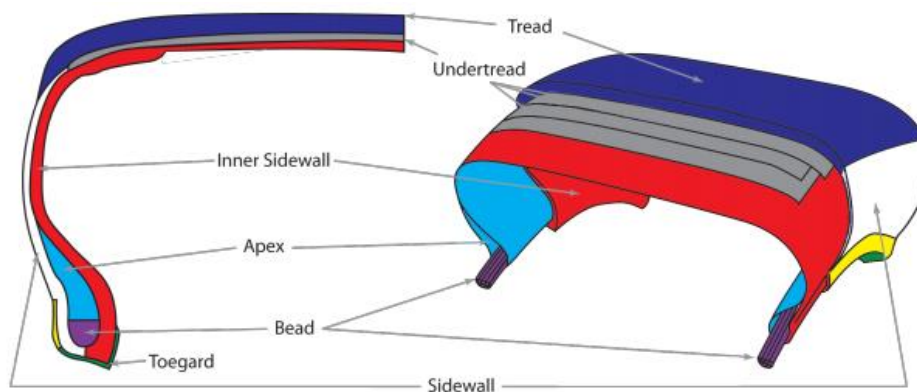


Figure 16 - Typical Tyre Construction (SPROT, 2013)

The use of a loaded tyre on a wind tunnel rolling road can cause excessive wear to the belt, which are expensive to maintain and replace. For this reason, many procedures are carried out using non-deformable tyres (solid) that have been pre-moulded to represent a loaded shape (*Figure 17*).

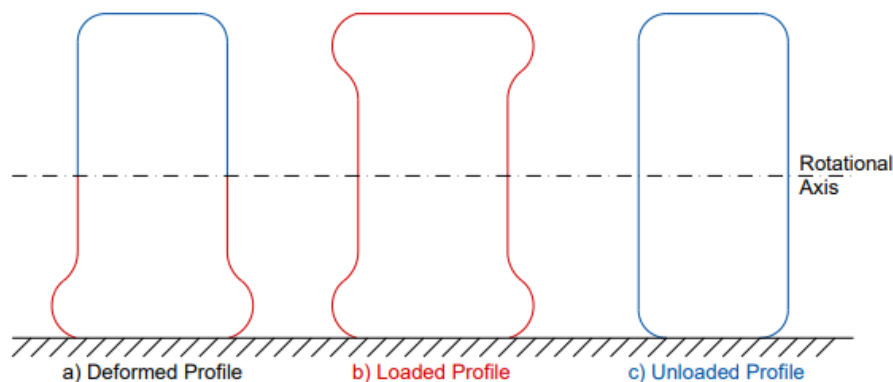


Figure 17 - Non-Deformable Tyre Profiles (Knowles, 2005)

Most wind tunnel test configurations require the vehicle to be supported using a sting. The sting is designed to be an aerodynamically neutral overhead support that is fixed to the vehicle and contains

a load cell. This mechanism suspends the vehicle over the rolling road, where the wheels can make contact and rotate, whilst it takes vertical and longitudinal load measurements.



Figure 18 - Wheels Off Wind Tunnel Test (Edmondson, 2015)

However, it will also measure the forces generated resulting from the rotation of the wheels, which is considered undesirable as regulations require that the wheels be fixed. Therefore a “wheels off” test structure is often used, which isolates the wheels from the vehicle body. The wheels are no longer supported by the suspension geometry, but are supported by individual wheel stings (*Figure 18*). It is thought, however, that the geometry of the wheel stings will now influence the flow regime generated by the rotating wheel, thus having a negative impact on data accuracy (Knowles, 2005).

Morelli is considered to be the pioneer of the aerodynamic testing of wheels and was the first to publish any work on the subject. His investigation involved the study of an isolated wheel, which he suspended above an already suspended ground plane. To further increase the accuracy of the procedure, an attempt was made to simulate the tyre deformation by protruding the tyre through a recess cut out in the ground plane (Morelli, 2000). After a significant number of studies, it was proposed that an isolated wheel would produce a vertical downward load, however, a test by the Motor Industry Research Association found that a wheel rotating in ground contact would produce an upward loading, to the contrary of Morelli. Stapleford and Carr discovered this by using an “idealised” Formula One model that housed motors to rotate the wheels, however the wheels could not be rotated when in contact with the ground, although a minor gap could be left between the ground plane and wheel. By using a pressure probe, static surface pressures could be measured on the surface of the tyre, and it was deemed that the wheels generated a significant amount of lift (Stapleford & Carr, 1969).

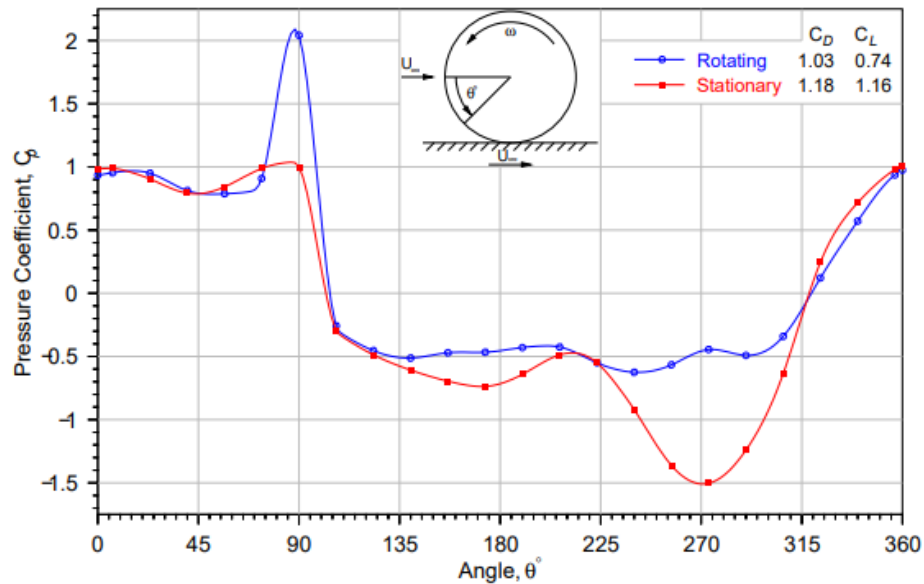


Figure 19 - Surface Pressure Distributions for Stationary and Rotating Wheels (Fackrell, 1972) (Knowles, 2005)

Fackrell introduced a non-intrusive surface pressure measurement for rotating wheels, which is illustrated in Figure 19. The above figure also presents a comparative case of a rotating and stationary wheel. It can be seen that the rotation of the wheel aids in the reduction of drag force and lift force. The graph also displays two key areas of interest, the contact patch (90°) and separation (280°) that are said to govern the flow field (Fackrell, 1972).

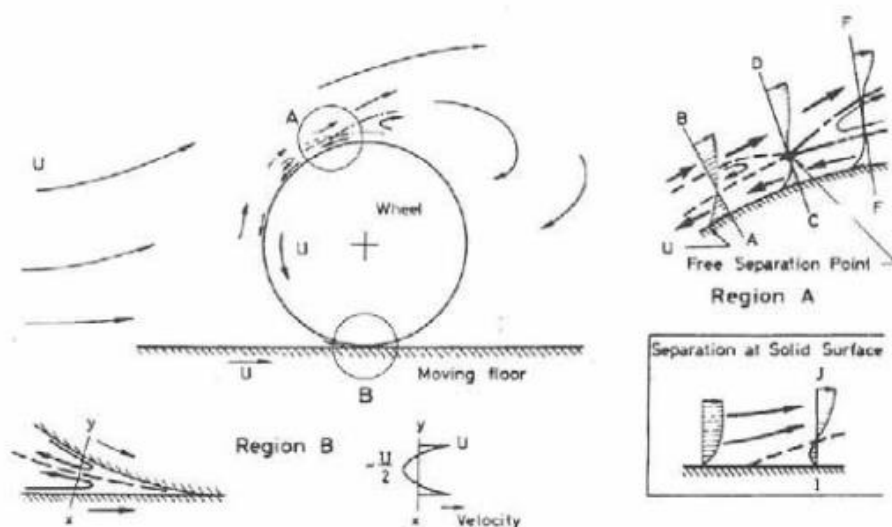


Figure 20 - Expected Flow Structures around a Rotating Wheel (Fackrell & Harvey, 1975)

With the rotation of the wheel, shear forces acting against the flow vector are radically increased, thus forming a much stronger pressure gradient and promoting separation to occur at an earlier point on the wheel compared to that of a stationary wheel. This then causes large areas of reverse flow to form within the wake structure, which are driven by the rotation of the wheel and the vortical structures (Croner, et al., 2012).

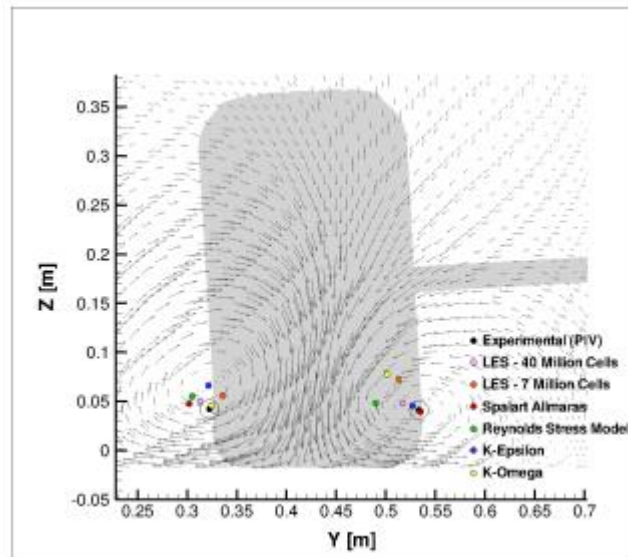


Figure 21 - Counter Rotating Vortical Structures (Issakhanian, et al., 2009)

In a computational study conducted by Issakhanian et al, two counter rotating vortical structures could be seen dominating the near-wake of a wheel, with the left side producing the stronger vortex due to the cambered nature of the wheel (*Figure 21*). It was found that as a result of the high air compressibility seen at the contact patch, a jetting would form as the compressed air would spill out of the contact patch leading edge and begin to interact with the near-wake, causing a vortex to form, following the shear layer path (Issakhanian, et al., 2009).

Whilst studying the near-wake structure of a stationary and rotating Formula One tyre, Wäschle was able to take laser Doppler anemometry measurements and compare them to Computational Fluid Dynamics (CFD) numerical codes. In the comparison, he was able to display regions of reversed flow, however, the 2 counter rotating vortices illustrated in *Figure 21*, were poorly represented due to the low resolution of the equipment used (Wäschle, et al., 2004).

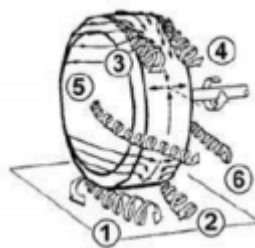


Figure 22 - The 6 Main Vortical Structures in the Near-Wake Flow Regime of a Rotating Wheel (Leśniewicz, et al., 2014)

When referring to *Figure 22*, there are six main vortical structures to consider within the near-wake region. The wake horseshoe vortex (1 and 2) which was already discussed in reference to *Figure 21*, the C-Shoulder vortex (3 and 4) which is caused by the onset separation forming over the top of the tyre and the squash vortex (5 and 6) (Leśniewicz, et al., 2014).

James, et al, demonstrated that a further downstream study of the wheel wake (*Figure 23*) exposes that the vortical structures begin to interact with each other. This is primarily due to the direction of rotation caused by the low pressure bubble, and presence of reverse flow seen behind the wheel (James, et al., 2014).

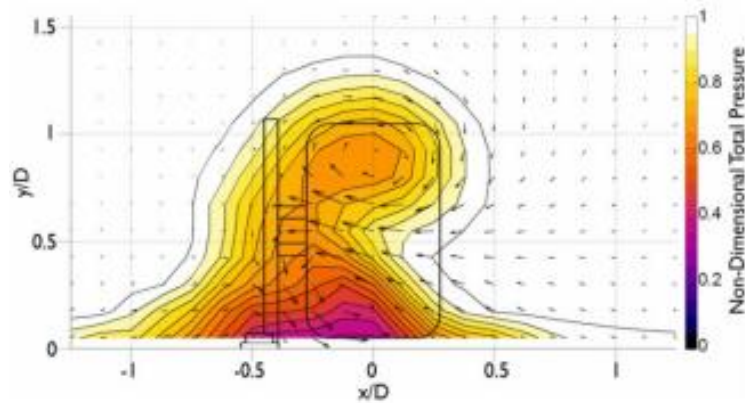


Figure 23 - 2.5m Downstream Total Pressure (Non-Dimensional) (James, et al., 2014)

2.4.0 Front Wing and Wheel Interaction

The original application of wings within motorsport was solely to generate downforce, however, with current simulation techniques and technological developments, the purpose of a front wing on an open wheel race car has progressed from the singular purpose of increasing grip.

By optimising the shape and aerodynamic properties of the wing, it can be used as a mechanism to control the flow regime of the wheel wake. Often, the front wing will work alongside other devices such as barge boards, to adequately manage the turbulent wake produced by the front wheels. As already stated, the wake generated by the front wheels has the potential to disrupt any aerodynamic components downstream of their location (TotalSim Ltd, 2016).

Due to the pressure gradient developed by a wing, the potential for vortex formation is highly likely, even when end plate structures are employed. S. Diasinos, et al, experimented with wing and wheel interactions using numerical computational methods. It was found that a wing with $\alpha=0^\circ$, and an equal span to the wheel width, that the vortex structures from the end plates were able to travel around the outer surface of the wheel. AoA was then increased to $\alpha=12^\circ$, which caused the lower vortex (the primary vortex) to travel inboard of the wheel, while the top vortex remained outboard. A third study found that reducing the span of the wing so the end plate was parallel to the inner wheel face resulted in both the upper and lower vortices travelling inboard of the wheel (Diasinos, et al., 2014).

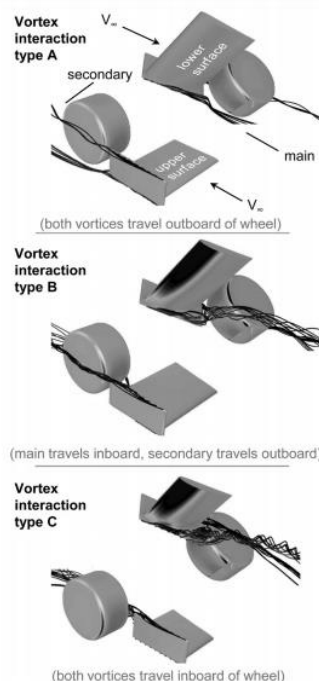


Figure 24 - Vortex Interaction Types (Diasinos, et al., 2014)

Figure 24, illustrates visually, the results produced by the experiments. However, the geometry of the end plate and wheel are general configurations and the distance between the wing and the wheel was fixed without prior experimentation. The wing profile used was an inverted Naca 4412. It was found

that in each of the experiments undertaken, including wing to ground proximity, the CI of the wing was always negatively affected when interacting with the wheel, compared to when in isolation.

Heyder-Bruckner found that the presence of a wheel situated behind a wing resulted in a blockage effect, forcing the air flow to accelerate between the trailing edge of the end plate and wheel, while also causing the wing edge vortices to deflect. The region of flow stagnation on the front surface of the wheel was also impacting the surface pressure distribution of the wing, which varied with ride height (Heyder-Bruckner, 2011).

Nakagawa, et al, also described the interaction between the front wing and wheels, referencing a Formula 1 car. It can be observed when examining *Figure 25*, that alterations to the front wing geometry can have a severe impact on the generated vortical structures, causing different trajectories of travel. By allowing the modification of the front wing to adjust the flow regime of the wheel wake, control can be gained depending on requirements (Nakagawa & Hojo, 2014).

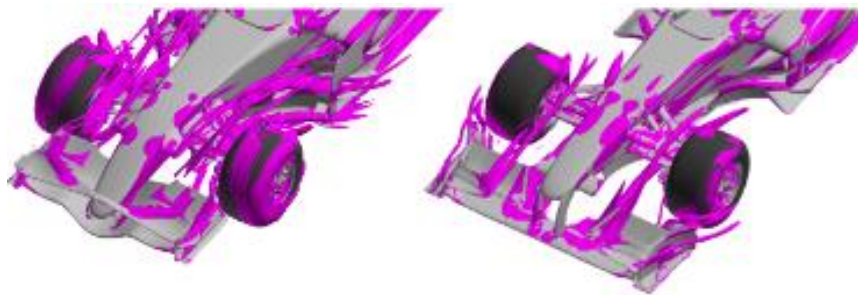


Figure 25 - Wing Geometry Impact of Vortex Formation (Nakagawa & Hojo, 2014)

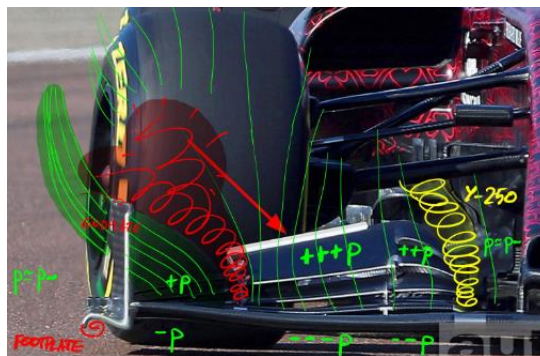


Figure 26 - Estimated 2019 Alfa Romeo C38 Formula 1 Car Front Wing Flow Regime (Susín, 2019)

When it comes to Formula 1, the created geometries for aerodynamic devices are often incredibly intricate and complex due to the chase to achieve any marginal gain in aerodynamic efficiency and overall performance. *Figure 26*, illustrates an estimated representation of the general flow regime developed on the 2019 Alfa Romeo C38 Formula 1 Car (Susín, 2019). It can be observed that by controlling the direction of vortex rotation, the vortex can be guided towards areas that require additional energy in order to maintain stability, or used as a blockage to stop the turbulent wheel wake disrupting downstream components or mixing with highly energised air that is being fed to the under floor area, as can be observed in *Figure 27*.

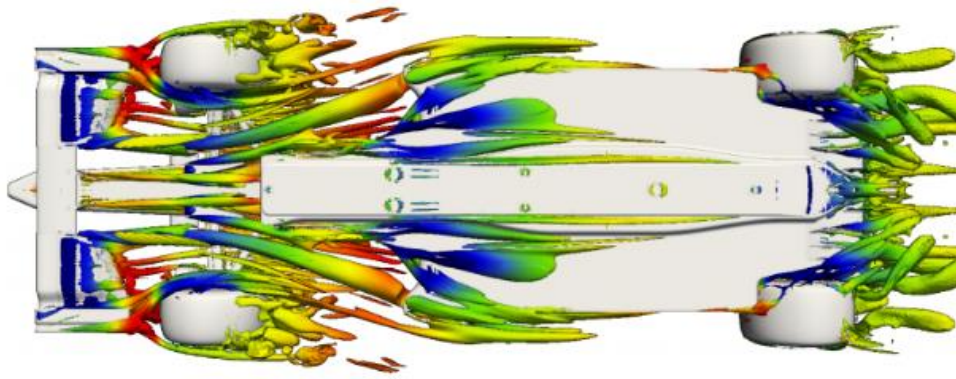


Figure 27 - Vortex Structures and Outwash used to Deflect Wheel Wake (TotalSim, 2016)

The rotational direction of the vortex can also be used to generate additional downforce if utilised properly. This is achieved by generating a strong downwash that can be fed to the floor section of the car.

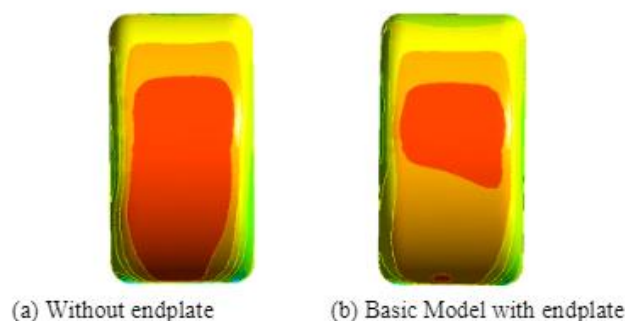


Figure 28 - Change in Front Wheel Stagnation Pressures, With and Without Endplates (Petkar, et al., 2014)

Petkar, et al, found the introduction of end plates had a dramatic effect on the stagnation zone of the front wheel. It can be observed in *Figure 28*, that the frontal area covered by high pressure is largely reduced with the introduction of end plates. This is as a result of the outwash effect accelerating the air past the leading surface of the wheel and generating a lower region of pressure. As a consequence, it was found that the inclusion of end plates led to a reduction in overall drag force and an increase in downforce. The increase in downforce is due to a pressure equalisation between the underside of the wing, and front surface of the wheel, therefore lowering the pressure gradient, causing less detrimental interaction (Petkar, et al., 2014).

In addition to understanding the fundamentals of wing theory and the aerodynamic characteristics of rotating wheels, this chapter highlights key areas of focus when considering design criteria for a front wing concept. The interactions between a front wing and wheel are highly complex and compromises always have to be made. These maybe in the form of model accuracy in order to decrease computational expense or to maintain the goals of the design outcome. The design philosophy of one car may differ greatly to that of another car, even if they are both single seater, open wheel race cars.

3.0.0 Methodology

This section of the report will outline the numerical methods used during the course of this investigation. Following the referenced literature in the previous chapter, it has been concluded that a series of validation studies should be undertaken.

CFD methods are purely numerical and are a function of a series of algorithms and therefore obey predetermined laws, including the conservation of mass, momentum, and energy (Navier-Stokes). In addition to these, a turbulence model is also required to be defined, which will therefore increase the solver complexity.

3.1.0 Validation Study

In order to provide full confidence in the results produced, three validation studies were conducted, each monitoring static surface pressures taken from experimental methods (Naca 0012, MIRA Notchback reference car and Fackrells Wheel) and then replicated with the use of numerical methods. The three validation studies are viewable in chapter 7.0.0 Appendix A.

Meshing strategy and specific turbulence models will also be discussed within the following chapters.

3.1.1 Dimensionless First Cell Height (Y^+)

Y^+ is a dimensionless height given for the first cell within the build grid. For the purposes of automotive CFD, the majority of the cells used are generally a tetrahedral shape. The reason for this is that they can easily be moulded and orientated without significantly impairing the mesh quality, which makes the use of the tetrahedron beneficial for complex geometries (Ordaz, et al., 2014). Its benefits can be demonstrated in freestream flow regime and eddy capture, however, it may be considered inadequate for boundary layer definition.

Particular attention is required to be given to the boundary layer. Refinement of the build grid needs to be improved in the close to wall region of the model in order to produce accurate modelling of the boundary layer. The reason for this is that the boundary layer can determine the flow regime immediately parallel to the body of interest.

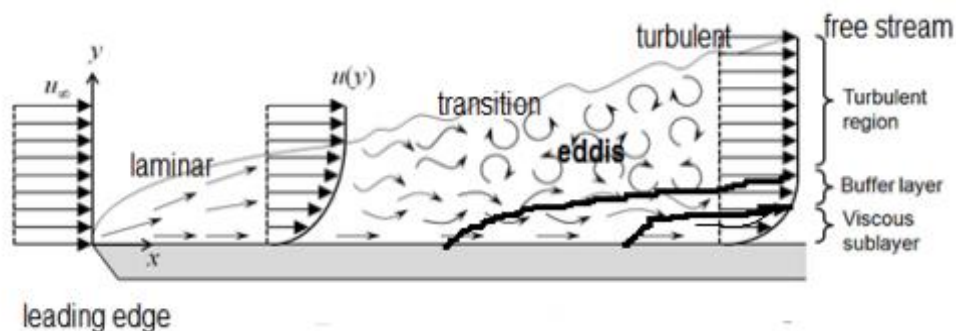


Figure 29 - Boundary Layer Construction (Mishra & Aharwal, 2018)

The boundary layer consists of three longitudinal regions and three vertical regions and is a consequence of the no shear conditions produced by the fluid viscosity, therefore allowing no slip

($U=0\text{m/s}$) immediately parallel to the body. *Figure 29*, illustrates these regions on a flat plate. It can be observed that as the flow regime travels further along the plate, it transitions from a laminar state to a turbulent state. This is as a result of the energy losses within the boundary layer as energy transforms into heat due to friction (while overcoming pressure gradients) which can lead to onset boundary separation from the planar surface. With an increase in laminar boundary conditions, friction is reduced resulting in a reduction of pressure drag, thus improving efficiency (Kim, et al., 2017), as pressure drag often outweighs skin friction drag.

During the transitional and turbulent states, three layers exist: the viscous sublayer, the buffer layer and the turbulent / log law region. Depending upon the turbulence model required (whether, or not, wall functions are implemented), the build grid needs to be highly refined to capture these regions.

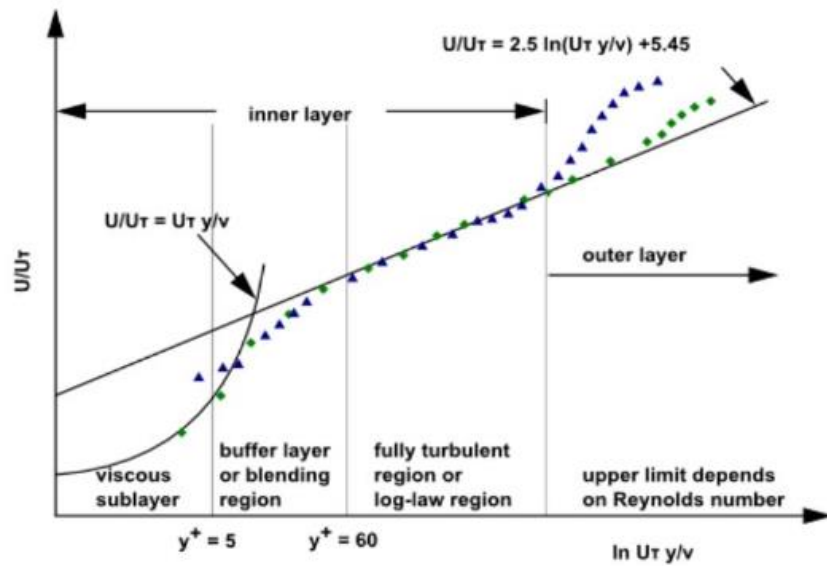


Figure 30 - Sub-Divisions of the Near-Wall Region (KRÁLIK, 2016)

Figure 30, illustrates the velocity profile within a boundary layer. It can be observed that during a low state of U/U_τ , there is an exponential increase in Y^+ which then develops a linear state as the log law region is entered. The state of the buffer layer is one of a transition period between the viscous sublayer and log law region. This produces difficulties when modelling due to the transfer of exponential to linear and where that transition begins, therefore particular turbulence models will apply wall functions to “reproduce or predict” this area (Liu, 2016).

To begin to accurately represent the near-wall regions, the application of the dimensionless Reynolds Number must be utilised. The Reynolds Number is stated as:

$$R_e = \frac{\rho UL}{\mu}$$

Equation 4 - Reynolds Number

Where ρ is the fluid density, U is the velocity, L is the distance and μ is the dynamic viscosity of the fluid. It is the ratio of the inertial forces to the viscous forces that defines whether the boundary flow

regime is laminar or turbulent. If improperly described, the Reynolds number will have a critical impact on the outcome of the simulation results (Wood, 2015). In the case of external flows, $Re < 1 \times 10^5$ is a laminar state and $Re > 5 \times 10^5$ is a turbulent state (Incropera, et al., 2007).

With an acquired Re , the state of the near-wall flow condition can be described, therefore allowing the calculation of the boundary layer thickness δ .

$$\delta_{Laminar} = \frac{5L}{\sqrt{Re}}$$

$$\delta_{Turbulent} = \frac{0.37L}{Re^{\frac{1}{5}}}$$

Equation 5 - Boundary Layer Thickness (m)

Schlichting describes the formulas presented in Equation 5, as a representation of the laminar and turbulent conditions, where $n = 5$ or 0.37 and is a scaling function of the freestream velocity, distance travelled and height (Schlichting, 1979).

Sijal explains the relevance of Y^+ and why the correct use of it is important. As already stated, turbulence models depend greatly on the value of Y^+ to properly model the near-wall region. Figure 31, illustrates the near-wall velocity profiles taken at a distance of $X = -123\text{mm}$ on an Ahmed body.

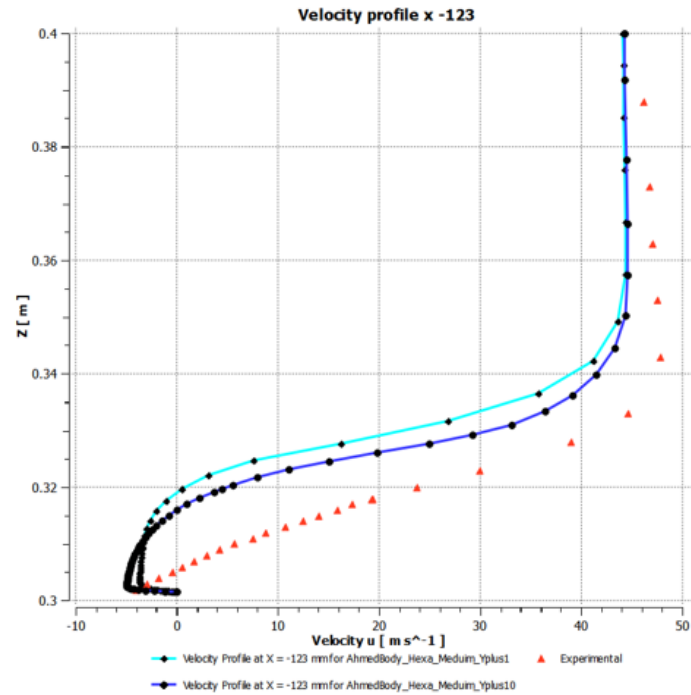


Figure 31 - Ahmed Body Velocity Profile at Distance X (Sijal, 2019)

Using Y^+ as a variable, the impact of the improper utilisation of Y^+ can clearly be observed in the velocity profiles, therefore inaccurately capturing the boundary layer, which could cause an over or under prediction of separation or surface pressure (skin friction) (Sijal, 2019).

$$C_f = \frac{0.058}{Re^{0.2}}$$

Equation 6 - Coefficient of Friction

$$\tau_w = 0.5C_f\rho U^2$$

Equation 7 - Wall Shear Stress (N/m²)

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}}$$

Equation 8 - Friction Velocity (m/s)

As the lower region of the boundary layer exists within a no slip condition, the fluid is subjected to shear stresses across its Y axis ($U \approx 0$ close to wall). *Equation 6*, indicates the utilisation of Re when calculating the coefficient of friction, therefore illustrating that Re is not only a representation of the boundary layer condition, but is also a function of friction within the near-wall region, which is subsequently used to calculate the wall shear stress (*Equation 7*). As the freestream velocity increases the frictional forces, thus the shearing forces will also increase. As a consequence, this will affect the velocity profile within the boundary layer, and is represented in *Equation 8* (Tachie, et al., 2002).

Therefore, Y_+ can be written as:

$$Y_+ = \frac{U_\tau \cdot \Delta Y_1 \cdot \rho}{\mu}$$

Equation 9 - Dimensionless First Cell Height

Where ΔY_1 is the first cell height (m) and μ is the dynamic viscosity of air (1.802e-5Kg/m-s at 15°C). By transposing *Equation 9*, to allow ΔY_1 to be the product, the desired Y_+ value can be used, and the first cell height can be found.

It is common practise to use prisms in the build grid within close to wall regions. The close to wall region can be refined using tetrahedrons, however, this method creates a mass density within the build grid and is computationally expensive. Prisms introduce 2 extra nodes per cell and are already orientated to be parallel with the near-wall flow trajectory, thus introducing further accuracy and saving computational time (efficiency increase) (Kapoor, 2016).

Having established the boundary layer thickness and the first cell height, the number of prism layers required can be calculated. An expansion rate of 120% is commonly used to allow a smooth transition between layers (Kapoor, 2016).

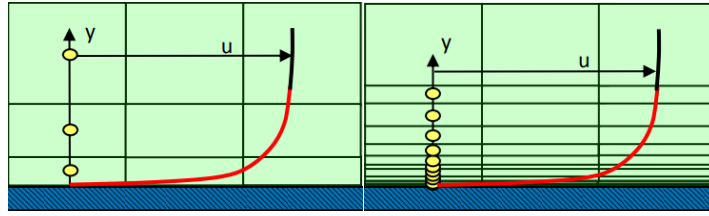


Figure 32 - (Left) Logarithmic Based Wall Functions Resolution, (Right) Viscous Sublayer Based Wall Functions Resolution

By refining Y^+ , the viscous sublayer can be resolved by introducing more nodes within the boundary layer region defined by δ (Figure 32). However, the value of Y^+ evolves depending on the required turbulence model due to the inclusion of wall treatments (Ansys Inc, 2014).

3.1.2 Turbulence Model

Turbulence models are a supplement to Navier Stokes equations. The Navier Stokes equation is the governing equation that calculates or predicts the motion of the fluid flow in three dimensions and are made up of five equations that govern the conservation of mass, momentum and energy, as illustrated in *Equation 10*.

$$\begin{aligned}
 \text{Continuity: } & \frac{dp}{dt} + \frac{d(\rho u)}{dx} + \frac{d(\rho v)}{dy} + \frac{d(\rho w)}{dz} = 0 \\
 X \text{ Momentum: } & \frac{d(\rho u)}{dt} + \frac{d(\rho u^2)}{dx} + \frac{d(\rho uv)}{dy} + \frac{d(\rho uw)}{dz} = \frac{dp}{dx} + \frac{1}{Re_r} \left(\frac{d\tau_{xx}}{dx} + \frac{d\tau_{xy}}{dy} + \frac{d\tau_{xz}}{dz} \right) \\
 Y \text{ Momentum: } & \frac{d(\rho v)}{dt} + \frac{d(\rho uv)}{dx} + \frac{d(\rho v^2)}{dy} + \frac{d(\rho vw)}{dz} = \frac{dp}{dy} + \frac{1}{Re_r} \left(\frac{d\tau_{yx}}{dx} + \frac{d\tau_{yy}}{dy} + \frac{d\tau_{yz}}{dz} \right) \\
 Z \text{ Momentum: } & \frac{d(\rho w)}{dt} + \frac{d(\rho uw)}{dx} + \frac{d(\rho vw)}{dy} + \frac{d(\rho w^2)}{dz} = \frac{dp}{dz} + \frac{1}{Re_r} \left(\frac{d\tau_{zx}}{dx} + \frac{d\tau_{zy}}{dy} + \frac{d\tau_{zz}}{dz} \right) \\
 \text{Energy: } & \frac{d(E_t)}{dt} + \frac{d(uE_t)}{dx} + \frac{d(vE_t)}{dy} = - \frac{d(up)}{dx} \\
 & = - \frac{d(up)}{dx} - \frac{d(vp)}{dy} - \frac{d(wp)}{dz} - \frac{1}{Re_r Pr_r} \left(\frac{dq_x}{dx} + \frac{dq_y}{dy} + \frac{dq_z}{dz} \right) \\
 & + \frac{1}{Re_r} \left(\frac{d}{dx} (u\tau_{xx} + v\tau_{xy} + w\tau_{xz}) + \frac{d}{dy} (u\tau_{yx} + v\tau_{yy} + w\tau_{yz}) + \frac{d}{dz} (u\tau_{zx} + v\tau_{zy} + w\tau_{zz}) \right)
 \end{aligned}$$

Equation 10 - Navier Stokes Equation

Equation 10, yields the local accelerations, convection, pressure gradients and viscosity terms. These equations are used to calculate the fluid flow for a Newtonian fluid with respect to time. By using the continuity of mass flow, the fluid velocity can be equated, therefore the pressures around a body are given, which can then be resolved as forces (Versteeg & Malalasekera, 2007).

Most external fluid regimes encounter high peaks of inertial forces and extreme gradients of velocity resulting from high viscosity interactions. This would require great amounts of computational power to resolve using a time dependant equation, therefore an application of a Reynolds Averaged Navier Stokes (RANS) equation is utilised. The RANS model effectively uses an average of the fluid velocity, pressure and stress per x, y and z coordinate and applies it to an average unit of time. This method requires far less computational power, thus the time required to resolve the equations is decreased (Sodja , 2007).

Due to the time averaging (with respect to the momentum equations) instantaneous turbulent fluctuations are lost, however, a description of the effects of turbulence on the mean flow are still essential. To allow for the computation of turbulent flows, specific models have to be applied to predict the Reynolds stresses (velocity components averaged and the divergence of that term is

used) and scalar transport terms to allow the mean flow system to be completed (Versteeg & Malalasekera, 2007).

RANS turbulence models are classified by the number of additional transport equations they apply to the RANS flow equations.

For the purposes of this investigation, two turbulence models will be investigated:

1. K-Epsilon ($k\varepsilon$)
2. K-Omega ($k\omega$)

Both models apply two additional transport equations, turbulent kinetic energy and the turbulence dissipation rate and are the most commonly used models for automotive external aerodynamics (Launder & Spalding, 1973).

Using the Boussinesq hypothesis, it can be observed that the Reynolds stresses are related to the turbulence viscosity (eddy viscosity) which is presented as μ_t . The turbulence model is used to solve μ_t which is the viscous term located at the end of each momentum formula in, *Equation 10*.

Early RANS models used a mixing length (l_m) to calculate μ_t . The mixing length is a representation of the size of the turbulent eddies that exist within the fluid flow. For areas with large eddies and high energy, the mixing length will be large in size, and for areas with small eddies and low energy, the mixing length will be small in size. This was initially proposed by Prandtl who produced the hypothesis:

$$l_m = Ky$$

Equation 11 - Prandtl Mixing Length

Where $K = 0.41$ and y is the distance away from the wall, the maximum size of the eddies is therefore limited by the presence of the wall and distance y away from the wall (Cebeci, 2004). However, viscous properties also exist close to the wall which have a dampening effect on the eddies which reduces the dissipation that they produce. Van Driest further extended Prandtl's hypothesis to include this, which is illustrated in *Equation 12*.

$$l_m = Ky \left(1 - \exp \left(-\frac{Y^+}{A^+} \right) \right)$$

Equation 12 - Van Driest Hypothesis

In *Equation 12*, A^+ is a damping constant equal to 26.0 (Craft, 2011). When compared to Prandtl's linear approach, the Van Driest method demonstrates that as the wall is approached, the mixing length reduces. This, therefore, alters the mean velocity profile and produces the sub-divisions of the near-wall region (*Figure 30*). Although this approach is used to resolve the viscous sublayer, it produces very low values of Y^+ .

As turbulence is diffused through the fluid flow, the mixing length cannot be fixed as *Equation 12* would suggest, so rather than resolve l_m , it's is now suggested that the turbulence dissipation rate ε (for the K-Epsilon model) be resolved. However, *Equation 13* demonstrates that the ε transport equation is directly related to l_m .

$$l_m = \frac{C_\mu k^{\frac{3}{2}}}{\varepsilon}$$

Equation 13 - Mixing Length Relation to Transport Equations

This, therefore, suggests that solving the turbulence dissipation rate is an alternative means of calculating the mixing length (physical eddy size) (Toorman, 2000).

$$\frac{d(\rho k)}{dt} + \nabla \cdot (\rho k U) = \nabla \cdot \left(\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right) + P_k + P_b - \rho \varepsilon + S_k$$

Equation 14 - Turbulent Kinetic Energy

Equation 14, illustrates the transport equation used for turbulent kinetic energy. It is a form of rate of change for kinetic energy per unit time. However, it includes the transport of k by convection, pressure, viscous stresses, Reynolds stress, dissipation and rate of production, or more concisely;

$$Time + Convection = Diffusion + Sources$$

In this formula P_k is the production due to the mean velocity gradient, P_b is the production due to buoyancy and S_k is a user definition (Versteeg & Malalasekera, 2007). It can be observed that ε exists within the kinetic energy transport equation as it accounts for the dissipation of the kinetic energy. This transport equation remains the same for each variant of $k\varepsilon$ model, however, the dissipation rate equations vary per model variant.

$$\frac{d(\rho \varepsilon)}{dt} + \nabla \cdot (\rho U \varepsilon) = \nabla \cdot \left(\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \nabla \varepsilon \right) + C_1 \frac{\varepsilon}{k} (P_k + C_3 P_b) - C_2 \rho \frac{\varepsilon^2}{k} + S_\varepsilon$$

Equation 15 - Turbulence Dissipation Rate

Equation 15, illustrates the formula used within the $k\varepsilon$ model to describe the turbulence dissipation rate (Ansys Inc, 2009). When examined, it can be observed to possess a similar layout to the kinetic energy (*Equation 14*) in terms of time, convection, diffusion and sources. C_1 and C_2 are model coefficients which vary depending the variant of $k\varepsilon$ model and are modelled against experimental data (Stanford University, 2007). However, for the standard model, $C_1 = 1.44$ and $C_2 = 1.92$. (Versteeg & Malalasekera, 2007), (Lauder & Spalding, 1973).

Once *Equations 14 and 15* have been solved, they can be substituted into *Equation 16* to solve the eddy viscosity μ_t , where C_μ is a coefficient given as 0.09 (Lauder & Spalding, 1973).

$$\mu_t = \rho k^{1/2} l_m = C_\mu \frac{\rho k^2}{\varepsilon}$$

Equation 16 - Eddy Viscosity

As the model no longer solves for l_m but for ε , a method for damping the dissipation rate needs to be applied in order to account for the viscous effects on the eddy formulation. Damping function coefficients f_1 , f_2 and f_μ are therefore applied to the model coefficients. These functions are accounting for the turbulent Reynolds number (Re_T). The substitution of *Equations 13 and 16* into *Equation 4* now produces:

$$Re_T = \frac{\rho k^2}{\mu \varepsilon} = \frac{\text{Turbulent Forces}}{\text{Viscous Forces}}$$

Equation 17 - Turbulent Reynolds Number

When the value Re_T is low, viscous forces are dominating. This means that the model can be used to solve the whole way up to the wall, even if the first cell height is located within the viscous sublayer. This is referred to as the Low Re approach (Pope, 2015). It must be stated that the low Re approach should not be applied to the entire flow scheme, it should be applied only to the close to wall region.

However, Frei describes the majority of the available $k\varepsilon$ models as being inadequate within the close to wall region as they are not of the low Re approach. Low Re approach models require an extremely high resolution mesh in the near-wall region, which therefore increases the computational complexity and time (Frei, 2017).

The $k\omega$ models employ the low Re approach, and therefore dominate within the close to wall region. However, it has also been stated that their superiority lessens within the far from wall region and is far more computationally expensive compared to the bulk of $k\varepsilon$ models.

It is as a result of this, that $k\varepsilon$ models have now been adapted to include the use of wall functions, as previously stated. The wall functions predict the velocity flow field within the sublayers of the boundary and apply universal log-law behaviour. This method, however, completely ignores the buffer layer. An update from the Standard $k\varepsilon$ model is the Realizable $k\varepsilon$ model, which is considered to be a highly stable and robust solution that converges well, but still suffers within high adverse pressure gradients. This limits its ability to correctly predict separation areas and to model vortical structures (Salient Edge, 2018), although it is still said to be a vast improvement over the standard $k\varepsilon$ model due to its application of dissipation distortion and safety functions applied to the Reynold stresses.

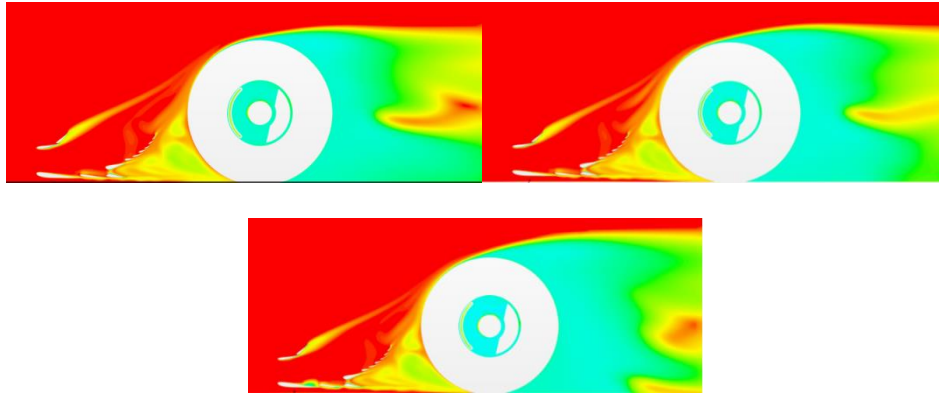


Figure 33 - Total C_p of Wheel Wake (Top Left – SKE) (Top Right – RKE) (Bottom – SST) (Salient Edge, 2018)

A variant of both the $k\varepsilon$ and $k\omega$ models is the $k\omega$ SST (shear stress transport) model. This is a model that inherits the close to wall superiority of the $k\omega$ model, but is not as computationally costly as it has adopted the far from wall modelling of the $k\varepsilon$ approach, therefore it is a blend of the two models. This model possesses the ability to account for the transport of shear stresses within adverse pressure gradient boundary layers (Fluent Inc, 2001). It also introduces the assumption that the shear stress is proportional to the turbulent kinetic energy, which is implemented within μ_r (Ansys Inc, 2006). Due to its superiority within the close to wall region, this model still requires a high resolution mesh in the near wall region (Figure 34).

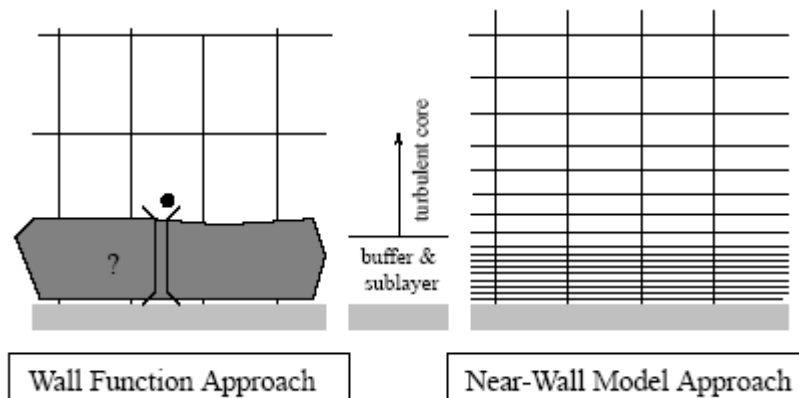


Figure 34 - Wall Function and Near-Wall Function Approaches (Ansys Inc, 2009)

It can be observed in Figure 33, that using the three turbulence models on the same study can introduce differences within the post processing results. Lanfrit suggests the use of the $k\varepsilon$ Realizable, Non Equilibrium Wall Functions model (Lanfrit, 2005). This model employs a wall function that utilises a two-layer concept within the boundary layer. It solves an economic amount of turbulent kinetic energy within the cells adjacent to the wall. The wall neighbouring cells are presumed to consist of a viscous sublayer and a fully turbulent layer, therefore assuming the profile of turbulent quantities. However, because of its capability to account for pressure gradient effects, this model is recommended for use where complex flow structures dominate, such as separation, reattachment and impingement where the mean fluid flow and turbulence are exposed to rapid changes in pressure gradient (Ansys Inc, 2009) which is contrary to Salient Edge cfd.

Due to computational power constraints, the decision was made to apply the recommended $k\varepsilon$ turbulence model for the purposes of this investigation, as refining the close to wall region would be too computationally costly. In addition, LES, DES and DNS models were not considered because of their highly complex nature and costly approach. Both Keogh et al, and Ouahabi et al, found that if the build grid could not be highly refined in the near wall regions, that the $k\omega$ SST model would predict onset separation too early and near-wake velocities would be inaccurate, although Keogh et al, found that the Realizable model provided superior correlation to measured experimental results (Keogh , et al., 2014) (Ouahabi , et al., 2017).

3.2.0 SBD Dallara F399/01 CFD Investigation

An initial investigation into the baseline geometry of the Dallara F399 was conducted. The study was set up with a 40m/s freestream velocity. The ground plane and wheels were selected as moving boundaries, and an applied translational velocity vector of 40m/s was applied to the near-wall region of the ground. A relative rotational velocity vector of 137 Rad/s and 136Rad/s respectively was applied to the wheels. No yaw or slip angle was applied, however, the wings were modelled at the lowest baseline AoA with a minimum ground proximity of 40mm being adhered to at the splitter, and 60mm marginally forward of the rear wheels, which results in a front wing H/c of 0.24.

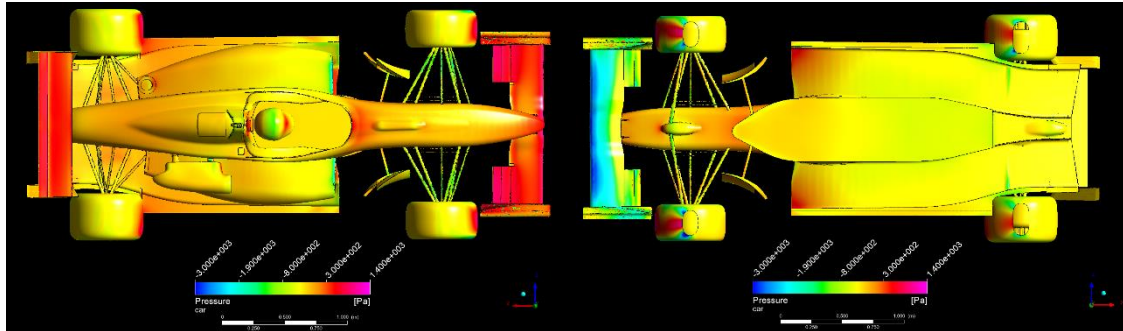


Figure 35 - SBD Dallara F399 Static Pressure

Viewed from the underside, *Figure 35* illustrates that there is a highly concentrated area of negative pressure located at the leading edge on the suction side of the front wing main plane (likely induced by the rake angle of the vehicle and wing profile geometry). This phenomenon can cause the aerodynamic behaviour of the wing to become increasingly unpredictable when in a dynamic situation. This is largely due to the profile geometry being used (*Figure 36*). As the free stream flow begins to interact with the leading edge, the air velocity must increase between the ground plane and suction side to maintain the conservation of mass.

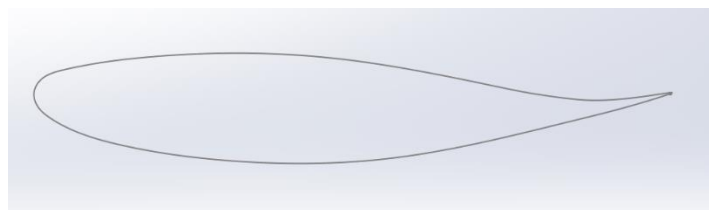


Figure 36 - Main Plane Profile

Having a large radius leading edge, causes a rapid increase in velocity, resulting in a peak pressure drop when adhering to Benoulli's hypothesis. As the radius begins to increase, the pressure begins to normalise and then begins to increase within the expansion zone as the air returns to atmospheric gauge pressure. Barbarino et al, describes this phenomenon with an isolated wing in freestream air (Barbarino, et al., 2010). With the wing located within recommended H/c for ground effect wings, it would be assumed that the lowest tangential surface to the ground would be subjected to the greatest negative pressure (Katz, 2006). However, the wing is also being subjected to an increased adverse pressure gradient due the vehicle body and the location of the wheels producing prominent stagnation

areas. This adversely affects the pressure distribution along the aerofoils chord length, allowing the peak negative pressure to be located earlier on the chord length. Toet also displays this within a concept case study discussing trailing edge separation, however, the aerofoil profile used greatly differs from the profile employed in this case (Toet, 2019).

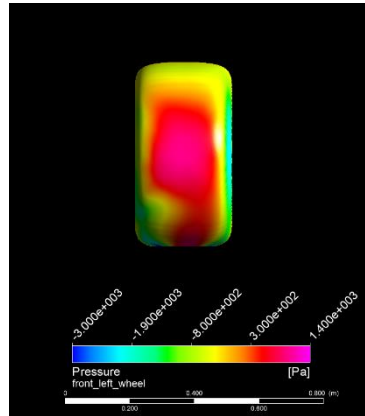


Figure 37 - Front Wheel Stagnation Zone

Large stagnation zones are also displayed on the front face of the front wheels (*Figure 37*). This is not uncommon and to be expected. It is also noticeable, however, that a small bubble of reduced pressure exists on the lower inner shoulder, merging onto the front face. This is as a result of a developed vortex on the bottom edge of the end plate.

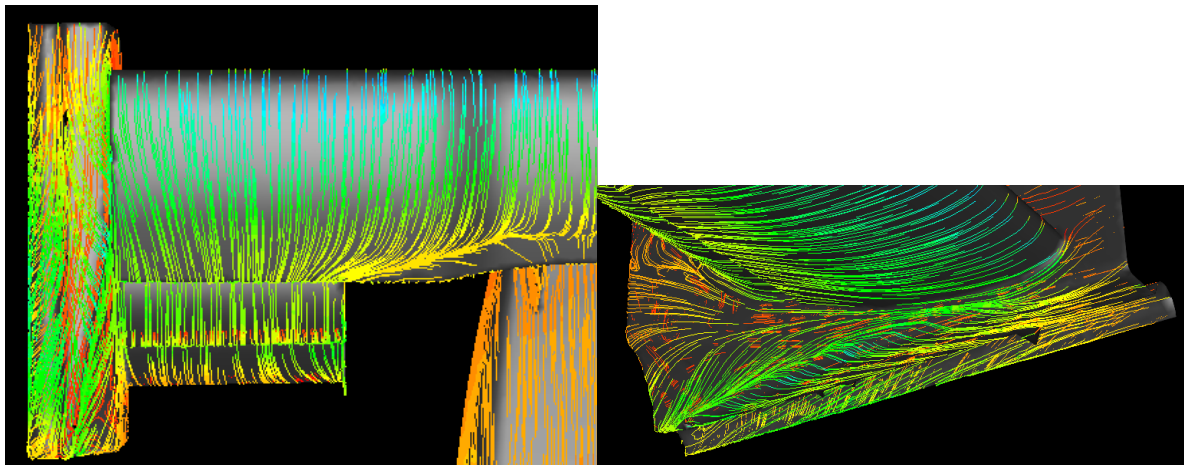


Figure 38 - Oil Flow Front Wing End Plate (Static Pressure)

Typically, the primary vortex would be located just below the suction surface on the inner side of the end plate (Keogh, et al., 2015). The foot plate is used to house the developed vortex (primary vortex), although this particular design incorporates a region of expansion (*Figure 38*). The expansion of the vortex allows a greater mass flow and an increased drop in pressure and the vortical structure is then passed around the inner shoulder of the front wheel. A similar end plate design is utilised in the aerodynamic package for the Indycar series (*Figure 39*). It is used, however, to accelerate air around the outer shoulder of the wheel due to the greater allowable span of main plane, thus dissipating a greater area of the stagnation, and reducing pressure drag (Sandoval, 2019).

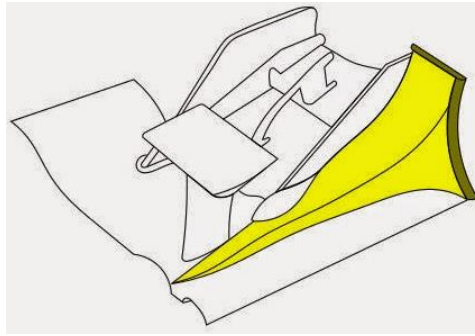


Figure 39 - 2015 IndyCar Front Wing End Plate (Somerfield, 2015)

The greater expansion volume also allows the secondary elements to perform better when a wheel lock is applied, due to the increased mass flow and a further pressure drop over the suction side (Somerfield, 2015) as well as the end plates usual purpose of increasing the working area of the wing by hindering pressure bleeding.

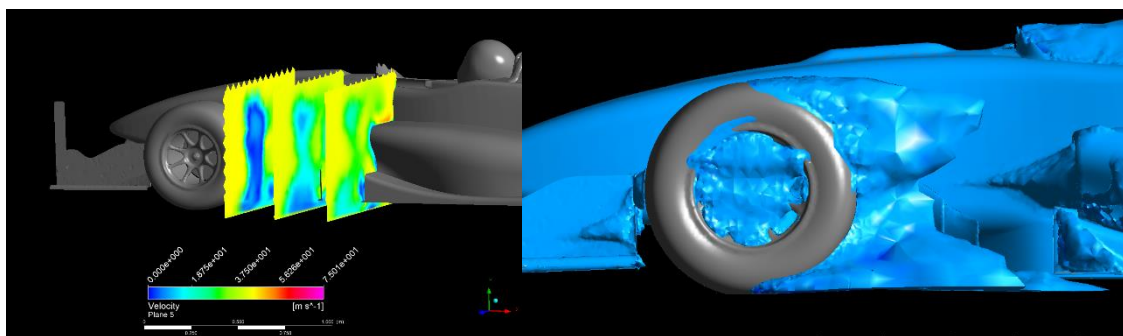


Figure 40 - Front Wheel Wake

The front wheel wake is similar to that described by James et al, in Figure 23. The left hand image in Figure 40, illustrates the velocity profile of the wheel wake, displayed on planes set at intervals of 0.3m respectively. It can be observed, that as the turbulent wake field travels further aft of the vehicle, its interaction with the side pods increases.

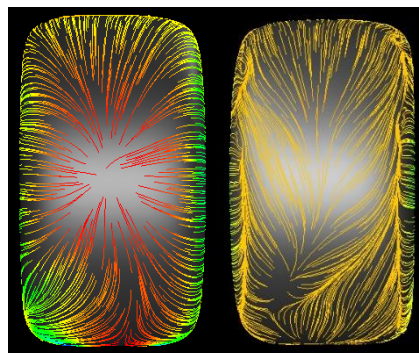


Figure 41 - Front Wheel Face Oil Flow in term of Static Pressure (Left - Front, Right - Rear)

Figure 41, clearly illustrates the area of recirculation occurring in the near-wake. This is, in part, due to shearing forces generated from the rear face of the wheel producing an upwash, with the upwash

being enhanced by the produced tyre jetting (vortices) located either side of the contact patch. This is also apparent when examining *Figure 42*.

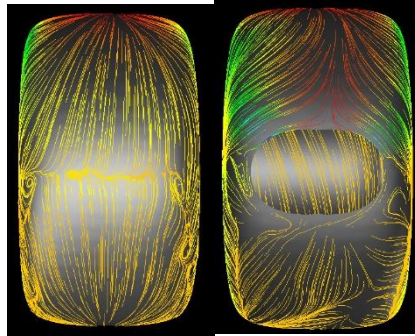


Figure 42 - Front Wheel Face Oil Flow in terms of Static Pressure (Left - Top, Right - Bottom)

Figure 42, illustrates the top face and contact patch face of the wheel. The top face displays the interaction of the upwash on the rear face of the wheel with the onset flow on the front face of the wheel. The defined line of mixing is located approximately where the boundary layer separation occurs. Two areas of recirculating flow are apparent on both shoulders of the wheel when viewed from above and from below. These are the vortical structures explained by Leśniewicz, in *Figure 22*.

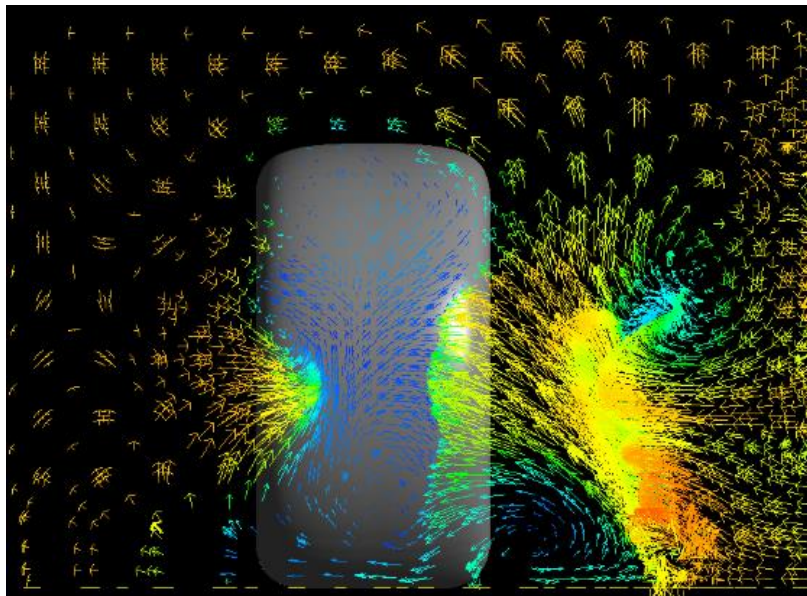


Figure 43 - Front Wheel Near-Wake Vortex Interaction

As previously referenced, and expressed by Mouffouk, the generated vortices then begin to interact with each other in the reverse flow (Mouffouk, 2014). This results in the wake structure illustrated in *Figure 40*. *Figure 43*, illustrates the mixing of the vortices within the wake field at a distance of 0.4m from the axle centre. It is also apparent that two additional vortices located at the inner shoulder boundary are beginning to interact within the wheel wake.

The lower vortex travelling immediately adjacent to the wheel is that which is generated by the front wing end plate. The second vortex, located further inboard, is as a consequence of the secondary elements. This vortex has an opposing direction of rotation compared to that of the end plate, due to

the direction in which the positive pressure gradients from the corresponding components can combine.

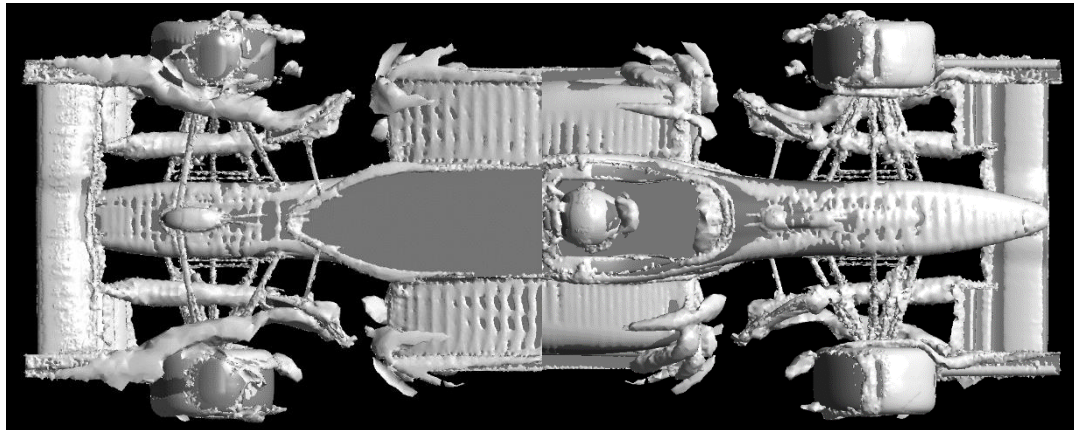


Figure 44 - SBD F399 Vortex Structures

These vortex structures are visualised in *Figure 44*. The inboard vortex structures are typically known as the Y250 vortices. The reason for this is with regard to the location in which they exist and are a highly used structure within Formula 1 aerodynamics. Due to the rotation of the Y250, a strong downwash effect is generated that can be utilised to guide high velocity flows to the floor section of the car (Nakagawa, et al., 2016).

The vortex generated from the main plane end plates can also be observed circulating around the inner shoulder of the wheel. As the vortex passes the rear face of the wheel, it is pushed inwards and begins to interact with the barge boards which subsequently guide the vortex outwards generating an outwash effect. This vortex, however, can later be seen colliding with the leading edge on the side of the venturi tunnel and collapsing after interacting with the Y250. This occurrence is also illustrated in *Figure 44*, with the impact being represented as a high pressure zone in *Figure 35*. The collapsing (bursting) of the vortex can produce aerodynamic instability as it is affecting the flow field with the random shedding of structures, resulting in abrupt changes of downforce and drag, and an aerodynamically unstable vehicle (Moet, et al., 2004).

While vortex generation can be a useful tool if utilised correctly, it can also result in catastrophic implications, resulting in the vehicle not working aerodynamically efficiently (Marchman, 1973).

Typical vortex breakdown results from a loss of energy within the structure, causing the rotation of the vortex to slow due to a growth of stagnation on the vortex axis, rather than the presented collision illustrated in *Figure 44* (Sidney, 1978).

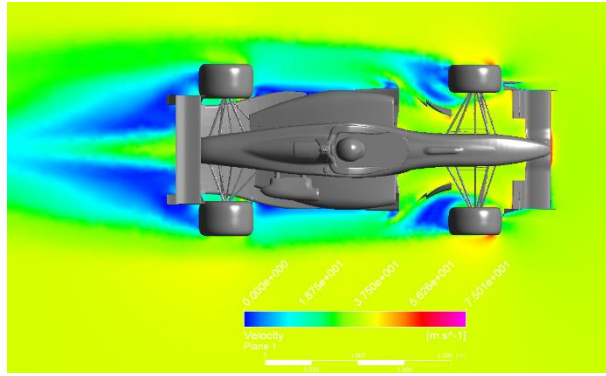


Figure 45 - SBD F399 Outwash (Velocity)

Notwithstanding this, *Figure 45*, illustrates that outwash is being generated (areas of asymmetry are due to the way in which the CAD model was constructed), although a marginal amount of the wake is still interacting with the floor just in front of the rear wheels, while the majority is dissipated within the generated outwash from the barge boards. The primary vortex can also be observed interacting on the outer lower edge of the barge boards. Early onset separation on the inner shoulder of both wheels is also apparent. This likely results from the interaction of the primary vortex colliding with the front face of the wheel and depleting energy on its surface, as shearing forces collide. This subsequently results in large wake formations being present inboard and aft of the wheel.

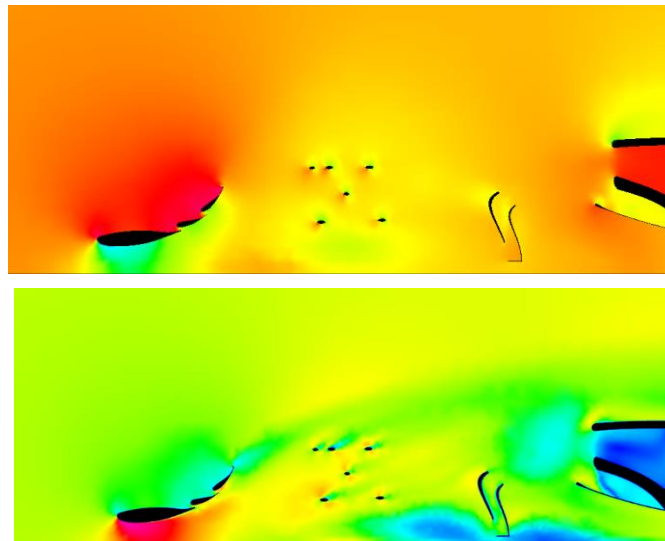


Figure 46 - Front Wing Flow Field (Top – Static Pressure, Bottom - Velocity)

Figure 46, presents the pressure and velocity fields of the generated upwash from the front wing, on a plane located 0.4m outboard of the central axis of the car. The results indicate an impact of trailing edge separation located on the third element. This could be as a result of the close proximity of the front wheel creating a stronger adverse pressure gradient due to the stagnation and potential downwash on its front face.

3.3.0 Baseline Isolated Wing Parameter Sweeps

To fully understand the function of the front wing, a series of studies were conducted with the full wing assembly in isolation. This isolates interference from other bodies and saves computational time.

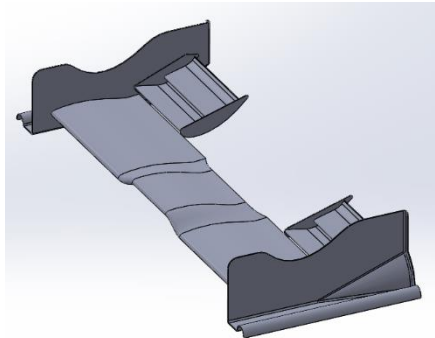


Figure 47 - Baseline Front Wing CAD Assembly

A full breakdown of isolated baseline front wing results can be viewed in chapter 8.0.0 Appendix B.

3.3.1 Secondary Elements AoA

The investigation commenced with the baseline AoA of the secondary elements. On the original wing assembly, only the second and third elements can be adjusted (at the same rate), and each angle adjustment is defined by a series of set angle positions, illustrated in *Figure 48*.



Figure 48 - F399 Front Wing Flap Adjustment

Figure 48, indicates fourteen angle adjustments, with the lowest angle being $\alpha = 15.42^\circ$ and the greatest angle being $\alpha = 20.42^\circ$.

Each case study was constructed using a translational ground plane at 40m/s with the relative freestream velocity, with an H/c of 0.24. A symmetry plane was also utilised for computational ease, although Toet states that caution should be exercised, as symmetry planes can produce irregularities in the convergence criterion when using RANS models (Toet, 2019).

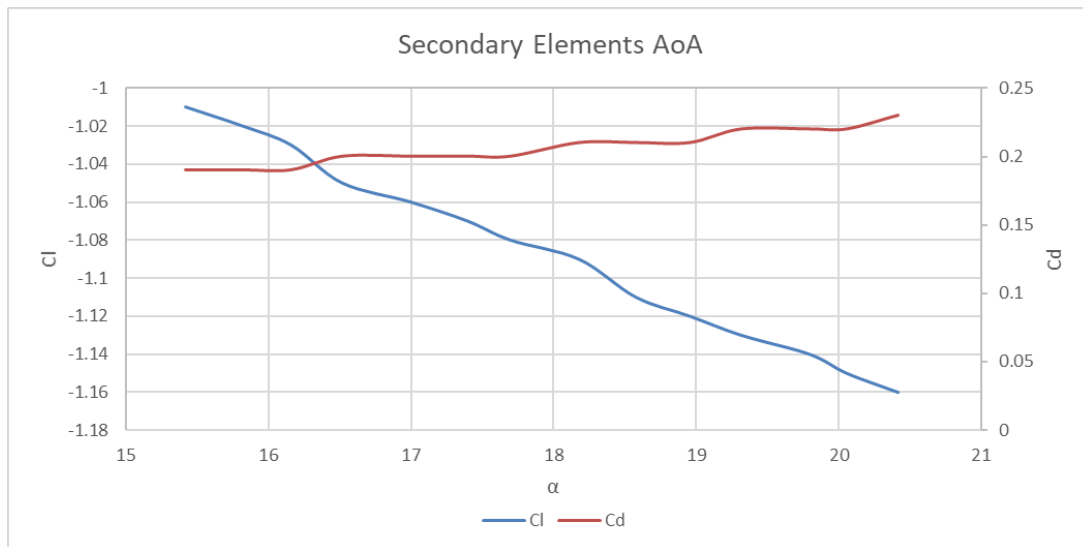


Figure 49 - Cl and Cd Values Taken from Baseline AoA

Figure 49, illustrates the typical characteristics of Cl and Cd when α is increased. This study was not taken at the point when the wing effectively stalls, as the purpose was solely to investigate the baseline adjustment of the wing flaps. It can be observed that a constant down trend in negative lift is achieved, however the drag component increases in apparent stepped increments. This has led to a stepped configuration within the Cl/Cd, although it does show a maximum efficiency of 5.42 at $\alpha = 16.16^\circ$. Minimum Cl and Cd values were found at the lowest adjustment of the flap, and maximum values were produced at the highest adjustment.

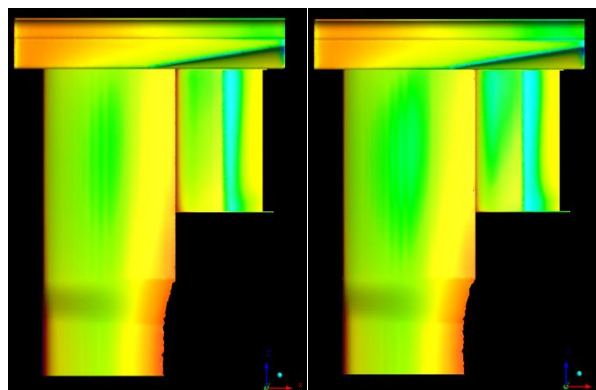


Figure 50 – Surface Static Pressure Contours (Left – $\alpha = 15.42^\circ$, Right – $\alpha = 20.42^\circ$)

With an increase in AoA, the pressure differential from suction surface to top surface should increase (geometry depending), with a redistribution of pressure. Figure 50, illustrates this typical trend. It can be observed that when comparing minimum α and maximum α , increasing the AoA also induces a greater negative pressure. This is particularly apparent on the second element, which seems to have gained the greatest pressure differential when examining Figure 51.

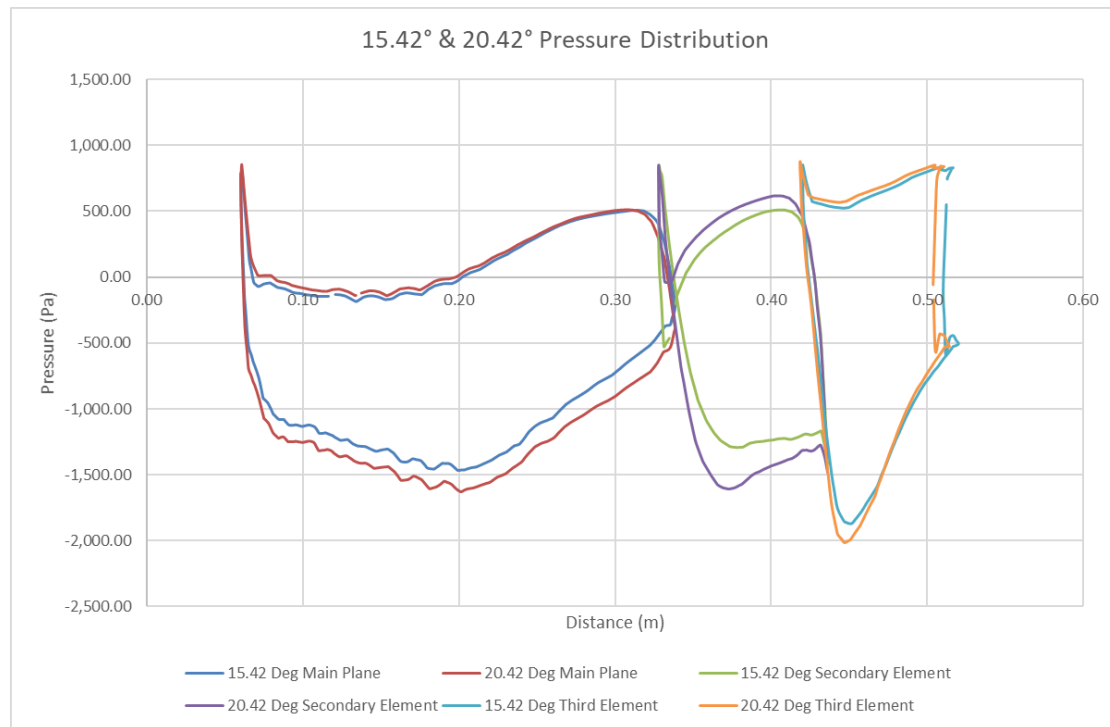


Figure 51 - Minimum and Maximum AoA Pressure Distribution

The increase in flap angle creates a greater expansion volume, therefore allowing a greater mass flow to be induced on the underside of the wing, therefore lowering the pressure across the lower surface of each element. As previously referenced, the secondary element is subjected to the greatest increase in pressure differential. The reason for this is that as the flap angle is increased, the suction side of the secondary flap begins to follow the flow trajectory path of the main element. At its lowest AoA, the flap elements create a blockage against the main plane upwash, which is represented as an area of high pressure located at the main planes trailing edge (*Figure 52*). A lesser negative pressure is, therefore, apparent on the secondary element, which consequently increases the strength of an adverse pressure gradient.

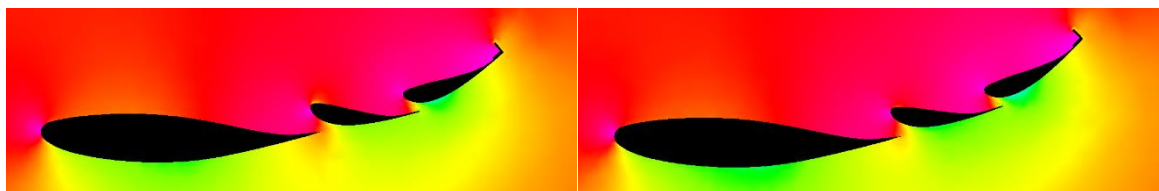


Figure 52 - Pressure (Left – $\alpha = 15.42^\circ$, Right – $\alpha = 20.42^\circ$)

Figure 52, illustrates pressure contours presented on a plane 0.5m from the central axis of the wing. In addition to an increase in pressure differential on the second element, the distribution of that pressure has also increased, with an increase of negative pressure located towards its leading edge. With this increase a greater area is now being utilised to create the increase in vertical force.

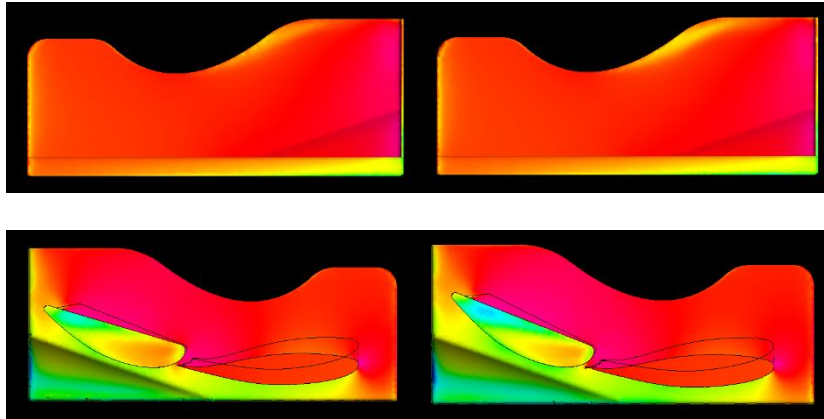


Figure 53 - End Plate Surface Static Pressures (Left – $\alpha = 15.42^\circ$, Right – $\alpha = 20.42^\circ$)

An increase in wing surface pressure differential can also increase the possibility of further induced drag. *Figure 49*, illustrates, that a rise in α will increase the value of C_d in the form of pressure drag. However, *Figure 53*, illustrates areas of interest located on the end plates, created as a result of the formation of vortices. Whilst vortices can be useful, as previously discussed, they can induce a significant amount of induced drag force. It can be observed that areas of low pressure exist on the outboard end plate, particularly on the top edge. This is due to the positive pressure on the top surface of the wing bleeding into the atmospheric pressure, and then being further excited by the exposed negative pressure at the trailing edge. The same occurrence can be observed on the smaller inboard end plate, which has led to an increase in swirling strength of the Y250 vortex. It can also be observed that the surface pressure within the lateral expansion zone has also further decreased, leading to a much stronger primary vortex. The size of the increase of the primary and Y250 vortices can be observed in *Figure 54*.

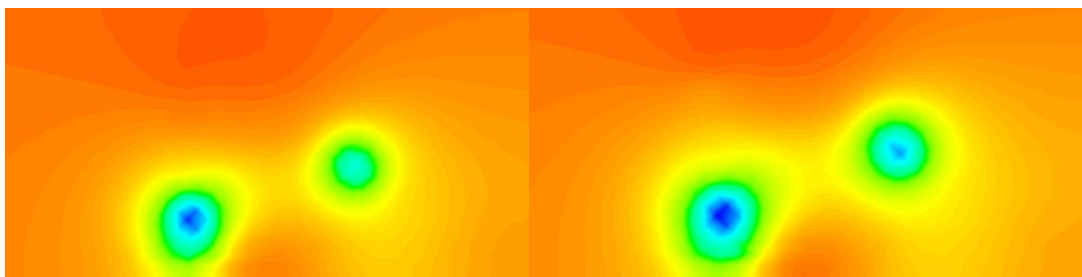


Figure 54 – Static Pressure and Size Comparison of the Primary and Y250 Vortices

The diameter increase of the vortices will dictate a larger diameter of low velocity in its centre. This phenomenon is proposed by Rankine, who proposed the Rankine theory, where a vortex has a rigidly rotating core (constant angular velocity), and as the radius increases away from the core, free motion is allowed (Katopodes, 2019), however the flow outside of the core is considered not to have any rotation.

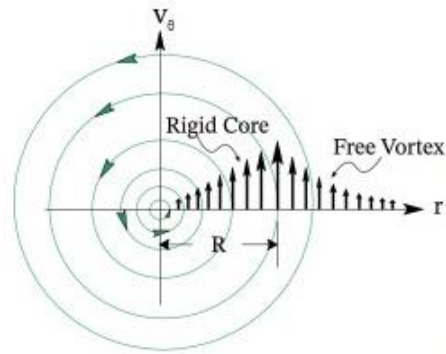


Figure 55 - Vortex Velocity Profile (Katopodes, 2019)

Figure 55, illustrates the Rankine theory vortex velocity profile. It can be observed that for distance $r < R$, the core is rigid and $r > R$, the model returns into a free vortex. This prevents the centre of rotation having an infinite velocity and allows the velocity to decay at greater distances. Due to its nature, the velocity profile can be presented as a sinusoidal wave form, although due to the viscosity of air, the sharp changes in velocity are smoothed, as presented in Figure 56 (Agrawal & Prasad, 2003).

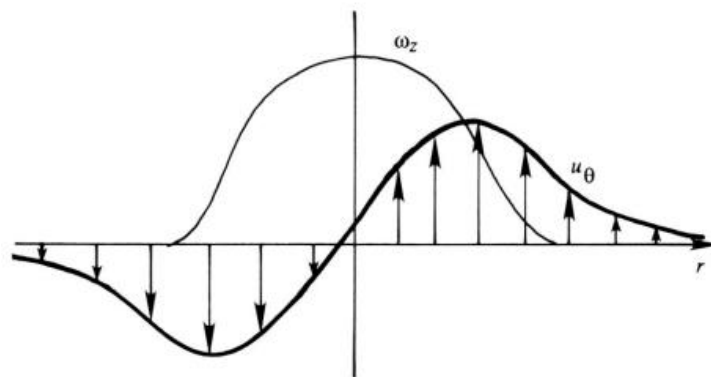


Figure 56 - Real Vortex Velocity Profile (Agrawal & Prasad, 2003)

3.3.2 Whole Assembly AoA

It has already been discussed that changing the incline angle of a wing alters the static pressure distribution across its surfaces, and this was evident in the previous chapter.

This case study involves changing the entire wing assembly angle, however, the end plate position is to be maintained, together with the baseline ground proximity. The wing flaps were left at baseline α relative to the main plane.

1° increments were utilised from 0° to 10°, these cases being constructed using the same boundary conditions as the previous study.

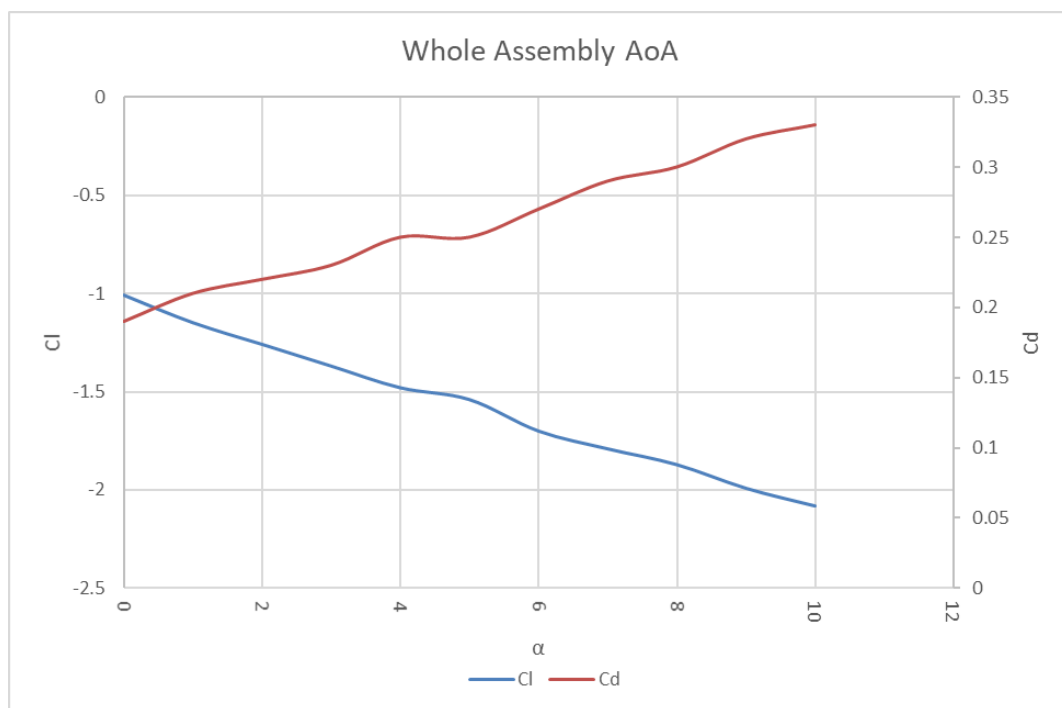


Figure 57 - C_L and C_D (Whole Assembly AoA)

Figure 57, illustrates the lift and drag coefficients per α . It can be observed that both $-C_L$ and C_D rise almost linearly, as α is increased. This results in minimum coefficient values at an incline angle of 0° and maximum values at 10°.

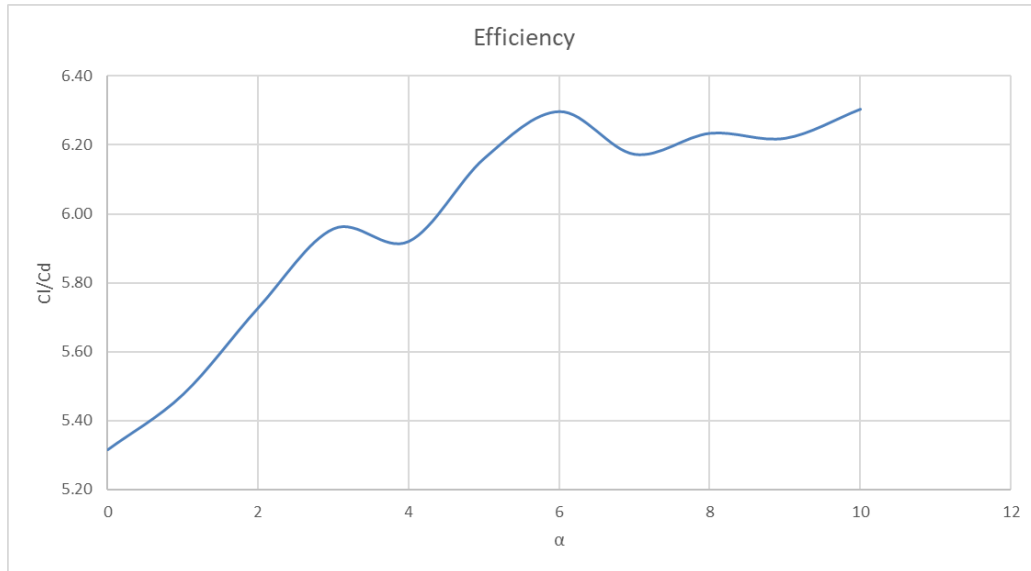


Figure 58 - Cl/Cd (Whole Assembly AoA)

Peak efficiency is achieved at 6° , which corresponds with a 0.16 increase in $-Cl$ and only a 0.02 rise in Cd , as observed in Figure 57.

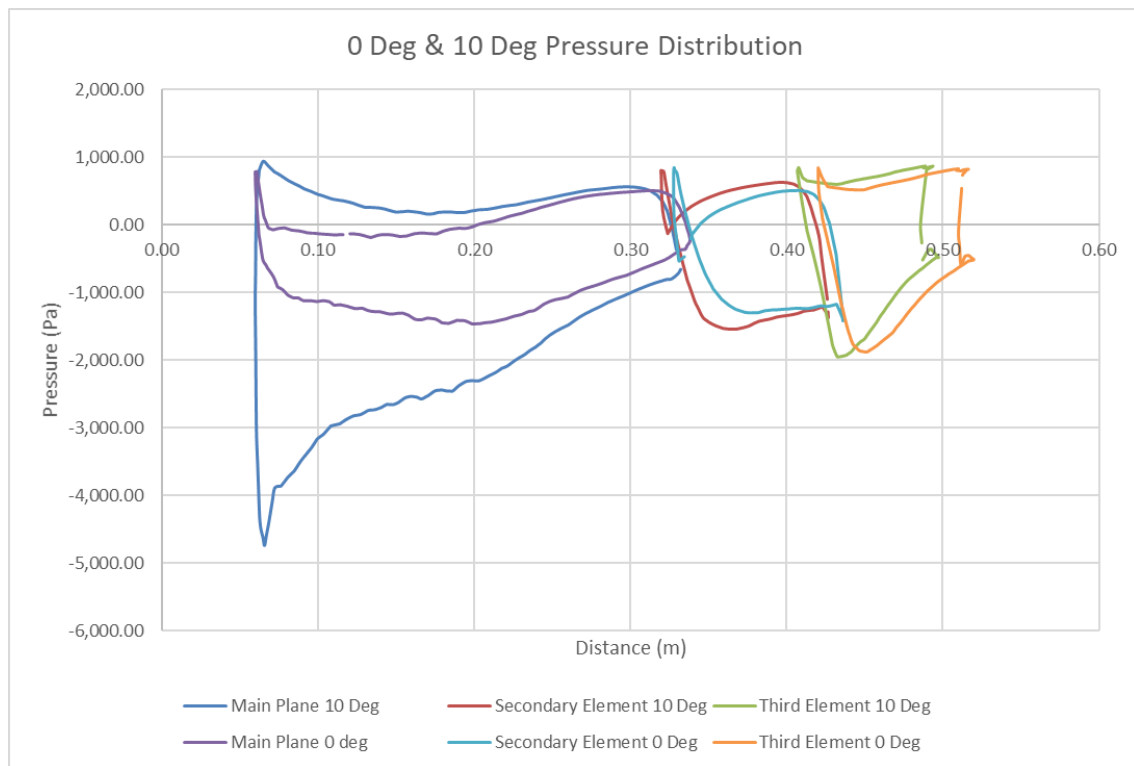


Figure 59 - Maximum and Minimum AoA Pressure Distribution (Whole Assembly)

Figure 59, illustrates the change in static pressure distribution as the AoA of the wing assembly is increased. The main element succumbs to the majority of the pressure change and distribution on its suction surface. It can be observed that at 0° , a peak negative pressure of -1460Pa is located at a distance of 0.2m from the axis origin. As the expansion volume increases, the peak negative pressure drops further to -4740Pa at 0.065m behind the axis origin. This illustrates that the peak suction force

will be redistributed with an angle change. The change in α does not result in flow separation on the leading edge. This can be observed in *Figure 60*, as the static pressure returns to atmospheric pressure, although a sharp gradient is present within the pressure distribution, which is as a result of a highly concentrated area of negative pressure.

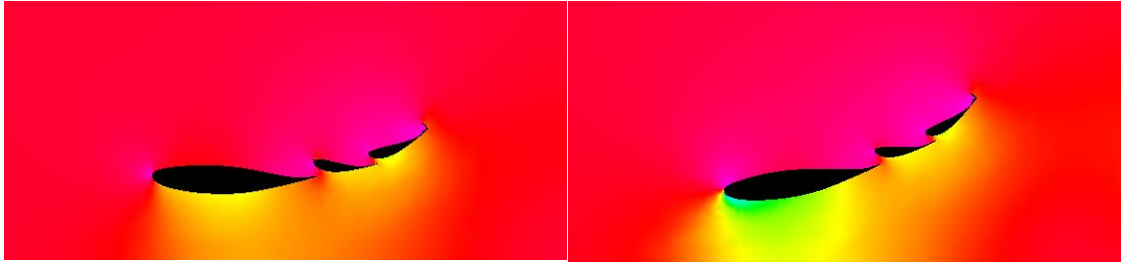


Figure 60 - Pressure (Left - $\alpha = 0^\circ$, Right - $\alpha = 10^\circ$)

Achieving large incident angles could result in an unpredictability of the wing assembly due to the highly concentrated area of peak negative pressure, shifting the CoP towards its leading edge. In a dynamic situation, this could result in vehicle handling issues, especially at high velocities and applied lateral accelerations.

Changes in vortex formation are apparent in *Figure 61*, however, these are similar to the changes observed and referenced in the previous chapter, although a greater magnitude of swirling strength is achieved within the Y250 vortex. It can be observed, however, that the primary vortex has decreased in length, although its diameter has increased.

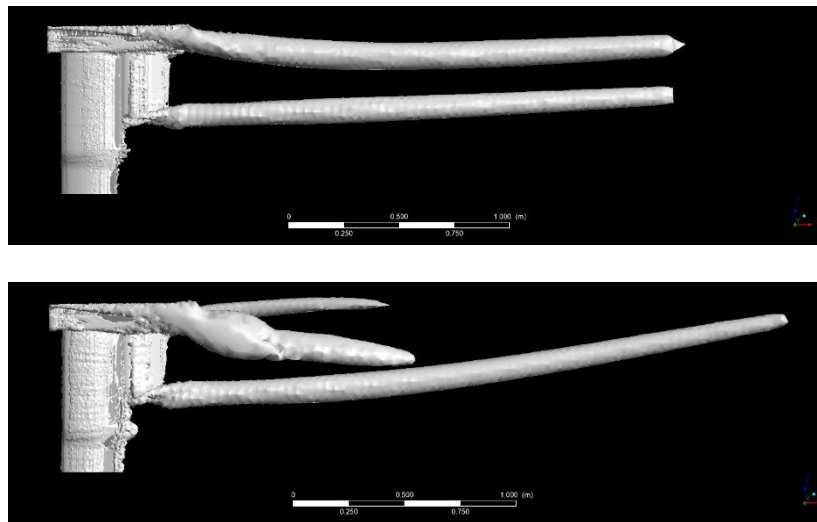


Figure 61 - Whole Wing Assembly Vortex Structure (Top - $\alpha = 0^\circ$, Bottom - $\alpha = 10^\circ$)

Figure 62, illustrates the static pressures present within the vortex structures on a plane located 0.8m behind the axis origin. Examining the pressures reveals that the larger diameter primary vortex yields an increase in pressure within the vortex core. This suggests that the vortex could be in a state of stagnation, resulting in a vortex breakdown, thus achieving a shorter length and a reduced swirling strength.

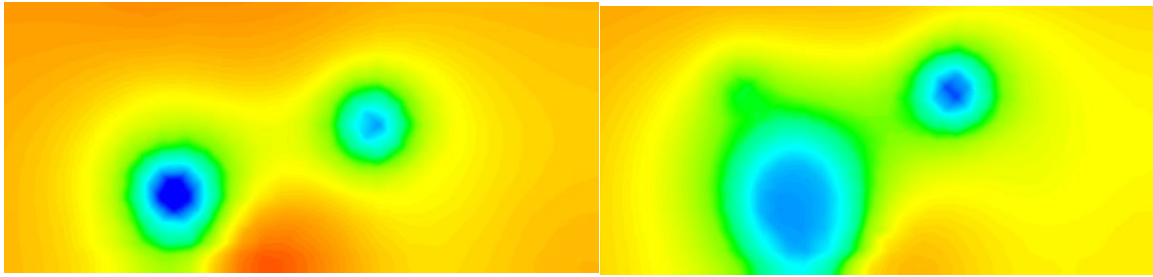


Figure 62 - Static Pressures located within the Vortical Structures (Left – $\alpha = 0^\circ$, Right – $\alpha = 10^\circ$)

3.3.3 Proximity to Ground

McBeath describes the phenomenon of ground effect on front wings, stating that the front wing C_l may increase by as much as 50 – 60% as the ground proximity is reduced from 6in to 2in. However, there is a dependency on the profile geometry to be able to produce such an increase (McBeath, 2001).

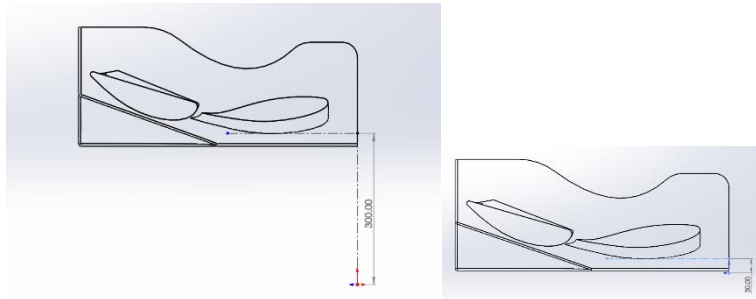


Figure 63 - Wing Ground Proximity

The front wing geometry was simulated at an H/c of 0.64 (300mm), with height reduction increments of 10mm, with a final H/c of 0.06 (30mm) as illustrated in Figure 63. The initial boundary conditions are maintained relative to the previous studies. Due to the end plate under hang, the ground proximity could not be reduced below an H/c of 0.06, therefore boundary layer interaction could not be observed within the results produced. The minimum flap angle of 15.42° was also implemented.

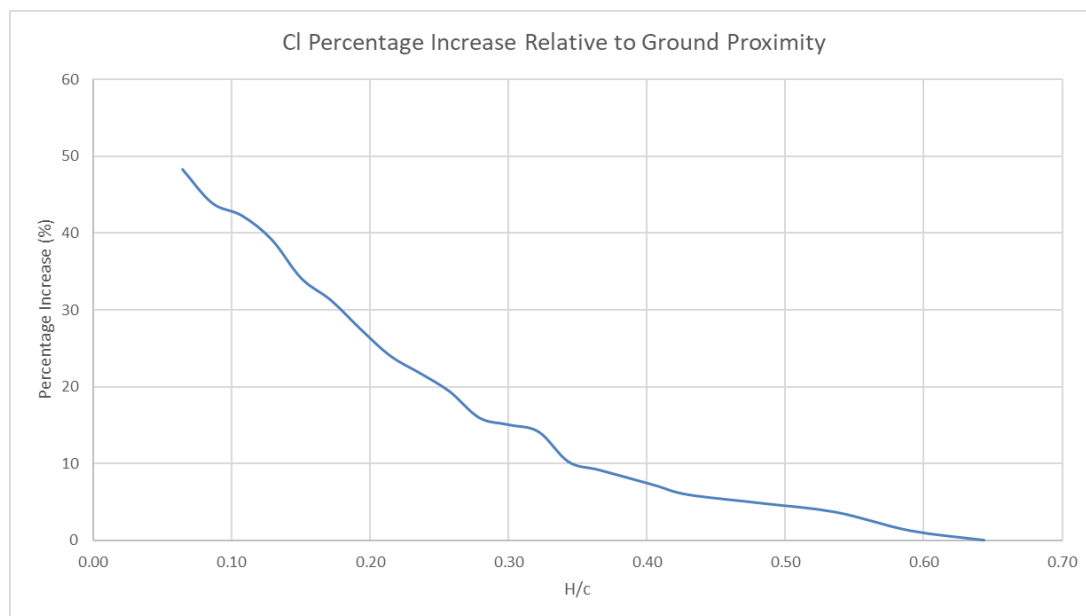


Figure 64 - C_l Percentage Increase Relative to Ground Proximity

Figure 64, illustrates an exponential percentage increase in C_l value as ground proximity is reduced. The increase in C_l is as a result of the reduction in throat section between the suction side of the wing and ground plane, which accelerates the air and decreases the local pressure (Figure 65), thus a greater vertical force is produced. Katz and Zhang et al, both express that the increase in $-C_l$ does not come freely, with both displaying an increase in C_d (Figure 67). However, this increase is not as

severe as increasing the assembly angle (*Figure 57*), thus presenting a much greater efficiency (*Figure 66*). This is because the main contributor to the drag increase is skin friction resulting from the increase in velocity beneath the wing, whereas increasing flap angle introduces additional pressure drag, which often outweighs the effects of skin friction drag (Katz, 2006) (Zhang & Zerihan, 2003).

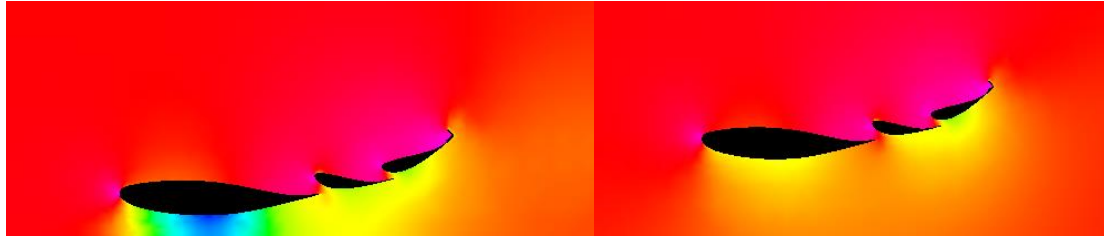


Figure 65 - Static Pressure (Left - H/c 0.06, Right - H/c 0.64)

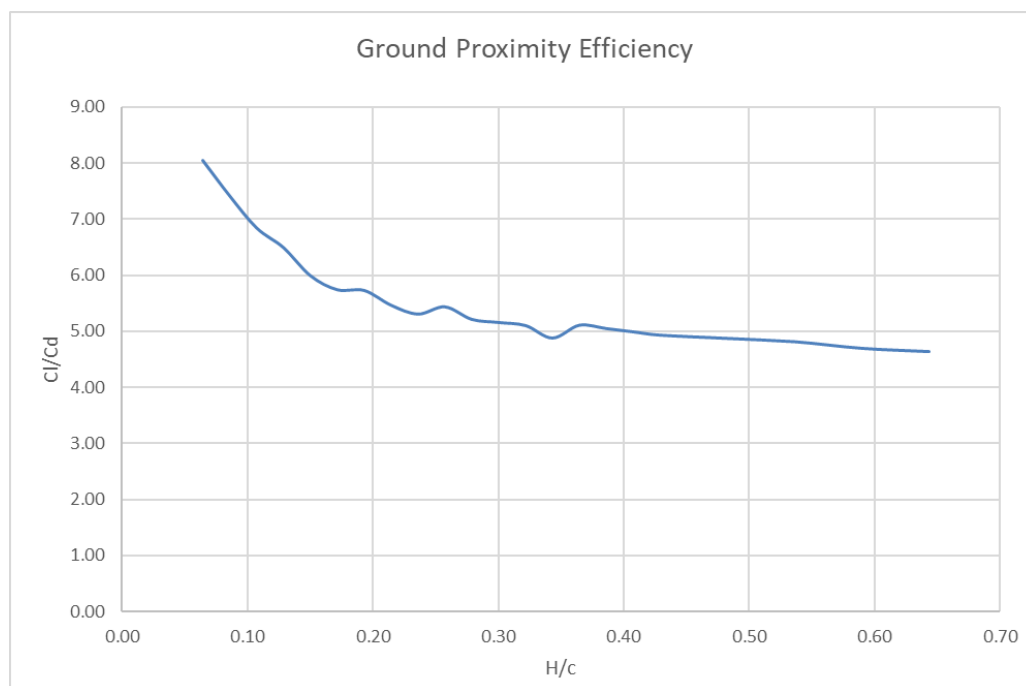


Figure 66 - Efficiency Relative to Ground Proximity

As ground proximity is reduced, the aerodynamic efficiency of the front wing increases exponentially, with the lowest H/c of 0.06 resulting in the greatest Cl/Cd of 80.5%. This results in a 21% increase in efficiency compared to that observed by the whole assembly AoA and 33% increase from flap AoA.

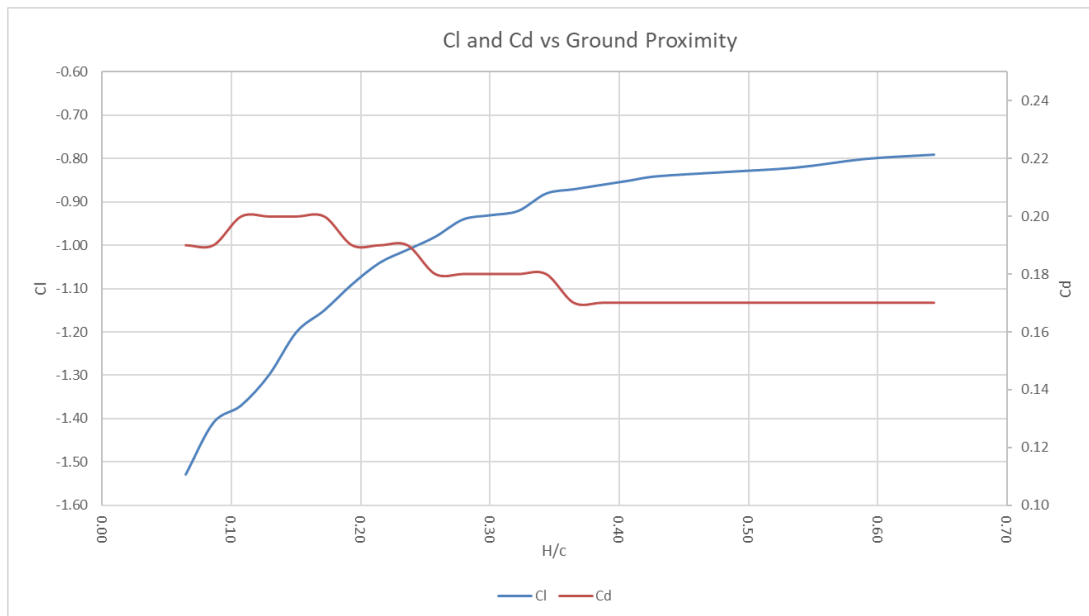


Figure 67 - C_l and C_d as a Function of Ground Proximity

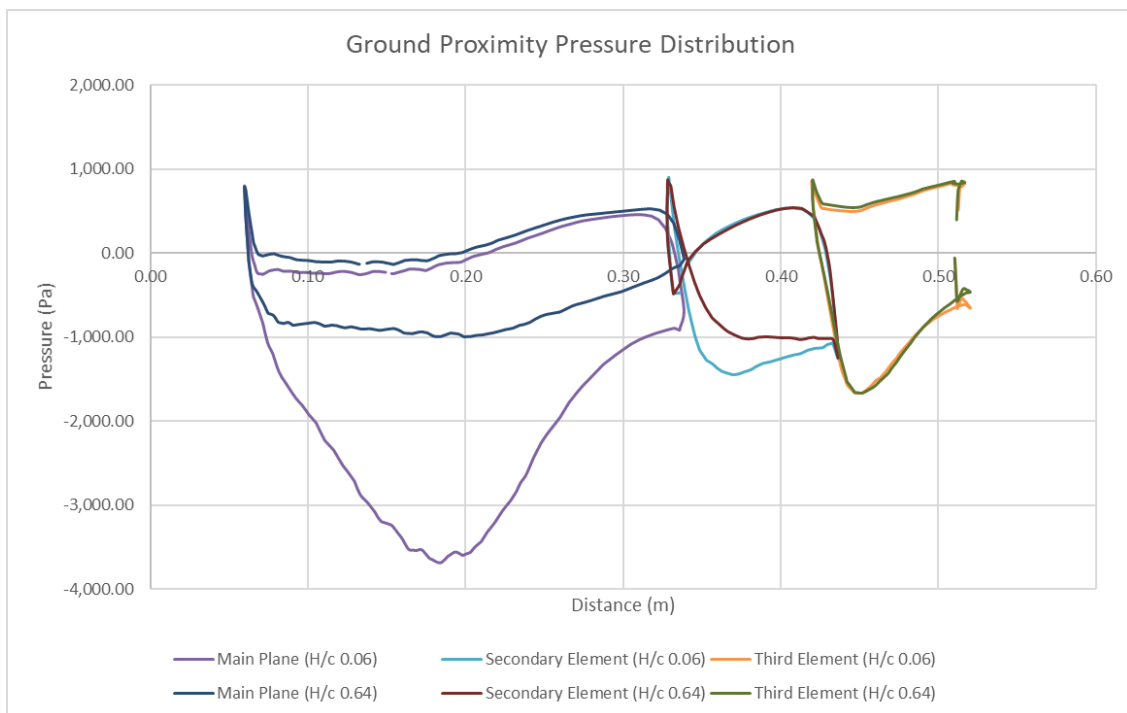


Figure 68 - Pressure Distribution Relative to Ground Proximity

As previously described, it is noticeable that as the ground proximity is reduced from H/c 0.64 to 0.06, the peak negative pressure on the main plane increases significantly by 2686.22Pa (Figure 68). The magnitude of negative pressure is also distributed over the entire main plane, providing further suction force. The secondary element also produces an increase in negative pressure, however the third element experiences little to no difference. Traditionally, each element of the wing should produce a reduction in negative pressure in order to maintain a smooth transition back to atmospheric pressure,

whilst reintroducing energy to help maintain the boundary layer within the adverse pressure gradient (Bansal & Selig, 2016).

Figure 69, illustrates the peak negative pressures for each element at $H/c = 0.64$ and 0.06 . It can be observed that at the greater height, the peak negative pressure actually increases, causing a downward trend. This results in a 76% bias in rear loading at the greater height compared to 57% rear bias at the lower height.

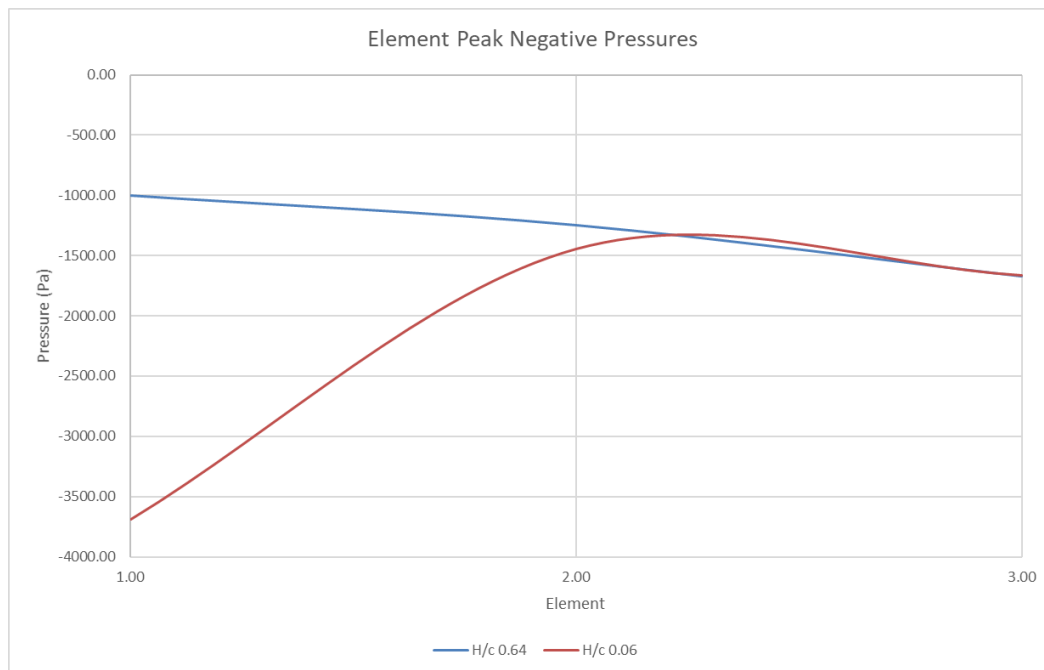


Figure 69 – Peak Negative Pressures for Each Element at $H/c = 0.64$ and 0.06

Such bias results from the slot gap size between the second and third elements being reduced compared to the gap between the main plane and second element. McBeath explains that this is a common occurrence, however it does somewhat depend on the profiles utilised within the multi element design (McBeath, 2001).

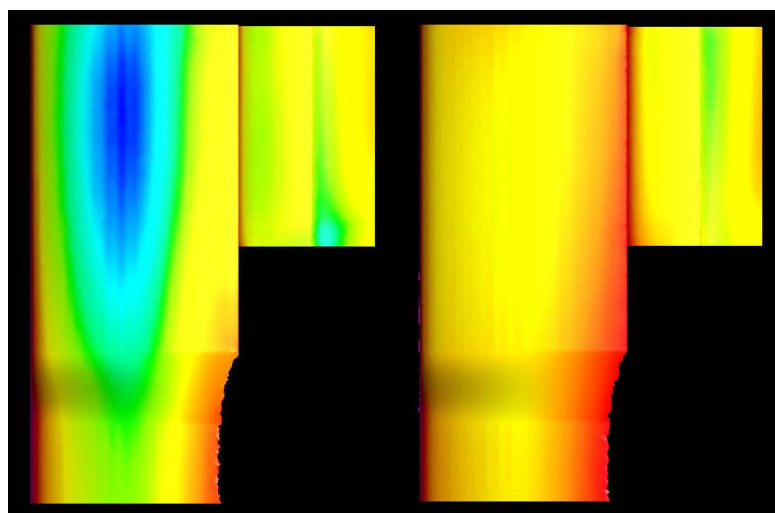


Figure 70 - Surface Pressure Contours (Left - $H/c = 0.06$, Right - $H/c = 0.64$)

The static pressure contours illustrated in *Figure 70* indicate a clear increase in negative pressure across the entire planar area of the main plane, at $H/c = 0.06$. Observing the third element shows a small area of considerable negative pressure. This is as a result of the increased pressure differential between the upper and lower surfaces, generating a much stronger Y250 vortex (*Figure 71*), as discussed in the previous chapter.

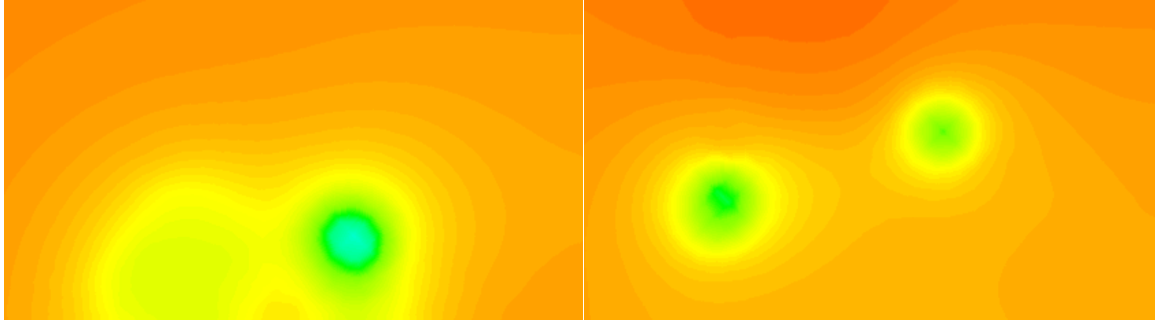


Figure 71 – Vortex Pressure (Left – H/c 0.06, Right – H/c 0.64)

The pressure contours illustrated in *Figure 71* are projected on a plane 0.7m rear of the axis origin, normal to the freestream. In addition to the increase in size of the Y250 vortex, it can also be observed that the primary vortex is unable to suitably assemble due to the end plate ground proximity.

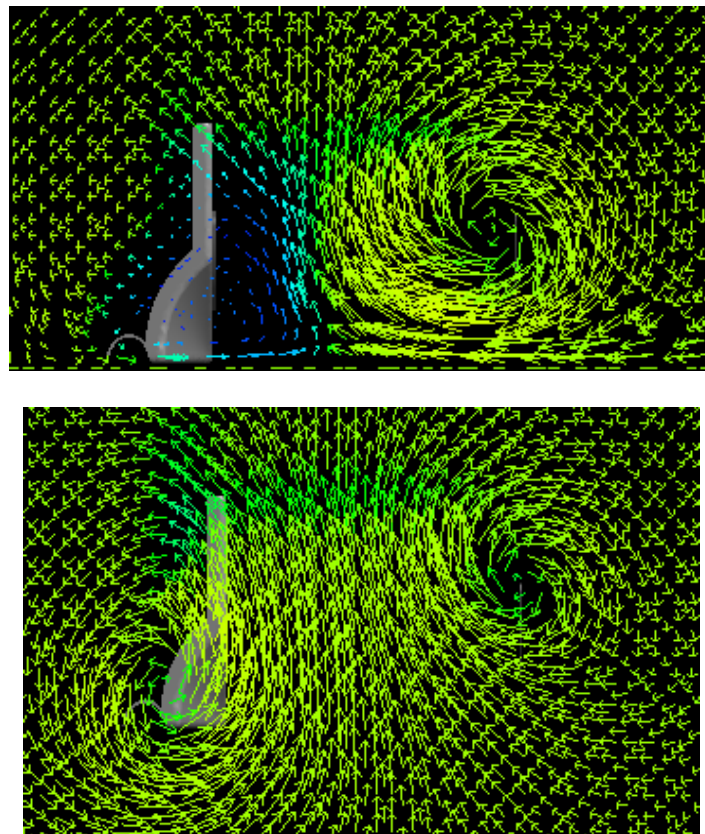


Figure 72 - Velocity Vector Plane (0.7m Rear of the Axis Origin)(Top – H/c 0.06, Bottom – H/c 0.64)

Figure 72, clearly demonstrates the blockage that has formed between the foot plate and the ground plane at $H/c = 0.06$. It has been discussed, previously, that the primary vortex could be utilised as a

useful tool for controlling the near-wake structure of the front wheels, therefore it is suggested that the end plate ground proximity is increased. An H/c of 0.06 contravenes the outlined technical regulations, therefore the ground proximity needs to be increased for the purposes of this design development.

3.3.4 Slot Gap Height

Both the second and third elements were elevated in 5mm increments, starting from 0mm ($H/c = 0$) tangential to the main plane trailing edge, and finishing at 70mm ($H/c = 0.15$) (Figure 73).

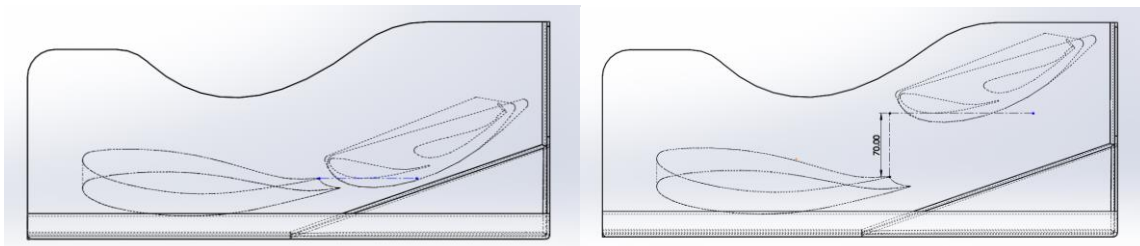


Figure 73 - Secondary Element Elevation

It has already been discussed how important slot gap sizing can be in order to achieve greater profile camber angles and reduce the potential for trailing edge separation, while energising the entire assembly (McBeath, 2001) (Corfield, et al., 2018). If the slot gap is too small, a choking effect will develop on the suction side of the secondary elements, which reduces the potential for boundary layer energisation and increases induced drag force. If the gap is too large, the secondary elements will no longer drive the upwash that is generated from the main plane, and the assembly will effectively become two isolated devices. The Formula 1 drag reduction system is designed using a similar principle, however, rather than altering the height uniformly, the height of the leading edge is altered by reducing the secondary elements AoA, thus stalling the wing and reducing drag and downforce (Wordley, et al., 2014) (Collins, 2018).

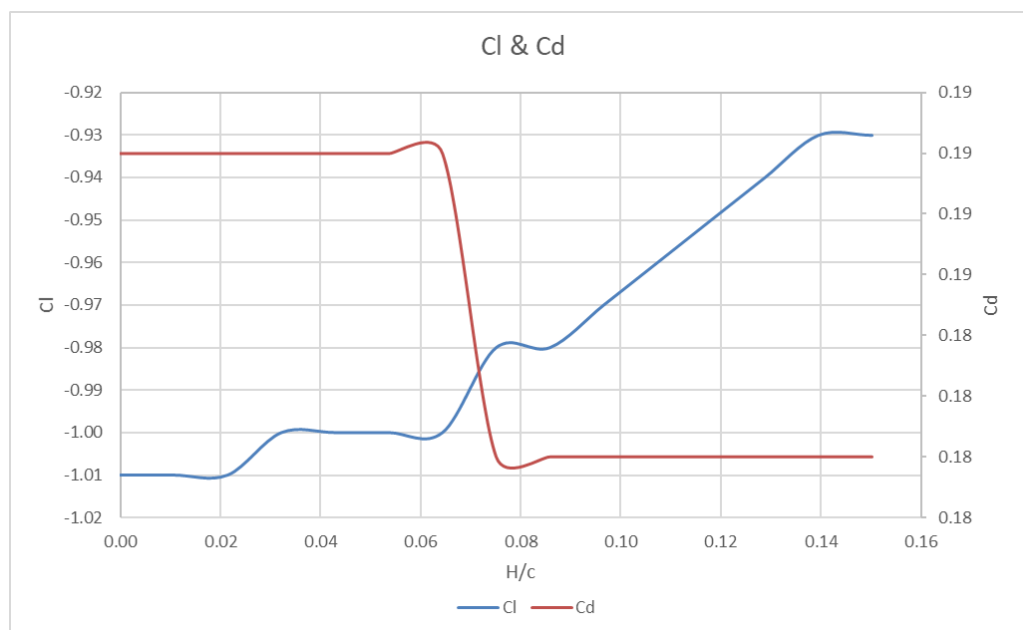


Figure 74 - Slot Gap Height Cl and Cd

Figure 74, illustrates that a severe decrease in Cd is apparent at a slot gap size $H/c = 0.08$ and consequently, a reduction in $-Cl$ at the same point, although this is highlighted in Figure 76, with a sharp increase in efficiency, denoting a greater drag reduction with respect to negative lift. At further

height increments, the drag remains consistent, however, negative lift presents an almost linear reduction until $H/c = 0.14$, and then flattens.

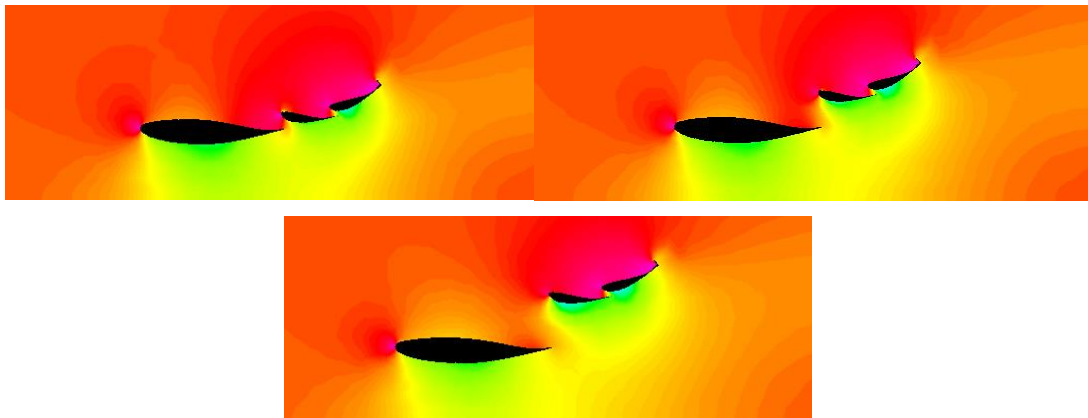


Figure 75 - Pressure Contours (Top Left - $H/c = 0$, Top Right - $H/c = 0.08$, Bottom - $H/c = 0.15$)

The sudden decrease in negative lift can be associated with a reduction in main plane interaction. Figure 75, illustrates that a bubble of high pressure is beginning to develop between the two suction sides of the main plane and flaps, and generating a split in negative pressure, which increases in size as the slot gap increases.

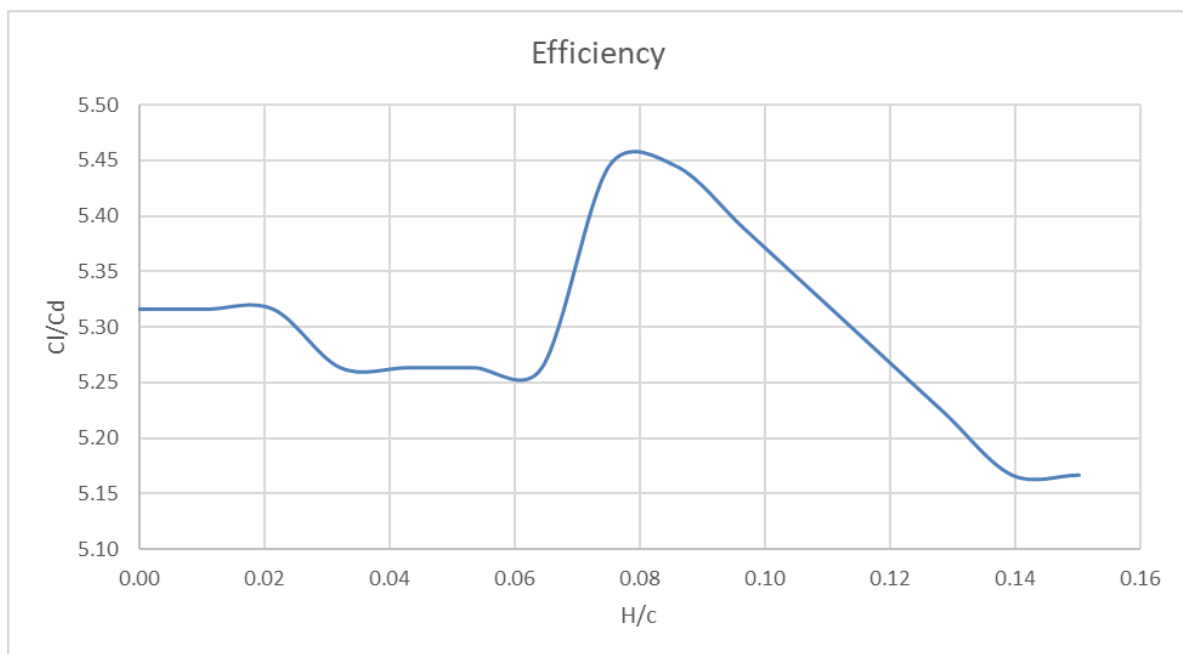


Figure 76 - Slot Gap Height Efficiency

The increase in slot gap height has allowed a greater mass flow of freestream air with an increase in pressure due to the absence of the venturi effect between the main plane and second element.

The lack of pressure interaction between the flap elements and main plane is apparent when examining Figure 77. A reduction in pressure differential can be observed on the main plane as the secondary elements are further elevated, however, the second element begins to increase its pressure differential.

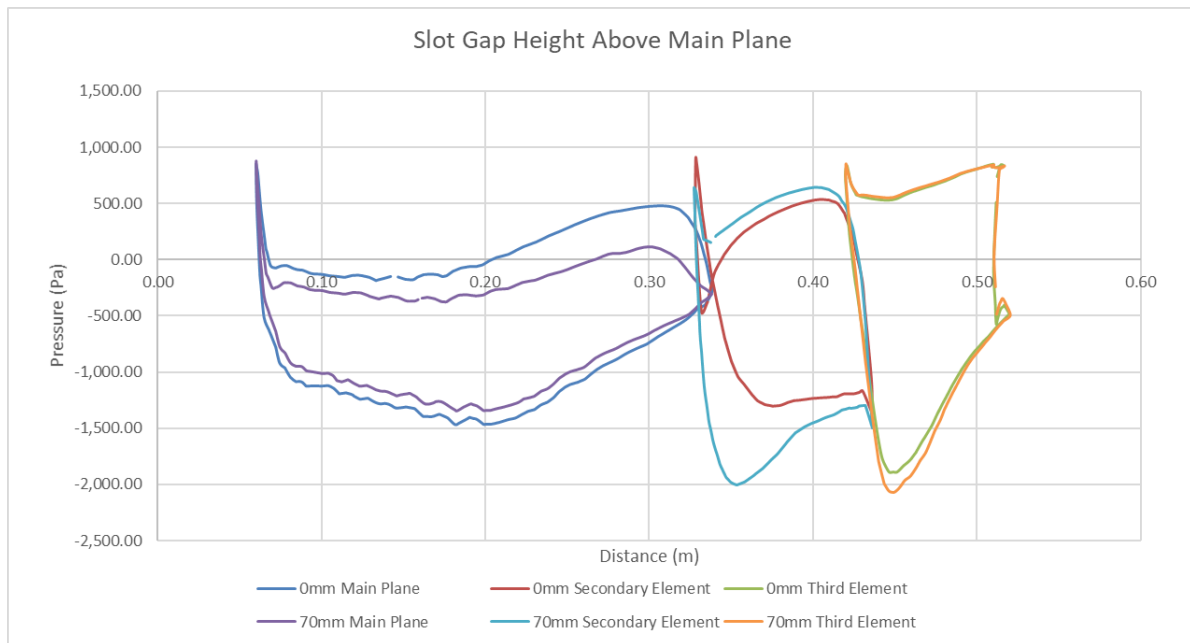


Figure 77 - Slot Gap Height Pressure Distribution

This increase is as a result of the onset air flow having an increase in energy as its velocity has not been depleted by interference from the main plane. The majority of the main plane pressure reduction is observed on the positive pressure side. A consequence of the slot gap increase is the second element stagnation pressure not bleeding onto to upper surface of the main plane. The conservative main plane profile, therefore, develops an increase in velocity and drop in pressure, thus reducing the differential in pressure.

The main plane is the largest working area of the wing assembly, therefore as the pressure differential is decreased across its planar area, the generated vertical force must also reduce, as force is relative to pressure and working area (*Equation 18*).

$$Force = P * A$$

Equation 18 - Force (N)

By integrating the total surface pressure across its planar area, an accurate representation of the generated force can be produced (van Garrel, et al., 2017).

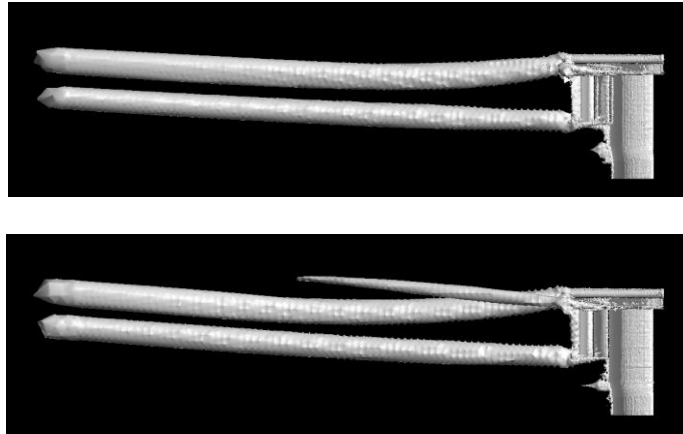


Figure 78 - Slot Gap Height Vortex Formation

As a consequence of increasing the slot gap height, the developed positive pressure on the flap elements is allowed to spill over the top edge of the end plate, as was observed with the increase of the AoA. This produces a vortex (*Figure 78*), although with a smaller rotation diameter compared to that of the Y250 and primary vortical structures.

3.3.5 Secondary Elements Positive Overhang

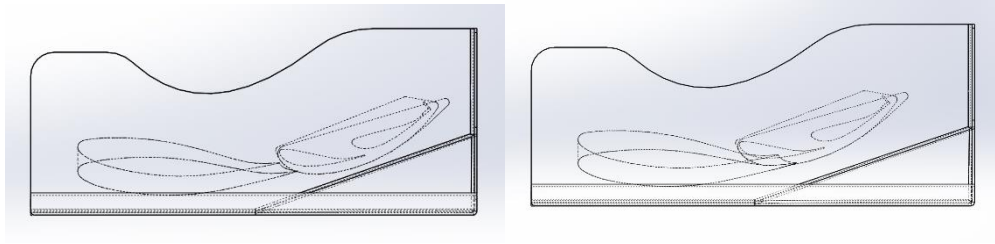


Figure 79 - Positive Overhang Positioning (Left - $S/c = 0$, Right - $S/c = 0.123$)

An increase in positive overhang was investigated. Moving the secondary elements over the main plane in 5mm increments, starting from 0mm ($S/c = 0$) and finishing at 50mm ($S/c = 0.123$). A minimum flap angle of 15.42° was maintained, with $H/c = 0.24$. Suitable boundary conditions were applied with a translational ground plane having a relative velocity to the free stream flow.

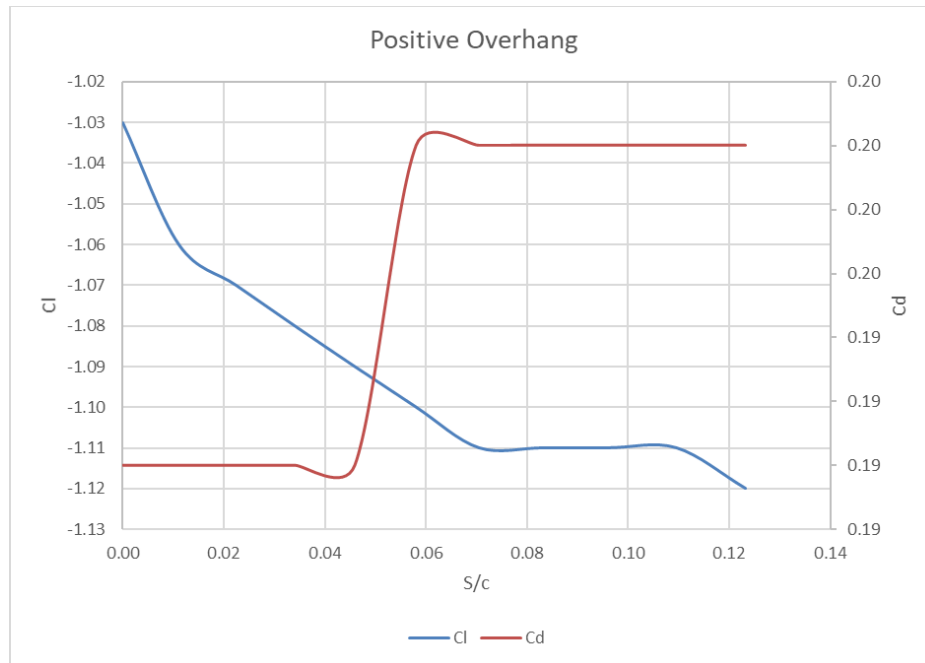


Figure 80 - Positive Overhang C_l and C_d

Figure 80, suggests that a gradual increase in negative C_l is apparent with an increase in S/c , however it is also evident that a spike in C_d exists when S/c of 0.058 is achieved. Unlike a bluff body, calculating the lift and drag forces for a wing involves the use of the planar area variable (Benson, 2014). With an increase of S/c , the effective planar area of the wing is reduced, resulting in a gradual decrease in downforce after a peak is reached at $S/c = 0.07$ (Figure 81).

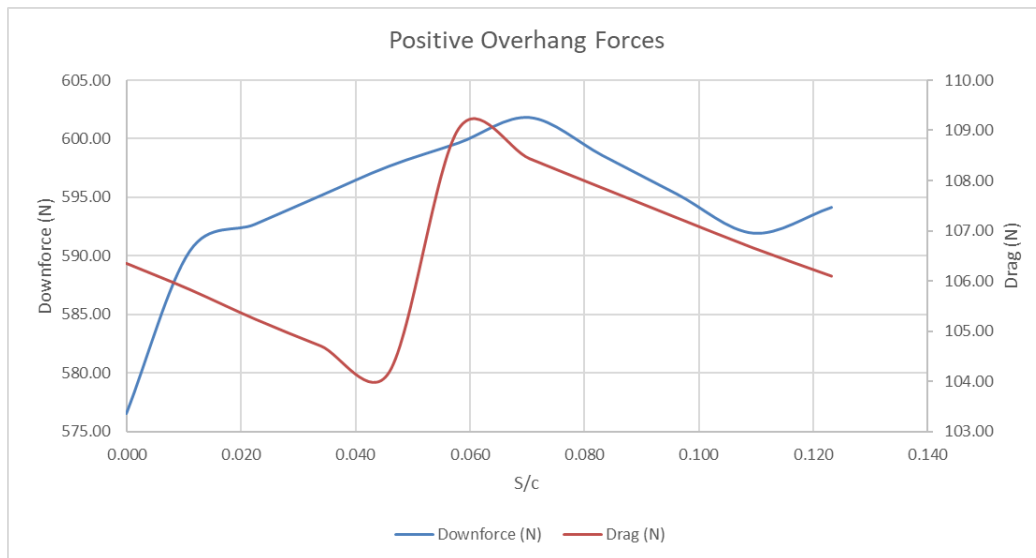


Figure 81 - Positive Overhang Forces

An initial reduction in overall drag force can be observed until an abrupt spike occurs at $S/c = 0.058$, achieving a peak of 109.03N. With the decreasing planar area, the subsequent drag force subsequently follows the same trend as the reducing downforce.

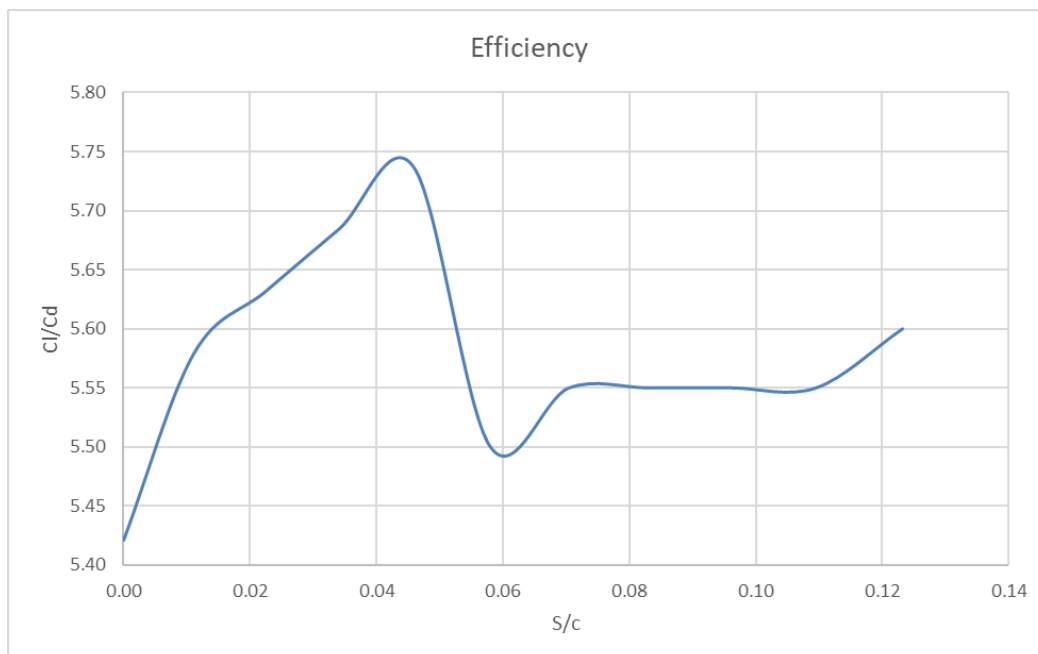


Figure 82 - Efficiency with Respect to S/c

Minimum efficiency is achieved with $S/c = 0$ ($Cl/Cd = 5.42$), as illustrated in Figure 82, with an almost linear increase until $S/c = 0.058$, where a dramatic reduction in efficiency can be observed as a result of the spike in Cd value. Thereafter a consistent value of efficiency is attained, which corresponds with the similar linear trends of Cl and Cd . Peak efficiency is reached at $S/c = 0.046$, with a value of 5.74, which is a 6% increase in efficiency over the baseline positioning, having achieved an 8.6% reduction in drag force with only a 2.2% decrease in downforce.

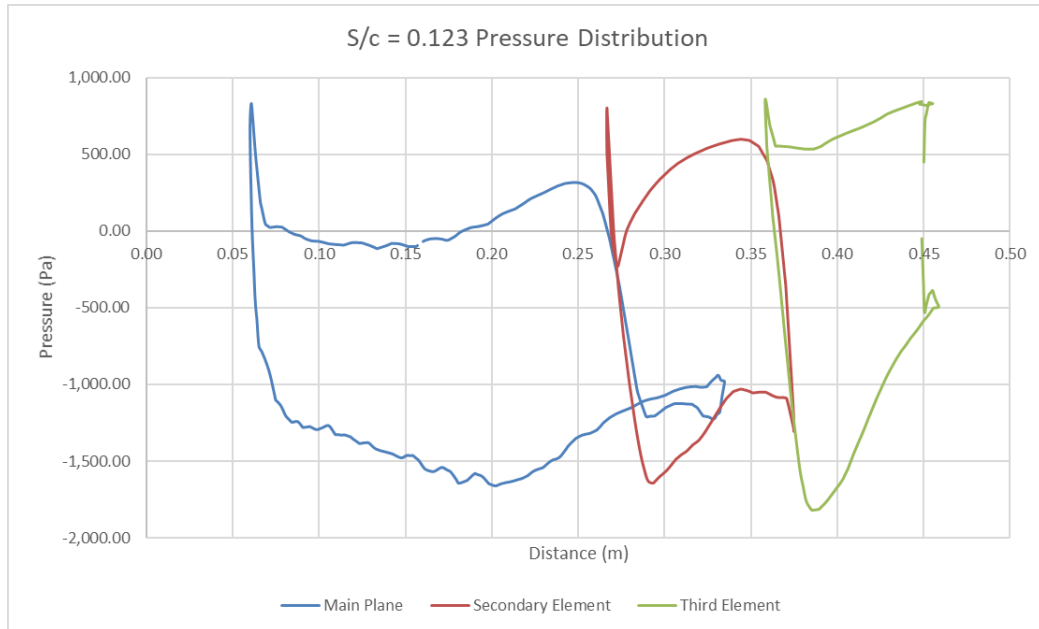


Figure 83 - $S/c = 0.123$ Pressure Distribution

The pressure distribution across every element for $S/c = 0.123$ can be observed in Figure 83. It can be observed that the increased positive overhang has an impact upon the trailing edge of the main plane, as interaction with the secondary flap occurs.

The effect of increasing the overhang results in an increased venturi effect, as a tunnel passage is developed. The consequences of this is the production of an increase in velocity over the upper surface of the main plane trailing edge and suction surface of the secondary flap, allowing a pressure drop to be developed in this region. This causes a cancelling effect between upper surface of the main plane and suction surface of the secondary element and reduces the working area, resulting in downforce reduction.

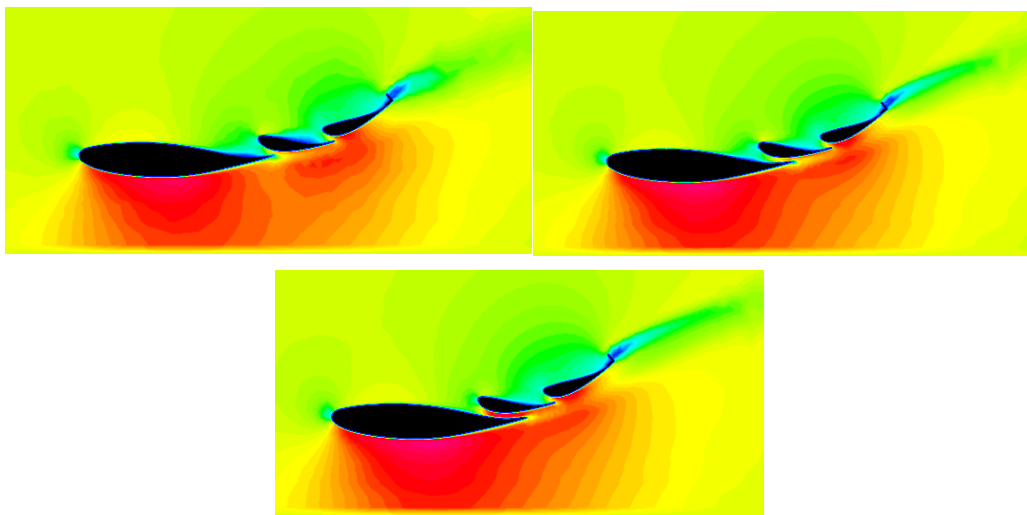


Figure 84 - Velocity Contours (Top Left - $S/c = 0$, Top Right - $S/c = 0.07$, Bottom - $S/c = 0.123$)

Figure 84, illustrates velocity contours projected onto a plane 0.5m outboard of the wings centreline. It can be observed that at $S/c = 0$, there are two strong velocity profiles located under the suction

surface. An increase in S/c results in these velocity profiles merging. The top right image presented in *Figure 84* illustrates the velocity profile at $S/c = 0.07$, where peak $-C_l$ exists. It can be further observed that the secondary elements are now driving the suction force of the main plane, while the pressure differential is maintained. The bottom image illustrates the velocity profile at the maximum, $S/c = 0.123$. It can be observed that the secondary flaps are still driving the main plane, however, the increase in velocity between the second flap suction surface and main plane trailing edge has resulted in the development of a venturi passage as described, thus causing a reduction in working area on the main plane.

3.3.6 Secondary Elements Negative Overhang

The final study conducted was the investigation into negative overhang. This is simply a reversal of the previous study, starting with $S/c = 0$ and finishing at $S/c = -0.099$.

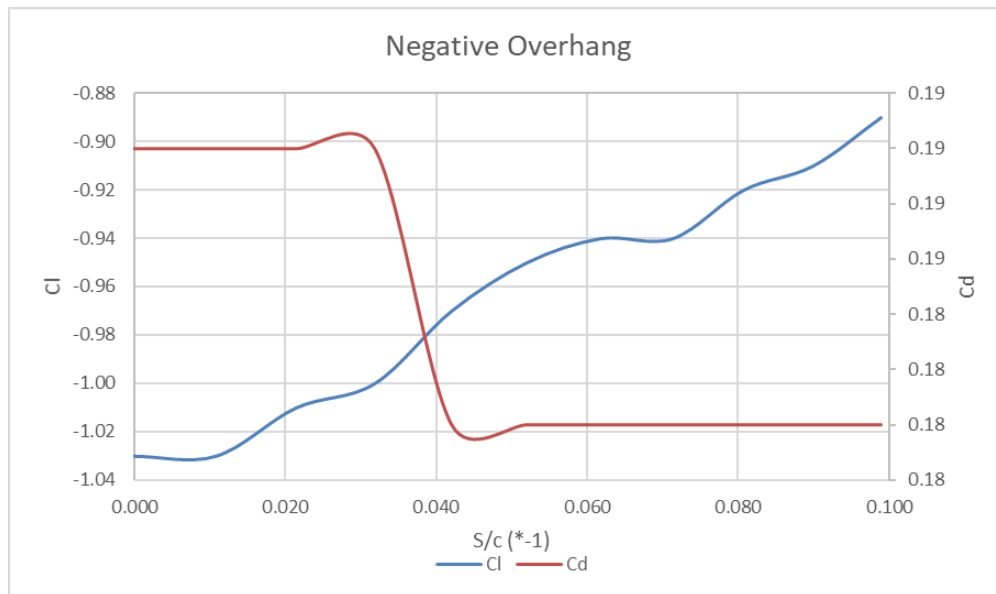


Figure 85 - Negative Overhang C_l and C_d

Figure 85, indicates the lift and drag coefficients for each incremental step of negative overhang. The general trend is a loss of $-C_l$ as the negative overhang increases, starting at -1.03 and finishing at -0.89. This results in a loss of 65.6N of total downforce, even when accounting for the increase in planar area, although a peak downforce of 580.37N is apparent at $S/c = -0.011$, resulting in peak efficiency which then begins to diminish (Figure 87). Figure 86, illustrates a similar trend in drag to that seen in Figure 81, however, there is a sharp decline in drag force rather than increase.

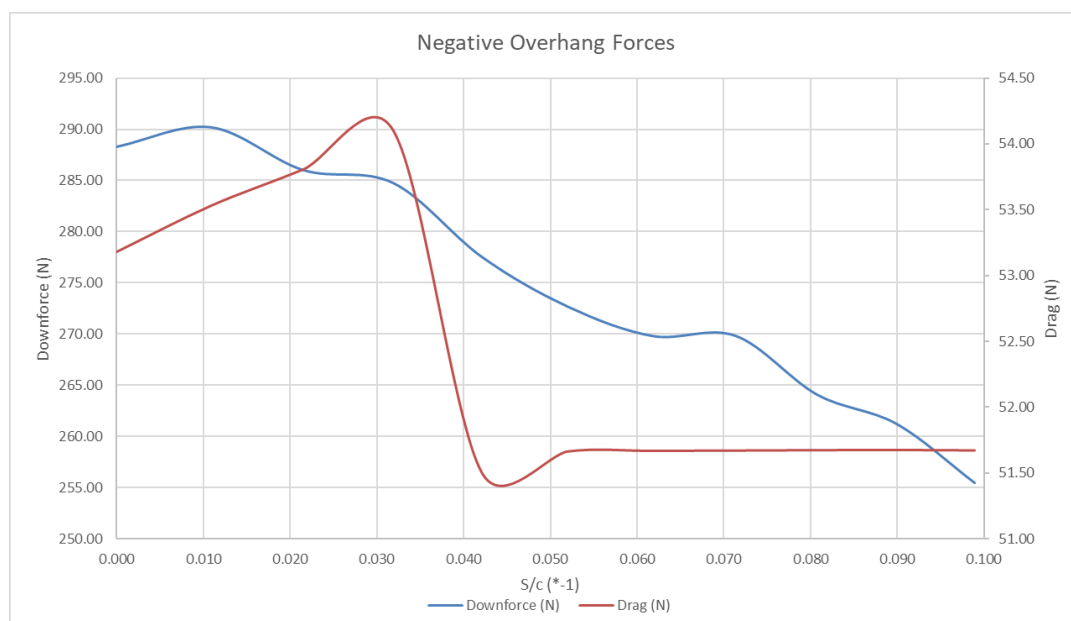


Figure 86 - Negative Overhang Forces

A peak drag force value of 108.2N is achieved at $S/c = -0.032$. Subsequently, a 4.49% decline in drag force is observed, which then begins to stabilise. This is as a result of the secondary elements no longer having a strong impact upon the main plane.

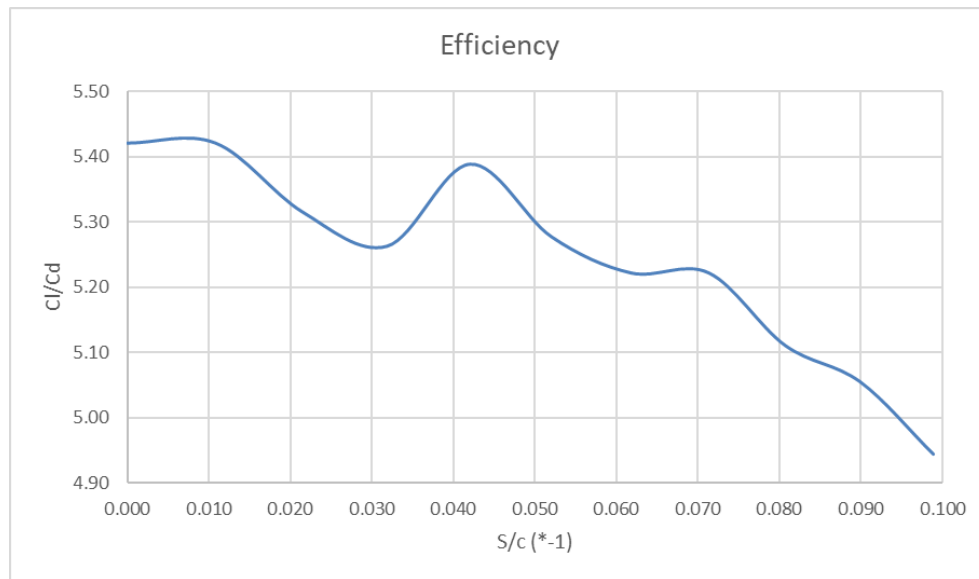


Figure 87 - Negative Overhang Efficiency

Skin friction drag is also reduced on the main plane with respect to wall shear, however, under examination it was found that a marginal increase in wall shear was displayed on the second flap. This is as a result of its further interaction with the onset free stream flow, thus more energy becomes available, however, the reduction in pressure drag outweighs this small increase, consequently resulting in a net loss of drag force.

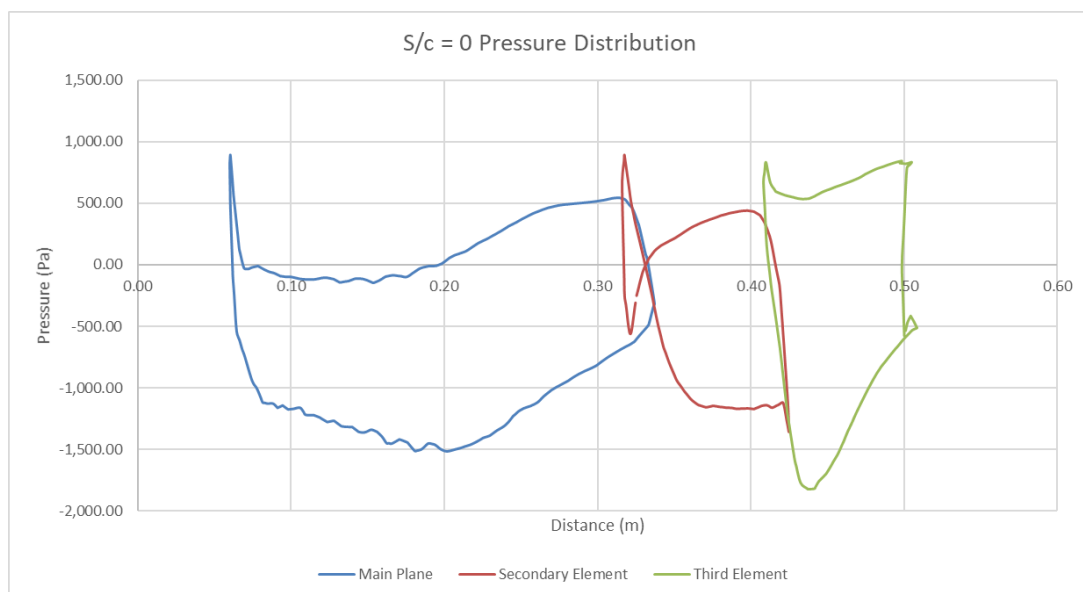


Figure 88 - $S/c = 0$ Pressure Distribution

Figures 88 and 89, illustrate the pressure distribution for both minimum and maximum negative overhang positions. It can be observed that as the secondary elements are positioned further away

from the main plane trailing edge, their influence on the main plane is reduced. This decreases its pressure differential and the generated adverse pressure gradient, however, it can be observed that the pressure differential increases on the secondary flap. Shear stress is a function of velocity (*Equation 7*), therefore an increase in velocity must increase the wall shear effect. This will result in a rise of net drag force for the secondary element (if it was to be independently studied) (Aguilar, et al., 2006).

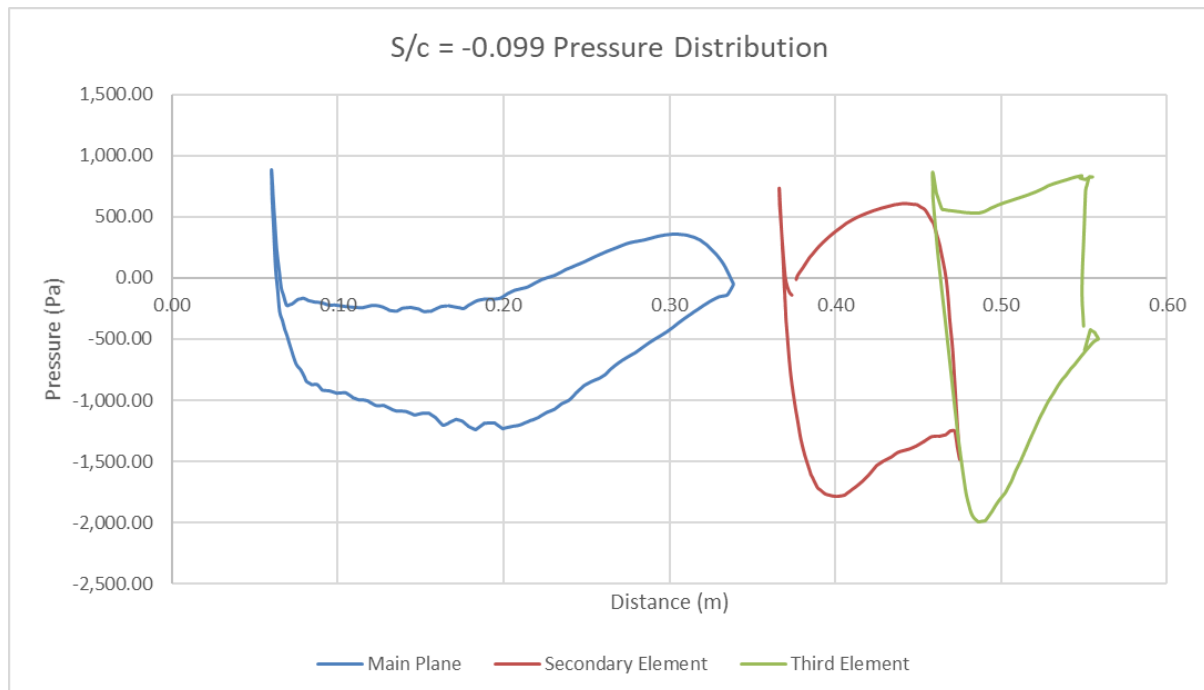


Figure 89 - $S/c = -0.099$ Pressure Distribution

The pressure contours given in *Figure 90*, illustrate the loss of influence on the main plane as the secondary elements are withdrawn downstream, with a bubble of high static pressure forming at the trailing edge of the main plane. This occurs as the leading stagnation pressure on the secondary element bleeds into the low pressure zone located under the main plane (positive pressure gradient). This results in a blockage and a decrease in the velocity and total downforce, which subsequently increases the adverse pressure gradient seen on the suction side of the main plane, and trailing edge separation can be observed.



Figure 90 – Pressure Contours (Left – $S/c = 0$, Right – $S/c = -0.099$)

3.3.7 Gurney Flap

A Gurney flap is a device commonly used in automotive aerodynamics but it has begun to find its way into aerospace and aviation technology. Originally designed by racing driver Dan Gurney, it involves the use of a flap that is located on the trailing edge of a wing, and sits perpendicular to the chord line (Smith, 2013).

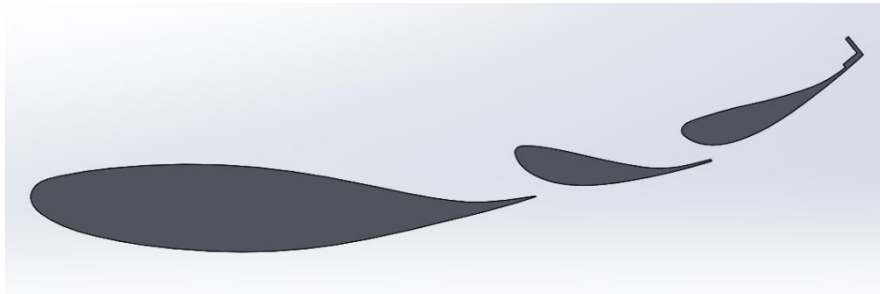


Figure 91 - Baseline Front Wing with Gurney Flap

The baseline configuration of the Dallara's F399 front wing utilises the use of a Gurney flap on its third flap of $H/c = 0.22$, (Figure 91).

The purpose of the GF (Gurney flap) is to alter the trailing edge characteristics of the flow field, by modifying its Kutta condition (Houghton, et al., 2017).

Numerous studies have been conducted with regard to GF's, however the general trend is that they increase negative lift with only a small increase in drag. McBeath also states that a Gurney flap of around 2% of chord can add as much as 0.3 to the C_L value, although this depends highly upon the profile geometry. (McBeath, 2001),

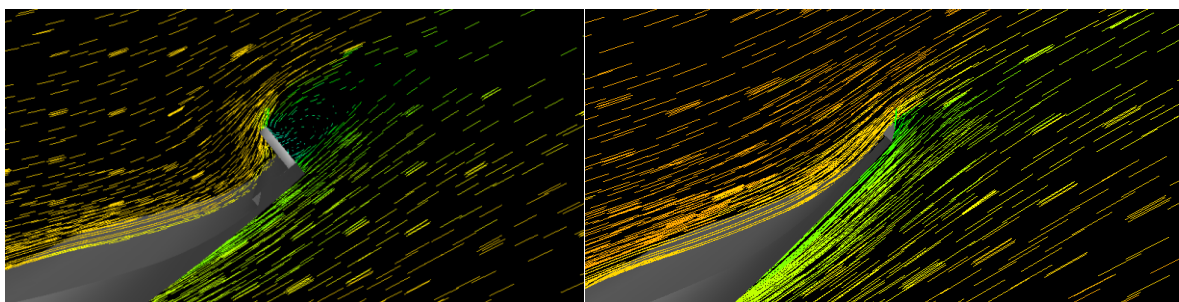


Figure 92 - Vectors of Total Pressure (Left – Gurney Flap, Right – No Gurney Flap)

The principle by which a GF works is to generate an area of localised high pressure on the wings upper surface, causing separation. This results in two counter rotating vortices immediately after the GF, with the lower vortex having an increased suction effect on the negative pressure side due to its rotation vector (Wang, et al., 2008). Figure 92, illustrates the flow vectors with and without a GF. It can be observed that using the GF moves the point at which the upper and lower flow fields re-join. This has the effect of changing the chord length of the aerofoil, whilst also generating the potential for further upwash and delaying flow separation, allowing for a greater AoA (He, et al., 2016).

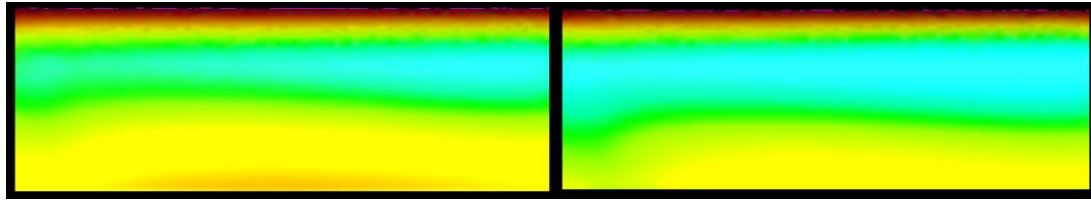


Figure 93 – Suction Surface Pressure Contours (Right – No Gurney Flap, Left - Gurney Flap)

The delay in trailing edge separation is visualised in *Figure 93*. The addition of the GF increases the negative pressure on the suction surface, and positive pressure on the upper surface. However, *Figure 94*, illustrates contours of total pressure for both configurations and demonstrates that the upwash characteristics show very little change from one configuration to another.



Figure 94 - Total Pressure Contours (Left - Gurney Flap, Right - No Gurney Flap)

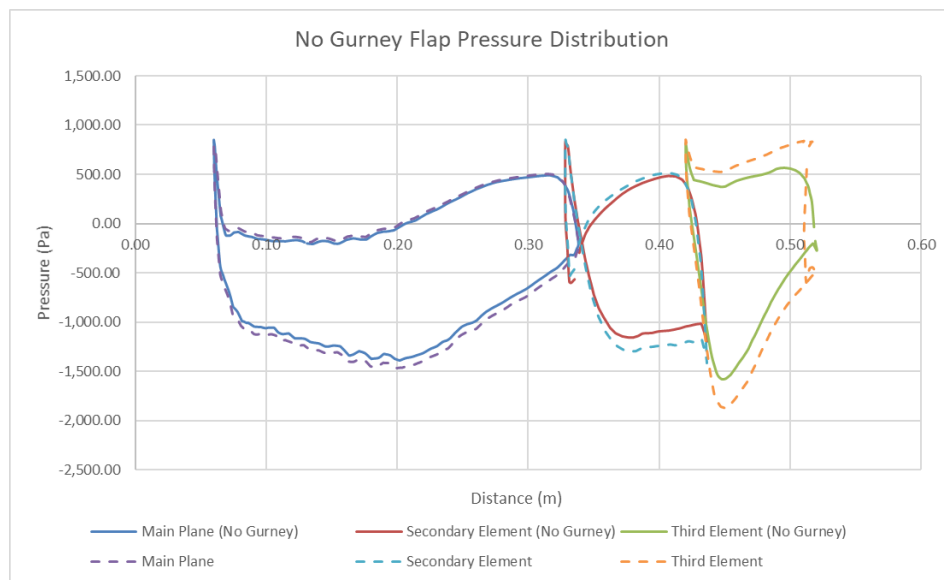


Figure 95 - No Gurney Flap Pressure Distribution (Baseline)

The change in surface pressure due to the GF is illustrated in *Figure 95*. It is evident that the majority of the pressure change is located at the third flap, with the pressure differential decreasing on the forward elements. This generates a 15.8% increase in downforce (when using a GF), however, it also suffers from a 21% increase in drag force, therefore reducing the lift to drag ratio (*Table 1*).

Table 1 Comparison of Baseline Front Wing with and without a Gurney Flap

	Cl	Cd	Cl/Cd
No Gurney Flap	-0.85	0.15	5.67
With Gurney Flap	-1.01	0.19	5.32
$\Delta(\%)$	15.84	21.05	-6.58

As a result of the increased pressure differential, stronger vortex formations are developed. *Figure 96*, illustrates the difference in vortex pressure between the two configurations on a plane 0.65m downstream of the axis origin, normal to the freestream.

The core areas of both primary and Y250 vortices increase with the introduction of the GF. The increase in core area displaces the Rankine theory over a greater cross-sectional area, while the angular velocity profile increases at a greater distance from its core.

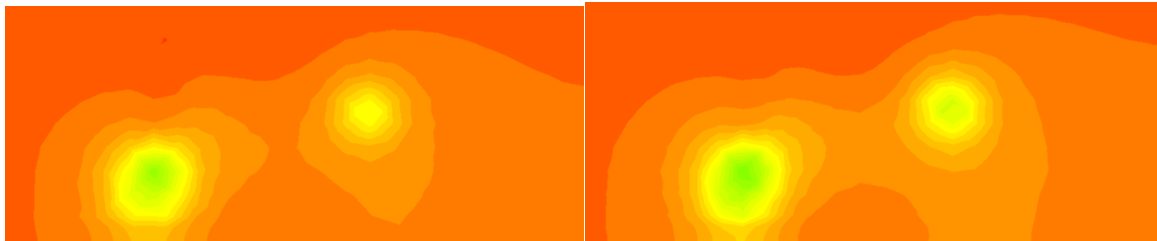


Figure 96 - Pressure Contours (Left - No Gurney Flap, Right - Gurney Flap)

This displacement of angular velocity dissipates the energy at a greater rate, and results in shorter length vortical structures using the GF (*Figure 97*), although it does result in an increase in induced drag.



Figure 97 - Vortex Swirling Strength (Top - No Gurney Flap, Bottom - Gurney Flap) (0.01Hz)

3.4.0 Parameter Sweep Conclusion

The manipulation of the secondary elements has a significant impact on the functionality and aerodynamic characteristics of the main plane, although changing the AoA of the secondary elements alone has little impact.

It was also found that the main plane is the primary generator of vertical force within the wing assembly.

Increasing the pressure differentials was found to form large vortical formations (primary, Y250 and tip), which increased the induced drag. This was especially apparent when increasing the AoA of the entire assembly.

It was established that by inducing further ground effect, by decreasing ground proximity, was the most efficient way of generating downforce.

Positioning of the fore and aft secondary elements requires particular attention in order to avoid the wing assembly from dissolving and becoming two isolated assemblies, or producing adverse effects on the main plane.

Further investigations could be conducted through the use of design of experiment methods as proposed by d'Apollonia et al, in order to facilitate the surface fitting of multiple variables and provide compound results, enabling controlled optimisation (d'Apollonia, et al., 2004). However, this approach was deemed unnecessary for the purposes of this development study.

3.5.0 Front Wheel Study

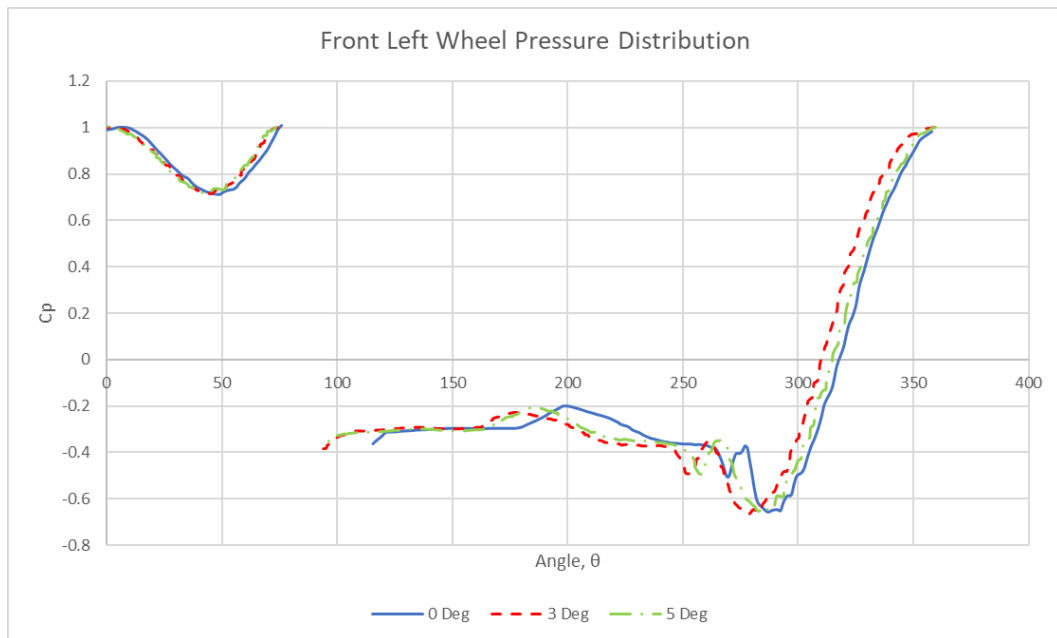


Figure 98 - Front Left Wheel Pressure Distribution

Figure 98, illustrates the pressure distribution on the front left wheel (in isolation) when orientated in the cars “lock to lock” positions. The pressures are taken from a plane perpendicular to the velocity inlet (freestream flow) rather than parallel to the wheels outboard or inboard face.

When examined, it can be observed that the initial surface pressures are consistent, although after the contact patch they begin to vary. When an angle is applied, it can be determined that the contact patch size changes, therefore enabling pressures to be established subsequent to 90° (115° for straight ahead), although this could be as a result of the solid nature of the CAD model, and may not represent a real life scenario as the tyre is unable to deform.

The trends in pressure are the same for each angle, although it can be observed that when a steering lock is applied, the point of separation relocates earlier on the circumference of the wheel, though this would be represented as a later point of separation if θ were clockwise.

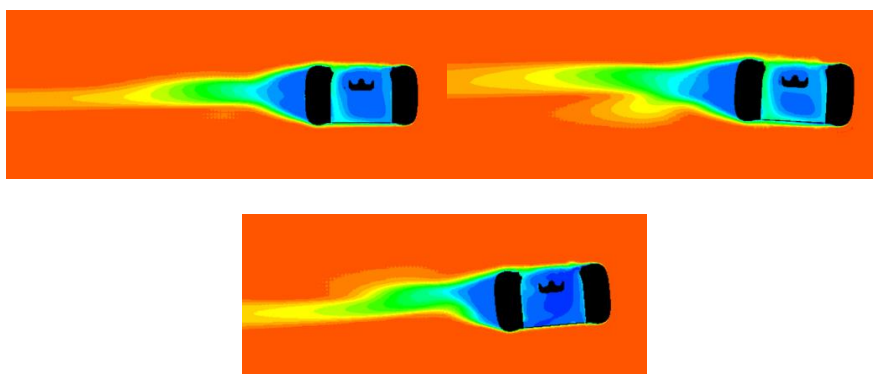


Figure 99 - Wheel Wake Total Pressure (Top Left - 0° , Top Right - 3° , Bottom - 5°)

Figure 99, represents the total pressure contours projected on a plane 0.2807m (wheel radius) from the ground. It can be observed that when the wheel is orientated with its outer face becoming normal to the freestream flow (3° , as if it was the outside wheel during a corner), the wake structure that is produced has the largest negative pressure zone. This results from the onset separation on its inboard edge and possible disruption due to the wheel spokes, which would suggest that the outside front wheel would have a greater impact upon the car, which would be further accelerated when a yaw angle is applied.

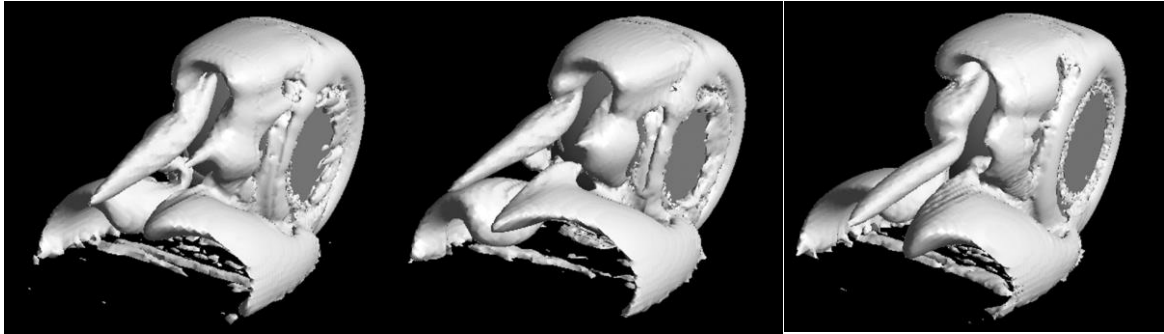


Figure 100 - Front Wheel Vortex Structures (Left - 0° , Middle - 3° , Right - 5°)

The vortex formations illustrated in Figure 100 correlate well with the findings provided in Figures 21 and 22. Six counter rotating vortices are produced, with the two largest structures originating from the contact patch. The two vortices generated from the top surface interact with each other, and merge as one. The middle vortex on the inboard side of the wheel can be observed to reduce in size when an angle is applied. This is most apparent at 5° and is due to the reduction in leading edge separation as illustrated in Figure 99. The most noticeable change can be observed at 3° , with a rapid increase in size of the lower inboard vortex, further suggesting the front outside wheel would be the most aerodynamically disruptive.

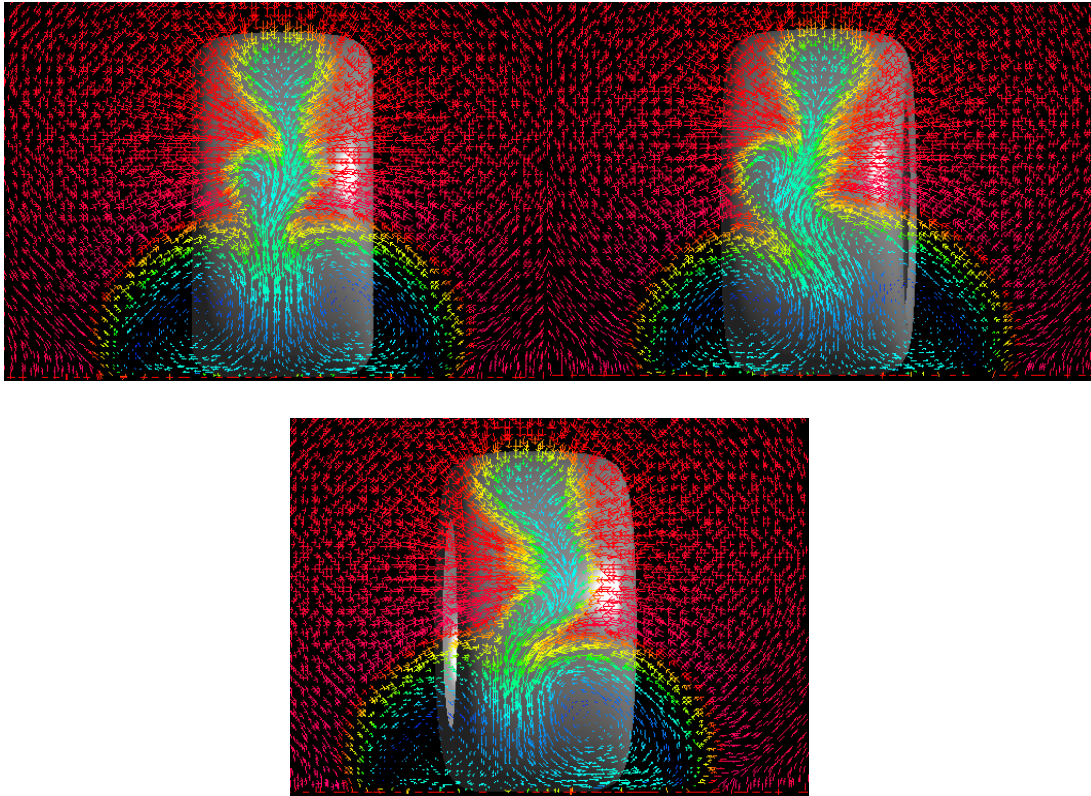


Figure 101 - Velocity Vectors (Top Left - 0°, Top Right - 3°, Bottom - 5°)

The two counter rotating vortices that are developed on the top of the wheel allow a bubble of low pressure air to be generated at the top of the wake formation, as illustrated in Figure 101. This is apparent for each wheel angle, although the position of the bubbles changes with wheel angle.

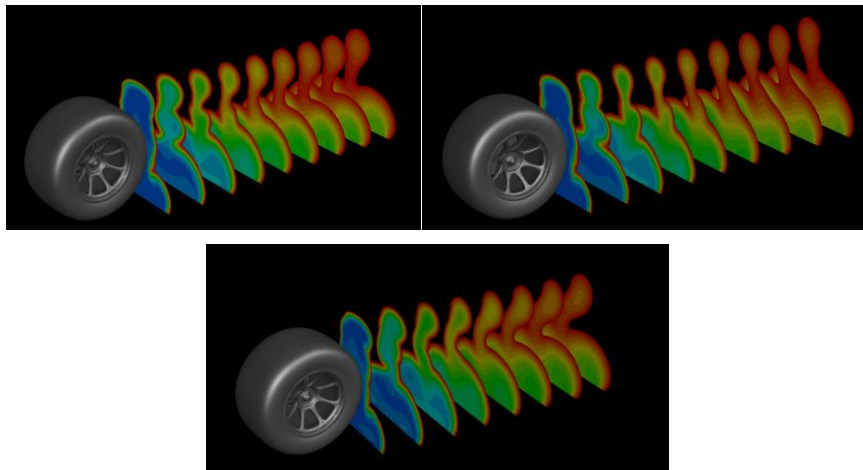


Figure 102 - Total Pressure Wake Structure (Top Left - 0°, Top Right - 3°, Bottom - 5°)

Wheel vortices have the greatest influence upon the wake structure, as illustrated in Figure 100, where it can be observed that the wheel orientation manipulates the vorticial structures and consequently, the shape and overall size of the deposited wake, as illustrated in Figure 102.

3.6.0 Design Development

Previous investigations have shown that the primary vertical force generator originates at the main plane due to its greater planar area. It has been decided, therefore, that the main plane shall be the first area of design development and followed by the secondary flaps.

3.6.1 Main Plane Standardised Aerofoils

Three standardised aerofoils will be investigated with comparisons being made to the baseline Dallara aerofoil with regard to main plane design (*Figure 103*).

All aerofoils have been scaled to match the baseline chord length of 0.279m and have been modelled with the baseline end plate and span (including the centre twist section), however, the secondary elements have not been included. This method was employed in order to provide a true comparison of the main plane characteristics. Two heights were also considered, $H/c = 0.39$ (Baseline) and $H/c = 0.14$ (measured from a tangential line to the suction surface), which would have the effect of introducing a minor ground effect and strong ground effect. Each wing was studied in isolation at 40m/s in order to decrease computational expense.

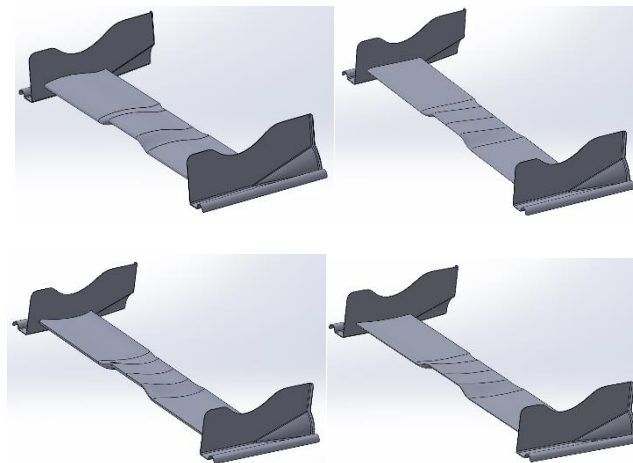


Figure 103 - Main Plane Geometries (Top Left - Dallara Baseline, Top Right - Naca 4412, Bottom Left - S1223, Bottom Right - Naca 6409)

3.6.1.1 Main Plane Aerofoil Study ($H/c = 0.39$)

The first aerofoil to be investigated was the Naca 4412 aerofoil. A study conducted by Hossain et al, indicated that the Naca 4412 has superior efficiency when in a ground effect scenario (Hossain, et al., 2014), and is often used for ground effect applications (Abdullah, et al., 2017).

Subsequently, the S1223 profile designed by Selig et al, was studied. This profile is designed primarily for high lift applications (Selig & Guglielmo, 1997), and closely follows the profiles of Seagull and Merganser wings (Liu, et al., 2006).

As a result of subsequent findings, which will be discussed, a third profile was investigated, being the Naca 6409. This profile is suggested as having notable lift capabilities, however, a study by Joshi et al, demonstrated that its efficiency declines rapidly at high AoA (Joshi, et al., 2019).

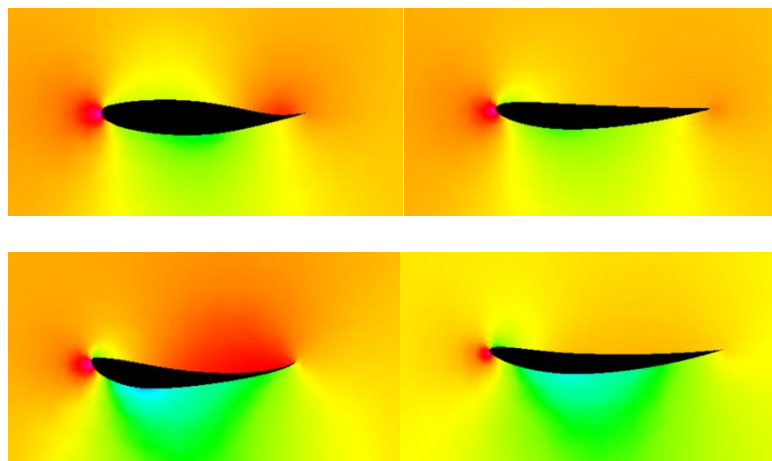


Figure 104 - Pressure Contours (Top Left - Dallara, Top Right - Naca 4412, Bottom Left - S1223, Bottom Right - Naca 6409) ($H/c = 0.39$)

Figure 104, illustrates pressure contours projected on to a plane 0.5m from the wing centre.

Examining the Dallara profile, it can be observed that a significant area of low pressure exists on the top surface, close to the leading edge. This is offset by the trailing edge including a small pressure recovery along its length, although this does have the potential to cause adverse effects as a result of the pressure gradient.

The Naca 4412 develops a lower negative pressure on its suction surface, however, the generated adverse pressure gradient is not as aggressive when compared to that of the Dallara profile. This results from its peak negative pressure being located further upstream and is therefore allowed to return at a lesser rate. However, due to the flat nature of the top surface, very little pressure differential is generated, resulting in a minimal amount of generated downforce.

The S1223 produces the largest pressure differential of all of the selected profiles, in turn, producing the greatest downforce by a factor of 3.3. As a consequence, it has also generated the highest drag force value (33% increase from the lowest), however it still maintains the highest efficiency. As with the Dallara profile, it can be observed to include a concave pressure recovery on its top surface, generating a considerable area of high pressure, helping to maximise the pressure differential.

As the previous profile studies resulted in two extremes, it was decided that a third profile should be investigated. The Naca 6409 was selected due to its profile geometry having a combination of traits from both the S1223 and the Naca 4412. It can be observed to also have a concave top surface, but without the aggression of the S1223. Its suction surface follows that of the Naca 4412, therefore producing a steady return to atmospheric pressure. This profile produced the second highest value of downforce, with only a 0.5% increase in drag (with respect to the lowest value), which is assisted by having a thin leading edge, thus generating the smallest stagnation zone.

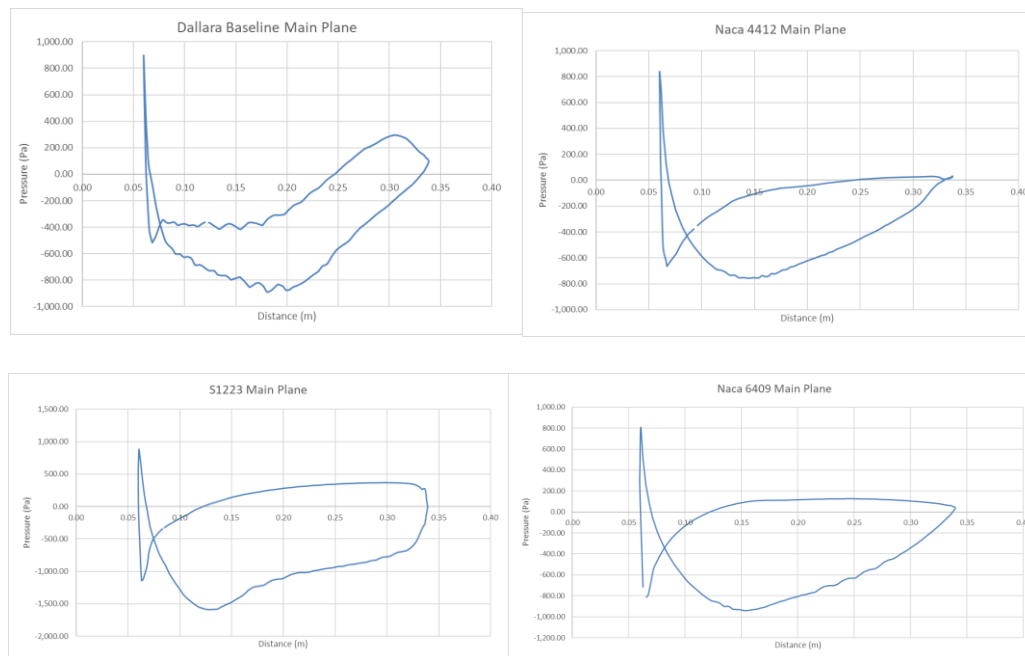


Figure 105 - Pressure Distribution (Top Left - Dallara Baseline, Top Right - Naca 4412, Bottom Left - S1223, Bottom Right - Naca 6409) ($H/c = 0.39$)

Figure 105, illustrates the pressure distributions around each of the studied geometries, 0.5m from the centreline. Examining each graph, it can be observed that the Dallara baseline utilises the most aggressive pressure gradient of all of the tested geometries, reaching a peak negative pressure of -890.2Pa and a maximum of 296.03Pa (disregarding the stagnation zone). This could be as a result of the exclusion of the secondary elements as Figure 51 indicates a far gentler form of positive pressure distribution. This would suggest that this profile could have been designed to be used with secondary elements.

The Naca 4412 produces a much smoother pressure gradient, however, it achieves a peak negative pressure of only -757.83Pa and a peak positive pressure of 29.88Pa (disregarding the stagnation zone). Due to the reduced pressure differential, this profile generates the lowest -Cl of all the profiles, thus it is the least efficient (Table 2).

The Selig designed S1223 generates the greatest amount of downforce due to its extreme pressure differential, reaching a peak negative of -1593.99Pa and a peak positive of 369.55Pa, generating a differential of 1963.54Pa, although it produces the greatest drag value (Table 2), however, as a result, is still considerably more efficient.

Even though the S1223 produces the greatest downforce, it was considered that the drag penalty may cause the car (if implemented) to suffer from adverse effects and reduce its aerodynamic performance. This would result in slower straight line speeds and increased lap times, therefore further consideration was given to the Naca 6409. As previously mentioned, its geometry contains attributes from both the Naca 4412 and S1223. Its thin leading edge allows it to generate the smallest stagnation area along with the lowest stagnation pressure. It achieves a peak negative pressure of -941.18Pa, located 0.153m along its chord length. A peak positive pressure of 126.77Pa is produced, although it can be observed, when referring to *Figure 105*, that this pressure is consistently maintained on its upper surface, which therefore reduces the potential for aerodynamic instability. It also attains the second highest pressure differential of 1067.95Pa, therefore making it the second highest producer of downforce, without the drag penalty of the S1223.

Table 2 - Main Plane Profile Study (H/c = 0.39)

Main Plane Profile Study (H/c = 0.39)			
Profile	Cl	Cd	Cl/Cd
Dallara Baseline	-0.33	0.06	5.50
Naca 4412	-0.29	0.06	4.83
S1223	-0.98	0.09	10.89
Naca 6409	-0.51	0.06	8.50

As *Table 2* indicates, the S1223 produces the greatest efficiency due to the considerable downforce generated, although it suffers from significant induced drag resulting from its concaved upper surface. Whilst the Naca 6409 does not achieve the greatest efficiency, it produces less drag than the S1223 and more downforce than the Dallara baseline and Naca 4412, which suggests that it would not disrupt straight line speeds through induced drag, and will still generate more front downforce than the baseline, therefore enhancing front aerodynamic grip in fast corners.

3.6.1.2 Main Plane Aerofoil Study ($H/c = 0.14$)

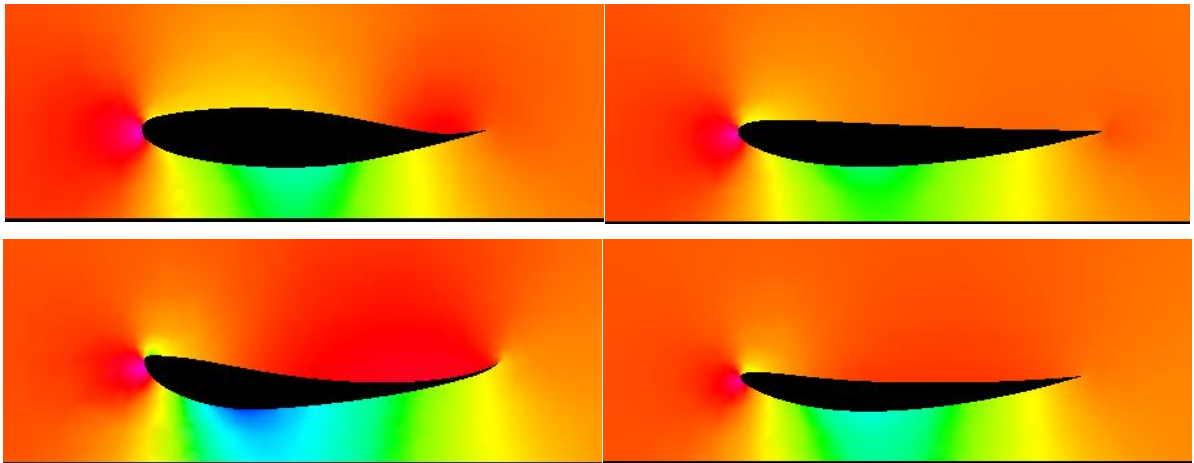


Figure 106 - Pressure Contours (Top Left - Dallara Baseline, Top Right - Naca 4412, Bottom Left - S1223, Bottom Right - Naca 6409) ($H/c = 0.14$)

As already discussed, the reduction of ground proximity has amplified the venturi effect, thus the generated downforce has increased.

Figure 106, illustrates the pressure contours displayed on a plane 0.5m from the wing's centreline. It is clear that the S1223 still produces the greatest peak pressure at $H/c = 0.14$, with the Naca 4412 displaying the lowest.

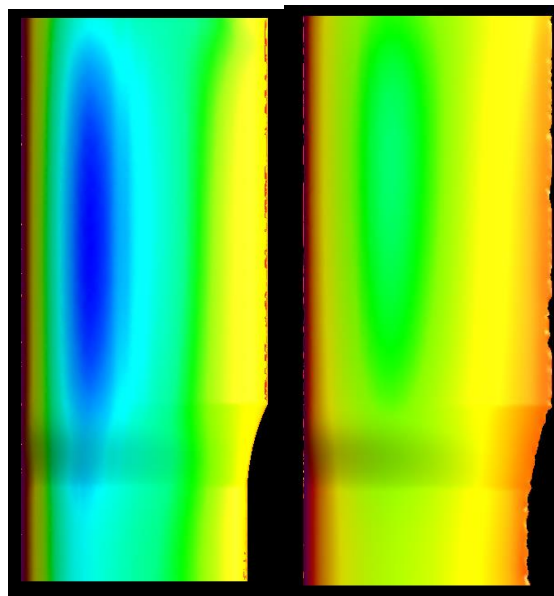


Figure 107 - Suction Surface Static Pressure (Left - S1223, Right - Naca 4412) ($H/c = 0.14$)

This phenomenon is indicated on the surface pressures illustrated in Figure 107. The aggressive geometry of the S1223 has produced a much greater peak negative pressure, however, it is only covered in a very localised area. This produces disadvantages when utilised in a dynamic situation, having the potential to generate porpoising as a consequence. This would suggest an extreme adverse pressure gradient, therefore risking separation and resulting in unpredictability (Nirooei, 2017).

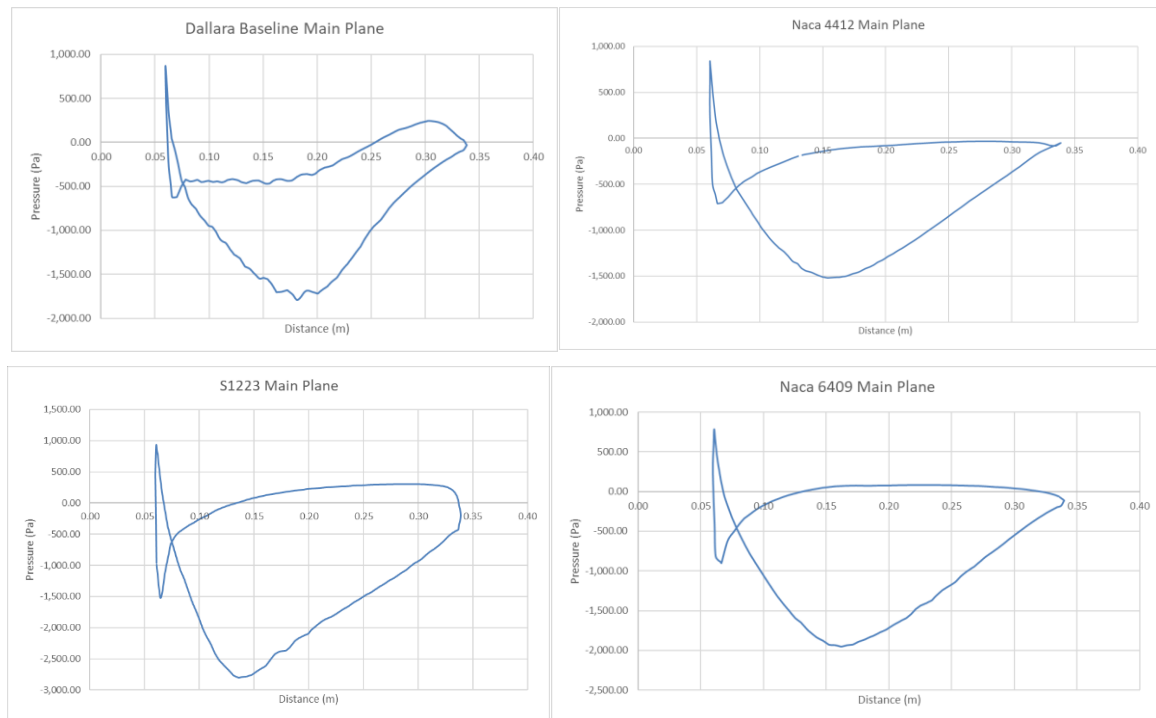


Figure 108 - Pressure Distribution (Top Left - Dallara Baseline, Top Right - Naca 4412, Bottom Left - S1223, Bottom Right - Naca 6409) ($H/c = 0.14$)

Figure 108, illustrates the pressure distribution for each of the studied profiles at $H/c = 0.14$. As expected and previously referred to in Figure 106, all peak negative pressures have increased with a reduction in ground proximity, however, Figure 109 indicates that peak positive pressures have decreased as a consequence of further induced ground effect.

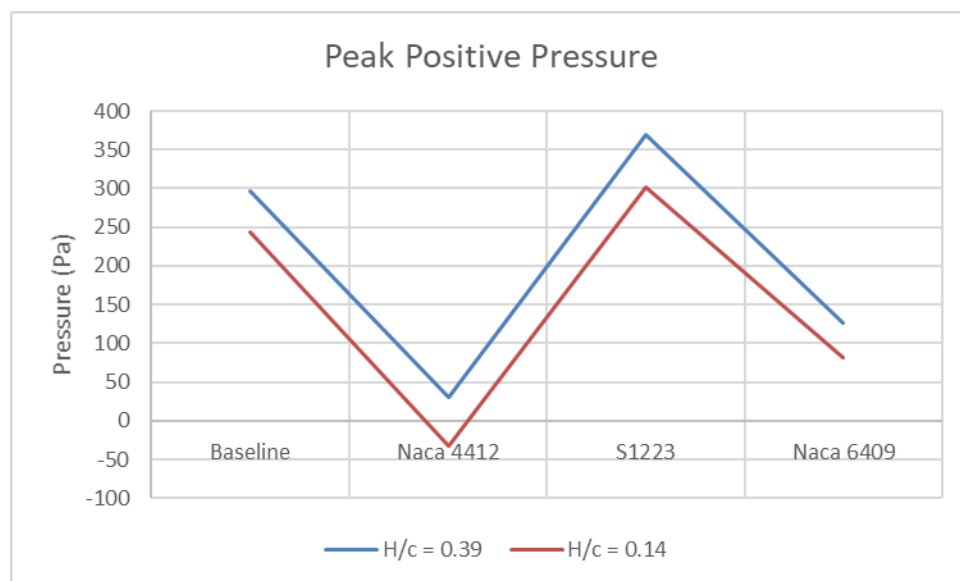


Figure 109 – Peak Positive Pressure

Despite this, the increase in peak negative pressure results in the pressure differentials increasing for every profile.

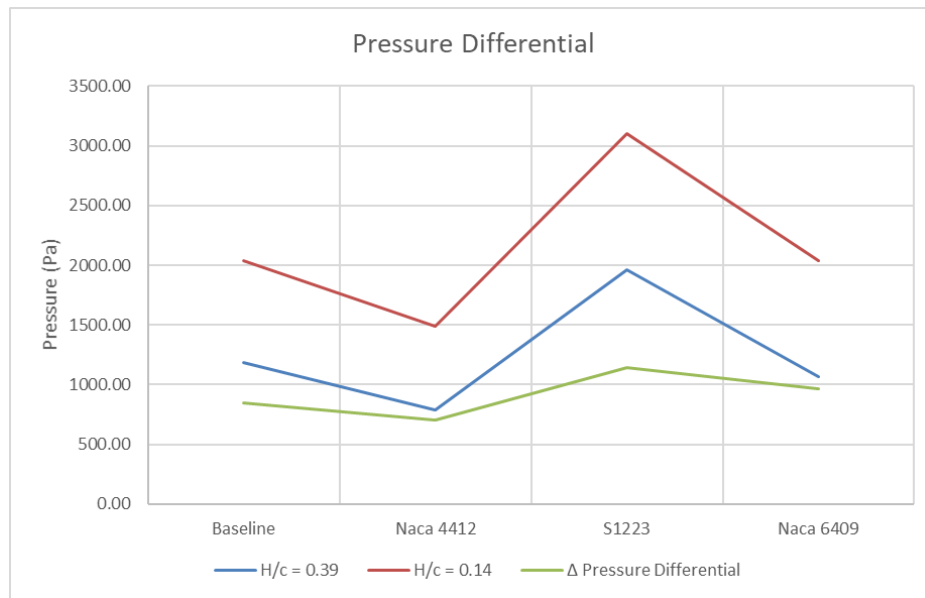


Figure 110 – Pressure Differential Increase

Figure 110, further confirms this. The green line indicates that the S1223 profile generates the greatest increase in pressure differential with the Naca 6409 producing the second greatest.

Table 3 - Main Plane Profile Study ($H/c = 0.14$)

Main Plane Profile Study ($H/c = 0.14$)			
Profile	Cl	Cd	Cl/Cd
Dallara Baseline	-0.64	0.07	9.14
Naca 4412	-0.59	0.07	8.43
S1223	-1.44	0.11	13.09
Naca 6409	-0.91	0.07	13.00

The increase in pressure differential has resulted in the S1223 having the greatest -Cl, although it has also achieved the greatest increase in Cd, it being double the rate of increase produced by the Naca 6409. This results in a greater rate of efficiency increase by the Naca 6409 as indicated in Table 4, with the S1223 producing the lowest rate of efficiency increase.

Table 4 - Main Plane Profile Study ($\Delta H/c$)

Main Plane Profile Study ($\Delta H/c$)			
	ΔCl	ΔCd	$\Delta Cl/Cd$
Dallara Baseline	-0.31	0.01	3.6
Naca 4412	-0.30	0.01	3.6
S1223	-0.46	0.02	2.2
Naca 6409	-0.40	0.01	4.5

This would suggest that the Naca 6409 maybe the most efficient profile to use within a dynamic situation, therefore it should be considered for main plane profile use.

3.6.2 Main Plane Design Process

Each front wing design iteration will be studied on the car in order to monitor the produced interactions, however, only half car case studies will be utilised to provide an increase in computational efficiency, however, this contradicts the advice given by Toet (Toet, 2019). In these case studies, however, every design iteration will remain relative to the previous.

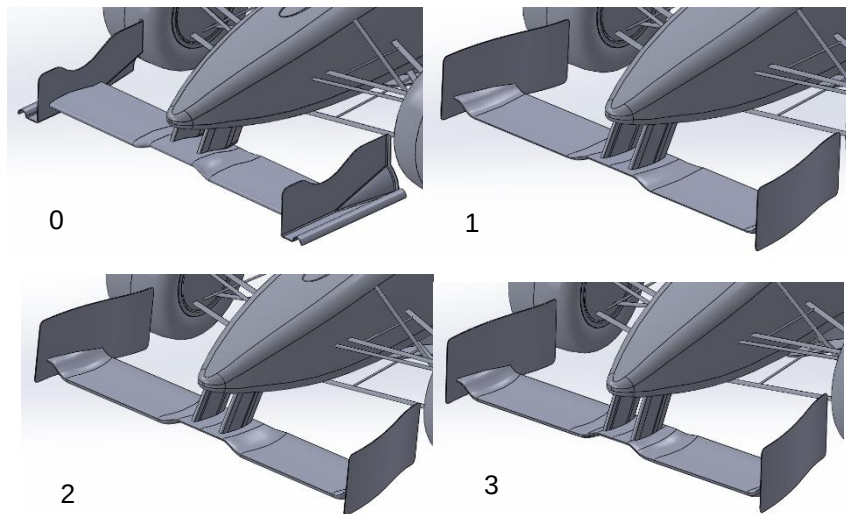


Figure 111 - Front Wing Main Plane Designs

Figure 111, illustrates the baseline main plane geometry (0) followed by three design iterations. The first design (1) utilises the Naca 6409 profile throughout its entire span. This decision was based upon the findings in the previous chapter. The second and third designs (2 and 3) employ both the Naca 6409 and S1223 profiles. Each design follows the same compound nature of the baseline main plane (raised centre) having the same chord length and allowing uninterrupted air to enter the floor area. All main plane design cases are conducted using an $AoA = 0^\circ$.

Other alterations have been made in accordance with previous investigations. The baseline front wing is located at $H/c = 0.39$, however, the new designs are situated at $H/c = 0.21$. After careful consideration and examination of previous results, it was decided not to reduce the ground proximity any further in order to help improve aerodynamic stability but to also reduce the risk of damage to the wing when in a dynamic condition (Hillam, 2020), although an $H/c = 0.21$ still falls within the recommended ground proximity range recommended by Katz (Katz, 2006).

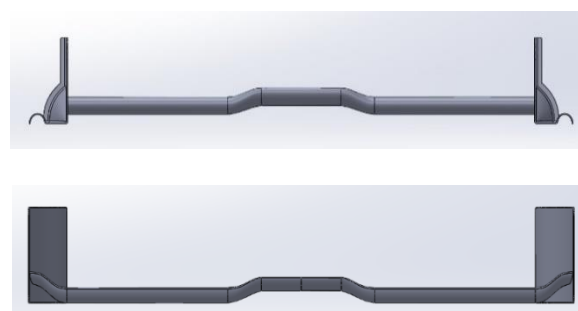


Figure 112 - Front Wing Main Plane Front View

In order to achieve this ground proximity, the outboard edges of the main plane have been raised and follow a curved contour which allow the lower face of the end plates to protrude below (*Figure 112*), and reduce any pressure bleed without hindering the reduction in ground proximity. This will, therefore, allow a greater proportion of the main plane to be utilised for an increased ground effect, thus inducing greater downforce efficiently. This technique is commonly used within Formula 1, although is also used to allow outboard pressure off load in order to reduce aerodynamic sensitivity when the front wheels are rotated around the Z axis. This is more critical in Formula 1 due to the greater allowable wing span, therefore more of the front wing is interacting with the front wheels (Anderson, 2020).

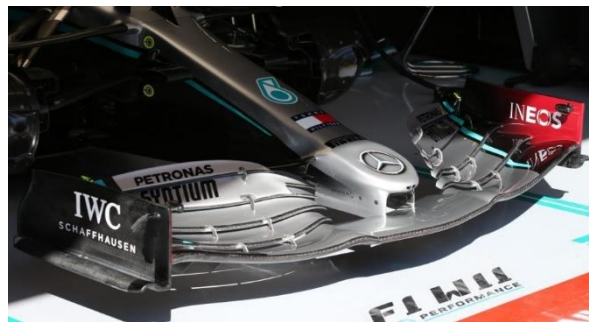


Figure 113 - 2019 Mercedes W10 Front Wing (Anderson, 2020)

The end plate geometry has also been simplified, although it still matches the same X and Y dimensions of the baseline end plates. As *Figure 114* illustrates, the new end plate is now being used to generate in-wash rather than the out-wash produced by the baseline. It was found that an out-wash inducing end plate was causing separation on the inboard front edge of the wheel (refer to *Figure 45*). This results from the restricted span of the front wing and the distance between the end plate trailing edge and the wheel front face not providing an out-wash flow condition.

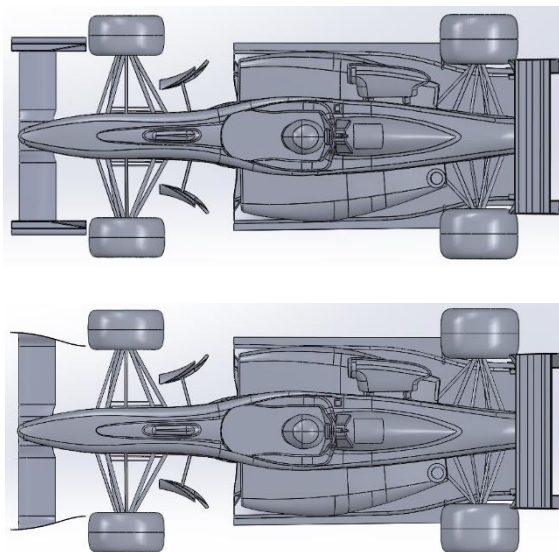


Figure 114 - Main Plane Top View

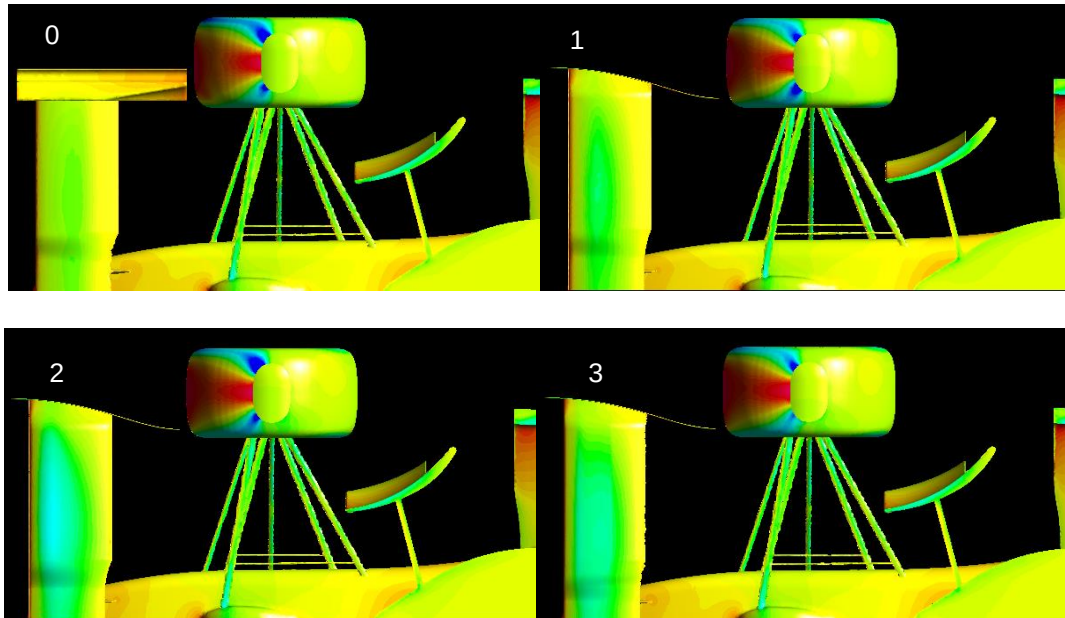


Figure 115 - Main Plane Design Iterations - Suction Side Static Pressure

As previously discussed, the main plane is the main producer of downforce due to its greater planar area, therefore, this study will be focussing on investigating C_l , C_d and the CoP location, however, flow conditions will be monitored.

Figure 115, illustrates static pressure contours for each main plane design iteration. Design 1, utilises the Naca 6409 across its entire span. This resulted in only a 7.3% increase in downforce and a 3.9% increase in drag relative to the baseline (0). The increase in frontal downforce shifts the CoP location 0.37m forward relative to the front axle, although the results indicate a frontal aerodynamic balance of -8.22%, suggesting that the rear downforce is far outweighing that of the front, producing front end lift. Despite the Naca 6409 producing more downforce than the baseline, its interaction with the car is hindering its performance compared to the results observed in the previous chapter.

In an attempt overcome this, the S1223 profile was employed for the outboard section of the wing due to its superior downforce production and high efficiency, but the Naca 6409 remained for the centre section to reduce the promotion of upwash, but to still generate downforce. This modification increased the downforce by 28.5% and reduced the drag force by 0.14%, which achieved a front end balance of 8.97%.

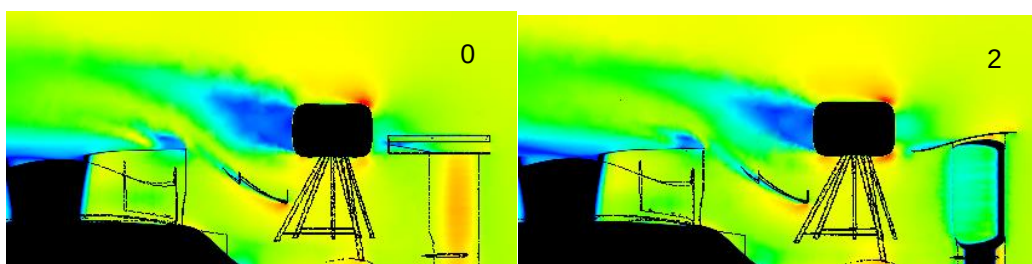


Figure 116 - Velocity Contours of Front Wheel Wake

The reduction in drag force is as a consequence of the reduction in size of the lower half of the front wheel wake. *Figure 116*, illustrates this using velocity contours projected on a plane 0.1m from the ground plane. The reduction in size and marginal increase in outwash results from the increase in velocity around the inboard leading edge of the wheel. The increase in ground effect using the S1223 profile and in-wash end plates has resulted in a strong primary vortex formation that travels further inboard of the front wheel without interference, therefore removing the separation on the inboard leading edge of the wheel as experienced with the baseline geometry.

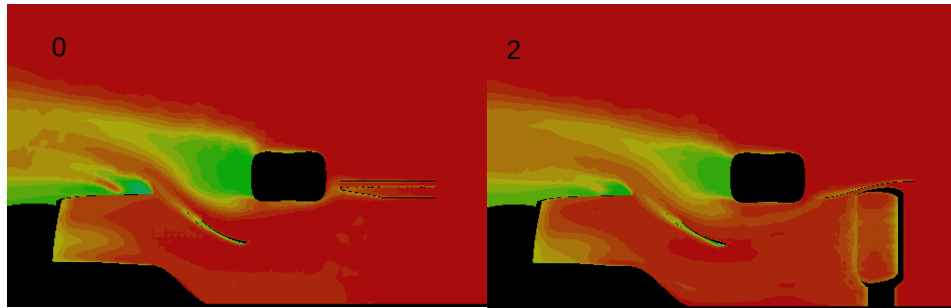


Figure 117 - Total Pressure Contours of Front Wheel Wake

Figure 117, illustrates the total pressure contours on a plane 0.1m above the ground. The vortex structure rolling off the trailing edge of the in-wash end plate can clearly be observed located inboard of the wheel, which is in contrast to the baseline end plate, where it can be observed to be interacting heavily with the front face of the wheel.

The vortex produces a blockage between the freestream flow and the wake generated by the wing. The freestream air having more energy, then interacts with the wheel wake and increases its total pressure, thus reducing its size. The separation bubble on the leading edge of the venturi tunnel has also reduced in size and intensity.

The aim of Design 3, was to try to reduce the peak in negative pressure on the S1223 profile, as has already been described. This peak in localised negative pressure in Design 2 is visible in *Figure 115*, however, it does not reach the magnitude observed when examined in isolation.

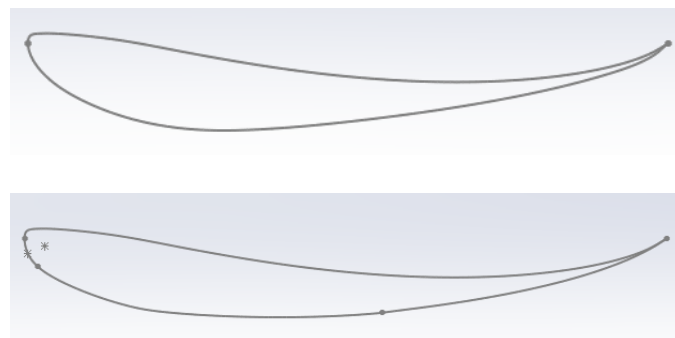


Figure 118 - S1223 Profile Modification

Figure 118, illustrates the change made to the suction side of the S1223, in an attempt to reduce the peak negative pressure. This change in profile allowed it to sit further down relative to the end plate, producing a greater surface area that would be subjected to an increase in ground effect, with a

greater consistency in negative pressure. As *Figure 115* illustrates, a decrease in negative pressure peak is achieved, however, the negative pressure distribution has not increased along the whole chord length. This has resulted in only a 15.23% increase in frontal downforce relative to the baseline (13.27% decrease from design 2) and a 0.73% increase in drag relative to the baseline.

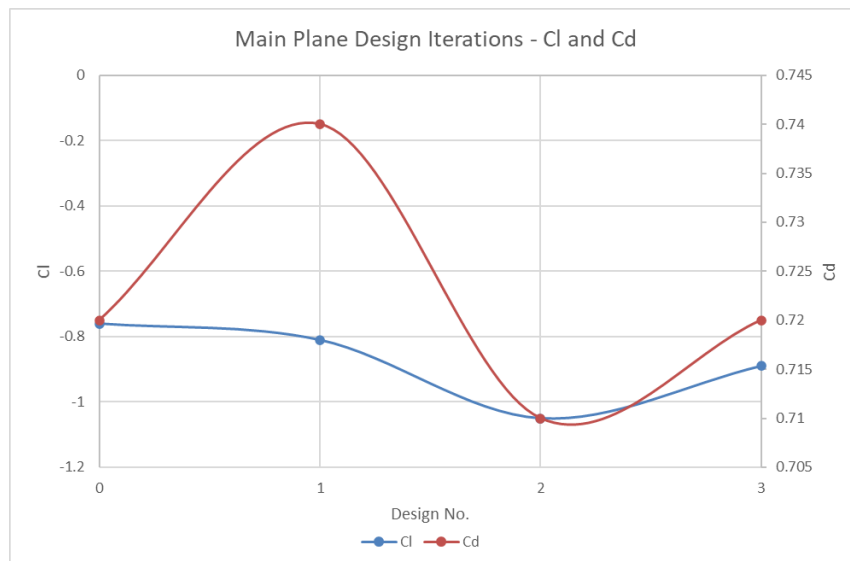


Figure 119 - Main Plane Design Iterations (Cl and Cd)

Figure 119, illustrates the Cl and Cd results for each main plane design iteration. It can be observed, that by utilising the Naca 6409 for the entire wing span produced the greatest increase in drag force. Introducing the S1223 for the second design yielded improved results, with an increase in -Cl and the largest decrease in Cd, thereby producing the most efficient concept. The increase in frontal downforce also allowed for the CoP point to be located further forward, as illustrated in *Figure 120*.

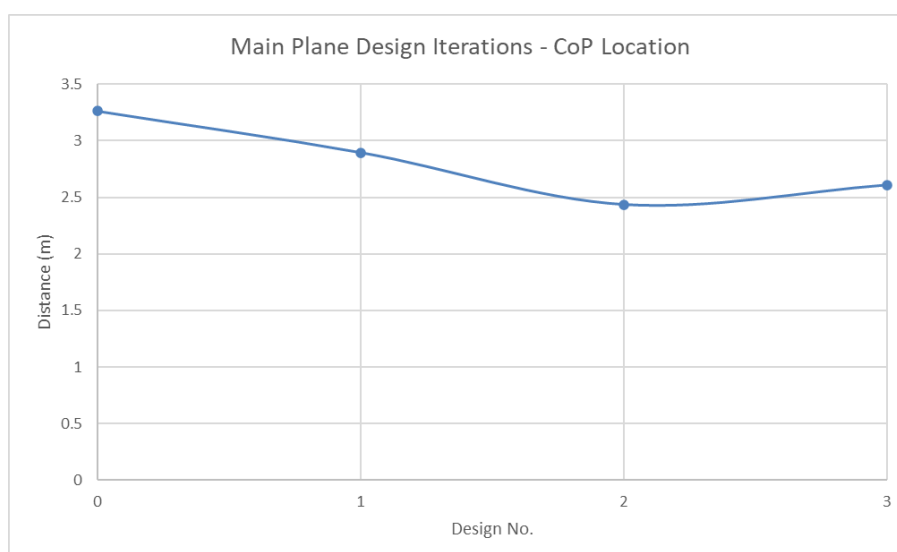


Figure 120 - Main Plane Design Iterations (CoP Location)

3.6.3 Secondary Elements Design Process

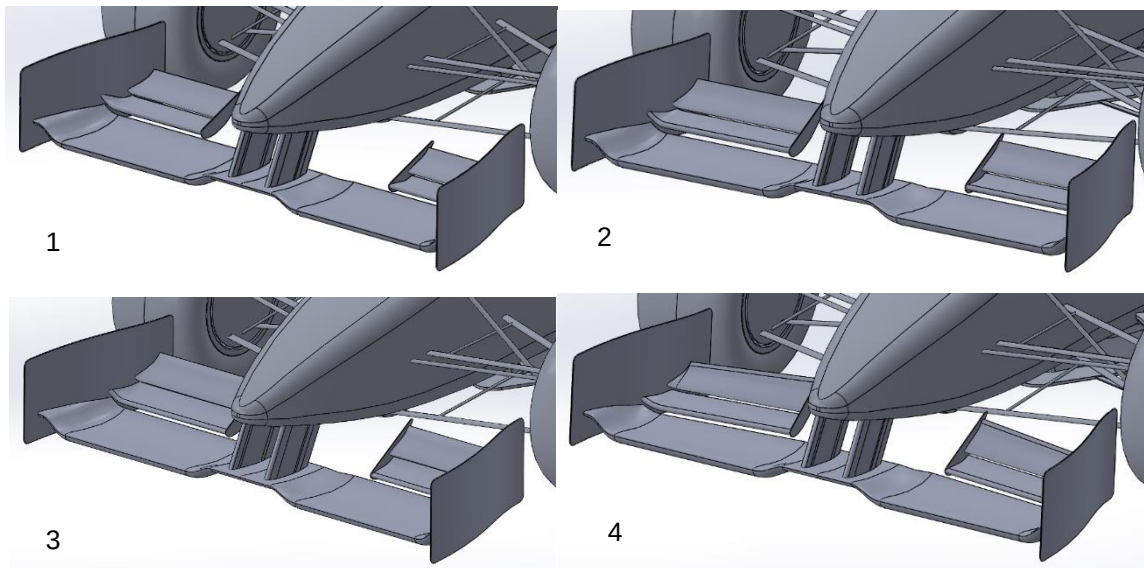


Figure 121 - Front Wing Secondary Element Designs

Figure 121, illustrates four design iterations utilising secondary elements. As previously stated, the use of secondary elements is employed to further drive the main plane and generate an increase in vertical force, consequently moving the CoP further forward. For convenience, it was decided to utilise the baseline secondary element profiles for this study and maintain the main plane H/c of 0.21. In addition, the outboard edges of the secondary elements have also been adjusted so as to follow the same contour as the main plane. Introducing this change allows the leading edge of the secondary elements to maintain the baseline height across the entire span, without creating an outboard blockage, and risking a severe decrease in component efficiency.

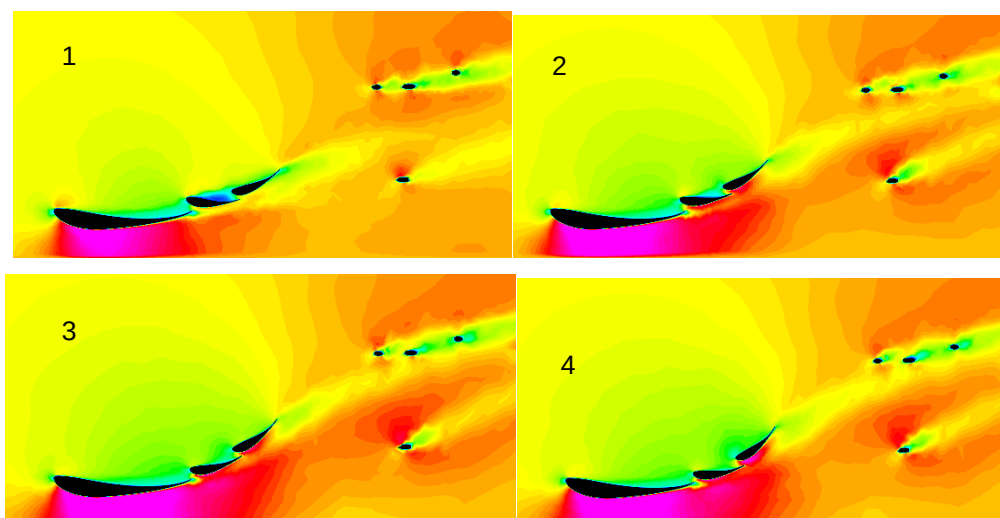


Figure 122 - Velocity Contours (0.35m from Axis Origin)

Design 1, utilises the baseline secondary elements with the same geometry dimensions as can be observed on the original front wing (apart from the outboard edge). An AoA of 15.42° was also employed as it was considered that this would provide an appropriate starting point.

Design 2, introduces an increase in span of 70mm and an AoA of 24° , although the chord length has remained the same. The concave radius on the upper surface of the first element has also been reduced in order to avoid boundary separation (*Figure 122*). This increases the pressure differential on the first element and further drives the second element within the assembly and has an impact of a 17.75% increase in downforce and a drag reduction of 1.41%.

When examining *Figure 122 – 2*, it can be observed that the velocity profile generated within the main plane upwash could further interact with the secondary elements to induce more energy and increase the negative pressure on the suction side (*Figure 123 – 3*). For this reason, the secondary elements were relocated 0.005m downstream for Design 3, with a further increase in AoA to 30° . As *Figure 122 – 3* illustrates, adequate boundary attachment is achieved, although this only results in a 1.59% increase in downforce relative to Design 2. The increase in AoA has also generated further induced pressure drag, resulting in a 2.47% increase in drag.

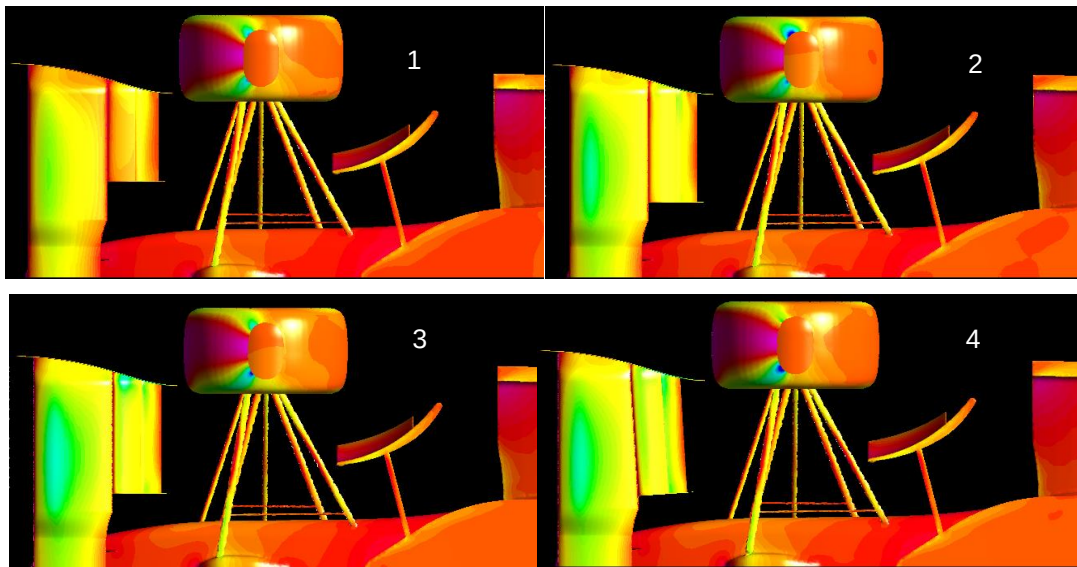


Figure 123 - Static Surface Pressures (Suction Side)

Figure 123, illustrates the static surface pressures of each design iteration. Design 3 illustrates an area of localised negative pressure outboard of the first element. This is as a result of ambient pressure bleeding under the end plate due to the increased pressure gradient. A stronger primary vortex is subsequently induced, which then interacts with the inboard leading edge of the wheel (*Figure 125*).

To negate this, Design 4 utilises the employment of twisted secondary elements. The outboard edge AoA is reduced back to 24° , therefore reducing the pressure gradient which will, however, result in a loss of vertical force. To counteract this, the inboard edge maintains a 30° AoA, while the span of the secondary elements is increased by a further 20mm. The increase in span results from an attempt to energise an increased area of the main plane, resulting in additional front downforce and a reduction in spikes of negative pressure, thereby achieving greater aerodynamic stability.

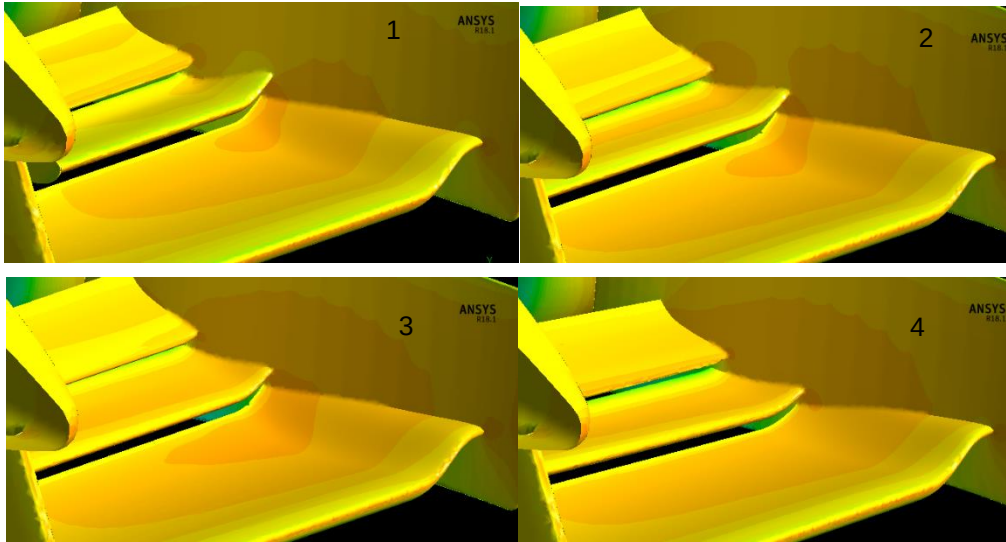


Figure 124 - Static Surface Pressure (Upper Surface)

To further decrease the outboard pressure gradient, the outboard chord length of the main plane has been reduced, which reduces the aggressive nature of the profile. The upturned outboard leading edge of the first element has also been reduced in order to maintain a consistent height. The reduction in chord length has allowed the secondary elements to be rotated by 3° around the Z axis so as to preserve its span interaction with the main plane.

The result of these changes in geometry is evident in *Figure 124*. It can be observed that Design 4 has a significantly more evenly distributed positive pressure along the main plane surface and second element surface. The localised area in positive pressure that can be observed in Design 3 that bleeds onto the outboard leading edge of the first element has greatly reduced in size in Design 4. This decreases the energy being fed into the developed vortex, generating a 1.39% reduction in drag force and increasing downforce by 1.16%.

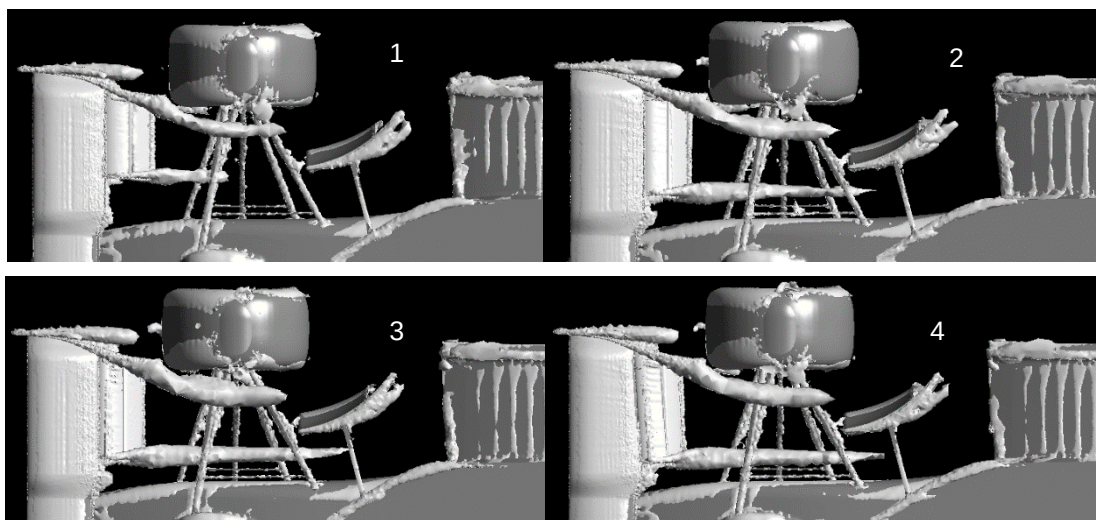


Figure 125 - Vortex Swirling Strength (Up to 252.5 Hz)

Figure 125, illustrates the swirling strengths of the developed vortices for each design iteration. The reduction in vortex diameter is apparent in Design 4, reducing its interaction with the front wheel and providing for an increase in freestream air to pass the inboard edge.

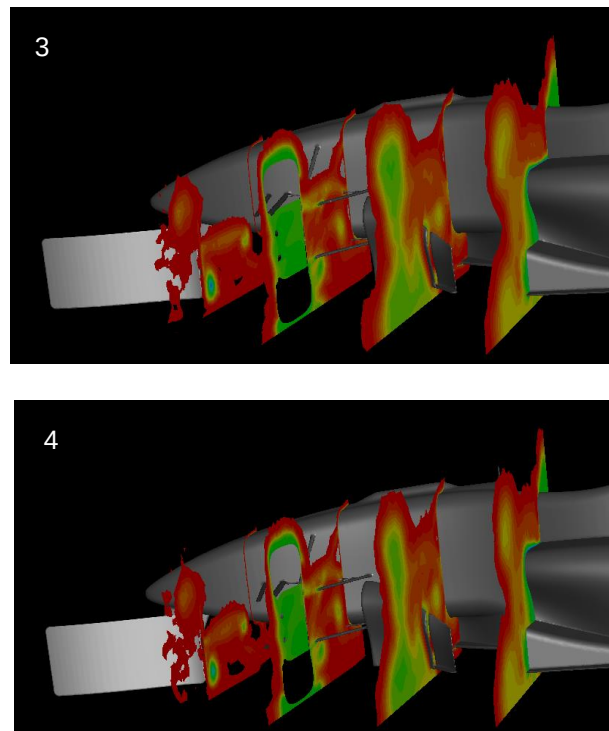


Figure 126 - Total Pressure Contours of the Wheel Wake

Figure 126, demonstrates the reduction in interaction between the wheel and primary vortex. The vortex can still be observed travelling immediately parallel to the wheel, its reduction in diameter allowing a greater volume of freestream air to pass beneath it.

This results in a minor decrease in the size of the lower portion of the wake, with a small increase in total pressure within its inboard region, parallel to the barge board (Figure 127). It can also be observed, however, that a decrease in pressure is apparent within the centre on the lower portion of the wake.

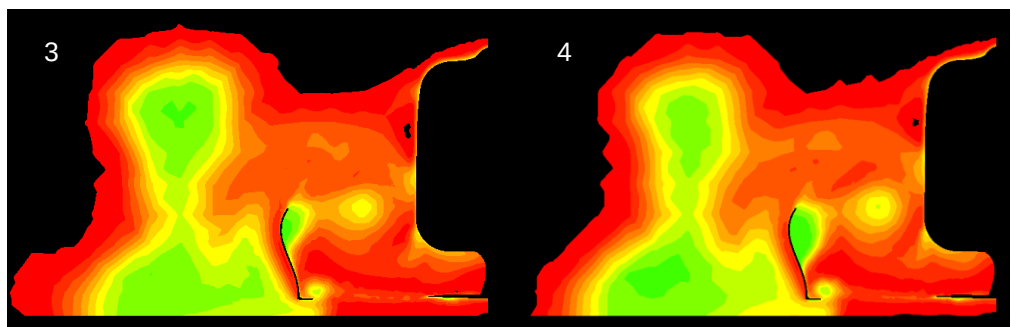


Figure 127 – Wheel Wake Total Pressure Contours (0.5m Downstream of Axis Origin)

This is as a result of the two generated contact patch vortices interacting with each other as the inboard vortex is pushed out. Allowing the wing to be twisted has allowed the upwash from it to be ejected outward, thereby re-energising the wheels wake region.

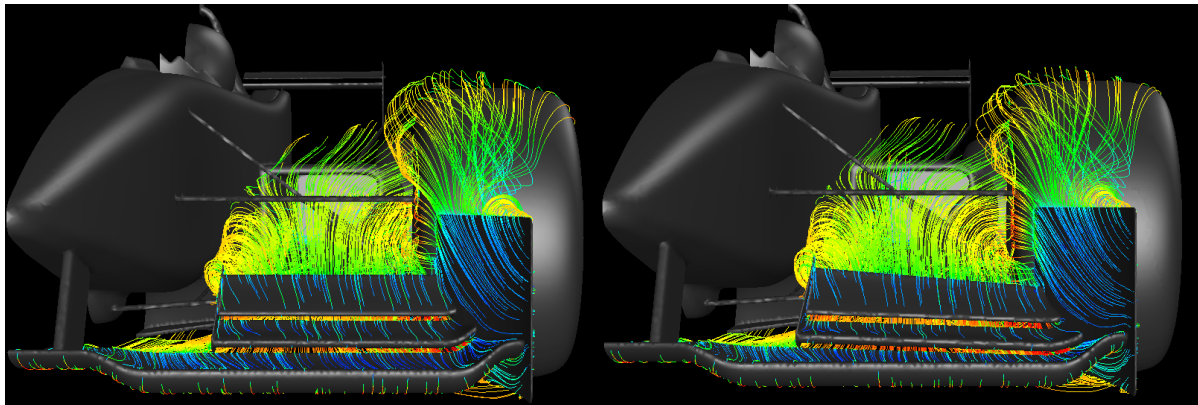


Figure 128 - Secondary Element Design Velocity Pathlines

The resulting wing outwash is clearly visible in Figure 128. Twisting wings in this manner is commonly employed in the aerospace industry in order to increase wing span efficiency and generate further aerodynamic stability whilst reducing induced drag (Ismail, et al., 2018). The twist combined with the rotation about the Z axis has directed the upwash outboard, towards the rear of the wheel and has resulted in reducing the diameter of the upper portion of the wake.

The increase in element span utilised for Design 4 has consequently relocated the initiation point of the Y250 vortex. The original geometry utilised the barge boards to alter the trajectory of the Y250 towards the outboard region, which pushes the wheel wake away and seals the floor.

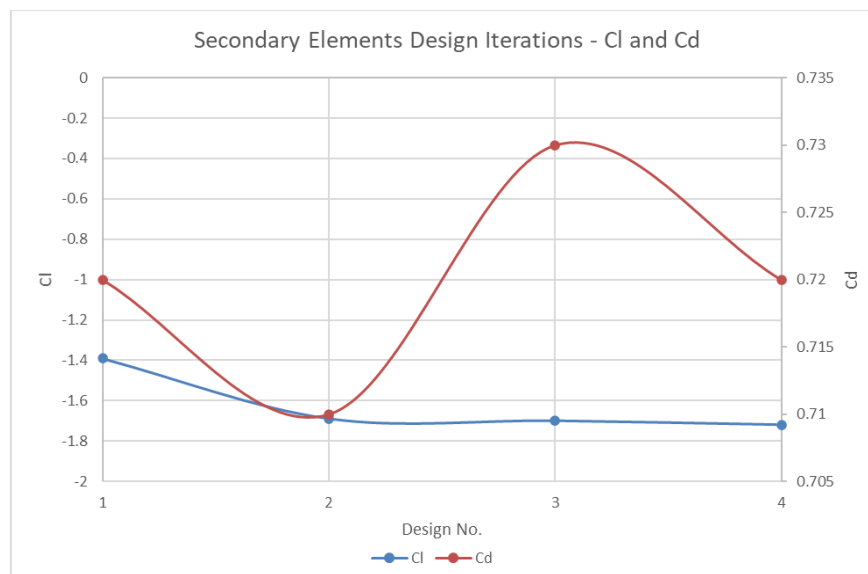


Figure 129 - Secondary Elements Design Iterations (Cl and Cd)

As Figure 129, illustrates that a minor increase in $-Cl$ is produced (design 4 $Cl = -0.72$), however, this is still less than that produced by the baseline geometry (-0.84). Cd , however, has also decreased

from that of the baseline by 2.78%, although when examining *Figure 130*, it can be observed that the CoP has relocated further downstream.

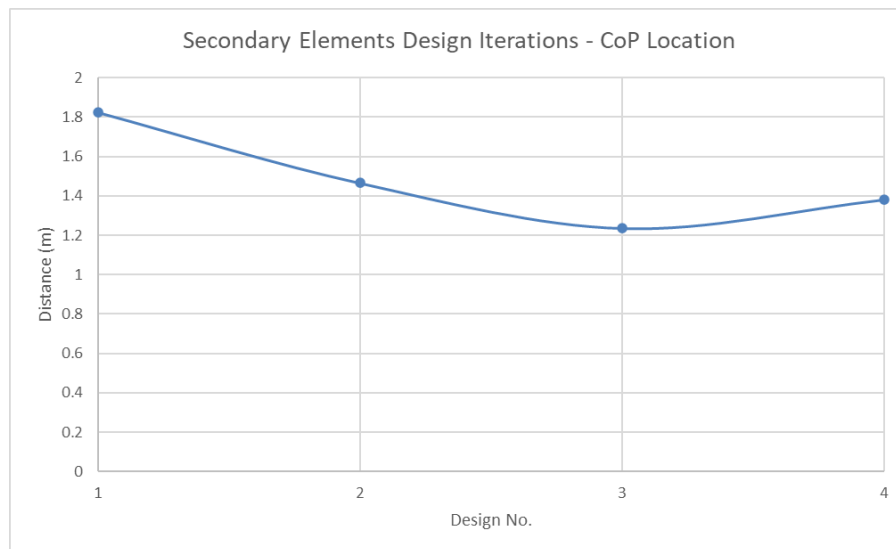


Figure 130 - Secondary Elements Design Iterations (CoP Location)

This could be as a result of the Y250 vortex being ejected into the venturi tunnels, increasing the throat velocity and lowering the pressure. This action, combined with the reduction in the negative pressure spike observed on the front wing, redistributes the aerodynamic load further downstream, producing 48.34% front load, which is 4.43% less than the baseline geometry.

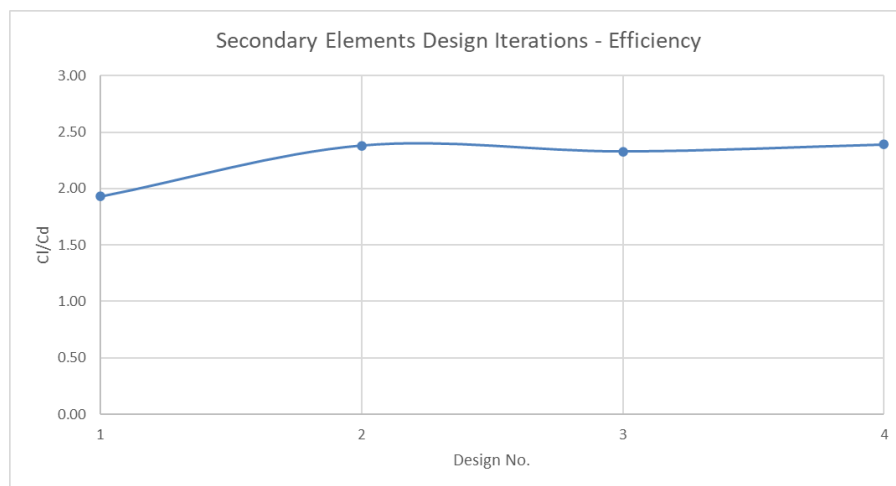


Figure 131 - Secondary Elements Design Iteration (Efficiency)

Although the CoP has moved rearwards, the aerodynamic efficiency of the fourth design has proven to be the more beneficial, however, only by a minor amount (2.52% relative to Design 3), although it is considered that the out washing capabilities of the fourth design could be utilised and improved by introduction of an end plate and turning vane design.

3.6.4 Design Iteration and Baseline Analysis

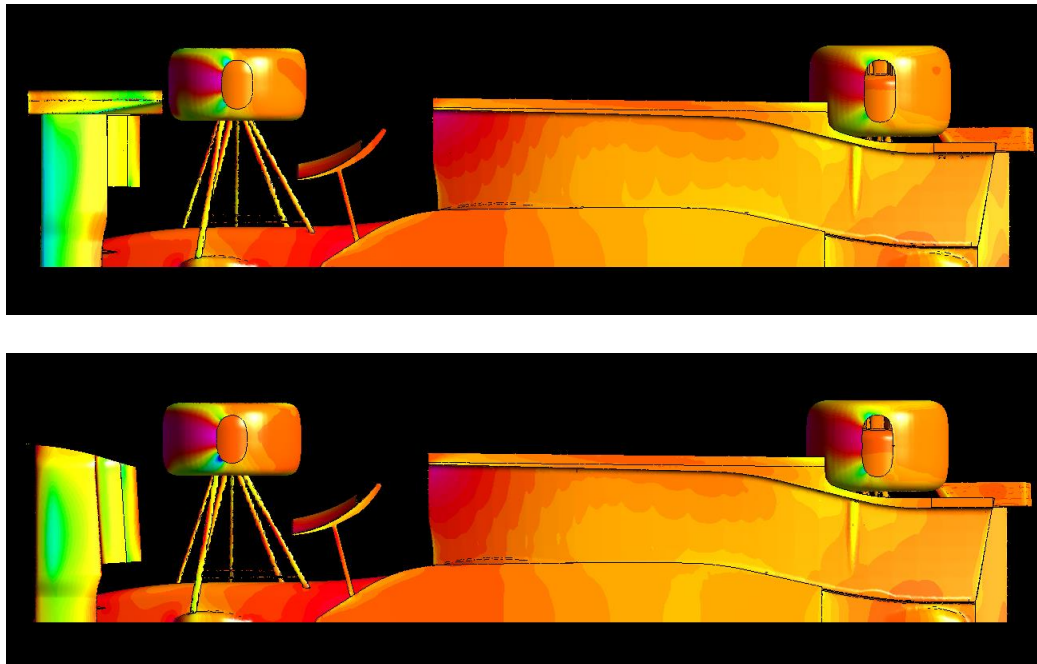


Figure 132 - Static Surface Pressure Contours (Top - Baseline, Bottom - Design 7)

Figure 132, illustrates static pressure contours on the underside of the F399 using both baseline and Design 7 (Design 4 in the previous chapter) front wings. It can be clearly observed that the baseline main plane achieves a significantly greater negative pressure along the entire span of the main plane. A peak negative pressure of over -3000Pa is achieved on the baseline main plane, however, this remains localised along its leading edge. This is in contrast to the peak negative pressure observed on Design 7, producing merely -2250Pa on its main plane and does not achieve an even spread across the wing span.

This could be as a result of the employment of the inwash style end plates, used to produce a greater aspect ratio and to control the end plate vortices. The raised outboard edge will also reduce the ground effect phenomenon, assisted by the twisted style flaps.

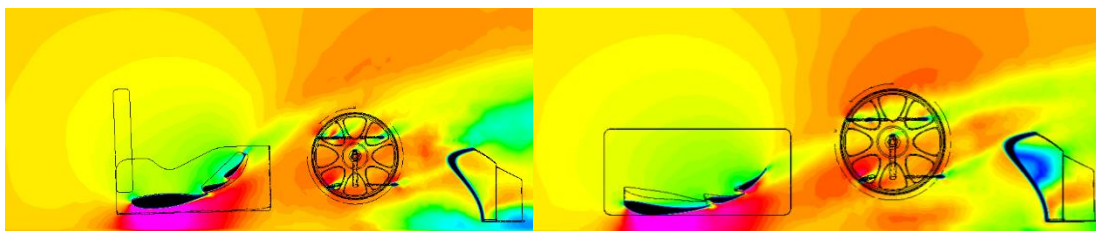


Figure 133 – Front Wing Velocity Contours on a Plane 0.45m Lateral of the Centreline (Left - Baseline, Right - Design 7)

The velocity contours in Figure 133 also illustrate the upwash generated by both front wings. It can be observed that the baseline produces a much greater angle of upwash compared to that in Design 7, although Design 7 demonstrates a significantly greater length of velocity profile resulting from it not being dispersed through such a great angle, therefore the wake is far more concentrated.

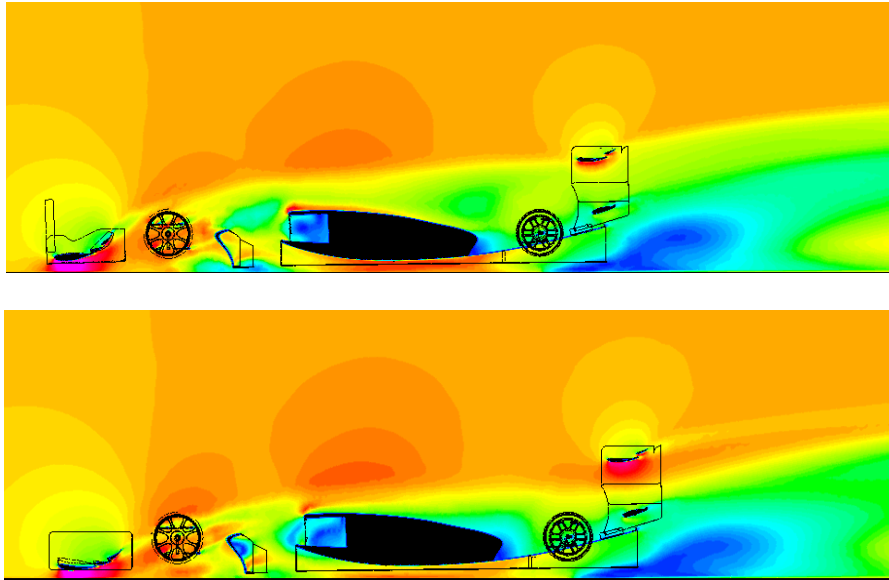


Figure 134 - Whole Car Velocity Contours on a Plane 0.45m Lateral of the Centreline (Top – Baseline, Bottom – Design 7)

However, the lower upwash has a consequence of not impacting the performance of the rear wing, with Figure 134 showing a much greater velocity on its suction side. Despite the rear wing performing harder, the low ground proximity of Design 7 has a significant adverse effect on the floor, as is presented in the above figure.

Table 5 - Average Velocity and Pressure at Venturi Entrance

	V (m/s)	P (Pa)
Baseline	30.06	71.36
Design 7	27.79	97.72

Table 5, presents the average values of velocity and pressure observed at the venturi entry. As stated, the ground proximity of Design 7 restricts the onset airflow to the floor section.

Table 6 - Average Velocity and Pressure at Venturi Throat

	V (m/s)	P (Pa)
Baseline	39.29	-458.07
Design 7	37.46	-437.28

This then has a “knock on” effect further downstream within the throat section of the floor, as indicated in Table 6. As such a large working area has been adversely affected, this results in the loss of total downforce and the CoP location moving downstream. Design 7 achieves a 48.34% forward bias of total downforce, whereas the baseline achieves 52.76%.

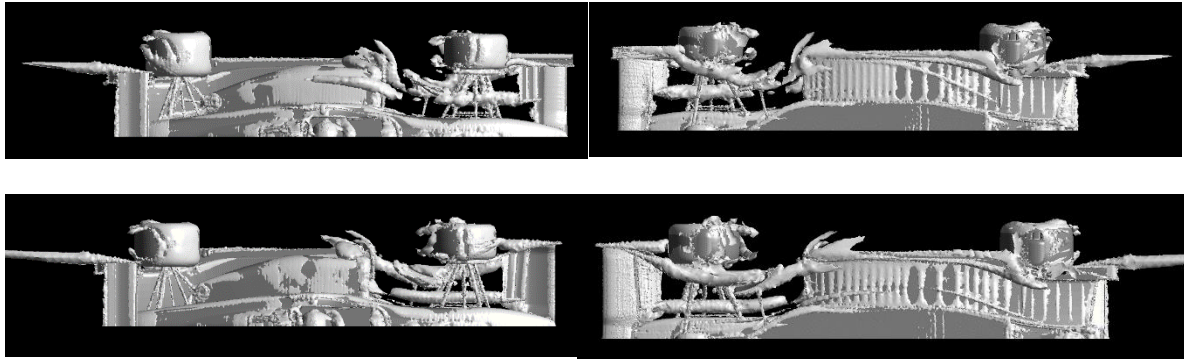


Figure 135 - Vortex Swirling Strength (1Hz) (Top - Baseline, Bottom - Design 7)

As Design 7 utilises an extension of the flap elements in order to achieve a greater aspect ratio and improve efficiency, this also has the effect of moving the Y250 vortex further inboard (Figure 135), therefore producing no interaction with the barge boards. This results in the vortex colliding with the leading edge of the side pod and creating an area of separation over its leading edge (Figure 134), whereas the baseline illustrates the vortex manoeuvring over the leading edge of the side pod, reenergising the boundary layer and achieving increased flow attachment. This can also be adjusted by decreasing the flap angle, allowing the suction side of the barge boards to carry the vortex outboard and around the side of the car.

The primary vortices within both designs are inherently different. It can be observed that the baseline primary vortex has a significantly stronger interaction with the front wheel. As a result of its proximity to the ground and rotation direction, the primary vortex will naturally travel inboard of the front wheel, therefore employing a foot plate restricts this movement (Pegrum, 2006).

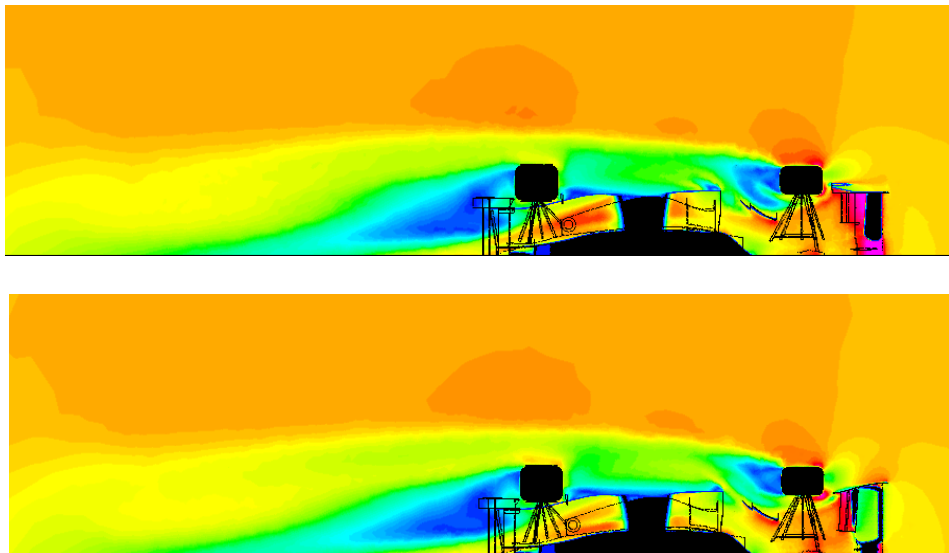


Figure 136 - Wake Velocity Contours 0.1m from the Ground Plane (Top - Baseline, Bottom - Design 7)

As Design 7 allows the vortex to travel inboard, its energy losses are not as great, therefore its interaction with the barge boards is far superior, generating a reduction in front wheel wake size and increasing the outwash effect. This can be observed in Figure 136. It must also be stated that a greater AoA will also have the effect of increasing the vortex diameter at a much greater rate, due to

the pressure gradient, therefore dispersing its energy at a greater rate. The twisted flap elements utilised in Design 7 assist in limiting this, therefore, the vortex retains its rotational energy for a longer duration.

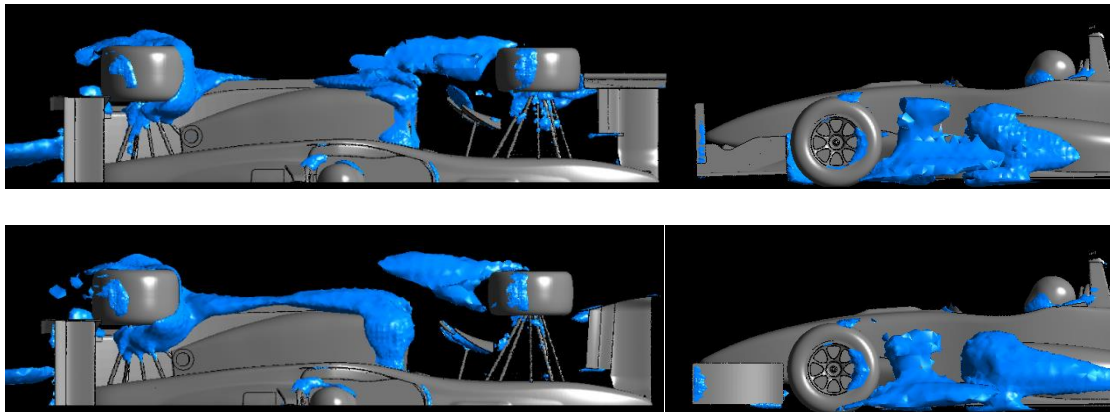


Figure 137 - Turbulent Kinetic Energy (60J/Kg) (Top – Baseline, Bottom – Design 7)

The isosurface illustrated in *Figure 137* presents the wake formations in terms of turbulent kinetic energy. The outwash effect and wake size reduction of Design 7 is clearly apparent when comparing the two wings, although it must be acknowledged that utilising Design 7 results in significantly more turbulent energy around the outboard face of the side pod. This is as a result of the inboard proximity and height of the Y250 vortex.

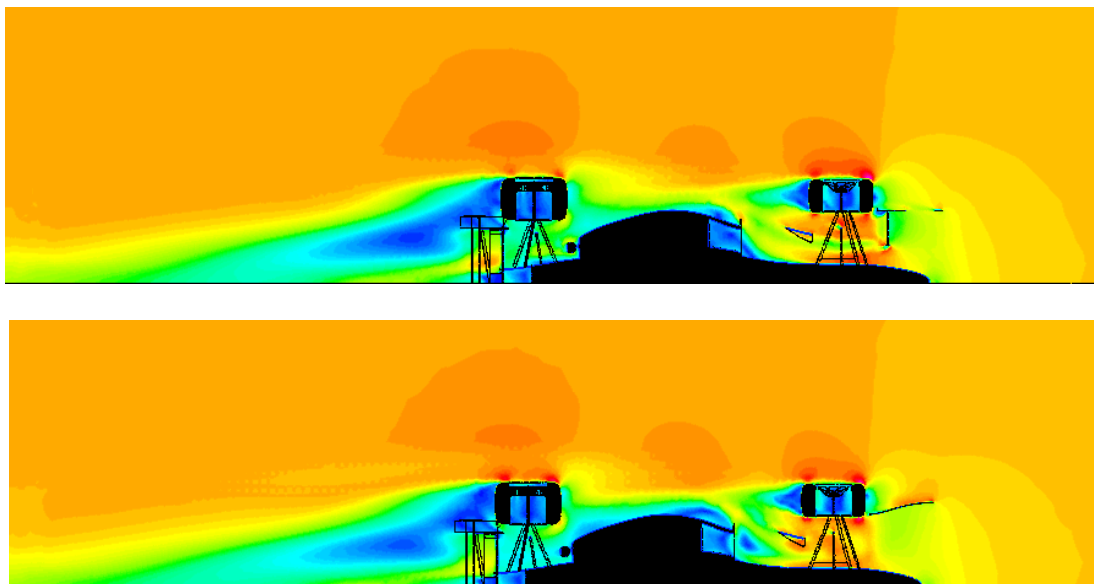


Figure 138 - Wake Velocity Contours 0.2807m from the Ground Plane (Top - Baseline, Bottom - Design 7)

Figure 138, illustrates how the Y250 vortex interacts with the barge boards on the baseline design. The relocation of the Y250 vortex results in an area of extreme negative pressure forward of the side pod entry. This reduces the boundary attachment flowing around the outboard edge of the side pod resulting from the large reduction in energy, thus separation is produced, causing the increase in turbulence.

3.7.0 Concept Development

In order to develop the concept further, ten concept designs were constructed, each a progression of the original Design 7, presented in the previous chapter.

Concepts 1, 2 and 3 were selected for further analysis based on the efficiencies and CoP locations obtained (*Figures 141 and 142*), with comparisons being made to the baseline geometry. A visual representation of each concept geometry is illustrated in chapter 9.0.0 Appendix C.

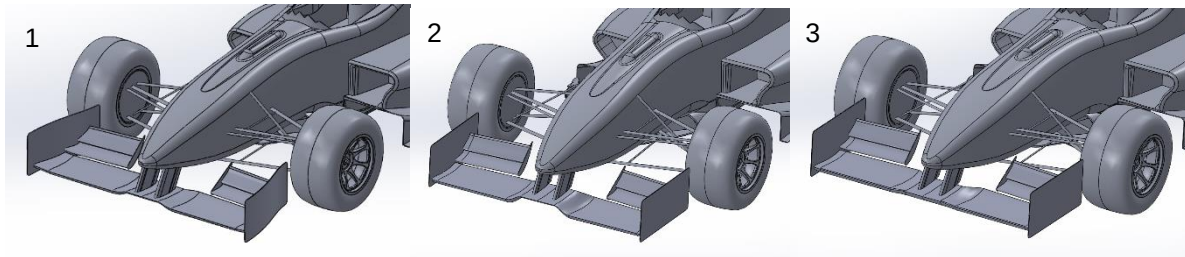


Figure 139 - Concepts 1, 2 and 3

Figure 139 illustrates the concept designs concerned.

Concept 1 is primarily based on Design 7, however the main plane centre and out board edges have a reduced ground proximity, for greater span ground effect, with $H/c = 0.21$ and a 2 degree increase in AoA. In order to compensate for the additional pressure drag induced by the AoA, the span of the flap elements has been reduced by 40mm, which should also help the Y250 vortex interact with the barge boards.

Concept 2 includes a 10% decrease in main plane centre chord length, a 40mm increase in centre height and a 3 degree increase in centre AoA. Outboard of the centre section, the main plane chord has been increased by 15% in order to increase the working area, but this does reduce the aspect ratio. Reducing the aspect ratio can lead to lower lift values, however, it can produce further aerodynamic stability when AoA is increased (Scott, 2006). This geometry change has ultimately lowered the value of H/c to 0.2. The outboard end plate geometry has been simplified in order to further increase the planar area and reduce the risk of adverse pressure effects of the suction sides of the wing assembly.

Concept 3 employs a 30mm reduction in centre height relative to Concept 2. Subsequent further investigation, found that the baseline main profile was that of the Nasa GA(W)-1, which is considered to be a profile having low speed and high lift characteristics, having an increase in leading edge radius in order to reduce peak pressure (Zhang, et al., 2011). However, this is contrary to previous findings, which displayed high localised pressure peaks at the leading edge. Based on an experiment conducted by Sharma, it was decided that increasing the aerofoil thickness may yield positive results by increasing peak pressures (Sharma, 2017). The wing assembly AoA was also increased by 2 degrees to further increase the pressure gradient and generate further suction force.

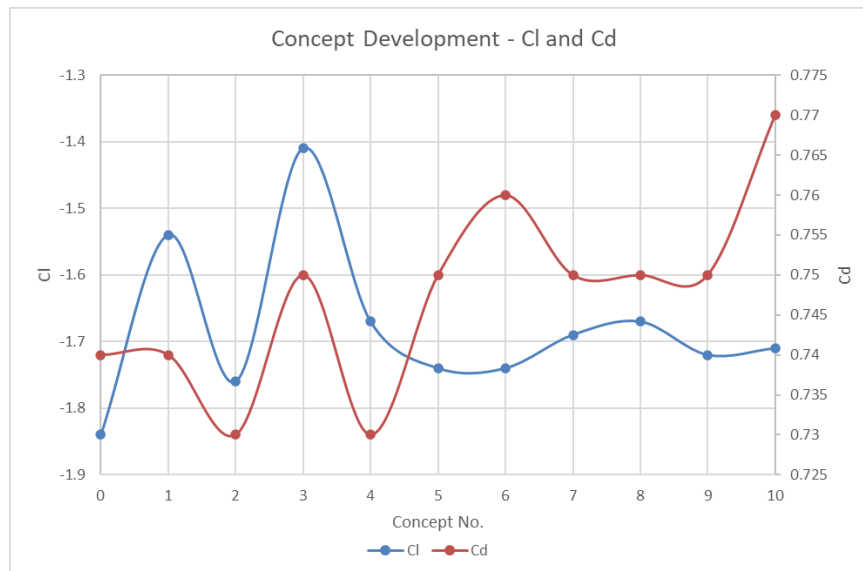


Figure 140 - Concept Lift and Drag Coefficients

Figure 140 presents the resulting lift and drag coefficients produced by each concept design (Baseline = Concept 0). Despite the development of ten concepts, none reach the Cl value achieved by the baseline. Concept 2 indicates the lowest error, achieving a 4.35% reduction, while Concept 3 produces a 23.37% reduction.

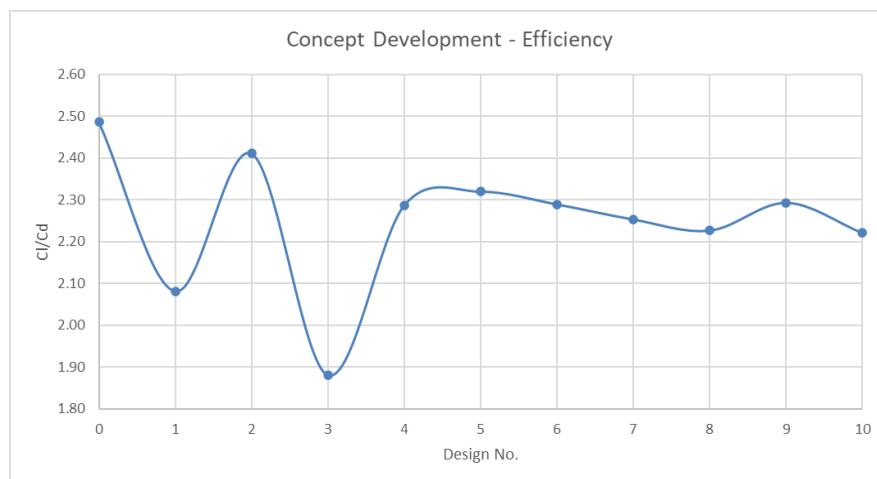


Figure 141 - Concept Aerodynamic Efficiency

This results from Concept 2 achieving the highest aerodynamic efficiency of all the concepts, while Concept 3 produces the poorest results, although Concept 2 still falls short of the efficiency produced by the baseline geometry (3% reduction).

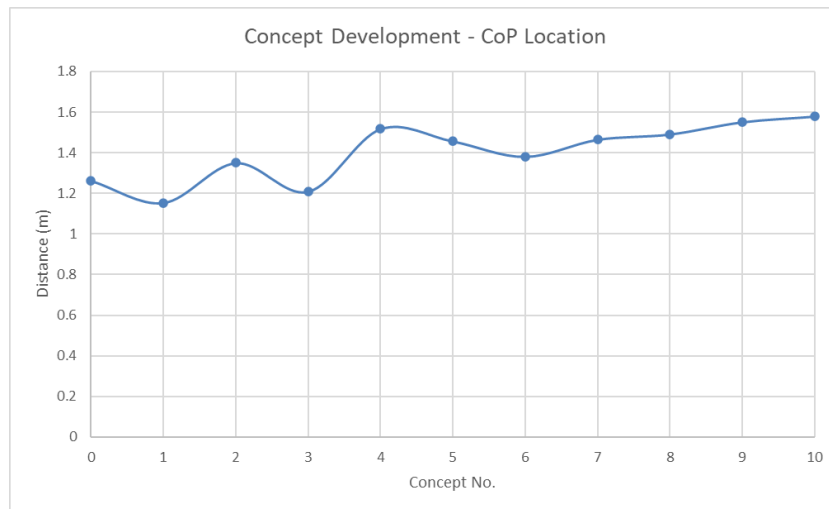


Figure 142 - Concept CoP Location

When referring to *Figure 142*, it can be observed that Concept 1 yields the greatest CoP relocation distance, followed by Concept 3 and then Concept 2.

Table 7 - Concept Design Downforce Balance

	Percentage Forward (%)	Front Downforce (N)	Rear Downforce (N)	Total Downforce (N)
Baseline	52.79	1065	952	2017
Concept 1	56.86	960	728	1688
Concept 2	49.43	956	974	1930
Concept 3	54.80	847	699	1546

The result being that Concept 1 achieves the greatest forward percentage of downforce, however, developing only the second lowest total downforce (*Table 7*). Although Concept 3 generates the second highest value of percentage forward, its frontal and rear force values are the lowest of all three concepts due to its lack in generated total downforce.

This demonstrates that moving the CoP forward is achievable, although thus far this has proved detrimental in achieving adequate total or rear downforce. This results in the net results of downforce falling short of that achieved by the baseline front wing.

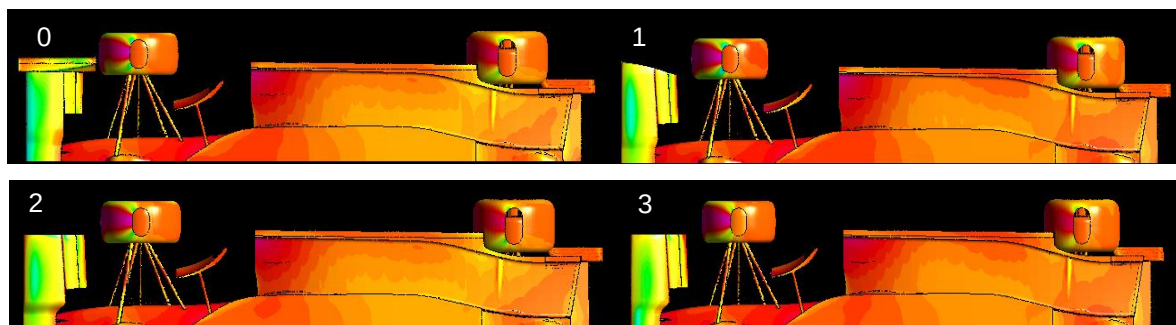


Figure 143 - Concept Suction Surface Static Pressure Contours

Examination of *Figure 143* reveals that Concepts 1 and 3 show an increase in pressure achieved on the floor surface, while the baseline and Concept 2 show similar underfloor pressure profiles, although

the area in which the lowest pressure is achieved on Concept 2 is marginally reduced compared to that of the baseline.



Figure 144 - Floor Static Pressure Contours

However, observing the floor region located closest to the barge boards (Figure 144) indicates that Concept 2 has an overall lower pressure covering this area.

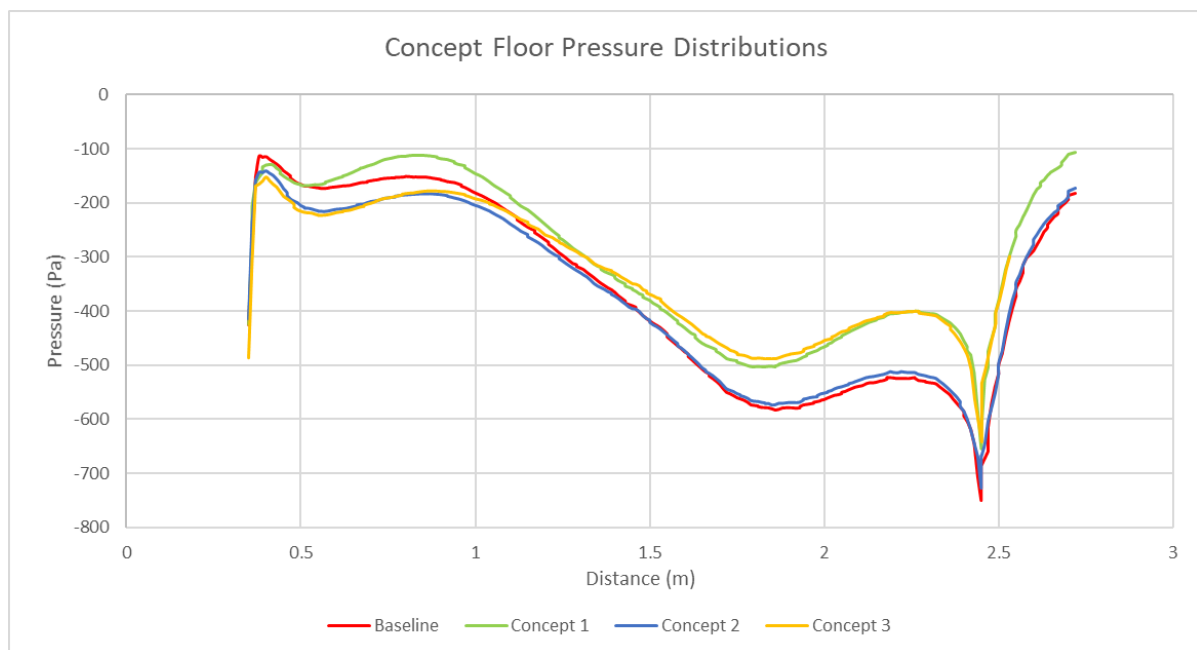


Figure 145 - Concept Floor Pressure Distributions

This is further identified when examining the pressure distributions located along the floor illustrated in Figure 145. Although Concept 2 presents a lower initial pressure along the floor, it can be observed to begin to follow the same trend as the baseline, although it does begin to deviate at 1.7m. Concept 3 also generates a lower pressure at the forward section of the floor, however, this rapidly deviates from the desired trend after 1m.

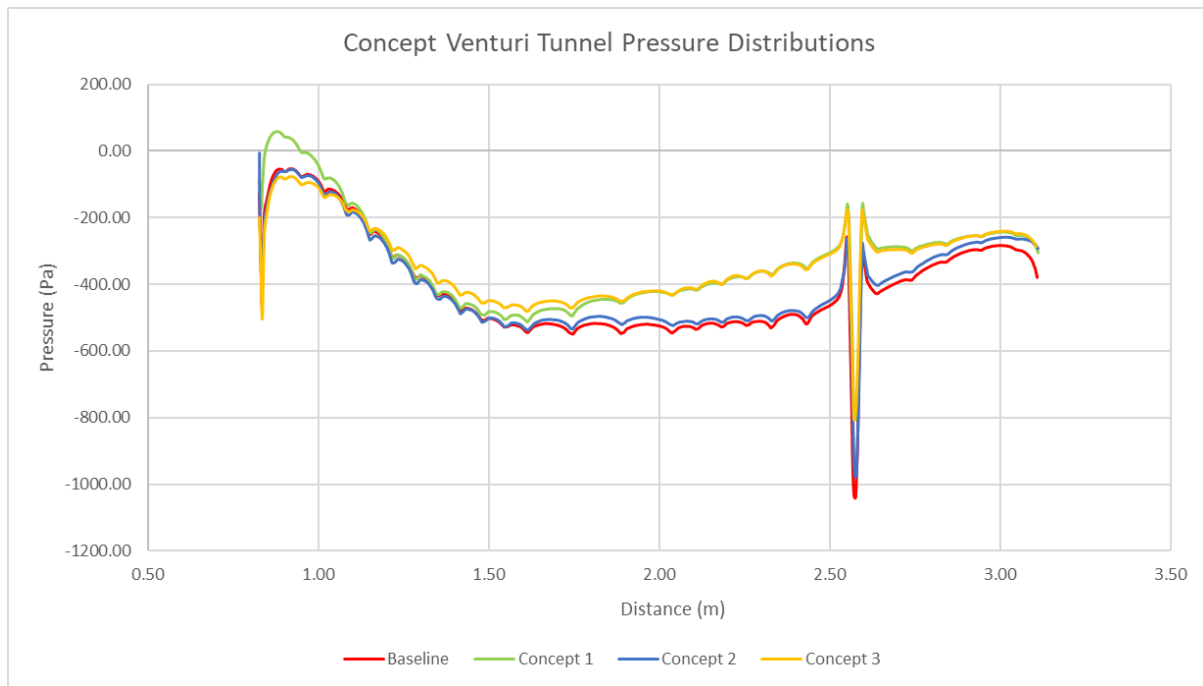


Figure 146 - Concept Venturi Tunnel Pressure Distributions

A similar trend to that illustrated in Figure 145 is presented when observing the pressure distributions generated in the venturi tunnels (Figure 146).

Table 8 - Concept Venturi Tunnel Entry Average Velocity and Pressure

	Velocity (m/s)	Pressure (Pa)
Baseline	29.08	71.01
Concept 1	28.29	152.37
Concept 2	29.62	41.81
Concept 3	28.12	2.36

Table 8, indicates the average air velocity and pressure developed at the venturi tunnel entrance. However, the existence of an anomaly can be identified within Concept 3, with a large pressure reduction observed at the venturi entry.

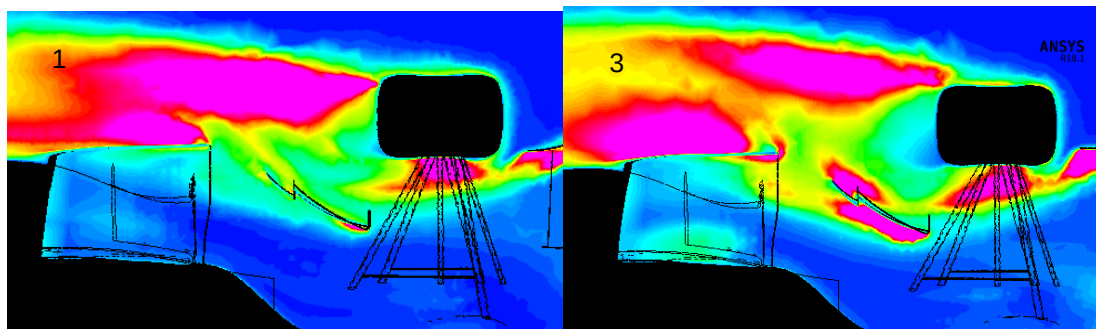


Figure 147 - Concept 3 Turbulent Kinetic Energy Contours (60J/Kg)

The dramatic reduction in pressure is as a consequence of large areas of high turbulence being produced at the venturi entrance (Figure 147). This phenomenon is as a result of boundary separation

on the suction side of the barge board, which results from the shedding of the end plate primary vortex, and the production of turbulent energy interacting within the venturi tunnel. This destabilises the boundary flow within the tunnel section, thus reducing the static pressure resulting from the wake formation and separation, although this will result in the drag increase illustrated in *Figure 140*, arising from the increase in turbulent kinetic energy (non-laminar).

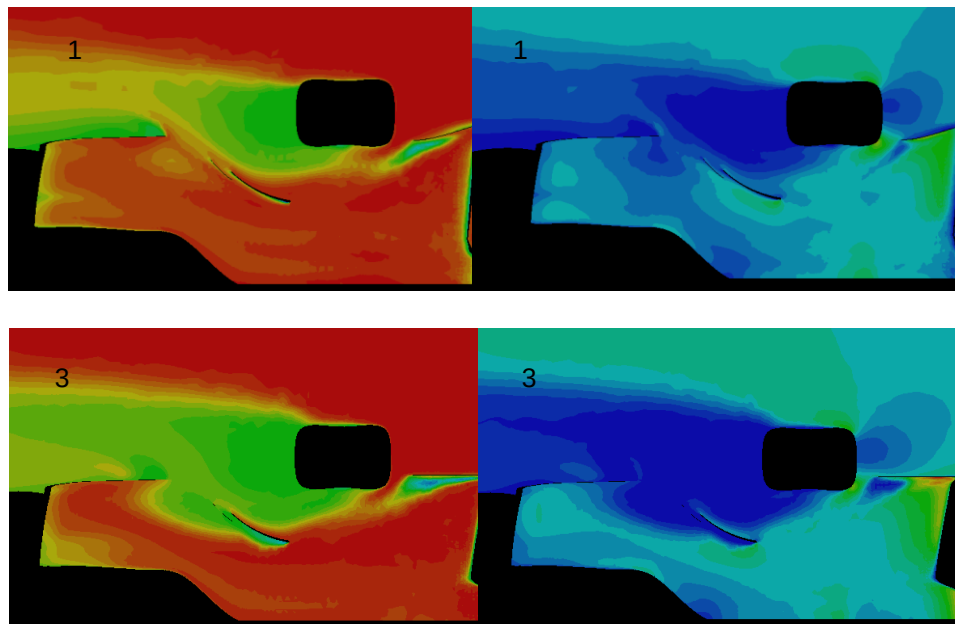


Figure 148 - Left - Total Pressure (Pa), Right - Dynamic Pressure (Pa)

This is further illustrated in *Figure 148*. It can be observed that an area of extremely low dynamic pressure is present at the venturi entry, which, combined with the low static pressure results in a low total pressure (separation), creating a blockage for the high energy freestream flow, impacting on the downstream surface pressures (*Figure 146*).

Table 9 - Concept Venturi Tunnel Throat Average Velocity and Pressure

	Velocity (m/s)	Pressure (Pa)
Baseline	40.84	-400.94
Concept 1	40.45	-383.65
Concept 2	40.28	-397.32
Concept 3	39.70	-363.77

Table 9, exhibits the described phenomenon, by indicating the average velocities and static pressures observed at the venturi throat. It can be observed that as a result of the blockage, Concept 3 has a reduction in air flow velocity, ultimately increasing the negative static pressure. This further indicates that the floor and venturi sections are being impacted by the design of the front wing.

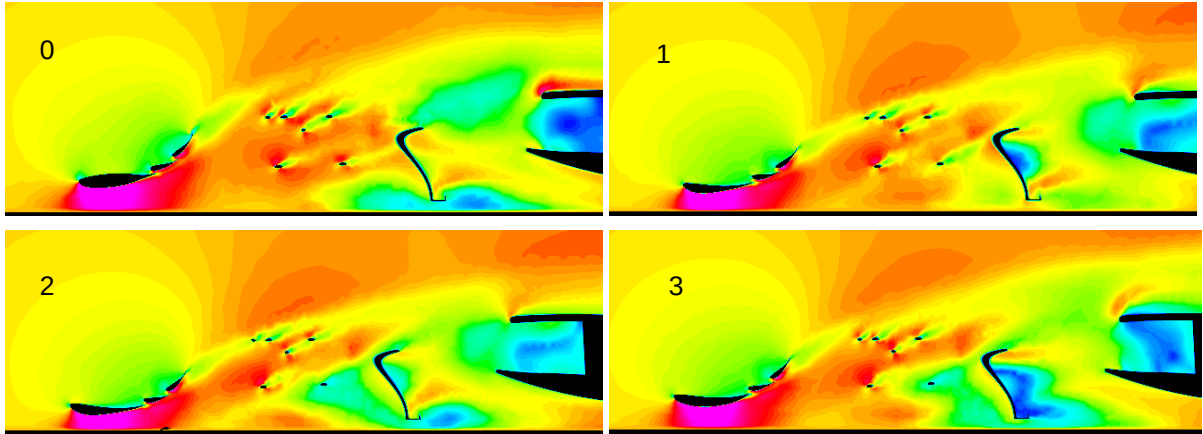


Figure 149 - Concept Front Wing Velocity Contours

Figure 149, illustrates velocity contours projected onto a plane situated 0.45m from the vehicle centreline. It can be observed that the baseline front wing continues to generate the greatest proportion of fluid upwash, as referred to in the previous chapter. This is achieved as a result of the main plane being able to utilise a greater AoA, as the baseline main plane employs a greater thickness/chord ratio. Rad et al, found that a greater thickness/chord ratio on symmetrical aerofoils (Nasa GA(W)-1 is almost symmetrical along its entire chord length for superior drag characteristics, however, utilises a small pressure recovery section at its trailing edge to further enhance its lifting potential) allowed a much greater AoA to be utilised, however, the lifting capabilities were reduced if H/c was too low, requiring it to be situated at a higher ground proximity for optimal use (Rad & Kazemi, 2001). The increase in ground proximity allows a greater mass flow of undisturbed freestream air to be fed to the floor section of the car.

Further observations can be made at the barge board area. As previously discussed, Concept 3 develops a large area of separation on the suction side of the barge board which is clearly visible in Figure 149. The same phenomenon can be examined on Concept 1, however, it is not as apparent to the magnitude of Concept 3.

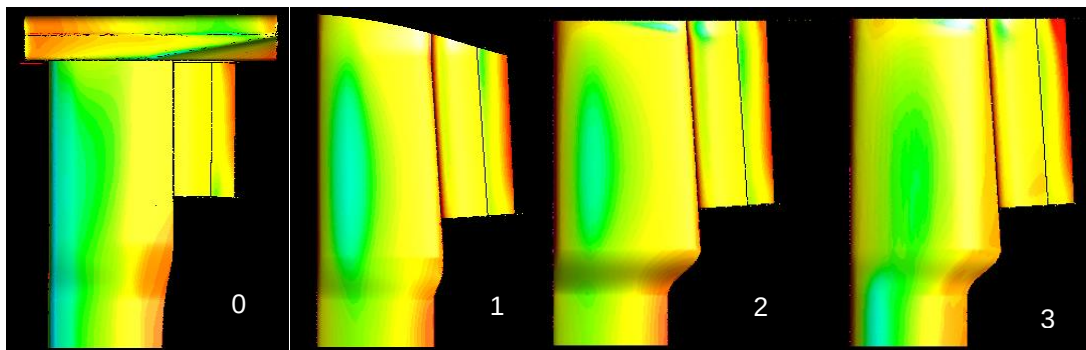


Figure 150 - Concept Front Wing Suction Surface Pressure Contours

Figure 150, illustrates the pressure profile of the wing concepts concerned. Concept 1 produces the greatest negative peak pressure of the three concept wings at -2750Pa on its outboard profile, followed by Concept 2 at -2400Pa and Concept 3 at -2000Pa (Figure 151).

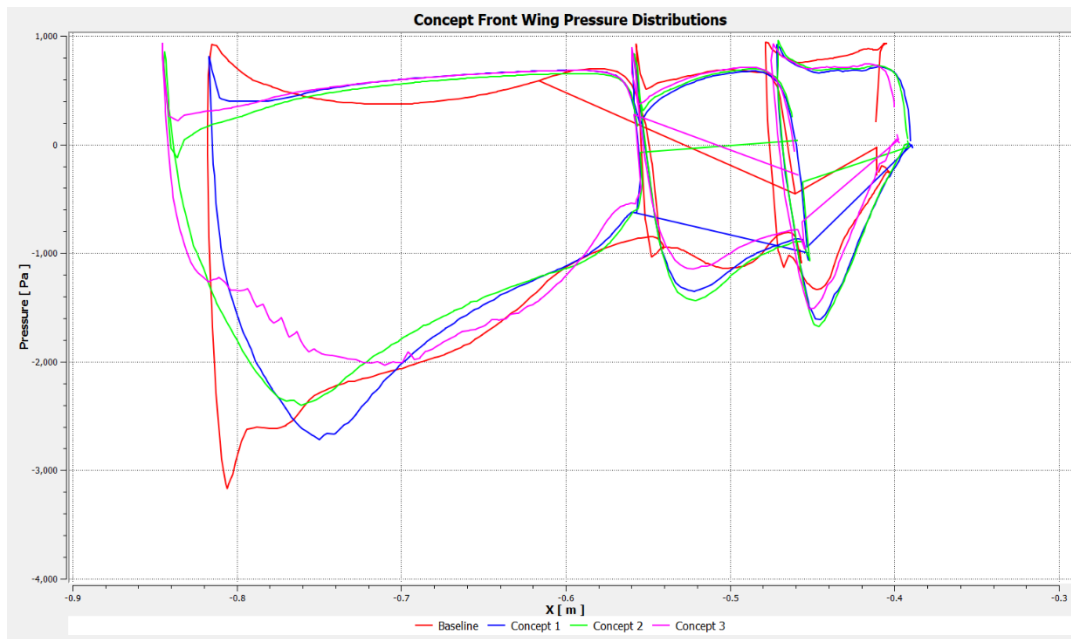


Figure 151 - Concept Front Wing Pressure Distributions

Extending the chord length has had the effect of centralising the area of maximum suction pressure (Figure 151), however, it has also started to produce a small localised area of negative pressure on its extreme outboard edge resulting from a pressure bleed developing a strong primary vortex. It can also be observed that Concept 3 generates the majority of its suction force on the central plane.

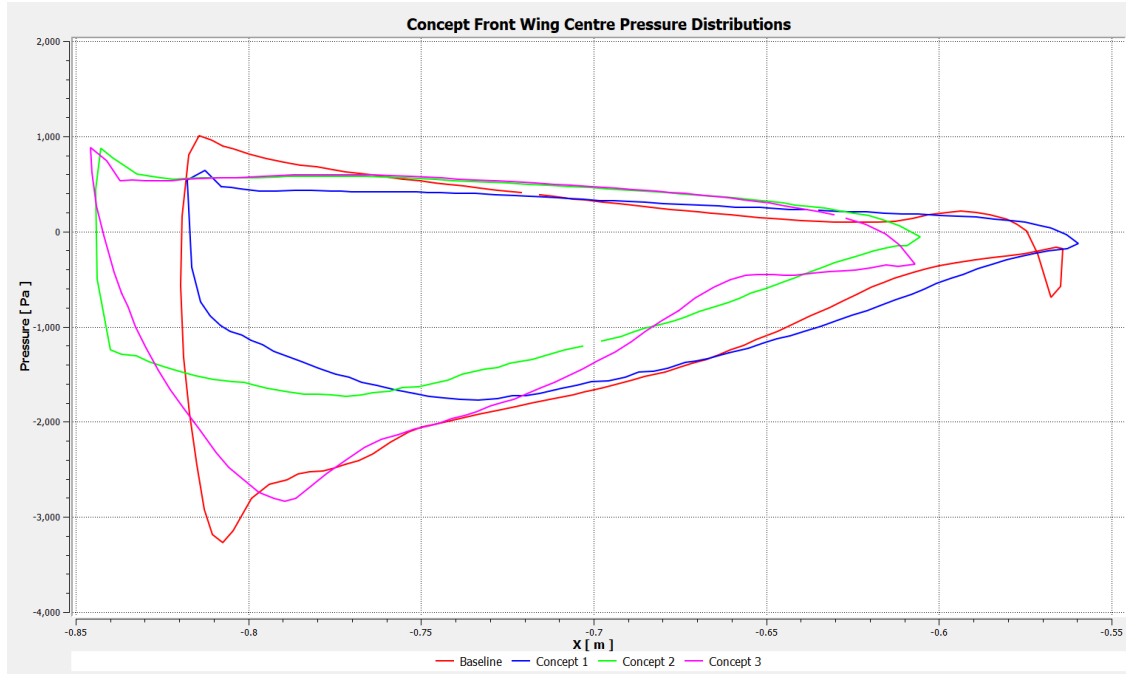


Figure 152 - Concept Front Wing Centre Pressure Distributions

This characteristic produces a significant central loading on the wing. The generated upwash increases the pressure on the underside of the nose cone, which further contributes to the vehicles lesser value of total downforce.

In examining *Figures 149, 150 and 151*, it can be readily ascertained that the secondary elements are working optimally on Concept 2, however the second flap on all three concepts is subjected to a greater amount of trailing edge separation compared to that of the baseline wing. Concept 3 displays the greatest rate of negative pressure reduction towards the trailing edge, with its outboard edge displaying the highest pressure. This is as a result of the increase in adverse pressure gradient due to the wing assembly AoA and outboard edge proximity to the front face of the wheel.

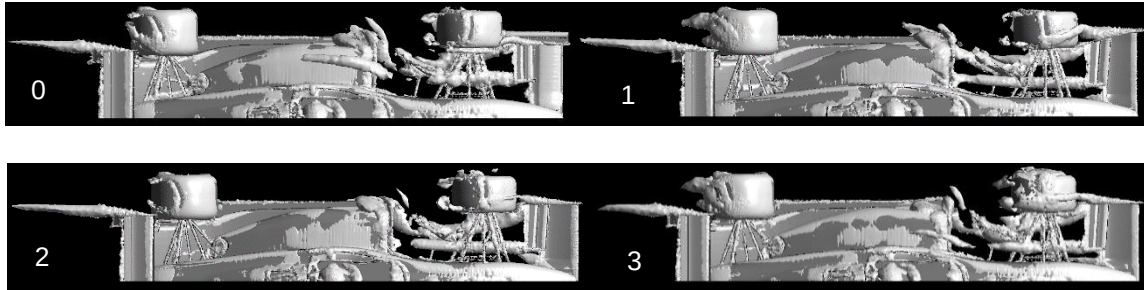


Figure 153 - Vortex Core Swirling Strength (1Hz)

The vortex structures of each concept are illustrated in *Figure 153*. The Y250 vortex developed on Concept 3 follows a similar trajectory to that of the baseline, whereas both Concepts 1 and 2 result in impacting on the front face of the side pod. Examination of *Figure 153* also indicates trends that have previously been discussed. As both the baseline and Concept 3 utilise a greater AoA, the developed Y250 vortical structures have an increase in diameter, however, the uniformity along their lengths is lacking consistency due to increased energy dissipation.

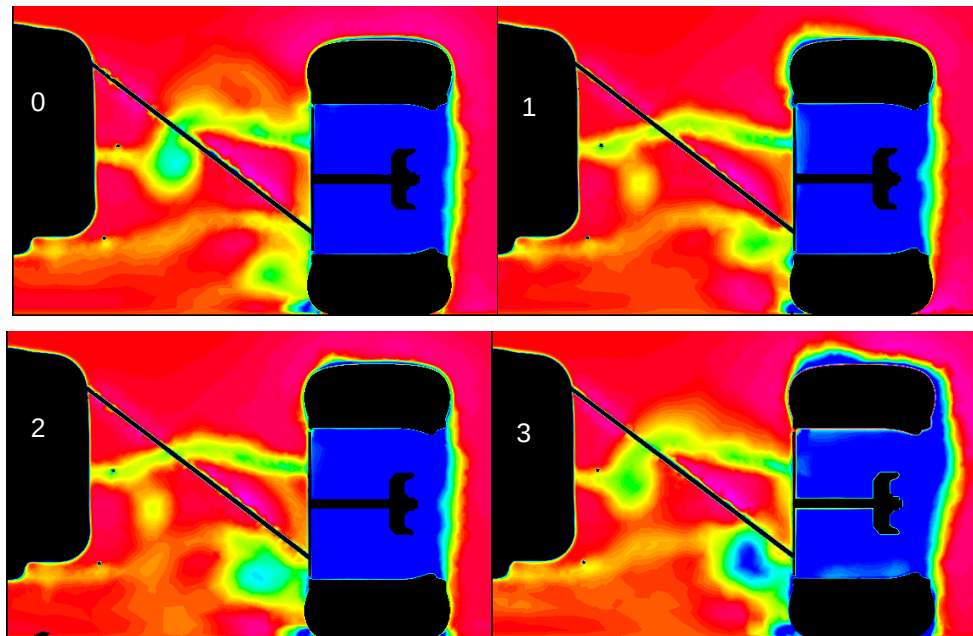


Figure 154 - Velocity Contours of Vortex Formation

Figure 154, illustrates scaled velocity contours, displaying the vortex formations on a plane located at the front wheel axle centre. The increased diameter of the Y250 vortices can be observed for the baseline and Concept 3. Concepts 2 and 3 also display much larger primary vortices, with Concept 3

illustrating the lowest core velocity. Its interaction with the inboard edge of the front wheel and tyre jet is significantly more extreme due to its increased size.

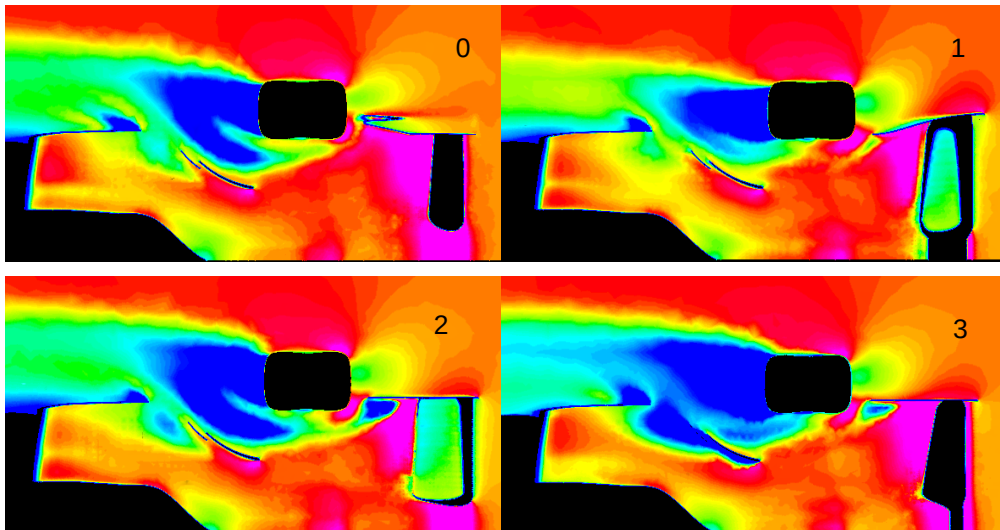


Figure 155 – Concept Front Wing Wheel Wake Velocity Contours

The vortex interaction and extreme energy dissipation, combined with the large separation zone on the barge board lead to a much greater wake formation developing. Concept 2 also indicates a small area of recirculating air at the barge board trailing edge, while Concept 1 shows the strongest flow attachment on the barge board suction side. Examination of *Figure 155* further reveals that despite Concept 1 producing the smallest wheel wake formation, its trajectory is pulled inboard. This is as a consequence of the rotational direction and energy retained within the primary vortex, drawing the lower portion of the wake inboard.

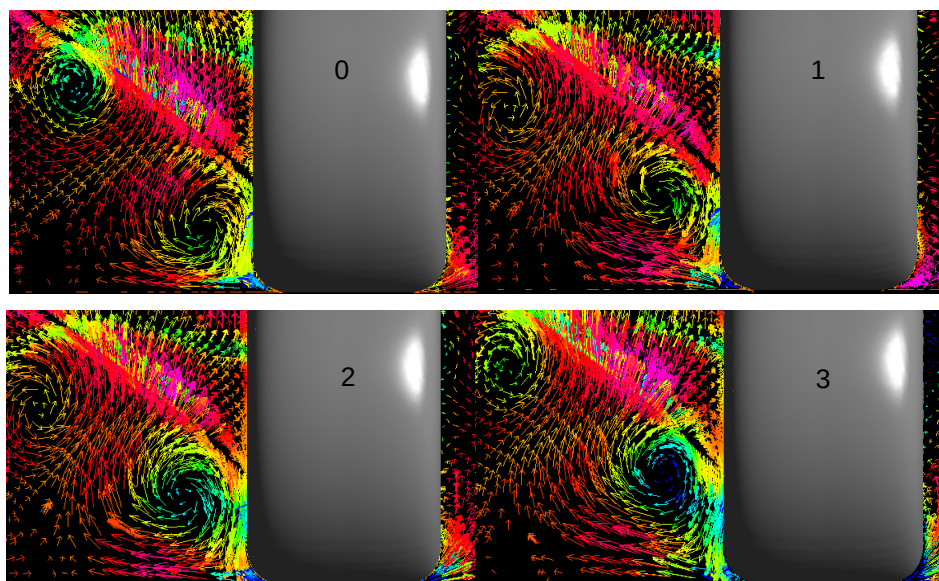


Figure 156 - Concept Front Wing Vortex Velocity Vectors

As *Figure 156* illustrates, the increased energy and ground proximity of the vortex draw inboard a much greater volume of air, which leads to the wheel wake being sucked inboard.

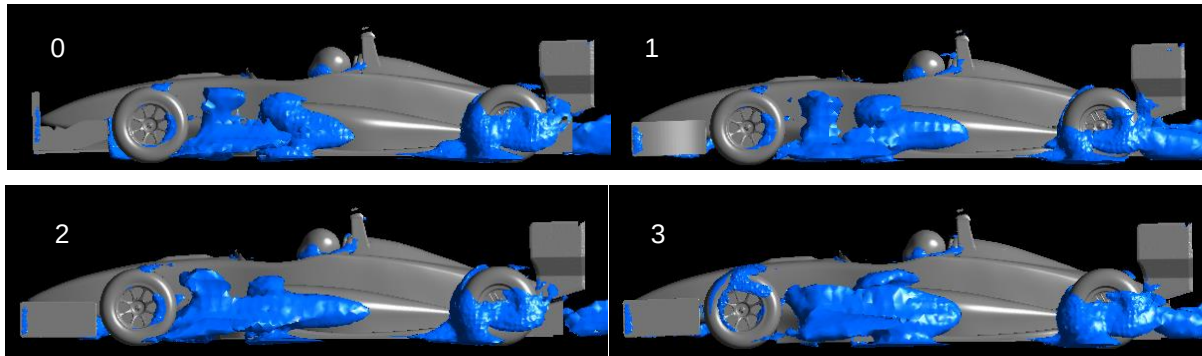


Figure 157 - Concept Front Wing Turbulent Kinetic Energy Isosurface (60J/Kg)

The front wheel wake structures are further illustrated in *Figure 157*. All three concepts display similar trends, however, some minor differences are apparent. The baseline and Concept 1 indicate the least amount of interaction between the wheel wake and the outboard surface of the side pod. Despite Concept 1 displaying inwashing trends, the reduced diameter of the primary vortex has produced the most contracted wake region. The near wake regions for the baseline, Concept 2 and 3 all display upwash traits.

Concept 3 has also produced the smallest vertically built wake area. In examining *Figure 153*, the top end plate vortex can be observed tangentially travelling over the top of the wheel. Concept 1 also displays similar traits to this, however, the increased AoA Concept 3 uses helps to carry a stronger vortical structure.

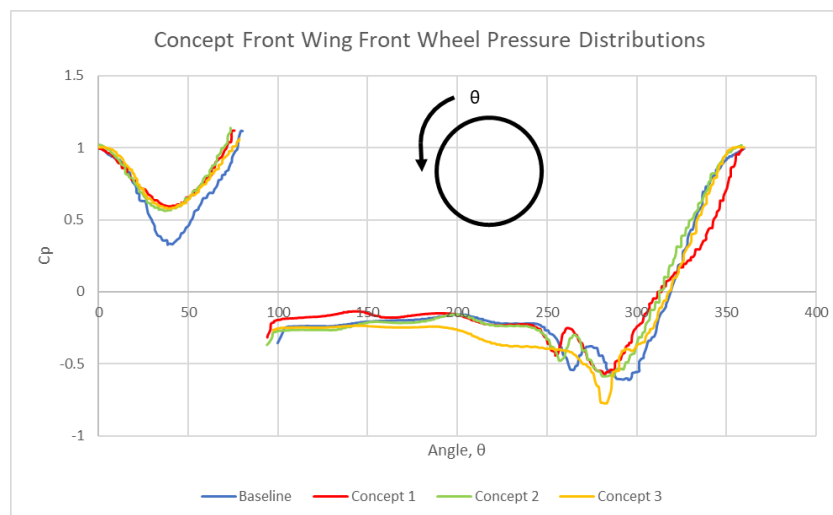


Figure 158 - Concept Front Wing Front Wheel Pressure Distributions

It appears that the vortex is creating boundary layer instability over the top surface of the wheel. This characteristic is highlighted in *Figure 158* which displays surface pressures taken from the centre circumference of the wheel. Examination of *Figure 158* suggests that Concept 3 develops a much lower negative pressure on the rear side of the wheel. Its point of boundary separation is far earlier with a stronger peak negative pressure within the separation bubble. This would characteristically

increase the developed drag force and induce additional lift force. The vortex then pushes the wake downwards as it follows the surface of the wheel.

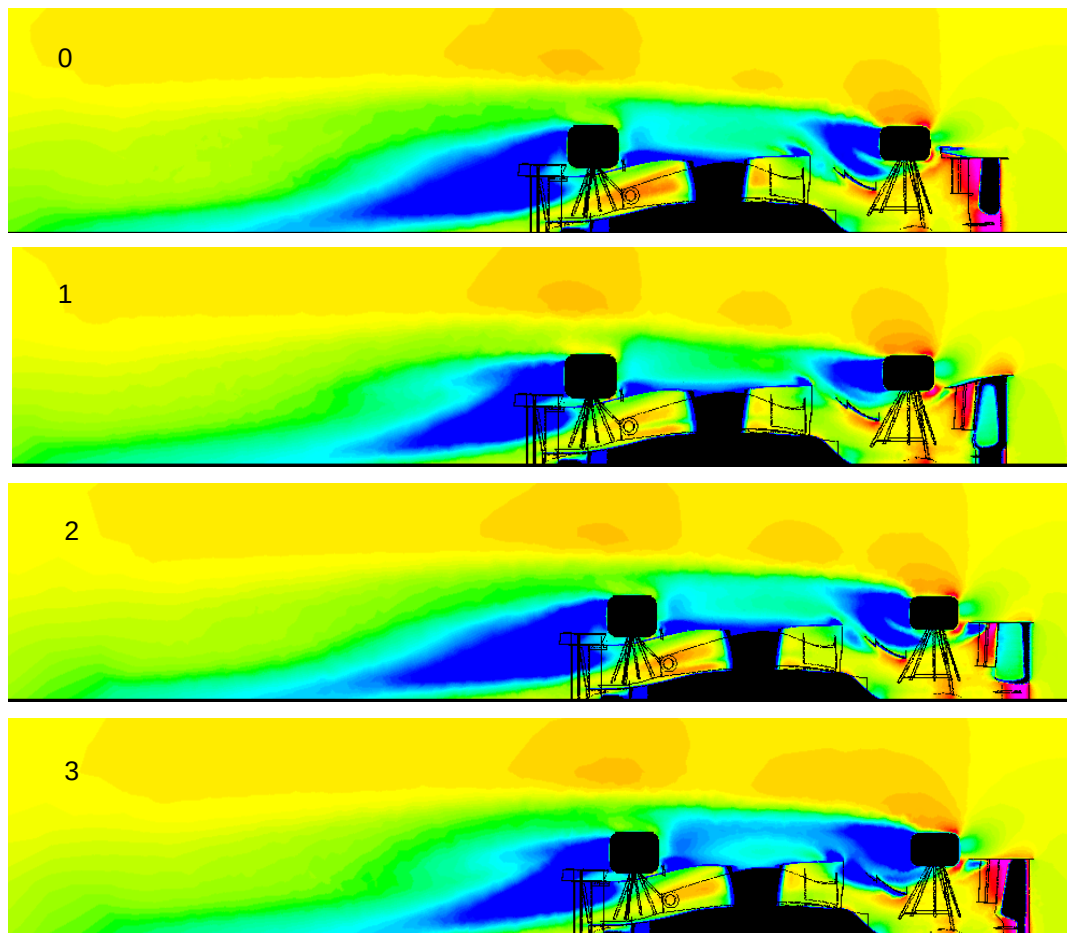


Figure 159 - Concept Front Wing Whole Car Velocity Contours

Studying *Figure 159*, the wheel wake produced by Concept 3 can be observed to have a significantly stronger interaction with the front face of the rear wheel.

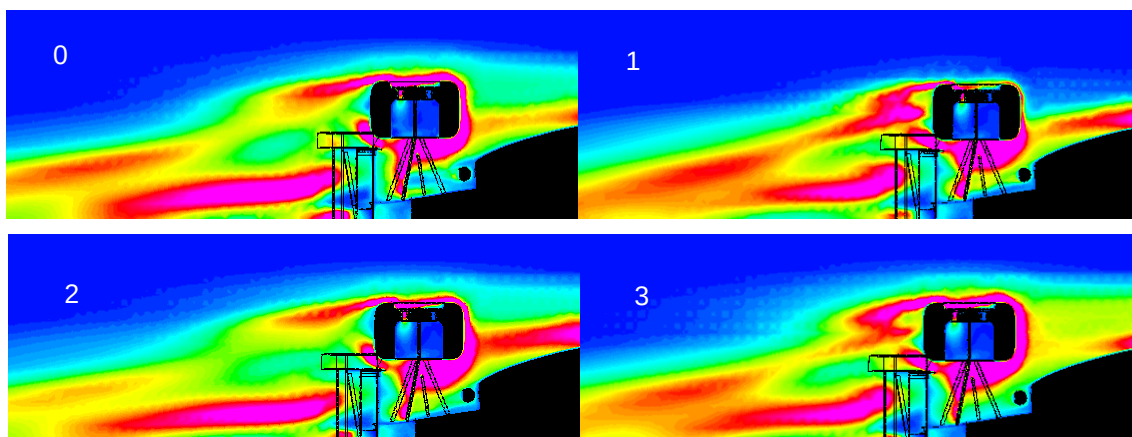


Figure 160 - Front Wing Concept Rear Wheel Wake Turbulent Kinetic Energy (60J/Kg)

The increased interaction subsequently allows further turbulence to be shed from the rear wheel as illustrated in *Figures 160 and 161*.

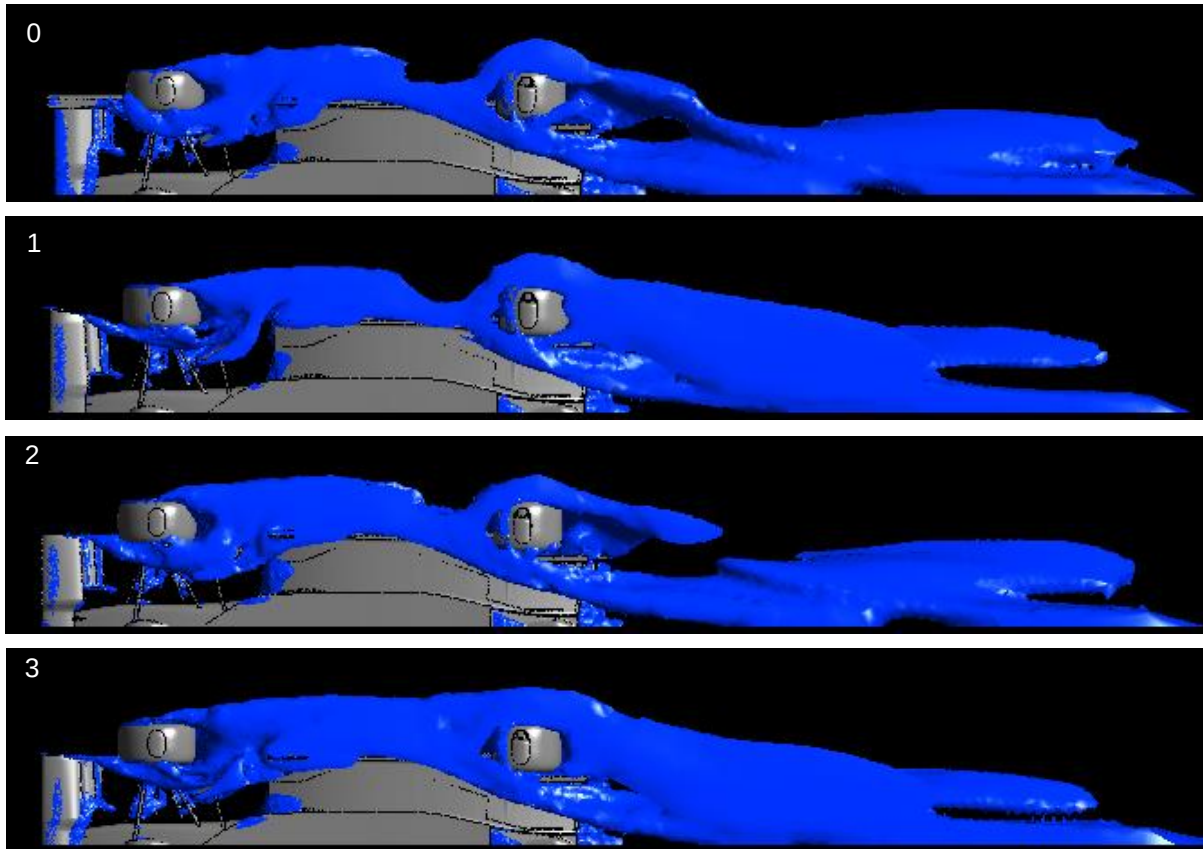


Figure 161 - Concept Front Wing Whole Car Isosurface (30J/Kg)

For each of the four illustrated cases, it is evident that the turbulent air being shed from the front wheel remains able to wash under the floor section, ahead of the rear wheel. This is probably exacerbated by the rotation of the rear wheel, however, will reduce rear end performance resulting from the diffuser's expansion zone being compromised.

Concept 1 displays the least amount of turbulent interaction at the venturi tunnel entrance on the outboard edge. This results from the end plate primary vortex feeding the vortex developed on the lower edge of the barge board and producing a robust sealing edge along the floor skirt, as illustrated in *Figure 162*.

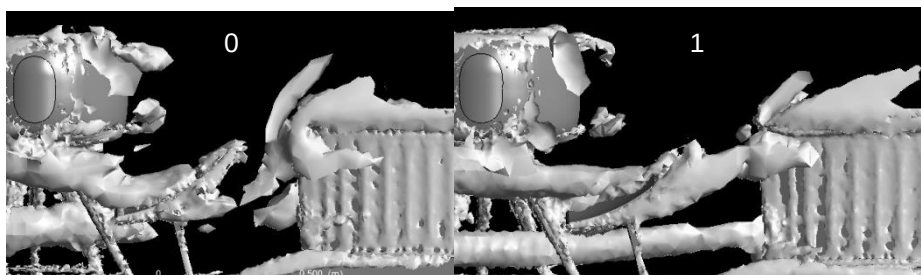


Figure 162 - Floor Sealing Vortex (1Hz)

4.0.0 Concluding Remarks

This report investigated the possibility of developing the front wing geometry on the SBD Ltd Dallara F399 British Sprint Championship race car, with the prospect of an outcome producing further front end performance while managing the generated front wheel wake.

The baseline front wing was subjected to a number of parameter sweeps in order to establish how it would react in isolation. It was found that altering the wing assemblies AoA would yield the greatest increase in negative lift. However, reducing the ground proximity ultimately produced the greatest efficiency for its generated downforce.

This was followed by investigating the flow regime of the front wheel in isolation. When applying the current maximum steer angles, it was found that the two counter rotating horseshoe vortices suffered the greatest impact depending on the orientation of the wheels. This ultimately altered the resulting near wake structure.

Four main plane aerofoils were then investigated. These were the baseline profiles, Naca 4412, Naca 6409 and the S1223. After examining their performance, the Naca 6409 was considered to be most appropriately suited, however, it was subsequently discovered that its performance is impeded when applied to the car. It was therefore substituted and the S1223 was selected because of its superior downforce characteristics while maintaining adequate efficiency.

A full design iteration study was conducted, with adjustments being introduced in order to improve overall performance. This resulted in the main plane comprising a compound profile structure, with a raised centre section employing the Naca 6409 element and outboard section using the S1223 at $H/c = 0.2$, falling within recommendations made by Katz (Katz, 2006).

The flap elements were also investigated, however, for simplicity the profiles remained as those of the baseline geometry. It was found that introducing a twist in the profile, with the inboard section utilising a greater AoA while sweeping forward on the outboard edge would produce minor outwash and reduce strong peaks of negative pressure providing further aerodynamic stability.

Concept 2 would ultimately provide the best results with a reduction in C_d relative to the baseline and other designs, however, suffered with a 4.35% decrease in total downforce and a 10.24% decrease in front end downforce, thus relocating the CoP further downstream, which is contrary to the desired outcome.

5.0.0 Recommended Further Work

Due to the Covid-19 pandemic (February 2020 and beyond the current period) restrictions impacted on available computational power, which increased simulation time and extended the length of geometry studies. It was, therefore, not possible for certain investigations to be carried out within the allocated time frame.

After thorough investigation, it was found that the front wing end plate would have the greatest impact upon the front wheel near wake structure and its resulting location. It is advised that further investigations are conducted into end plate geometry with the purpose of developing the optimum location for the subsequent vortical structures in order to facilitate the efficient management of the wake structure.

In the interest of simplicity, the decision was made to leave the original barge board geometry unaltered and maintain its current location. This resulted in further complications, therefore compromises had to be made. Due to the barge boards being located further downstream of the wing, it is suggested that novel geometries be constructed to perform efficiently with the new concept wing. This would allow the wing to generate the desired results and utilise the barge boards as flow management devices, optimised for the wing geometry.

Only four main plane profiles were investigated, the decision being based upon previous studies that had been carried out, however, it would be advisable to examine further profiles. It was found that the baseline main plane profile had similarities to that of the Nasa GA(W)-1 aerofoil, which is designed with low speed and high lift capabilities (Zhang, et al., 2011). As a result of its greater thickness to chord ratio, it is able to utilise an increased AoA and be located at a greater ground proximity, thus more high energy air can be fed to the floor section without incurring any blockage effects, which was found to play a significant role in the generation of superior results by the baseline vehicle geometry to any of the results produced by the design iterations.

As the resulting centre section of the wing achieves a greater H/c ratio than that of the outboard sections, and the outboard ends have a reduced chord length to reducing pressure spikes, these ideas require parameter sweeps in order to achieve effective optimisation. It is recommended that the compound geometry of the main plane also requires further investigation.

The flap elements also require additional examination. In the interest of simplicity, the baseline profiles were utilised, therefore it is recommended that additional profiles are investigated together with the resulting geometry.

Increasing the parameters of the design analysis would allow an improved expansion of design iteration, therefore, the optimum compromise between wing performance and flow regime could be achieved.

5.1.0 Design Concept 23

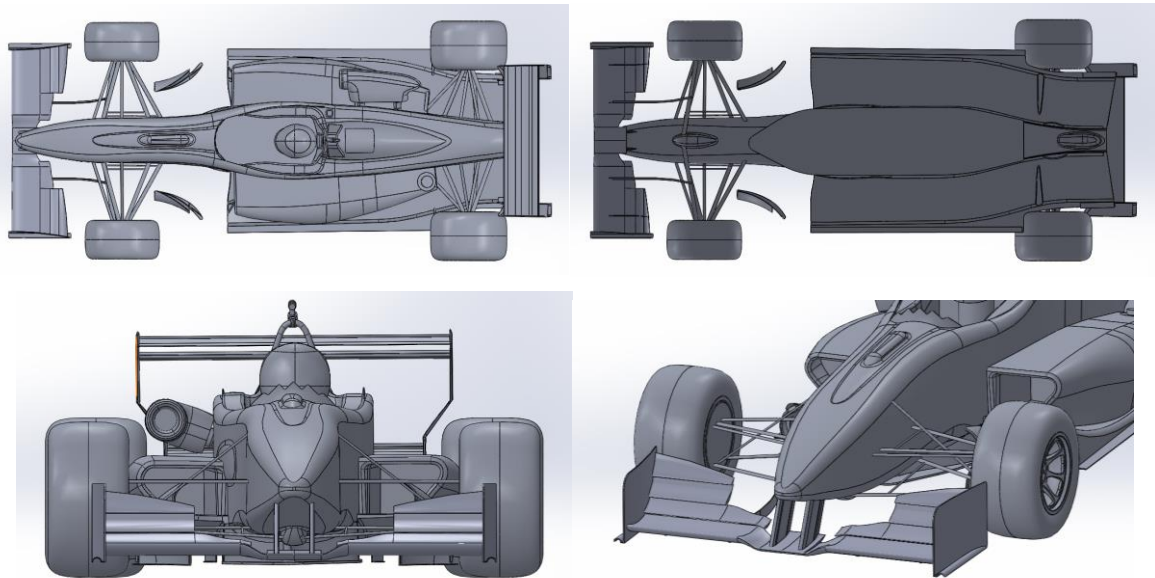


Figure 163 - Concept 23

After further consideration, it was decided that the front wing concept required re-examination in order to achieve the set aims and objectives.

Figure 163, illustrates the updated front wing design. The outboard main plane sides now utilise the Eppler 420 profile at $H/c = 0.32$, however, still employs the use of the Naca 6409 at $H/c = 0.23$ within its centre section. Aguilar et al, describes the Eppler 420 as achieving increased efficiency compared to that of the previously used S1223, and due to its thickness, is able to support much greater loads (Aguilar, et al., 2019).

As a result of the outboard sides of the mainplane having such a large ground proximity, a second fixed element using an Eppler 420 was employed to increase the expansion rate of the main plane. This was then supplemented by two additional elements (the same as the baseline secondary element profile) in order to achieve an increased camber and improved front end loading.

The barge boards were also moved a further 25mm from the cars central plane in an attempt to decrease, any blockage that might effect to the venturi tunnels and floor section.

Table 10 - Baseline and Concept 23 Comparison

	Baseline	Concept 23
Cl	-1.90	-1.97
Cd	0.77	0.77
Cl/Cd	2.47	2.56
% Forward	52.30	56.97

Table 10, indicates the relative numerical values for both the baseline and Concept 23 half car models. It can be observed that a 3.55% increase in total -Cl is achieved while maintaining Cd.

Consequently, this improves the aerodynamic efficiency of the entire car. It can also be observed that the forward percentage of downforce has increased further, with the new wing producing 545.3N of downforce compared to that of the baseline, 508.89N. However, the extra downforce comes at a cost of 7.18% increase in drag force generated by the front wing

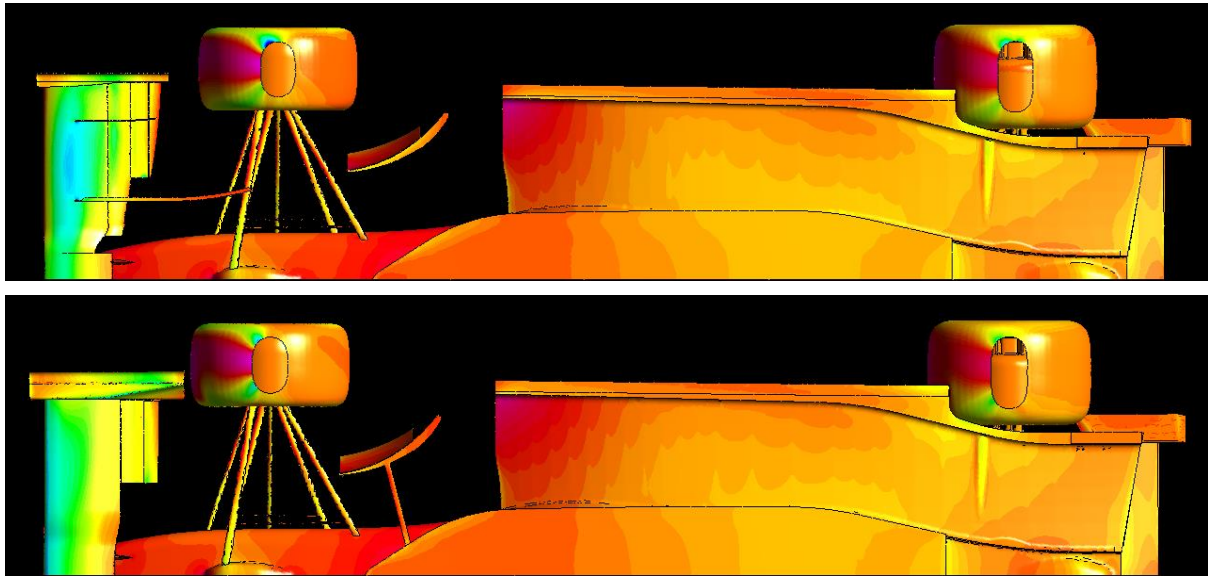


Figure 164 - Static Surface Pressures (Top - Concept 23, Bottom - Baseline)

Observing Figure 164, clearly demonstrates that a much greater distribution of negative pressure is achieved on the suction side of the front wing (Figure 165) and the floor section of the car. This results in the car (minus the front wing and wheels) achieving a 4.04% increase in downforce with only a 1.2% increase in drag force.

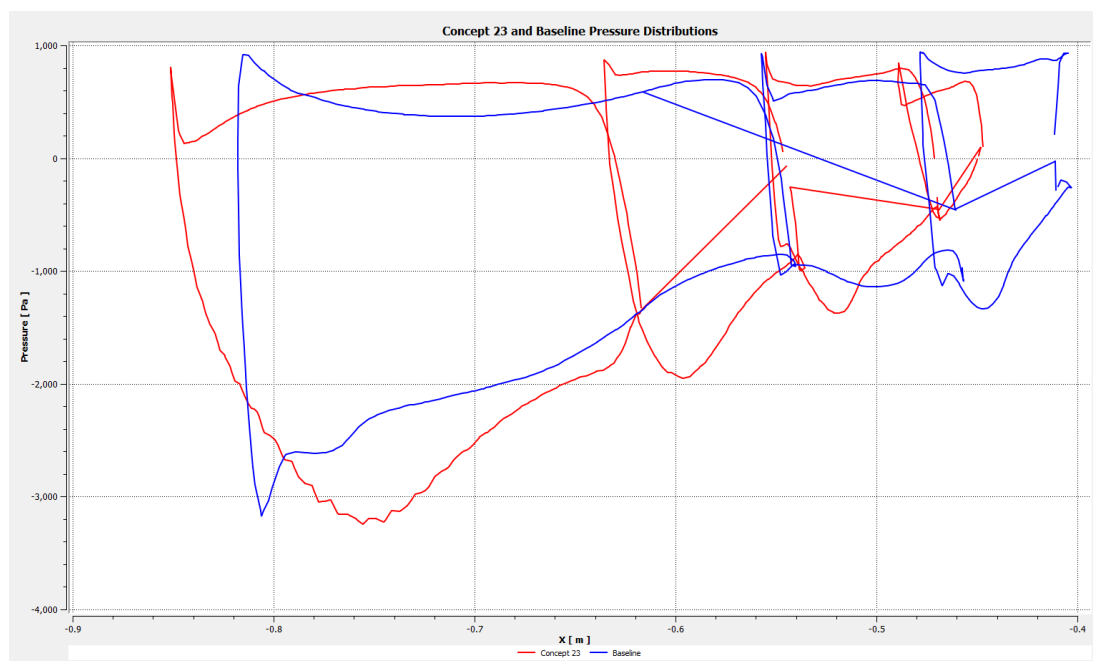


Figure 165 - Front Wing Pressure Distributions

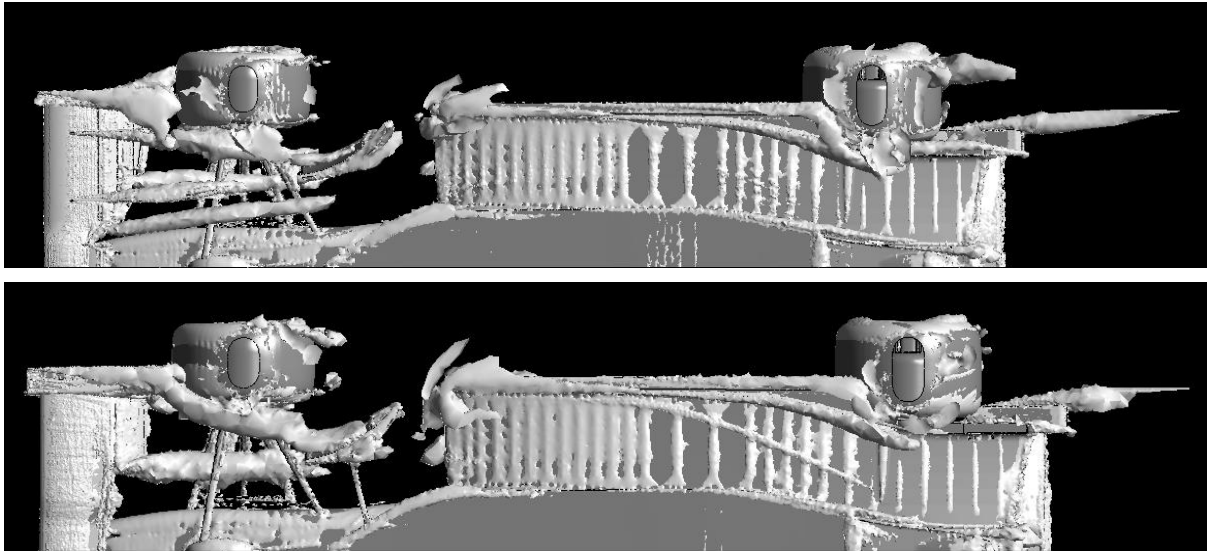


Figure 166 - Vortex Core Swirling Strength (1Hz) (Top - Concept 23, Bottom - Baseline)

The increase in front wing drag force is as a result of the developed vortical structures. Figure 166, demonstrates the increase in the number of vortices present. The increase in the number of vortices produces a much greater pressure drop on the suction side of the front wing while generating an increase in downwash and decrease in blockage. This is further fed to the floor section of the car with increased velocity due to the swirling strengths of the vortices. Feeding vortices to the floor section of the car is a technique commonly used within Formula One in order to produce a much larger pressure drop, which generates an increase in downforce (British Broadcasting Centre, 2020), which was also described by Nakagawa, et al (Nakagawa & Hojo, 2014).

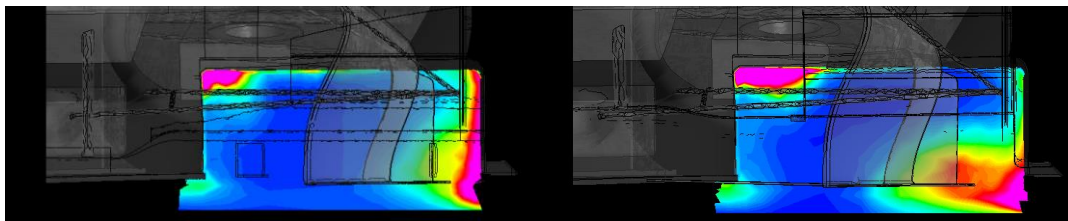


Figure 167 - Turbulent Kinetic Energy (60J/Kg) (Left – Concept 23, Right – Baseline)

Figure 167, illustrates the turbulent kinetic energy at the venturi entrance. It can be observed that Concept 23 has a significantly greater region of low turbulence. The increase in fed laminar air results in an 8.86% increase in negative pressure located at the venturi throat section due to the marginal increase in average air velocity (Figure 168).

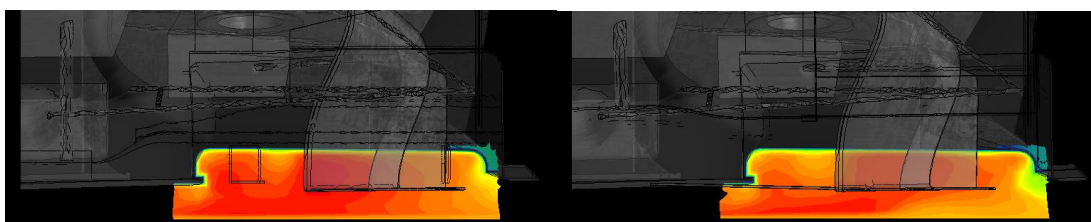


Figure 168 - Venturi Throat Velocity Contours (Left - Concept 23, Right - Baseline)

Due to the reduction in chord length of the inboard edge of the final wing element, the generated Y250 vortex has been lowered. This allows it to interact with the top surface of the side pod.

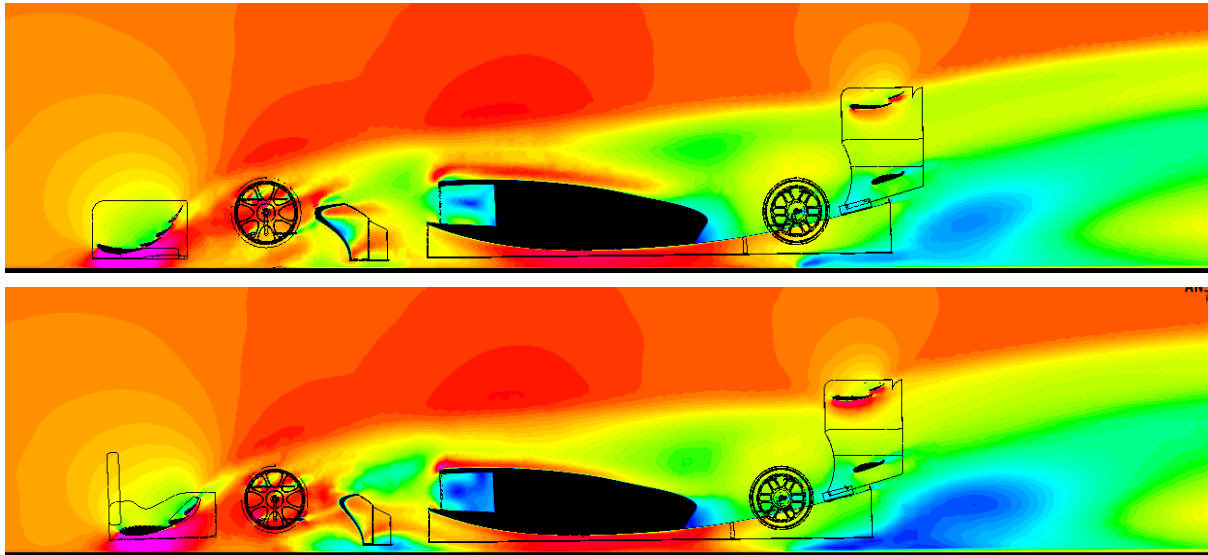


Figure 169 - Velocity Contours (Top - Concept 23, Bottom - Baseline)

Figure 169, illustrates velocity contours projected onto a plane 0.45m from the centre line of the car. An increase in duration and thickness of high velocity can be observed on the upper surface of the side pod when using Concept 23, resulting from the Y250 vortex. However, the baseline demonstrates significantly superior flow attachment on the side pod leading edge. It can also be observed that Concept 23 generates a rear wake region from the diffuser expansion that is somewhat reduced compared to that of the baseline. This is potentially due to the increased upwash generated by Concept 23 having a greater impact on the onset flow to the rear wing, thus reducing its upwash and interaction with the diffuser.

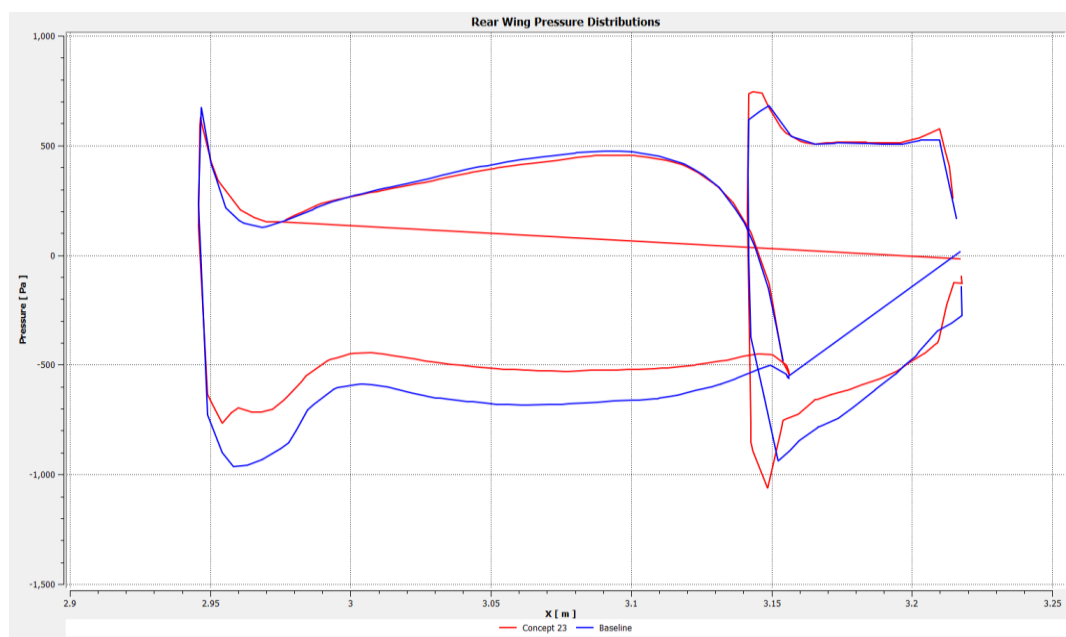


Figure 170 - Rear Wing Pressure Distributions

As *Figure 170* illustrates, the pressure distribution on the rear wing main plane differs considerably, with the main plane suction side for Concept 23 indicating an increase in pressure, resulting in less upwash. However, the loss of rear wing downforce is compensated for by the increase in floor downforce, which is considered to be a far more efficient method of generating vertical load (Gallego, 2013).

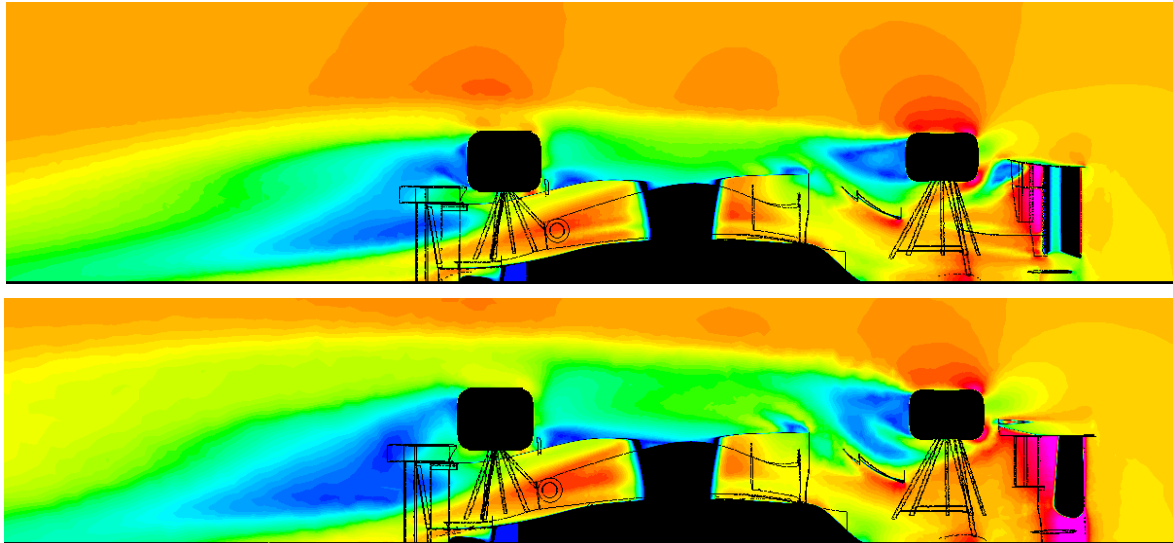


Figure 171 - Vehicle Wake Velocity Contours (Top - Concept 23, Bottom - Baseline)

Figure 171, illustrates velocity contours situated 0.1m above the ground plane. First observations indicate that the baseline front wing and barge board combination provide additional generated outwash compared to that of Concept 23. This results in a greater air velocity on the outer face of the rear wheel, which subsequently has increased the turbulent kinetic energy within its wake (*Figure 172*).

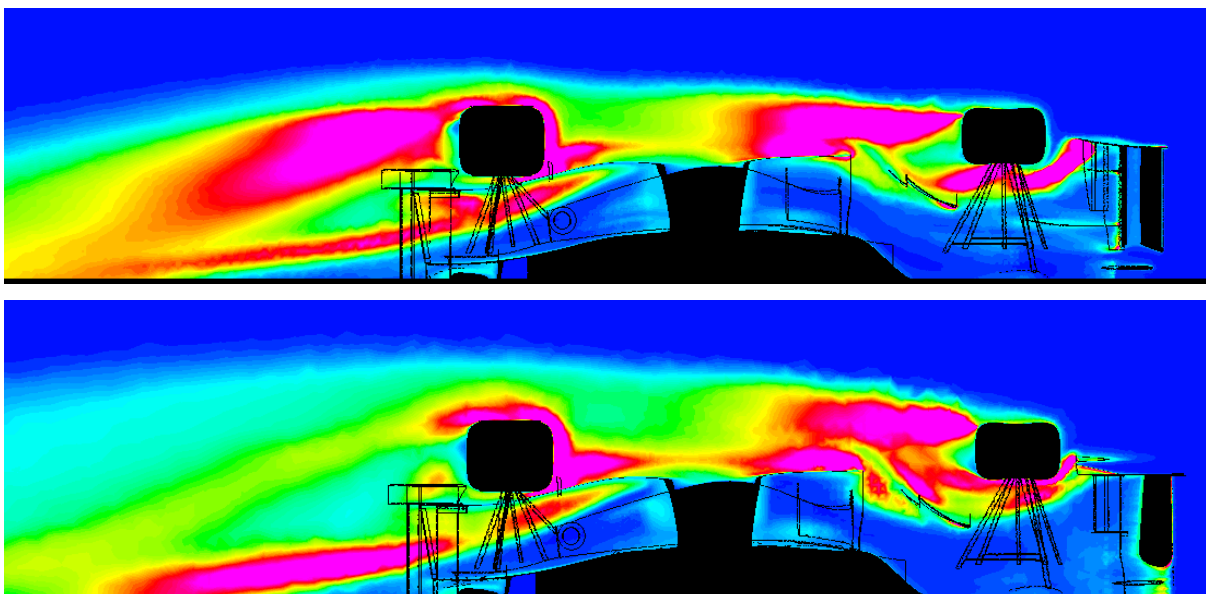


Figure 172 - Vehicle Wake Turbulent Kinetic Energy (60J/Kg) (Top - Concept 23, Bottom - Baseline)

This has led to an increase in induced drag and lift force from the rear wheel (*Table 11*), with a much greater stagnation pressure on its front face and a significant pressure drop on its upper surface and rear face (*Figure 173*).

Table 11 - Rear Wheel Forces

	Concept 23	Baseline
Lift (N)	36.01	16.00
Drag (N)	96.44	88.77

However, this runs contrary to the forces impacting on the front wheel.

Table 12 - Front Wheel Forces

	Concept 23	Baseline
Lift (N)	2.55	5.40
Drag (N)	34.93	47.31

As *Table 12* indicates that both drag and lift are reduced on the front wheel when Concept 23 is employed, which is as a result of a reduction in pressure gradient between its front face and rear face.

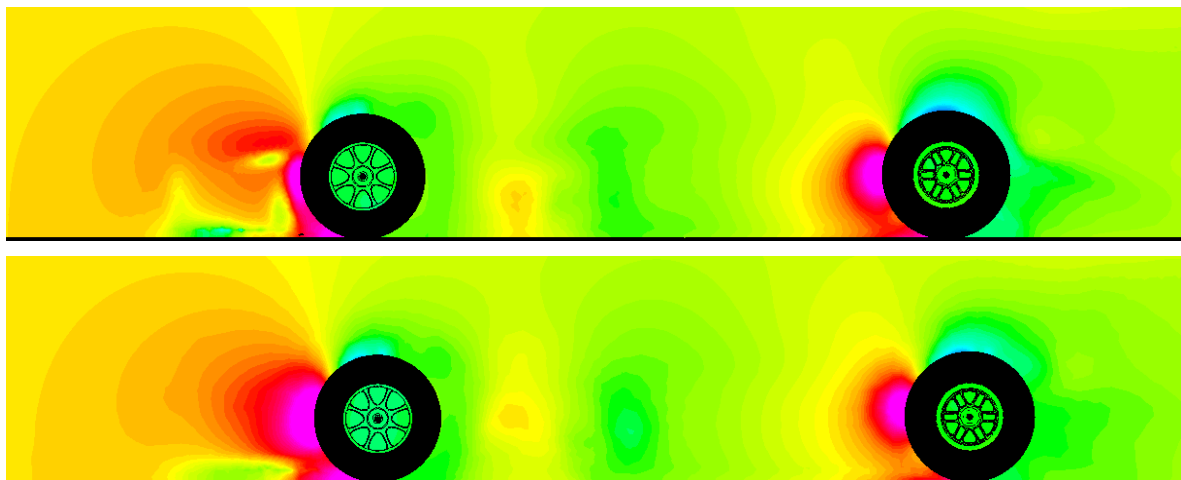


Figure 173 - Wheel Stagnation Pressure Contours (Top – Concept 23, Bottom – Baseline)

Figure 173, illustrates static pressures projected onto a plane 0.77m from the centre line of the car. When observing the front wheel stagnation zones, it is evident that the magnitude of positive pressure on the wheels front face using Concept 23 is significantly reduced. This is as a result of the increased span of the front wing producing a reduction in pressure due to its expansion, and turning the onset flow away from the majority of the front face of the wheel. The pressure drop across the upper surface of the wheel is also reduced, resulting in a decrease in lift force.

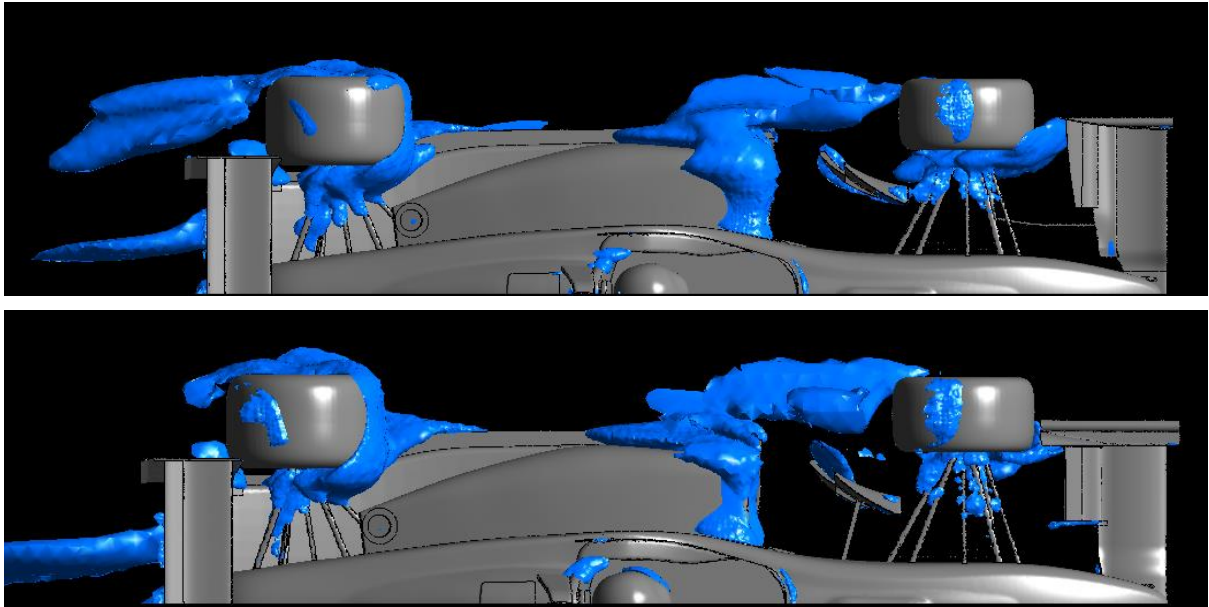


Figure 174 - Turbulent Kinetic Energy Isosurface (60J/Kg) (Top - Concept 23, Bottom - Baseline)

Despite the lack of outwash, the reduction in pressure gradient across the front wheel has reduced the lateral size of the near wake formation. This is highlighted in both *Figures 172, 174 and 175*. However, as *Figure 175* illustrates, the interaction of the wake with the outer face of the side pod is significantly increased, resulting in the effect on the rear wheel as previously referenced.

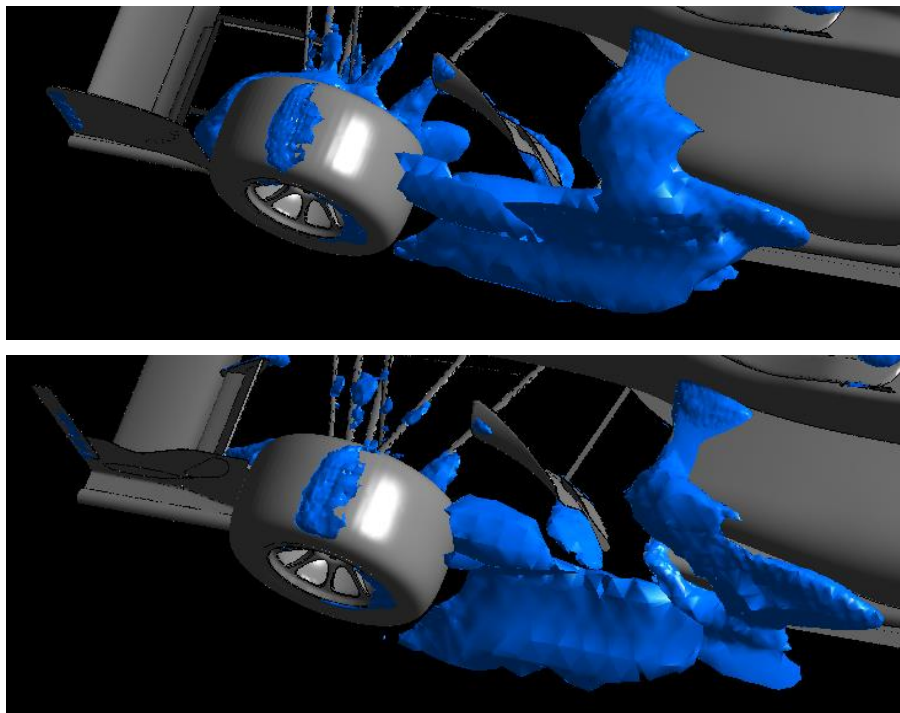


Figure 175 - Front Wheel Wake Side Pod Interaction Isosurface (60J/Kg) (Top - Concept 23, Bottom - Baseline)

6.0.0 References

- Abdullah, A. et al., 2017. *Ground proximity effect on the flow over NACA 4412 multi-element airfoil in clean configuration*. Johor, IOP Conf. Series: Journal of Physics: Conf. Series 914.
- Agrawal, A. & Prasad, A. K., 2003. Measurements Within Vortex Cores in a Turbulent Jet. *Journal of Fluids Engineering*, 125(1), pp. 561 - 568.
- Aguilar, G., Gasljevic, K. & Matthys, E. F., 2006. *Reduction of friction in fluid transport: experimental investigation*, Santa Barbara: University of California Santa Barbara.
- Aguilar, J., Rubio-Clemente, A., Velasquez, L. & Chica, E., 2019. *Design and Optimization of a Multi-Element Hydrofoil for a Horizontal-Axis Hydrokinetic Turbine*, Basel: Mdpi.
- Ahmed, S. R., 1981. Wake Structure of Typical Automobile Shapes. *Trans. ASME. Journal of Fluids Engineering*, 103(1), pp. 162 - 169.
- Anderson, G., 2020. *GARY ANDERSON EXPLAINS THE 2020 F1 FRONT WING AERO WAR*. s.l.:The Race 2020.
- Ansys Inc, 2006. *Modeling Turbulent Flows*, Canonsburg: Ansys Inc.
- Ansys Inc, 2009. 4.12.1 Overview. [Online]
Available at: www.afs.enea.it/project/neptunius/docs/fluent/html/th/node98.htm
[Accessed 2 7 2020].
- Ansys Inc, 2009. 4.12.3 Non-Equilibrium Wall Functions. [Online]
Available at: www.afs.enea.it/project/neptunius/docs/fluent/html/th/node100.htm
[Accessed 2 7 2020].
- Ansys Inc, 2009. *Turbulence*, Canonsburg: Ansys Inc.
- Ansys Inc, 2009. *Turbulence Modeling: Introductory Fluent Training*, Canonsburg: Ansys Inc.
- Ansys Inc, 2014. *Lecture 7: Turbulence Modeling*, Canonsburg: Ansys Inc.
- Bansal, S. & Selig, M., 2016. *Wake Deceleration of a Racecar Multielement Airfoil in Ground Effect*. Washington D.C., AIAA Aviation .
- Barbarino, S., Dettmer, W. G. & Friswell, M. I., 2010. *Morphing Trailing Edges with Shape Memory Alloy Rods*. Pennsylvania, ICAST.
- Benson, T., 2014. *Modern Drag Equation*. [Online]
Available at:
wright.nasa.gov/airplane/drageq.html#:~:text=The%20induced%20drag%20coefficient%20is,divided%20by%20the%20wing%20area.
[Accessed 13 7 2020].

British Broadcasting Centre, 2020. *Formula 1 2020: The secret aerodynamicist - are capes the key?*. London: BBC.

British Sprint, 2020. *THE SBD Motorsport UK HSA BRITISH SPRINT CHAMPIONSHIP*. [Online]
Available at:
www.britishsprint.org/#:~:text=THE%20SBD%20Motorsport%20UK%20HSA%20BRITISH%20SPRINT%20CHAMPIONSHIP,-The%202020%20SBD&text=The%20British%20and%20National%20championship,National%20and%20above%20competition%20licences.
[Accessed 27 6 2020].

Cai, J., Kapoor, S., Sikder, T. & He, Y., 2017. *Effects of Active Aerodynamic Wings on Handling Performance of High-Speed Vehicles*, Michigan: SAE International.

Castejon, D. V., Catalano, F. M. & Severi Junior, G., 2011. *Methods of Drag Reducing without Losing Aerodynamic Stability Applied on SAE Reference Model*, São Paulo: SAE International.

Cebeci, T., 2004. 4 - General Behavior of Turbulent Boundary Layers. In: T. CEBECI, ed. *Analysis of Turbulent Flows (Second Revised and Expanded Edition)*. Amsterdam: Elsevier, pp. 81 - 140.

Coch, M., 2018. *FIA unveils new generation Formula 3 car*. [Online]
Available at: www.speedcafe.com/2018/11/23/fia-unveils-new-generation-formula-3-car/
[Accessed 24 6 2020].

Collins, S., 2018. *DRS: The Drag Reduction System explained*. Market Harborough: Racecar Engineering.

Corfield, E., Hodgson, G. & Garmory, A., 2018. *A Computational and Experimental Investigation into the Effects of Debris on an Inverted Double Wing in Ground Effect*, Loughborough: SAE International.

Craft, T. J., 2011. *Near-Wall Modelling*. Manchester: University of Manchester.

CRONER, E., MOTHAY, G., SICOT, C. & B´EZARD, H., 2012. *AERODYNAMIC ANALYSIS OF AN ISOLATED WHEEL WAKE*, Ladoux: MICHELIN.

d'Apollonia, A. S., Granier, B. & Debaty, P., 2004. *Design of Experiments Methods Applied to CFD Simulations in Vehicle Aerodynamics*, Michigan: SAE International.

Davies, R., 2019. *FORMULA 3 STUDENT PROJECT PUTS SPOTLIGHT ON NEXT GENERATION OF ENGINEERING TALENT*. [Online]
Available at: www.uwtsd.ac.uk/news/press-releases/press-2019/formula-3-student-project-puts-spotlight-on-next-generation-of-engineering-talent.html?fbclid=IwAR07_tvJCNChhgWuv1mGGkDwPrdz6EmB9Kz50CzHgkvjO352DaihKQBkUpw
[Accessed 25 6 2020].

Diasinos, S., Barber, T. J. & Doig, G., 2015. The effects of simplifications on isolated wheel aerodynamics. *Journal of Wind Engineering and Industrial Aerodynamics*, 146(1), pp. 90 - 101.

Diasinos, S., Doig, G. & Barber, T. J., 2014. On the interaction of a racing car front wing and exposed wheel. *The Aeronautical Journal*, 118(1210), pp. 1385 - 1407.

Edmondson, L., 2015. *F1 without wind tunnels 'many years' away - Paddy Lowe*. [Online] Available at: www.espn.co.uk/f1/story/_/id/13820494/f1-wind-tunnels-many-years-away-paddy-lowe [Accessed 29 6 2020].

Fackrell, J. E., 1972. *The Aerodynamic Characteristics of an Isolated Wheel Rotating in Contact with the Ground.*, London: Imperial College.

Fackrell, J. E. & Harvey, J. K., 1975. "The Aerodynamics of an Isolated Road Wheel," *Proceedings of the 2nd Symposium on Aerodynamic of Sports and Competition Automobiles*. s.l., AIAA.

Fluent Inc, 2001. *Chapter 10. Modeling Turbulence*, s.l.: Fluent Inc.

Frei, W., 2017. *Which Turbulence Model Should I Choose for My CFD Application?*, Burlington: Cosmol.

Gallego , D. M. V. P., 2013. *Aerodynamics in race cars Design and characterization of a diffuser for a Formula Student race car* , Málaga: Escuela Ingenierías Industriales.

Håkansson, E. & Dube, B., 2014. *Racing at Bonneville: Determining the "center of pressure" and center of gravity of your vehicle*, Michigan: Land Racing.

Heisler, H., 2002. 14 - Vehicle body aerodynamics. In: H. Heisler, ed. *Advanced Vehicle Technology (Second Edition)*. Oxford: Butterworth Heinemann, pp. 584 - 634.

He, X. et al., 2016. Numerical simulation of Gurney flap on SFYT15 thick airfoil. *Theoretical and Applied Mechanics Letters*, 6(6), pp. 286 - 292.

Heyder-Bruckner, J., 2011. *The Aerodynamics of an Inverted Wing and a Rotating Wheel in Ground Effect*, Southampton: Southampton University.

Hillam, M., 2020. *Main Plane Ground Proximity* [Interview] (16 6 2020).

Hossain, S., Raiyan, M. F., Akanda, M. N. U. & Jony, N. H., 2014. A COMPARATIVE FLOW ANALYSIS OF NACA 6409 AND NACA 4412 AEROFOIL. *nternational Journal of Research in Engineering and Technology*, 3(10), pp. 1 - 9.

Houghton, E. L., Carpenter, P. W., Collicott, S. H. & Valentine, D. T., 2017. Chapter 6 - Thin Airfoil Theory. In: *Aerodynamics for Engineering Students (Seventh Edition)*. Oxford: Butterworth Heinemann, pp. 391 - 447.

Houghton, E. L., Carpenter, P. W., Collicott, S. H. & Valentine, D. T., 2017. Chapter 7 - Wing Theory. In: *Aerodynamics for Engineering Students (Seventh Edition)*. Oxford: Butterworth Heinemann, pp. 449 - 523.

INCROPERA, F. P., DEWITT, D. P., BERGMAN, T. L. & LAVINE, A. S., 2007. *Fundamentals of Heat and Mass Transfer*. 6 ed. Danvers: John Wiley & Sons.

Ismail, N. I. et al., 2018. Lift distribution of washout twist morphing MAV wing. *International Journal of Engineering & Technology*, 7(4), pp. 89 - 94.

Issakhanian, E., Lo, K., Elkins, C. & Eaton, J., 2009. *Computational and Experimental Investigation of the Flow Structure and Vortex Dynamics in the Wake of a Formula 1 Tire*, Stanford: Stanford University.

James, M. R., Watkins, S. & Watts, M., 2014. *The Effect of Upstream Turbulence on an Exposed*, Michigan: SAE International.

Jasinski, W. J. & Selig, M. S., 1998. *Experimental Study of Open-Wheel Race-Car Front Wings*, Warrendale: SAE Technical Paper Series.

Joshi, P. S., Pradhan, P. P., Joshi, T. N. & Sherje, N. P., 2019. Development and Implementation of Multi Element Aerodynamic Devices with Drag Reduction System for a Formula Student Vehicle. *JETIR*, 6(4), pp. 409 - 417.

Jung, J. H. et al., 2012. Mean flow characteristics of two-dimensional wings in ground effect. *International Journal of Naval Architecture and Ocean Engineering*, 4(2), pp. 151 - 161.

Kapoor, A., 2016. *A Quick Approach to Boundary Layer meshing - For Beginners*, s.l.: Airbus Digital Services.

Katopodes, N. D., 2019. Chapter 7 - Vorticity Dynamics. In: N. D. Katopodes, ed. *Free-Surface Flow - Environmental Fluid Mechanics*. Oxford: Butterworth Heinemann, pp. 516 - 565.

Katz, J., 2006. Aerodynamics of Race Cars. *Annual Review of Fluid Mechanics*, 27(63), pp. 27 - 63.

Keogh, J., Doig, G. & Diasinos, S., 2014. Flow compressibility effects around an open-wheel racing car. *The Aeronautical Journal*, 118(1210), pp. 1409 - 1431.

Keogh, J., Doig, G., Diasinos, S. & Barber, T., 2015. *The influence of cornering on the vortical wake structures of an inverted wing*, s.l.: IMECHE.

Kim, Y., Al-Abadi, A. & Delgado, A., 2017. *Airfoil boundary layer optimization towards HAWT aerodynamic efficiency by Genetic Algorithm*. Copenhagen, Wind Energy Science Conference.

Knowles, R. D., 2005. *Monoposto Racecar Wheel Aerodynamics: Investigation of Near-Wake Structure and Support-Sting Interference*, Cranfield: Cranfield University.

KRÁLIK, J., 2016. CFD SIMULATION OF AIR FLOW OVER AN OBJECT WITH GABLE ROOF, REVISED WITH Y+ APPROACH. *Transactions of the VŠB*, 16(2), pp. 85 - 94.

- Kundu, P. K., Cohen, I. M. & Dowling, D. R., 2016. Chapter 10 - Boundary Layers and Related Topics. In: P. K. Kundu, ed. *Fluid Mechanics (Sixth Edition)*. Massachusetts: Academic Press, pp. 469 - 532.
- Lanfrit, M., 2005. *Best practice guidelines for handling Automotive External Aerodynamics with FLUENT*, Darmstadt: Fluent Deutschland GmbH.
- LAUNDER, B. E. & SPALDING, D. B., 1973. *THE NUMERICAL COMPUTATION OF TURBULENT FLOWS*, London: NORTH-HOLLAND PUBLISHING COMPANY.
- Leśniewicz, P., Kulak, M. & Karczewski, M., 2014. *Aerodynamic analysis of an isolated vehicle wheel*, Lodz: IOP Publishing.
- Liu, F., 2016. *A Thorough Description Of How Wall Functions Are Implemented In OpenFOAM*, Chalmers: Chalmers University of Technology.
- Liu, T., Kuykendoll, K., Rhew, R. & Jones, S., 2006. Avian Wing Geometry and Kinematics. *AIAA JOURNAL*, 44(5), pp. 954 - 963.
- Li, Y., Visiedo, M. d. C. F., Paz, H. V. & de Aranaga, L. R. B., 2000. *Downforce without wings*, Göteborg: Institutionen för Termo-och Fluidmekanik.
- Marchman, J. F., 1973. *Wake Turbulence and Its Elimination*, New York: SOCIETY OF AUTOMOTIVE ENGINEERS, INC.
- McBeath, S., 2001. *Competition Downforce - A Practical Handbook*. 1 ed. Nr Yeovil: Haynes Publishing.
- Memon, M. O., 2017. *WINGTIP VORTICES AND FREE SHEAR LAYER INTERACTION IN THE VICINITY OF MAXIMUM LIFT TO DRAG RATIO LIFT CONDITION*, Dayton: UNIVERSITY OF DAYTON.
- Misar, A. S., Uddin, M. & Robinson, A., 2020. *Numerical Analysis of Flow around an Isolated Rotating Wheel Using a Sliding Mesh Technique*, Michigan: SAE International.
- Mishra, P. & Aharwal, K. R., 2018. *A review on selection of turbulence model for CFD analysis of air flow within a cold storage*, Bhopal: IOP Publishing.
- Moet, H., Laporte, F., Chevalier, G. & Poinot, T., 2004. *Wave propagation in vortices and vortex bursting*, Bremen: Airbus.
- Mokhtar, W. A., 2008. *Aerodynamics of High-Lift Wings with Ground Effect for Racecars*, Michigan: SAE International.
- Morelli, A., 2000. *A New Aerodynamic Approach to Advanced Automobile Basic Shapes*, Michigan: SAE International.
- Motorsport UK, 2020. *Motorsport UK Yearbook 2020*. 1 ed. Colnbrook: Motorsport UK.

- Mouffouk, M. A., 2014. *Aerodynamic Development of the Seguracing F1-R01 Prototype Using CFD*, Batna: University H.L of Batna.
- Nakagawa, M. & Hojo, T., 2014. *Typical Vortical Structures around a Formula 1 Car*. Nagoya, Conference: Invited Lecture, 6th International Conference on Vortex Flows and Vortex Models (ICVFM Nagoya 2014).
- Nakagawa, M., Kallweit, S., Michaux, F. & Hojo, T., 2016. *Typical Velocity Fields and Vortical Structures around a Formula One Car, based on Experimental Investigations using Particle Image Velocimetry*, Michigan: SAE International.
- Nirooei, M., 2017. Aerodynamic and static stability characteristics of airfoils in extremeground effect. *J Aerospace Engineering*, 0(0), pp. 1 - 15.
- Ordaz, I., Li, W. & Campbell, R., 2014. *Automated Tetrahedral Mesh Generation for CFD Analysis*, Reston: American Institute of Aeronautics and Astronautics .
- Ouahabi , M. H., Ichenial, M. M., Hajjaji, A. E. & Benabdelouahab, F., 2017. Application of the Turbulence Models at Low Reynolds Number for Horizontal-Axis Wind Turbine Design by using Computational Fluid Dynamics (CFD). *International Journal of Control and Automation*, 10(7), pp. 179 - 190.
- Palaskar, P. M., 2016. *Effect of Side Taper on Aerodynamics Drag of a Simple Body Shape with Diffuser and without Diffuser*, Michigan: SAE International.
- Palmer, F. S. et al., 2018. Towards an Airfoil Catalogue for Wind Turbine Blades at IDR/UPM Institute with OpenFOAM. *Journal of Aerospace Engineering and Mechanics*, 2(1), pp. 121 - 133.
- Pegrum, J. M., 2006. *Experimental Study of the Vortex System Generated by a Formula 1 Front Wing*, London: Imperial College, London.
- Petkar, R. K., Kolgiri, S. G. & Ragit, S. S., 2014. Study of Front-Body of Formula-One Car for Aerodynamics using CFD. *International Journal of Application or Innovation in Engineering & Management*, 3(3), pp. 353 - 359.
- Pope, S. B., 2015. *Turbulent Flows*, Cambridge: Cambridge University Press.
- Quintarelli, A., 2018. *Aerodynamic efficiency, balance and center of pressure*, s.l.: Racing Line.
- Rad, M. & Kazemi, F. J., 2001. *EFFECT OF CAMBER AND THICKNESS ON THE AERODYNAMIC PROPERTIES OF AN AIRFOIL IN GROUND PROXIMITY*, Tehran: SID.
- Ramalingaih, Manjunath, S., Kumar, A. B. S. & Glen, 2013. *VALIDATION OF WIND TUNNEL TEST RESULTS FOR NACA2415 AIRFOIL USING FINITE VOLUME BASEDCOMPUTATIONAL FLUID DYNAMICS SOFTWARE*, Karnataka: Institute of Technology Bangalore Karnataka.
- Ranzenbach, R. & Barlow, J., 1996. *Cambered Airfoil in Ground Effect - An Experimental and Computational Study*, Michigan: SAE International.

Salient Edge, 2018. *Turbulence Modelling For The Win - Part 1*, s.l.: Salient Edge.

SANDOVAL, S., 2019. *Where Science Meets Art*. Leicestershire: Racecar Engineering.

SBD Ltd, 2015. *Vauxhall 2.0L XE TP300bhp Kit*. [Online]

Available at: www.sbdmotorsport.co.uk/index.php/products/index/4429

[Accessed 27 6 2020].

SBD MOTORSPORT UK HSA BRITISH SPRINT CHAMPIONSHIP, 2020. *2020 SBD MOTORSPORT UK HSA BRITISH SPRINT CHAMPIONSHIP SPORTING & TECHNICAL REGULATIONS*, s.l.: SBD MOTORSPORT UK HSA BRITISH SPRINT CHAMPIONSHIP.

Schlichting, D. H., 1979. *Boundary Layer Theory*. 7 ed. New York: McGraw Hill.

Scott, J., 2006. *F-14 Wing Sweep & Aspect Ratio*. [Online]

Available at: www.aerospaceweb.org/question/aerodynamics/q0281c.shtml

[Accessed 10 8 2020].

Selig, M. S. & Guglielmo, J. J., 1997. High-Lift Low Reynolds Number Airfoil Design. *JOURNAL OF AIRCRAFT*, 34(1), pp. 72 - 79.

Sharma, A., 2017. *Airfoil Thickness Effects on Dynamic Stall Onset*, Ames: Iowa State University.

Sidney, L., 1978. The Structure of Vortex Breakdown. *Annual Review of Fluid Mechanics*, Volume 10, pp. 221 - 246.

Sijal, A., 2019. *Understanding Y+ for CFD Simulations*, s.l.: Advanced Engineering Solutions.

Smith, A. M. O., 1975. High-Lift Aerodynamics. *J. AIRCRAFT*, 12(6), pp. 501 - 530.

Smith, J., 2013. *Smith's Fundamentals of Motorsport Engineering*. 1 ed. Cheltenham: Nelson Thornes Ltd.

SOARES, R. F., 2015. *DRAG OF ROAD CARS: COST-EFFECTIVE CFD SETUP, PROPOSAL OF AN AERODYNAMIC CONCEPT AND CASE STUDIES*, Santa Mônica: UNIVERSIDADE FEDERAL DE UBERLÂNDIA.

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 Driver Model*, Cranfield: SAE International.

SODJA, J., 2007. *Turbulence models in CFD*, Ljubljana : University of Ljubljana .

Somerfield, M., 2015. *2015 INDYCAR AERO KITS - FRONT WING*. s.l.:SomersF1.

Soso, M. & Wilson, P., 2004. *Investigating Changes to the Downforce Curve of a Double Element Airfoil in Ground Effect*, Michigan: SAE International.

SPROT, A. J., 2013. *Open-Wheel Aerodynamics: Effects of Tyre Deformation and Internal Flow*, Durham: Durham University.

Stanford University, 2007. *Simulation of Turbulent Flows*. Stanford: Stanford University.

Stapleford, W. R. & Carr, G. W., 1969. *Aerodynamic Characteristics of Exposed Rotating*, Nuneaton: MIRA.

Susín, D. I., 2019. *Foundations, Methodologies and Design of the Formula UC3M Car Aerodynamics*, Madrid: Universidad Carlos III de Madrid.

Tachie, M. F., Bergstrom, D. J. & Balachandar, R., 2002. THE EFFECT OF WALL ROUGHNESS ON AN OPEN CHANNEL BOUNDARY LAYER. In: N. F. W. RODI, ed. *Engineering Turbulence Modelling and Experiments*. Mallorca: Elsevier, pp. 455 - 463.

Toet, W., 2019. *Willem Toet explains...CFD Post Processing*. London: Race Tech - Motorsport Engineering.

Toet, W., 2019. *Willem Toet explains...CFD Post Processing*. London: Race Tech Motorsport Engineering.

TOORMAN, E. A., 2000. *ANALYSIS OF NEAR-WALL TURBULENCE MODELLING WITH k-E MODELS*, Leuven: Katholieke Universiteit Leuven.

Total Simulation, 2016. *Secrets of Formula 1 Part 2 – Importance of Aerodynamics*. [Online] Available at: www.totalsimulation.co.uk/secrets-formula-1-part-2-importance-aerodynamics/ [Accessed 25 6 2020].

TotalSim, 2016. *Secrets of a Formula 1 Front Wing*, Brackley: Total Simulation Ltd.

TotalSimLtd, 2016. *Secrets of Formula 1 Part 3 – The role of the Front Wing*. Brackley: Total Sim Ltd.

van Garrel, A., Venner, C. H. & Hoeijmakers, H. W. M., 2017. *Fast Multilevel Panel Method for Wind Turbine Rotor Flow Simulations*, Enschede: University of Twente.

Versteeg, H. K. & Malalasekera, W., 2007. *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*. 2 ed. Harlow: Pearson Education Limited.

Wang, D. & Fan, L. S., 2013. 2 - Particle characterization and behavior relevant to fluidized bed combustion and gasification systems. In: F. Scala, ed. *Fluidized Bed Technologies for Near-Zero Emission Combustion and Gasification*. Cambridge: Woodhead Publishing, pp. 42 - 76.

Wang, J. J., Li, Y. C. & Choi, K. - S., 2008. Gurney flap - Lift enhancement, mechanisms and applications. *Progress in Aerospace Sciences*, 44(1), pp. 22 - 47.

Wang, Y. et al., 2014. Numerical and Experimental Investigations on the Aerodynamic Characteristic of Three Typical Passenger Vehicles. *Journal of Applied Fluid Mechanics*, 7(4), pp. 659 - 671.

- Wäschle, A., Cyr, S., Kuthada, T. & Wiedemann, J., 2004. *Flow around an Isolated Wheel – Experimental and Numerical Comparison of Two CFD Codes*, Michigan: SAE International.
- Wood, R., 2015. *Reynolds Number Impact on Commercial Vehicle Aerodynamics and Performance*, Michigan: SAE International.
- Wordley, S. et al., 2014. *Development of a Drag Reduction System (DRS) For Multi-Element Race Car Wings*, Melbourne: Monash University Clayton.
- Yang, X. & Olatunbosun, O. A., 2011. Optimization of reinforcement turn-up effect on tyre durability and operating characteristics for racing tyre design. *J. Materials and Design*, 35(1), pp. 709 - 798.
- Zahm, A. F. & Bear, B. S. R. M., 1921. Ground-plane influence on airplane wings. *Journal of the Franklin Institute*, 191(5), pp. 687 - 693.
- Zerihan, J. & Zhang, X., 2000. Aerodynamics of a single element wing in ground effect.. *Journal of Aircraft*, 37(6), p. 1058 – 1064.
- Zhang, X., Toet, W. & Zerihan, J., 2006. *Ground Effect Aerodynamics of Race Cars*, Southampton: University of Southampton.
- Zhang, X. & Zerihan, J., 2003. Aerodynamics of a Double-Element Wing in Ground Effect. *AIAA JOURNAL*, 41(6), pp. 1007 - 1016.
- Zhang, Y., Igarashi, T. & Hu, H., 2011. *Experimental Investigations on the Performance Degradation of a Low-Reynolds-Number Airfoil with Distributed Leading Edge Roughness*. Orlando, AIAA.

7.0.0 Appendix A

7.1.0 C2 Wind Tunnel

Validating the CFD methodology provides confidence in the results that are produced. This ensures the accuracy of the data produced and provides for a comprehensive understanding of both aerodynamic and computational principles.

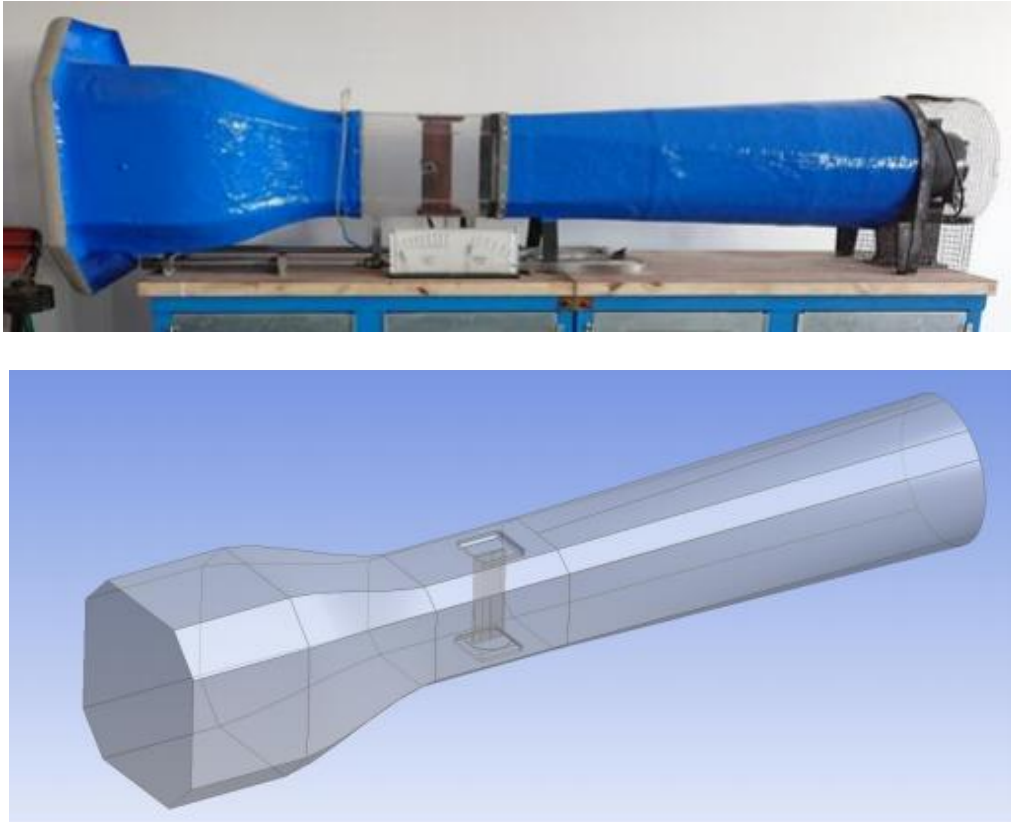


Figure 176 - C2 Open Loop Wind Tunnel (1/12 Scale)

Experimental data was produced using a Naca 0012 profile within a C2 open loop wind tunnel (*Figure 176*). Surface pressures were taken at four angle increments (0, 5, 10 and 15 degrees) with the use of eleven pressure taps at intervals along a 0.0101115m chord length. A velocity of 6.75m/s was measured and set using a hot wire anemometer.

The wind tunnel and aerofoil were, modelled using CAD software, however, the flow straighteners located at the velocity inlet of the tunnel were not modelled as it was believed that assigning the correct boundary conditions and user defined functions (UDFs) within the simulation case would provide a straight onset freestream flow, although it is not known whether this would be an accurate representation of the real wind tunnel inlet flow regime.

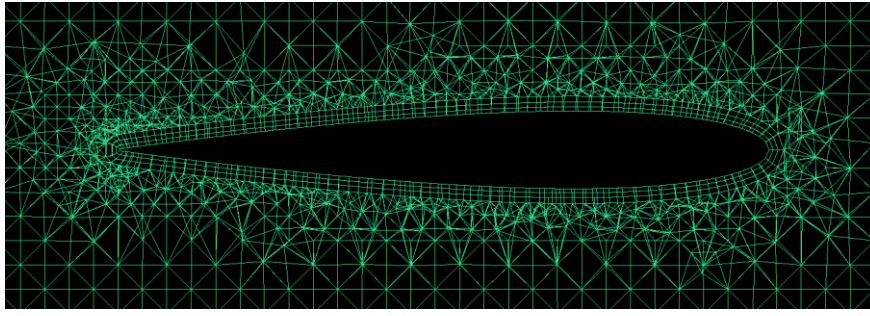


Figure 177 – Naca 0012 Near-Wall Mesh Resolution

The simulation case was established using the $k\varepsilon$, Realizable, Non Equilibrium Wall Functions turbulence model, with a first cell height of $Y^+ = 30$ with recommended values of $30 < Y^+ < 300$ (Ansys Inc, 2009). Each model was ran using $Re = 4.64E4$ at the same velocity used for the experimental data, producing a boundary layer thickness of $\delta = 0.0065m$, therefore promoting the use of three prism layers (using a growth rate of 120%) to resolve the boundary layer (Figure 177).

Within the UDFs of Fluent, the pressure outlet was given a gauge pressure of 0Pa, therefore the simulation results produced would have to be offset to provide a genuine comparison against the experimental data.

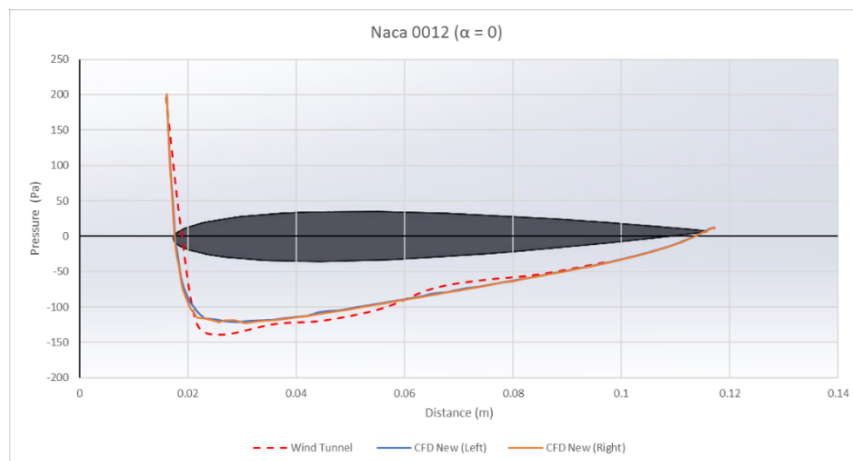


Figure 178 - Naca 0012 ($\alpha = 0$)

The symmetrical nature of the profile allows symmetry within the static pressures at $\alpha = 0$. Figure 178, illustrates the static pressures produced from the simulation and experimental data. Overall correlation appears adequate, however, a maximum error of 20.33Pa exists within the pressure drop at the leading edge (0.0261m). A deviation in static pressure exists along the chord length of the profile with a maximum error of 7.28Pa. It should also be noted that the pressure taps do not exist on the entire chord length of the profile, thus the presented results display a completed return to atmospheric pressure within the simulation results.

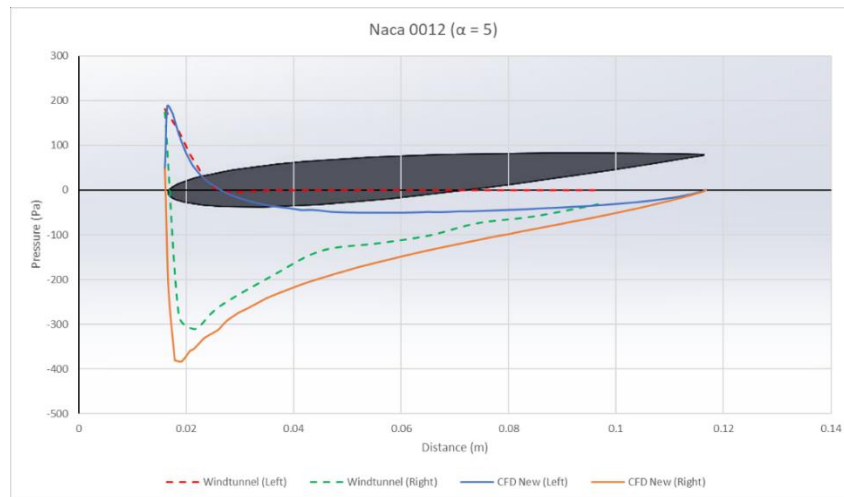


Figure 179 - Naca 0012 ($\alpha = 5$)

Figures 179, 180 and 181, indicate the subsequent results for the increasing AoA. An anomaly exists within $\alpha = 5$. It can be observed that the high pressure side (left) consistently dips below 0Pa, whereas the wind tunnel shows a consistent 0Pa on the same side. Trends are followed on the suction side, however the simulation has predicted a consistently lower pressure with a shaper transition of pressure around the leading edge. This could be as a result of mesh resolution within the near wall region, resulting in the turbulence model is incorrectly modelling the boundary layer, which has produced inaccurate surface pressures.

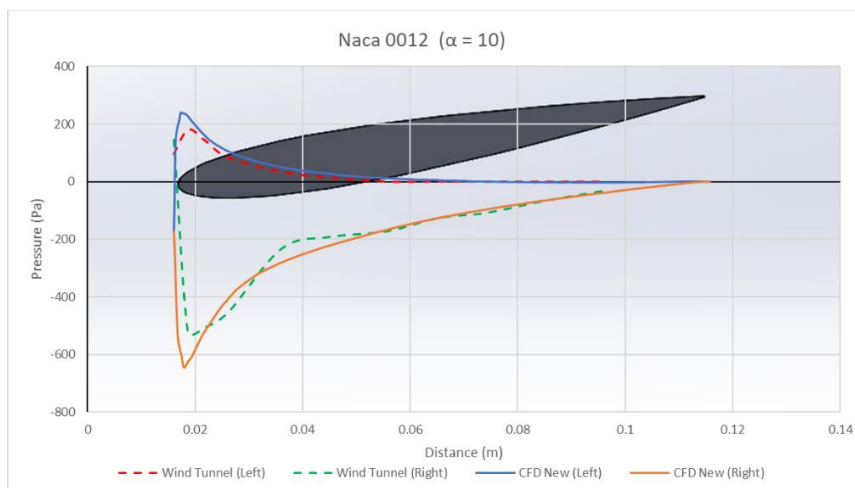


Figure 180 - Naca 0012 ($\alpha = 10$)

Improved correlation exists with $\alpha = 10$, although it must be noted that the turbulence model has predicted a much smoother transition to atmospheric pressure at 0.0361m on the suction side. This could, however, be as a result of the turbulence model failure to predict the beginnings of separation. Once more, the simulation shows a much greater peak of negative and positive pressures during the transition around the leading edge. This could be as a result of the limited number of available pressure taps along the surface of the Naca 0012 (11 pressure taps), whereas the simulation utilises the amount of available nodes within the near-wall region, which exceeds over 100 available “virtual” pressure taps, which is possibly the reason for the smoother transition of pressure.

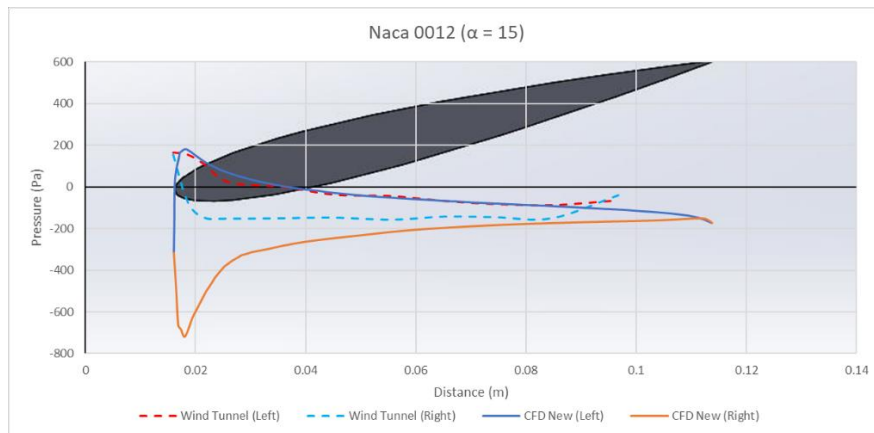


Figure 181 - Naca 0012 ($\alpha = 15$)

Figure 181, clearly illustrates boundary separation at the leading edge (0.0261m) when examining the wind tunnel data at $\alpha = 15$. The turbulence model has clearly under predicted the location of separation, which is clearly visible in Figure 181, as flow recirculation is present within the wake flow regime.



Figure 182 - Naca 0012 Wake Flow Circulation

It must be stated that the existing error within the CFD data reduces along the chord length distance, while the positive pressure sides match well, which again suggests the turbulence model struggles within strong adverse pressure gradients.

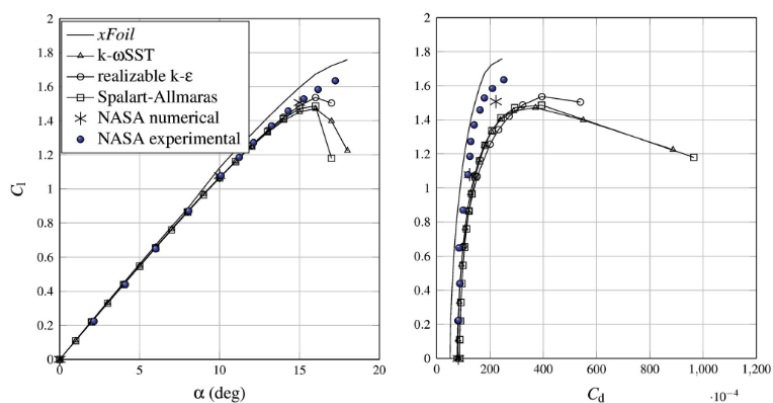


Figure 183 - Naca 0012 Performance at $Re = 6E+6$

Figure 183, illustrates data produced by Palmer et al, when conducting a correlation experiment. It can be observed that the $k\varepsilon$ Realizable model displays the closest correlation to the experimental data. In this case, however, it demonstrates that the model over predicts the occurrence of separation when studying CI, whereas the investigation for this study suggests under prediction of separation, although the relative Re values differ due to the greater aspect ratio used for the collected data displayed in Figure 183 (Palmer, et al., 2018). This would have an impact on the separation period, the near-wall velocity profile and the pressure gradient within the wake region.

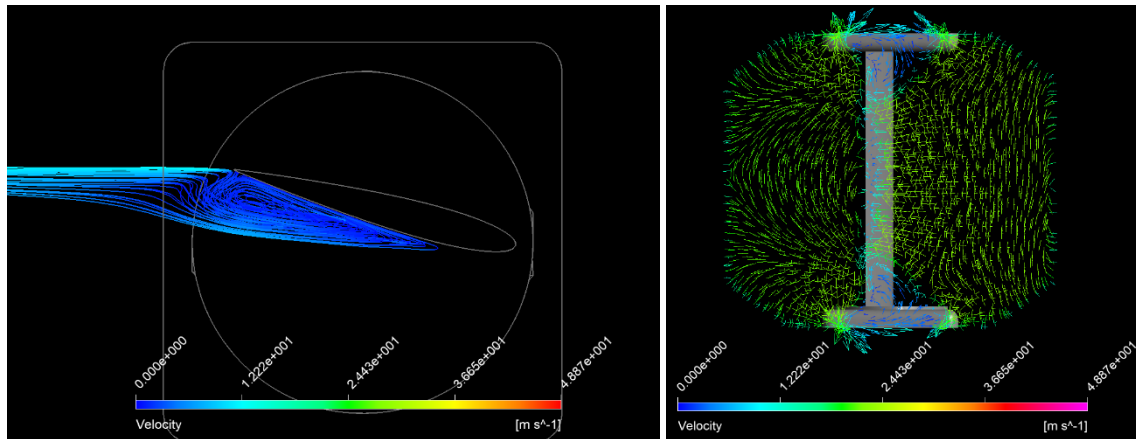


Figure 184 - Post Processing Wake Flow Regime

Figure 184, clearly illustrates that the near wake flow has been modelled and is comparable to that observed in Figure 182, with the clear presence of air recirculation. Two vortical structures have also developed at the top and bottom ends of the aerofoil which is as a result of the stagnation zone located at the leading edges of the mounting plates (Figure 185). The positive pressure gradient allows the high pressure to bleed into the suction side of the aerofoil, injecting it with kinetic energy and causing the swirling motion (Memon, 2017).

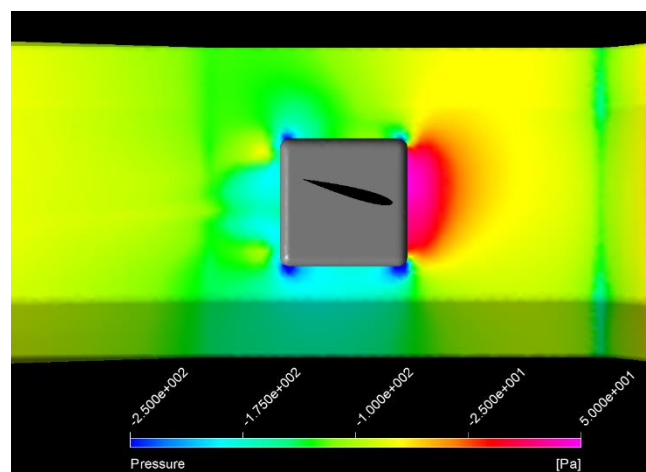


Figure 185 - Areas of Localised Pressure on the Wind Tunnel Sides

This issue is subsequently exacerbated at the trailing edge as the positive pressure side of the aerofoil allows air to bleed into the localised area of negative pressure at the trailing edge. This is caused by the mounting plate, which consequently feeds further kinetic energy back into the vortices.

However, this was not considered when collecting the experimental data, therefore the vortex structures were not monitored, for that reason it is unknown whether they had any impact upon the measured static pressures.

7.2.0 Horiba MIRA Reference Car

The MIRA reference vehicle geometry is often used as a basis for assessing the effect the rear geometry of the vehicle has on the rear wake structure and drag implications (Soares, 2015). Three basic reference models exist (notchback, fast back and hatch back), however, these have often been modified for the purposes of a particular application.

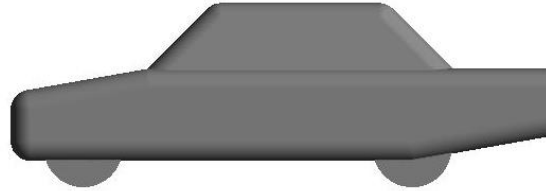


Figure 186 - MIRA Notchback Geometry

Ahmed first introduced the study of aerodynamic wake structure formation over simplified bluff bodies, such as the Ahmed body (Ahmed, 1981). However, the introduction of the MIRA reference car has allowed further experimentation to be carried out on an increasingly representative (though still simplified) geometry.

As further validation, measured static pressures produced from wind tunnel data were compared to the produced CFD results. The case domain was constructed using the same dimensions as the wind tunnel test area with the boundary sweep also being modelled, with a velocity of 35m/s and $Re = 9.95E6$.

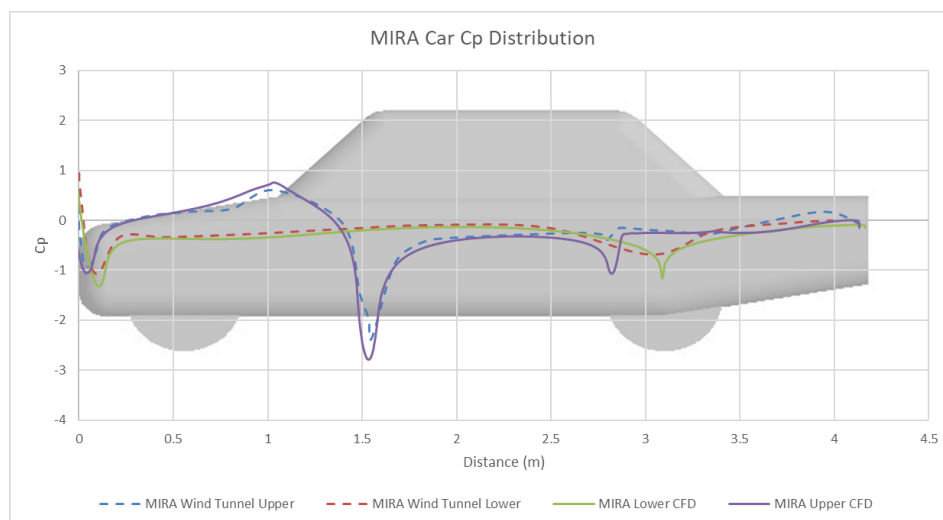


Figure 187 - MIRA Notchback Cp Distribution

Figure 187, illustrates the computational results as a comparison to the experimental data. The values of C_p have been captured on a central longitudinal plane.

It can be observed when examining the data, that the computational method has correlated well, although it does display harsher peaks of C_p within large pressure gradients. It can also be observed

that the computational data displays lower values of C_p aft of the rear slant when compared to the experimental data. This could be as a result of the refinement of the mesh build grid. As previously stated, an increased number of nodes within the build grid will provide further points for the measurement of surface pressure. The experimental data was measured using 30 pressure taps on the upper surface and 15 on the lower surface, therefore suggesting that the computational model is providing a much greater resolution in the results being presented, which produces the error variations.

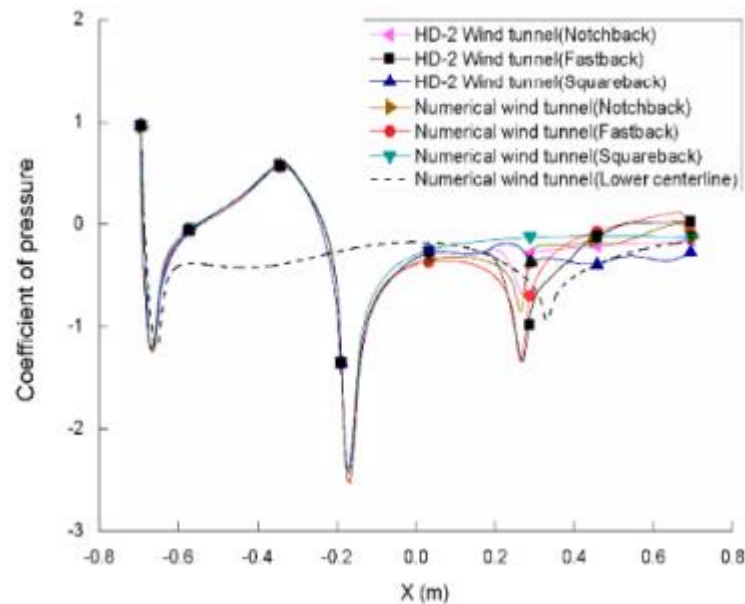


Figure 188 - MIRA Reference Car C_p Distribution (Wang, et al., 2014)

Wang et al, conducted an experiment using the MIRA reference vehicles to monitor the drag implications of varying rear geometries. Figure 188, illustrates the C_p results along the top surface of the varying geometries. It can be observed that the notchback C_p (HD-2 Wind tunnel Notchback) compares favourably with the results illustrated in Figure 187.

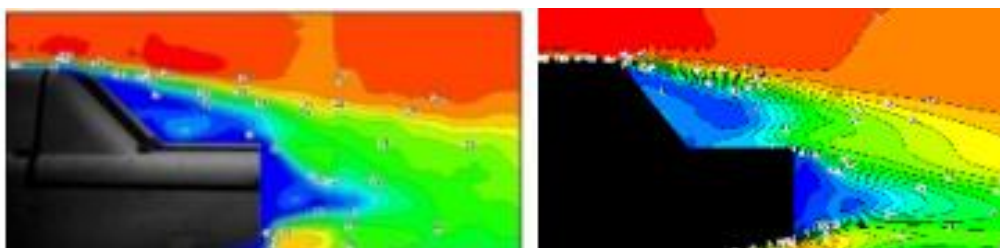


Figure 189 - MIRA Notchback Wake Velocity (Left - PIV) (Right - CFD) (Wang, et al., 2014)

Wang et al, went on to display the time averaged velocity along a central plane, illustrated in Figure 189. The turbulence model used was the $k\epsilon$ Realizable, however, it has not been established whether wall functions were applied. It can be observed that variations exist between the captured PIV (particle image velocimetry) and the CFD results, with the PIV displaying the top wake structure merging with the lower. Figure 190, illustrates the time averaged velocity along a central plane for the

experiment referenced in *Figure 187*. It highlights similar trends to that of the CFD results by Wang et al, however, the top wake structure is much more defined and similar in shape to that produced by the PIV in *Figure 189*, although it does still display two separate wake structures.

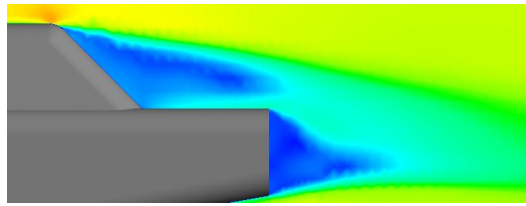


Figure 190 - MIRA Notchback Wake Velocity

However, results produced in a study conducted by Palaskar (*Figure 191*) show marked similarities with those illustrated in *Figure 190*. Palaskar used the $k\varepsilon$ Realizable, Non equilibrium wall functions model, thus correlation is present (Palaskar, 2016).

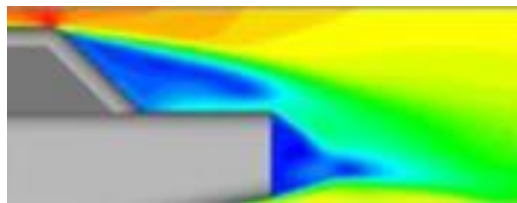


Figure 191 - MIRA Notchback Wake Velocity (Palaskar, 2016)

Fluid recirculation within the wake zone is also apparent (*Figure 192*). The post processing displays a large area of recirculation immediately behind the rear slant, and two small areas of recirculation immediately to the rear that possess opposing vectors of rotation due to the presented pressure gradients, which also correlates well with the findings from both Palaskar and Wang (*Figure 193*).

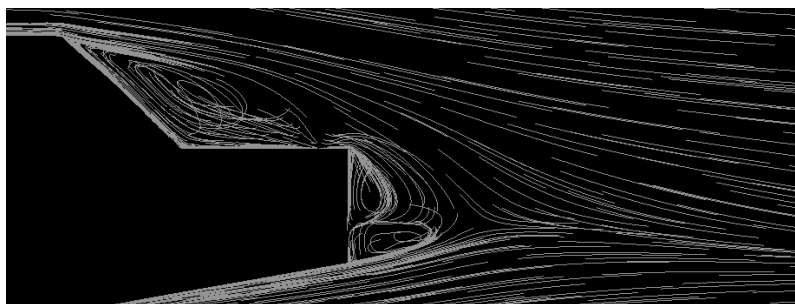


Figure 192 - MIRA Notchback Wake Structure Path Lines

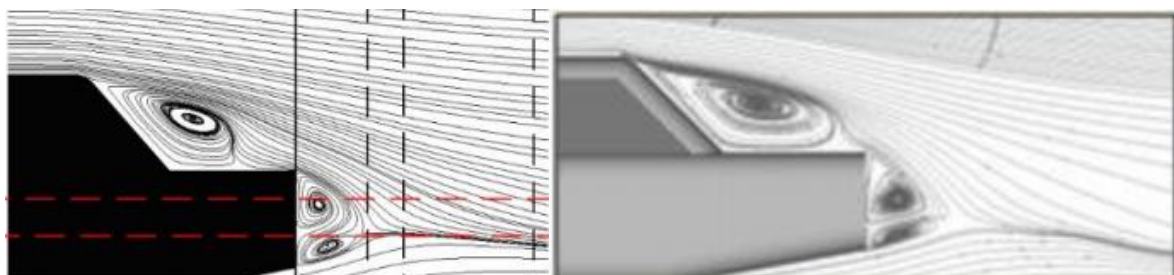


Figure 193 - MIRA Notchback Wake Structure Streamlines (Left - Wang et al) (Right - Palaskar)

7.3.0 Fackrells Wheel

Using the geometry dimensions outlined by Diasinos et al, Frackrells A2 wheel was modelled (Diasinos, et al., 2015). Fackrell and Harvey made time averaged pressure measurements using a Kiel tube (Zhang, et al., 2006). This allowed a case comparison to be conducted, to further validate and solidify the proposed computational method

The initial boundary conditions involved a freestream velocity of 18.6m/s. The ground plane was initialised with a no slip condition and a translational velocity the same as that of the freestream. This resulted in a wheel angular velocity of 89.42rad/s, which correlated well with the investigation conducted by Misar et al, (Misar, et al., 2020).

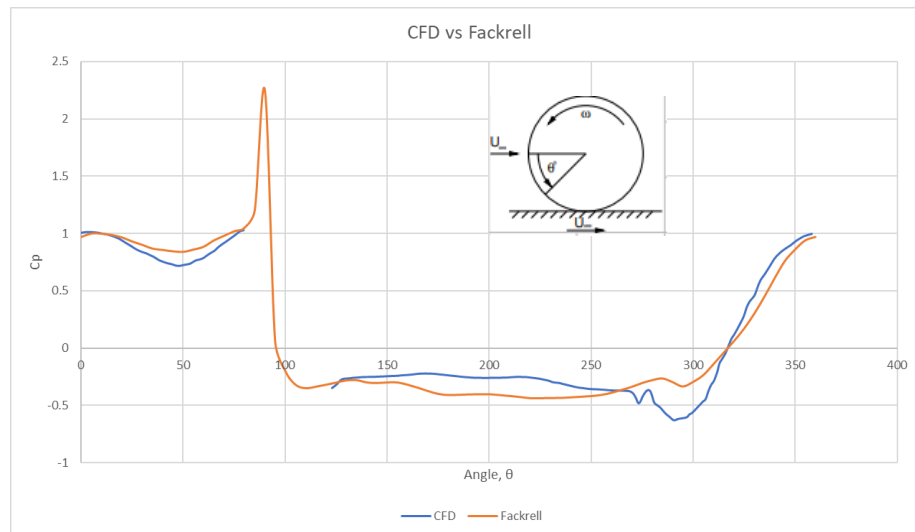


Figure 194 - Fackrell Wheel Pressure Distribution Measured on the Centreline

Figure 194, illustrates the plotted values for C_p around the curve length of the modelled wheel. The plot overlay indicates similar trends to Fackrell's findings. As discussed, with regard to Figure 19, the area of flow separation over the top surface shows the greatest margin of error, with the CFD methodology displaying a much greater pressure drop, however, the angle of separation for both cases correlates well. Investigations conducted by Leniewicz et al, also show similar trends, especially on the rear face of the wheel, with the computational method showing an under prediction in negative pressure (Leśniewicz, et al., 2014).

The use of the steady RANS model did however produce poor convergence within the residuals. This is due to the nature of the flow regime, causing large areas of recirculation, which is explained by Croner et al, (Croner, et al., 2012). However, as previously made reference to, the utilised $k\epsilon$ model struggles within high adverse pressure gradients.

Table 13 - C_l and C_d for Fackrells Wheel

	C_l	C_d
Fackrell	0.28	0.51
CFD	0.22	0.54
Difference (%)	21.4	5.6

Table 13, indicates the coefficients resulting from both Fackrells experimental data and the proposed CFD method. It bears noting that the error within the CI results is greatest, and potentially originates from the increased pressure drop observed at the separation point. However, C_d values show adequate correlation.

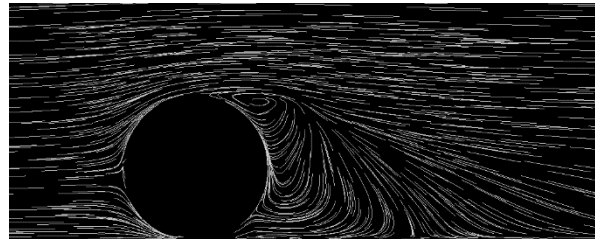


Figure 195 - Near-Wake Flow Recirculation

The area of recirculation with the near-wake region is represented in Figure 195. It can be observed that the rotation of wheel causes the flow to rotate back upon itself as a result of the applied shearing forces.

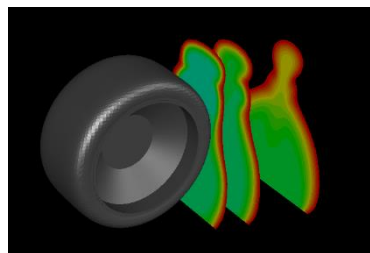


Figure 196 - Total Pressure Normal to Freestream

The horseshow vortices located at the contact pact of the wheel generate a larger area of negative pressure within the near-wake region at lower ground proximity (Issakhanian, et al., 2009). The total pressure contours displayed in Figure 196 also show further commonalities with the investigation conducted by Diasinos et al, which closely matched that of Fackrell and Harvey (Diasinos, et al., 2015).

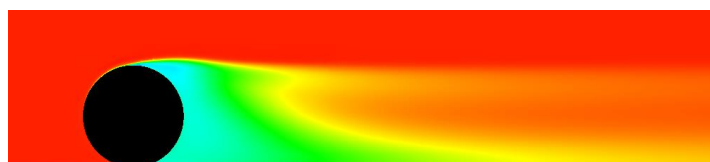


Figure 197 - Wheel Wake Total Pressure Perpendicular to the Freestream

Figure 197, further illustrates the total pressure located within the wheels wake region. The point of separation can also be observed with a very strong negative pressure bubble located within close proximity, promoting the large area of recirculation.

8.0.0 Appendix B

8.1.0 Breakdown of Study Results

Table 14 - Baseline Wing AoA

	Variable	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	CoP Distance Behind Coord (m)			Drag Cd	Lift Cl
Baseline AoA (Deg) (Height = 110mm)	15.42	107.83	573.18	58.43	5.32	32.72	187.55	385.63	0.331	0.67279	0.327211	0.19	-1.01
	15.82	107.68	578.05	58.92	5.37	32.72	189.14	388.90	0.331	0.67279	0.327211	0.19	-1.02
	16.16	107.72	583.94	59.52	5.42	32.72	191.07	392.87	0.331	0.67279	0.327211	0.19	-1.03
	16.52	113.36	595.16	60.67	5.25	32.89	195.77	399.40	0.33	0.67107	0.328933	0.20	-1.05
	17	113.28	600.36	61.20	5.30	32.89	197.48	402.88	0.33	0.67107	0.328933	0.20	-1.06
	17.4	113.25	605.87	61.76	5.35	32.89	199.29	406.58	0.33	0.67107	0.328933	0.20	-1.07
	17.7	113.14	610.97	62.28	5.40	32.89	200.97	410.00	0.33	0.67107	0.328933	0.20	-1.08
	18.18	118.77	616.47	62.84	5.19	32.89	202.78	413.69	0.33	0.67107	0.328933	0.21	-1.09
	18.58	118.72	627.52	63.97	5.29	32.98	206.95	420.57	0.3295	0.67021	0.329793	0.21	-1.11
	18.96	118.69	633.00	64.53	5.33	32.81	207.67	425.33	0.3305	0.67193	0.328072	0.21	-1.12
	19.32	124.22	638.06	65.04	5.14	33.07	210.98	427.08	0.329	0.66935	0.330654	0.22	-1.13
	19.8	124.19	643.53	65.60	5.18	33.07	212.78	430.74	0.329	0.66935	0.330654	0.22	-1.14
	20.06	124.21	649.27	66.18	5.23	33.07	214.68	434.59	0.329	0.66935	0.330654	0.22	-1.15
	20.42	129.74	654.34	66.70	5.04	33.07	216.36	437.98	0.329	0.66935	0.330654	0.23	-1.16

Table 15 - Baseline Wing Assembly AoA

	Variable	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	CoP Distance Behind Coord (m)			Drag Cd	Lift Cl
Whole Assembly AoA (Deg) (Height = 110mm) (End Plates Stay in Baseline Position)	0	107.83	573.18	58.43	5.32	32.72	187.55	385.63	0.331	0.67279	0.327211	0.19	-1.01
	1	118.45	648.66	66.12	5.48	34.61	224.53	424.13	0.32	0.65385	0.346145	0.21	-1.15
	2	123.95	709.91	72.37	5.73	36.51	259.17	450.73	0.309	0.63492	0.365079	0.22	-1.26
	3	129.38	770.65	78.56	5.96	38.32	295.28	475.38	0.2985	0.61685	0.383152	0.23	-1.37
	4	140.22	830.10	84.62	5.92	39.76	330.06	500.04	0.2901	0.60239	0.397611	0.25	-1.48
	5	140.25	863.97	88.07	6.16	41.16	355.57	508.40	0.282	0.58845	0.411553	0.25	-1.54
	6	150.98	950.61	96.90	6.30	42.27	401.86	548.75	0.2755	0.57726	0.422741	0.27	-1.70
	7	162.70	1004.22	102.37	6.17	43.22	434.03	570.19	0.27	0.56779	0.432208	0.29	-1.79
	8	166.30	1036.59	105.67	6.23	43.93	455.34	581.25	0.2659	0.56073	0.439265	0.30	-1.87
	9	177.12	1101.46	112.28	6.22	44.86	494.07	607.39	0.2605	0.55144	0.44856	0.32	-1.99
	10	181.97	1146.95	116.92	6.30	45.54	522.37	624.58	0.2565	0.54455	0.455445	0.33	-2.08

Table 16 - Baseline Wing Ground Proximity

	Variable	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	CoP Distance Behind Coord (m)			Drag Cd	Lift Cl
Main Plane Height from Ground (mm) (AoA = 15.42)(Baseline = 110mm)	300	96.48	448.33	45.70	4.65	23.60	105.80	342.53	0.384	0.76402	0.235985	0.17	-0.79
	275	96.48	454.01	46.28	4.71	24.63	111.83	342.18	0.378	0.75369	0.246312	0.17	-0.80
	250	96.48	465.36	47.44	4.82	25.84	120.23	345.13	0.371	0.74164	0.258361	0.17	-0.82
	225	96.48	471.03	48.02	4.88	26.87	126.56	344.47	0.365	0.73131	0.268689	0.17	-0.83
	200	96.48	476.71	48.59	4.94	27.90	133.01	343.70	0.359	0.72098	0.279016	0.17	-0.84
	190	96.48	482.38	49.17	5.00	28.07	135.42	346.96	0.358	0.71926	0.280737	0.17	-0.85
	180	96.48	488.06	49.75	5.06	28.59	139.54	348.52	0.355	0.7141	0.285901	0.17	-0.86
	170	96.48	493.73	50.33	5.12	28.93	142.86	350.87	0.353	0.71066	0.289344	0.17	-0.87
	160	102.15	499.41	50.91	4.89	29.45	147.08	352.33	0.35	0.70549	0.294507	0.18	-0.88
	150	102.15	522.11	53.22	5.11	30.57	159.61	362.50	0.3435	0.6943	0.305696	0.18	-0.92
	140	102.15	527.78	53.80	5.17	31.00	163.61	364.17	0.341	0.69	0.309999	0.18	-0.93
	130	102.15	533.46	54.38	5.22	31.24	166.66	366.80	0.3396	0.68759	0.312409	0.18	-0.94
	120	102.15	556.16	56.69	5.44	32.20	179.11	377.05	0.334	0.67795	0.322048	0.18	-0.98
	110	107.83	573.18	58.43	5.32	32.72	187.55	385.63	0.331	0.67279	0.327211	0.19	-1.01
	100	107.83	590.21	60.16	5.47	33.15	195.66	394.55	0.3285	0.66849	0.331515	0.19	-1.04
	90	107.83	618.59	63.06	5.74	33.93	209.86	408.72	0.324	0.66074	0.33926	0.19	-1.09
	80	113.50	652.64	66.53	5.75	34.53	225.35	427.29	0.3205	0.65472	0.345285	0.20	-1.15
	70	113.50	681.01	69.42	6.00	34.96	238.07	442.94	0.318	0.65041	0.349588	0.20	-1.20
	60	113.50	737.76	75.21	6.50	36.51	269.34	468.42	0.309	0.63492	0.365079	0.20	-1.30
	50	113.50	777.49	79.25	6.85	38.14	296.56	480.93	0.2995	0.61857	0.381431	0.20	-1.37
	40	107.83	800.19	81.57	7.42	39.95	319.68	480.51	0.289	0.6005	0.399504	0.19	-1.41
	30	107.83	868.29	88.51	8.05	42.70	370.80	497.49	0.273	0.57296	0.427044	0.19	-1.53

Table 17 - Baseline Wing Secondary Elements Height

	Variable	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	CoP Distance Behind Coord (m)			Drag Cd	Lift Cl
Secondary Elements Height Above Main Plane (mm)(Baseline = -4.39mm)	0	107.83	573.18	58.43	5.32	32.55	186.57	386.62	0.332	0.67451	0.32549	0.19	-1.01
	5	107.83	573.18	58.43	5.32	32.38	185.58	387.61	0.333	0.67623	0.323769	0.19	-1.01
	10	107.83	573.18	58.43	5.32	32.03	183.61	389.58	0.335	0.67967	0.320326	0.19	-1.01
	15	107.83	567.51	57.85	5.26	31.86	180.81	386.70	0.336	0.68139	0.318605	0.19	-1.00
	20	107.83	567.51	57.85	5.26	31.34	177.88	389.63	0.339	0.68656	0.313441	0.19	-1.00
	25	107.83	567.51	57.85	5.26	31.00	175.93	391.58	0.341	0.69	0.309999	0.19	-1.00
	30	107.83	567.51	57.85	5.26	30.66	173.97	393.54	0.343	0.69344	0.306556	0.19	-1.00
	35	102.15	556.16	56.69	5.44	30.31	168.58	387.58	0.345	0.69689	0.303114	0.18	-0.98
	40	102.15	556.16	56.69	5.44	29.97	166.67	389.49	0.347	0.70033	0.299671	0.18	-0.98
	45	102.15	550.48	56.11	5.39	29.45	162.12	388.36	0.35	0.70549	0.294507	0.18	-0.97
	50	102.15	544.81	55.54	5.33	29.11	158.57	386.23	0.352	0.70894	0.291065	0.18	-0.96
	55	102.15	539.13	54.96	5.28	28.76	155.07	384.07	0.354	0.71238	0.287622	0.18	-0.95
	60	102.15	533.46	54.38	5.22	28.42	151.60	381.86	0.356	0.71582	0.28418	0.18	-0.94
	65	102.15	527.78	53.80	5.17	28.07	148.17	379.62	0.358	0.71926	0.280737	0.18	-0.93
	70	102.15	527.78	53.80	5.17	27.73	146.35	381.43	0.36	0.72271	0.277295	0.18	-0.93

Table 18 - Baseline Wing Secondary Elements Forward Position

	Variable	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	CoP Distance Behind Coord (m)			Drag Cd	Lift Cl
Secondary Elements Forward Positioning (mm) (Changing Plannar Area!)	0	106.36	576.57	58.77	5.42	34.27	197.59	378.98	0.322	0.6573	0.342703	0.19	-1.03
	5	105.84	590.45	60.19	5.58	34.27	202.35	388.10	0.322	0.6573	0.342703	0.19	-1.06
	10	105.25	592.73	60.42	5.63	35.48	210.27	382.46	0.315	0.64525	0.354752	0.19	-1.07
	15	104.71	595.20	60.67	5.68	35.99	214.22	380.98	0.312	0.64008	0.359915	0.19	-1.08
	20	104.19	597.70	60.93	5.74	36.34	217.18	380.52	0.31	0.63664	0.363358	0.19	-1.09
	25	109.03	599.68	61.13	5.50	36.85	221.00	378.69	0.307	0.63148	0.368522	0.20	-1.10
	30	108.44	601.85	61.35	5.55	37.20	223.87	377.98	0.305	0.62804	0.371964	0.20	-1.11
	35	107.86	598.61	61.02	5.55	37.37	223.69	374.92	0.304	0.62631	0.373685	0.20	-1.11
	40	107.26	595.32	60.69	5.55	37.71	224.51	370.81	0.302	0.62287	0.377128	0.20	-1.11
	45	106.66	591.98	60.34	5.55	37.88	224.27	367.71	0.301	0.62115	0.378849	0.20	-1.11
	50	106.10	594.18	60.57	5.60	38.23	227.15	367.03	0.299	0.61771	0.382292	0.20	-1.12

Table 19 - Baseline Wing Secondary Elements Aft Position

	Variable	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	CoP Distance Behind Coord (m)			Drag Cd	Lift Cl
Secondary Elements Aft Positioning (mm) (Changing Plannar Area!)	0	106.36	576.57	58.77	5.42	34.27	197.59	378.98	0.322	0.6573	0.342703	0.19	-1.03
	5	107.06	580.37	59.16	5.42	33.75	195.90	384.47	0.325	0.66246	0.337539	0.19	-1.03
	10	107.61	572.02	58.31	5.32	32.89	188.16	383.87	0.33	0.67107	0.328933	0.19	-1.01
	15	108.20	569.48	58.05	5.26	32.03	182.42	387.06	0.335	0.67967	0.320326	0.19	-1.00
	20	103.01	555.10	56.59	5.39	31.17	173.04	382.06	0.34	0.68828	0.31172	0.18	-0.97
	25	103.33	545.33	55.59	5.28	30.31	165.30	380.04	0.345	0.69689	0.303114	0.18	-0.95
	30	103.33	539.63	55.01	5.22	29.28	158.00	381.64	0.351	0.70721	0.292786	0.18	-0.94
	35	103.34	539.66	55.01	5.22	28.42	153.36	386.30	0.356	0.71582	0.28418	0.18	-0.94
	40	103.34	528.21	53.84	5.11	27.56	145.56	382.65	0.361	0.72443	0.275574	0.18	-0.92
	45	103.35	522.48	53.26	5.06	26.70	139.49	382.99	0.366	0.73303	0.266967	0.18	-0.91
	50	103.34	510.97	52.09	4.94	25.84	132.01	378.95	0.371	0.74164	0.258361	0.18	-0.89

Table 20 - Main Plane Design Study (110mm)

	Variable	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	CoP Distance Behind Coord (m)			Drag Cd	Lift Cl
Main Plane Study (No Flaps!) (Baseline Height = 110mm)	Dallara	27.63	151.97	15.49	5.50	38.06	57.83	94.13	0.3	0.61943	0.38057	0.06	-0.33
	4412	27.64	133.60	13.62	4.83	39.09	52.23	81.38	0.294	0.6091	0.390898	0.06	-0.29
	51223	41.65	453.47	46.23	10.89	48.38	219.41	234.06	0.24	0.51615	0.483846	0.09	-0.98
	6409	27.77	236.04	24.06	8.50	45.46	107.30	128.74	0.257	0.54542	0.454585	0.06	-0.51

Table 21 - Main Plane Design Study (40mm)

	Variable	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	CoP Distance Behind Coord (m)			Drag Cd	Lift Cl
Main Plane Study (No Flaps!) (Height = 40mm)	Dallara	32.23	294.68	30.04	9.14	45.63	134.46	160.22	0.256	0.54369	0.456306	0.07	-0.64
	4412	32.17	271.18	27.64	8.43	47.18	127.94	143.24	0.247	0.5282	0.471797	0.07	-0.59
	51223	50.90	666.29	67.92	13.09	50.79	338.44	327.85	0.226	0.49206	0.507944	0.11	-1.44
	6409	32.39	421.11	42.93	13.00	49.59	208.83	212.28	0.233	0.50411	0.495895	0.07	-0.91

Table 22 - SBD Dallara F399 Baseline Study

	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	CoP Distance Behind Coord (m)			Drag Cd	Lift Cl
SBD Formula 3 Dallara 2.0 XE												
BASLINE (Full Car)	811	2017	206	2.49	52.79	1065	962	1.263	0.47215	0.52785	0.74	-1.84
BASLINE (Half Car) Neutral Whole Wing AoA	778	1644	168	2.11	26.24	431	1213	1.973	0.73757	0.26243	0.71	-1.5
BASLINE (Half Car) Whole Wing AoA 2.3°	844	2083	212	2.47	52.30	1089	993	1.276	0.47701	0.522991	0.77	-1.9
BASLINE (Half Car) (NO FLAPS)	789	833	85	1.06	-21.94	-183	1016	3.262	1.21944	-0.21944	0.72	-0.76

Table 23 - SBD Dallara F399 Main Plane Design Process

	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	CoP Distance Behind Coord (m)			Drag Cd	Lift Cl
SBD Formula 3 Dallara 2.0 XE												
Main Plane 1_60mm_In Wash End Plates (Half Car)	811	888	91	1.09	-8.22	-73	961	2.895	1.08224	-0.08224	0.74	-0.81
Main Plane 2_60mm_In Wash End Plates_Hybrid6409_51223 (Half Car)	778	1151	117	1.48	8.97	103	1048	2.435	0.91028	0.08972	0.71	-1.05
Main Plane 3_60mm_In Wash End Plates_Hybrid6409_51223 (Half Car)	789	976	99	1.24	2.47	24	951	2.609	0.97533	0.024673	0.72	-0.89

Table 24 - SBD Dallara F399 Secondary Elements Design Process

	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	CoP Distance Behind Coord (m)			Drag Cd	Lift Cl
SBD Formula 3 Dallara 2.0 XE												
Main Plane 2_60mm_In Wash End Plates_Baseline Flaps (15.42°) (Half Car)	789	1524	155	1.93	31.81	485	1039	1.824	0.68187	0.318131	0.72	-1.39
Main Plane 2_60mm_In Wash End Plates_Flaps 2 with 70mm Extension (24°) (Half Car)	778	1852	189	2.38	45.23	638	1015	1.485	0.54766	0.452336	0.71	-1.69
Main Plane 2_60mm_In Wash End Plates_Flaps 3 with 70mm Extension (30°) (Half Car)	800	1863	190	2.33	53.79	1002	861	1.236	0.46206	0.537944	0.73	-1.7
Main Plane 2_60mm_In Wash End Plates_Flaps 4 with 90mm Extension (30° In, 24° Out) (Half Car)	799	1883	192	2.39	48.34	911	974	1.382	0.51664	0.483364	0.72	-1.72
Main Plane 2_60mm_In Wash End Plates_Flaps 5 with 90mm Extension (24° In, 30° Out) (Half Car)	778	1710	174	2.20	51.21	876	894	1.305	0.48785	0.51215	0.71	-1.56

Table 25 - SBD Dallara F399 End Plate Design Process

	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	CoP Distance Behind Coord (m)			Drag Cd	Lift Cl
SBD Formula 3 Dallara 2.0 XE												
Out Plates 1_Flaps 4reduced 40mm_(32" in, 26" Out) (Half Car)	767	1820	185	2.37	45.08	820	999	1.469	0.54916	0.450841	0.7	-1.66
Out Plates 2_Flaps 4reduced 40mm_(32" in, 26" Out) Mid Span Fence (Half Car)	800	1754	179	2.19	44.86	787	967	1.475	0.5514	0.448586	0.73	-1.6
Out Plates 3_Flaps 4reduced 40mm_(32" in, 26" Out) Mid Span Fence (Half Car)	800	1480	151	1.85	47.44	702	778	1.406	0.52561	0.474383	0.73	-1.35

Table 26 - SBD Dallara F399 Front Wing Concept Design Process

	DRAG (N)	DOWNFORCE (N)	DOWNFORCE (KG)	EFFICIENCY	BALANCE (% Front)	FRONT DOWNFORCE (N)	REAR DOWNFORCE (N)	CoP Distance Behind Coord (m)			Drag Cd	Lift Cl
SBD Formula 3 Dallara 2.0 XE												
Concept 1	811	1688	172	2.08	56.86	960	728	1.154	0.4314	0.568598	0.74	-1.54
Concept 2	800	1929	197	2.41	49.53	956	974	1.35	0.50467	0.495327	0.73	-1.76
Concept 3	822	1546	158	1.88	54.80	847	699	1.209	0.45196	0.548037	0.75	-1.41
Concept 4	800	1831	187	2.29	43.29	792	1038	1.517	0.5671	0.432897	0.73	-1.67
Concept 5	822	1907	194	2.32	45.53	868	1039	1.457	0.54467	0.455327	0.75	-1.74
Concept 6	833	1907	194	2.29	48.37	923	885	1.381	0.51526	0.483738	0.76	-1.74
Concept 7	822	1852	189	2.25	45.23	839	1015	1.465	0.54766	0.452334	0.75	-1.69
Concept 8	822	1831	187	2.23	44.36	819	1020	1.491	0.55728	0.442627	0.75	-1.67
Concept 9	822	1885	192	2.29	42.02	792	1093	1.551	0.57981	0.420187	0.75	-1.72
Concept 10	844	1874	191	2.22	40.97	768	1106	1.579	0.59028	0.40972	0.77	-1.71

9.0.0 Appendix C

9.1.0 Concept Design Geometries

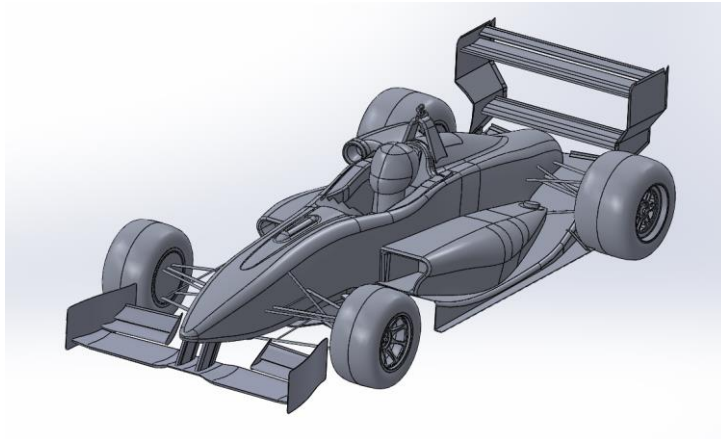


Figure 198 - Concept 1

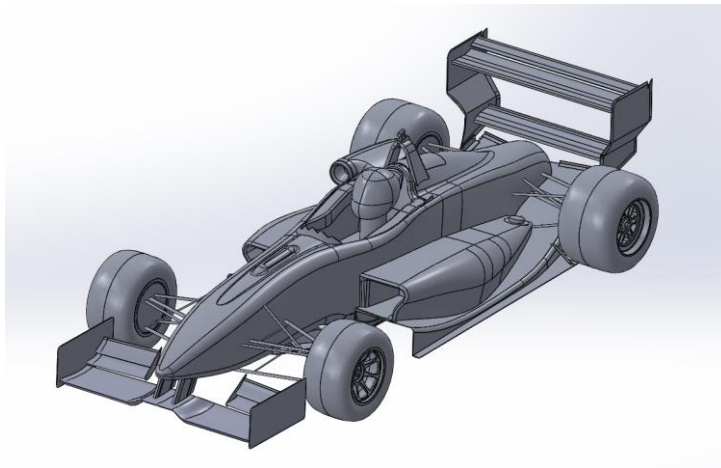


Figure 199 - Concept 2

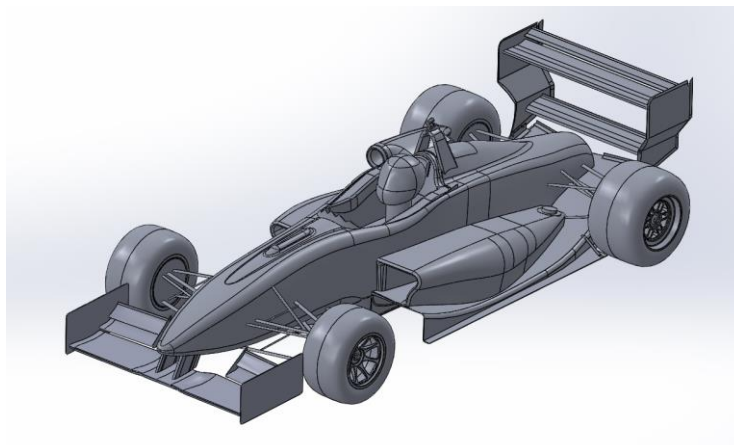


Figure 200 - Concept 3

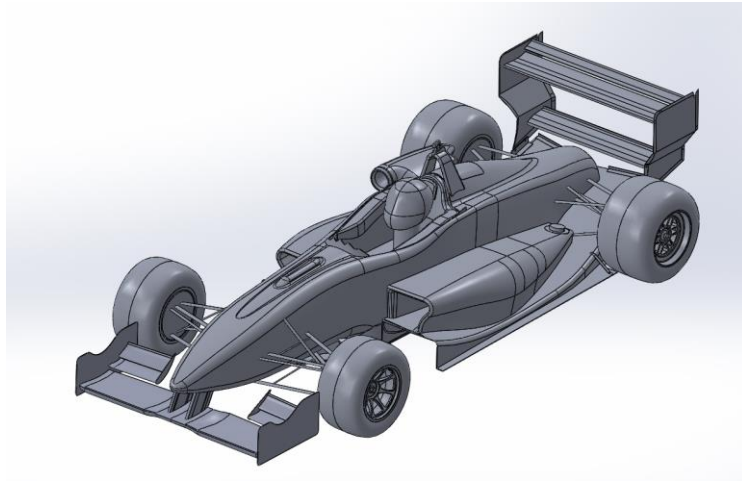


Figure 201 - Concept 4

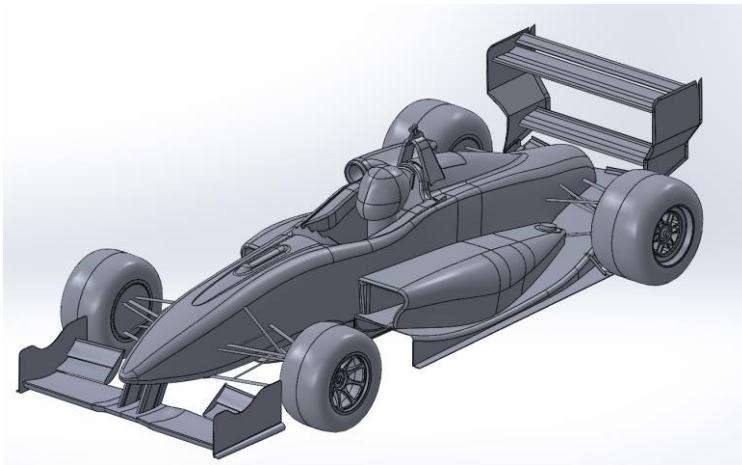


Figure 202 - Concept 5

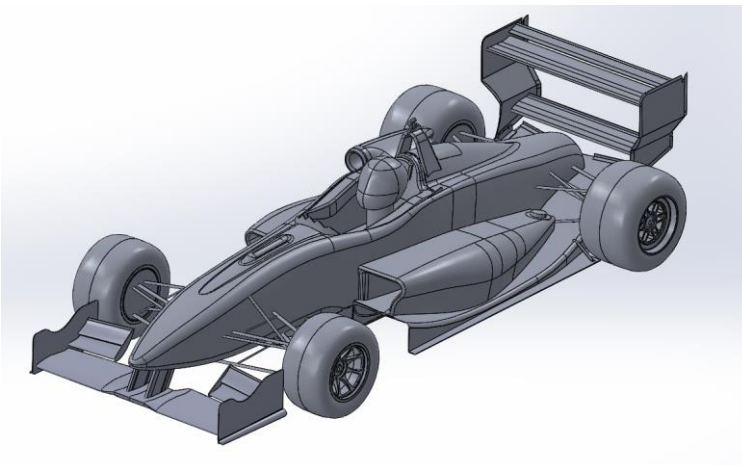


Figure 203 - Concept 6

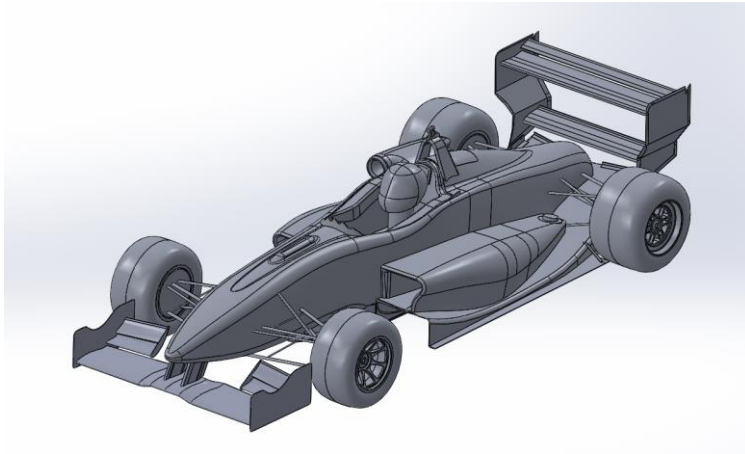


Figure 204 - Concept 7

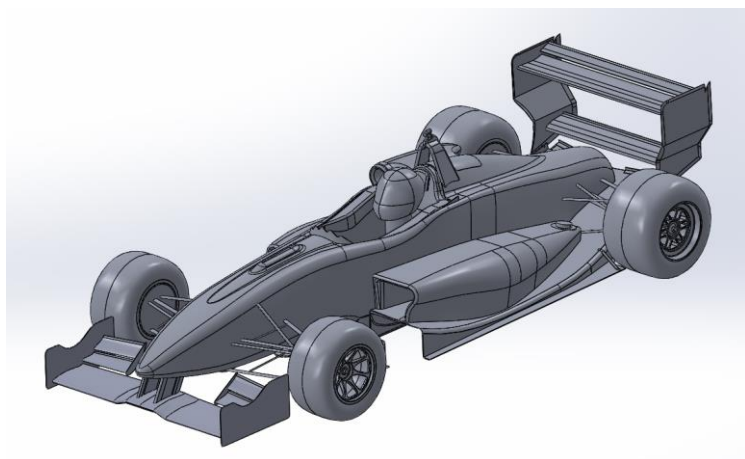


Figure 205 - Concept 8

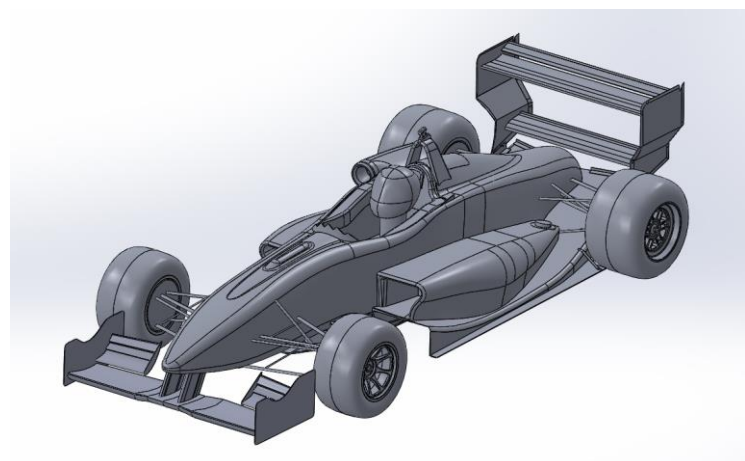


Figure 206 - Concept 9

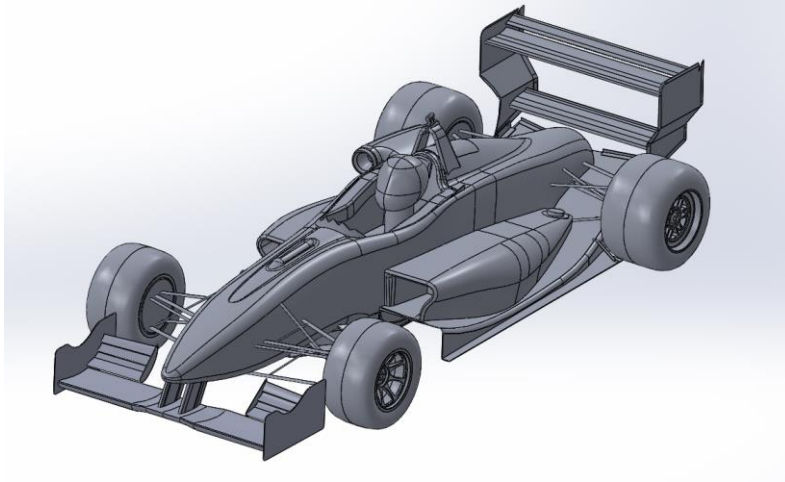


Figure 207 - Concept 10